SIMULIA

3DEXPERIENCE

ABAQUS 2025 FD04
KEYWORDS GUIDE

DASSAULT SYSTEMES

Contents

Trademarks and Legal Notices....16
Abaqus Keywords Guide....17
What's New....18
A....36
*ACOUSTIC CONTRIBUTION....37
*ACOUSTIC FLOW VELOCITY....39
*ACOUSTIC MEDIUM....41
*ACOUSTIC WAVE FORMULATION....45
*ACTIVATE ELEMENTS....46
*ADAPTIVE MESH....47
*ADAPTIVE MESH CONSTRAINT....49
*ADAPTIVE MESH CONTROLS....51
*ADAPTIVE MESH REFINEMENT....54
*ADJUST....56
Adjusting nodes so that the nodes lie on a given surface....56
Adjusting nodes using a distribution of adjustments....56
*ALLOWABLE STRESS....58
*AMPLITUDE....60
*ANISOTROPIC HYPERELASTIC....66
*ANNEAL....72
*ANNEAL TEMPERATURE....73
*AQUA....74
*ASSEMBLY....75
*ASYMMETRIC-AXISYMMETRIC....76
*AXIAL....77
B....79
*BASELINE CORRECTION....80
*BASE MOTION....81
*BEAM ADDED INERTIA....83
*BEAM FLUID INERTIA....85
*BEAM GENERAL SECTION....86
*BEAM SECTION....94
*BEAM SECTION GENERATE....98
*BEAM SECTION OFFSET....99
*BIAXIAL TEST DATA....100
Using biaxial test data to define a hyperelastic material....100
Using biaxial test data to define an elastomeric foam....101
Using biaxial test data to define the Mullins effect material model....102
*BLOCKAGE....103
*BOUNDARY....104
Prescribing boundary conditions at nodes....104
Defining primary and secondary bases for modal superposition procedures....109
Submodel boundary conditions....110

*BRITTLE CRACKING....112
*BRITTLE FAILURE....114
*BRITTLE SHEAR....116
*BUCKLE....118
*BUCKLING ENVELOPE....119
*BUCKLING LENGTH....120
*BUCKLING REDUCTION FACTORS....121
*BULK VISCOSITY....123
C....124
*C ADDED MASS....127
*CAPACITY....128
*CAP CREEP....130
*CAP HARDENING....133
*CAP PLASTICITY....134
*CAST IRON COMPRESSION HARDENING....136
*CAST IRON PLASTICITY....137
*CAST IRON TENSION HARDENING....138
*CAVITY DEFINITION....139
*CECHARGE....141
*CECURRENT....142
*CENTROID....143
*CFILM....144
*CFLOW....147
*CFLUX....149
*CHANGE FRICTION....152
*CHARACTERISTIC LENGTH....153
*CLAY HARDENING....154
*CLAY PLASTICITY....156
*CLEARANCE....158
*CLOAD....162
Applying concentrated loads....162
Applying Abaqus/Aqua loads....164
*CLUSTER MASS INERTIA TABLE....167
*COHESIVE BEHAVIOR....168
*COHESIVE SECTION....170
*COMBINATORIAL RULE....172
*COMBINED TEST DATA....173
*COMPLEX FREQUENCY....175
*COMPOSITE MODAL DAMPING....177
*CONCENTRATION TENSOR....178
*CONCRETE....181
*CONCRETE COMPRESSION DAMAGE....183
*CONCRETE COMPRESSION HARDENING....185
*CONCRETE DAMAGED PLASTICITY....187
*CONCRETE FAILURE....189
*CONCRETE TENSION DAMAGE....191
*CONCRETE TENSION STIFFENING....193
*CONDUCTIVITY....196

*CONNECTOR BEHAVIOR....199
*CONNECTOR CONSTITUTIVE REFERENCE....201
*CONNECTOR DAMAGE EVOLUTION....202
*CONNECTOR DAMAGE INITIATION....206
*CONNECTOR DAMPING....209
*CONNECTOR DERIVED COMPONENT....218
*CONNECTOR ELASTICITY....221
*CONNECTOR FAILURE....229
*CONNECTOR FRICTION....230
*CONNECTOR HARDENING....233
*CONNECTOR LOAD....237
*CONNECTOR LOCK....239
*CONNECTOR MOTION....241
*CONNECTOR PLASTICITY....244
*CONNECTOR POTENTIAL....245
*CONNECTOR SECTION....247
*CONNECTOR STOP....249
*CONNECTOR UNIAXIAL BEHAVIOR....250
*CONSTITUENT....251
*CONSTRAINT CONTROLS....254
*CONTACT....256
*CONTACT CLEARANCE....257
*CONTACT CLEARANCE ASSIGNMENT....259
*CONTACT CONTROLS....260
Specifying additional controls for contact in an Abaqus/Standard analysis....260
Specifying additional controls for contact in an Abaqus/Explicit analysis....262
*CONTACT CONTROLS ASSIGNMENT....264
*CONTACT DAMPING....267
*CONTACT EXCLUSIONS....269
*CONTACT FILE....270
*CONTACT FORMULATION....271
*CONTACT INCLUSIONS....274
*CONTACT INITIALIZATION ASSIGNMENT....275
*CONTACT INITIALIZATION DATA....276
*CONTACT INTERFERENCE....278
*CONTACT MASS SCALING....280
*CONTACT OUTPUT....281
*CONTACT PAIR....284
Defining contacting surfaces in an Abaqus/Standard analysis....284
Defining contacting surfaces in an Abaqus/Explicit analysis....287
*CONTACT PERMEABILITY....289
*CONTACT PRINT....291
*CONTACT PROPERTY ASSIGNMENT....292
*CONTACT RESPONSE....293
*CONTACT STABILIZATION....294
*CONTOUR INTEGRAL....296
*CONTROLS....300
*CONWEP CHARGE PROPERTY....307

*CORRELATION....308
*CO-SIMULATION....310
*CO-SIMULATION REGION....311
Defining a co-simulation region for *CO-SIMULATION, PROGRAM=MULTIPHYSICS....311
Defining a co-simulation region for *CO-SIMULATION, PROGRAM=ABAQUS....312
*COUPLED CONSTITUTIVE RESPONSE....313
*COUPLED TEMPERATURE-DISPLACEMENT....314
*COUPLED THERMAL-ELECTRICAL....318
*COUPLED THERMAL-ELECTROCHEMICAL....320
*COUPLING....321
*CRADIATE....322
*CREEP....324
*CREEP STRAIN RATE CONTROL....328
*CRUSHABLE FOAM....330
*CRUSHABLE FOAM HARDENING....332
*CRUSH STRESS....333
*CRUSH STRESS VELOCITY FACTOR....335
*CURE GLASS TRANSITION TEMPERATURE....336
*CURE HEAT GENERATION....338
*CURE KINETICS....339
*CURE MAX CONVERSION....342
*CURE SHRINKAGE....343
*CYCLED PLASTIC....345
*CYCLIC....346
*CYCLIC HARDENING....348
*CYCLIC SYMMETRY MODEL....350
D....351
*D ADDED MASS....353
*DAMAGE EVOLUTION....354
*DAMAGE INITIATION....367
Defining damage initiation as a material property for continuum damage modeling in ductile materials.367
Defining damage initiation as a material property for modeling instability in sheet metals....370
Defining damage initiation as a material property for continuum damage modeling in fiber-reinforced composites.373
Defining damage initiation as a material property for continuum damage modeling in fiber-reinforced composites using the multiscale material model in Abaqus/Standard.375
Defining general stress/strain-based damage initiation as a material property for continuum damage modeling in anisotropic materials.377
Defining damage initiation as a material property for ductile materials in low-cycle fatigue....383
Defining damage initiation as a traction-separation–based material property for cohesive elements....384
Defining damage initiation as a material property for enriched elements in Abaqus/Standard....386
Defining damage initiation as part of a contact property model 391
*DAMAGE STABILIZATION....394
*DAMPING....396
*DAMPING CONTROLS....400
*DASHPOT....402
*DEBOND....405
*DECHARGE....407
*DECURRENT....408
*DEFORMATION PLASTICITY....410
*D EM POTENTIAL....411

*DENSITY....413
*DEPVAR....415
*DESIGN GRADIENT....417
*DESIGN PARAMETER....418
*DESIGN RESPONSE....419
Specifying design responses for a direct design sensitivity analysis....419
Specifying design responses for an adjoint design sensitivity analysis....420
*DETONATION POINT....421
*DFLOW....422
*DFLUX....424
*DIAGNOSTICS....426
Defining diagnostics in an Abaqus/Explicit analysis....426
Defining diagnostics in an Abaqus/Standard analysis....428
*DIELECTRIC....429
*DIFFUSIVITY....431
Defining mass diffusivity of a material diffusing through a base material in a mass-diffusion analysis....431
Defining the diffusivity of particles in a slurry flow during a coupled pore fluid flow and stress analysis.433
*DIRECT CYCLIC....434
*DISCRETE ELASTICITY....437
*DISCRETE SECTION....438
*DISPLAY BODY....439
*DISTRIBUTING....440
*DISTRIBUTING COUPLING....442
*DISTRIBUTION....443
*DISTRIBUTION TABLE....451
*DLOAD....455
Applying distributed loads....455
Loads used by Abaqus/Aqua....461
*DOMAIN DECOMPOSITION....465
*DRAG CHAIN....467
*DRUCKER PRAGER....468
*DRUCKER PRAGER CREEP....471
*DRUCKER PRAGER HARDENING....474
*DSA CONTROLS....476
*DSECHARGE....477
*DSECURRENT....478
*DSFLOW....480
*DSFLUX....481
*DSLOAD....482
Applying distributed loads....482
Applying submodel boundary conditions (Abaqus/Standard only)....486
*DYNAMIC....488
Defining a dynamic analysis in Abaqus/Standard....488
Defining a dynamic analysis in Abaqus/Explicit....492
*DYNAMIC TEMPERATURE-DISPLACEMENT....494
E....496
*EIGENSTRAIN....498
*ELASTIC....499

*ELCOPY....506
*ELECTRICAL CONDUCTIVITY....507
*ELECTRICAL RESISTIVITY....511
*ELECTRIC MACHINE LOAD....513
*ELECTRIC MACHINE PROPERTY....515
*ELECTROMAGNETIC....516
*ELEMENT....518
*ELEMENT MATRIX OUTPUT....520
*ELEMENT OPERATOR OUTPUT....522
*ELEMENT OUTPUT....523
*ELEMENT PROGRESSIVE ACTIVATION....526
*ELEMENT RECOVERY MATRIX....528
*ELEMENT RESPONSE....529
*ELEMENT SOLUTION-DEPENDENT VARIABLES....532
*ELEMENT USER OUTPUT VARIABLES....533
*EL FILE....535
*ELGEN....538
*EL PRINT....539
*ELSET....541
*EMBEDDED ELEMENT....543
*EMISSIVITY....545
*END ASSEMBLY....546
*END INSTANCE....547
*END LOAD CASE....548
*END PART....549
*END STEP....550
*ENERGY FILE....551
*ENERGY OUTPUT....552
*ENERGY PRINT....554
*ENRICHMENT....555
*ENRICHMENT ACTIVATION....556
*EOS....557
*EOS COMPACTION....560
*EPJOINT....561
*EQUATION....562
*EQUIVALENT RADIATED SURFACE PROPERTIES....563
*EULERIAN BOUNDARY....564
*EULERIAN MESH MOTION....565
*EULERIAN SECTION....567
*EVENT SERIES....569
*EVENT SERIES TYPE....571
*EXPANSION....572
*EXTERNAL FIELD....577
Using an external field to define distributions and initial conditions....577
Using an external field to define history-dependent fields....579
Using an external field in a node-based submodel analysis....582
*EXTREME ELEMENT VALUE....584
*EXTREME NODE VALUE....586

*EXTREME VALUE....588
F....589
*FABRIC....591
*FAIL STRAIN....593
*FAIL STRESS....594
*FAILURE RATIOS....595
*FASTENER....596
*FASTENER PROPERTY....600
*FATIGUE....601
*FIBER DISPERSION....603
*FIELD....604
*FIELD IMPORT....608
*FIELD MAPPER CONTROLS....609
*FIELD OPERATIONS....612
*FILE FORMAT....614
*FILE OUTPUT....615
*FILM....616
*FILM PROPERTY....618
*FILTER....619
*FIXED MASS SCALING....622
*FLEXIBLE BODY....624
*FLOW....626
*FLUID BEHAVIOR....628
*FLUID BULK MODULUS....629
*FLUID CAVITY....630
*FLUID DENSITY....633
*FLUID EXCHANGE....634
*FLUID EXCHANGE ACTIVATION....636
*FLUID EXCHANGE PROPERTY....638
*FLUID EXPANSION....642
*FLUID FLUX....643
*FLUID ID FIELD....644
*FLUID INFLATOR....645
*FLUID INFLATOR ACTIVATION....646
*FLUID INFLATOR MIXTURE....647
*FLUID INFLATOR PROPERTY....648
*FLUID LEAKOFF....650
*FLUID PIPE CONNECTOR LOSS....652
*FLUID PIPE CONNECTOR SECTION....654
*FLUID PIPE CONNECTOR THERMAL LOSS....655
*FLUID PIPE CONNECTOR WALL....656
*FLUID PIPE FLOW LOSS....657
*FLUID PIPE SECTION....660
*FLUID PIPE THERMAL....661
*FLUID PIPE WALL....662
*FOUNDATION....663
*FRACTURE CRITERION....664
*FRAME SECTION....675

*FREQUENCY....679
*FRICTION....683
G....689
*GAP....690
*GAP CONDUCTANCE....692
*GAP CONVECTION....694
*GAP DIFFUSIVITY....695
*GAP ELECTRICAL CONDUCTANCE....697
*GAP FLOW....699
*GAP HEAT GENERATION....703
*GAP RADIATION....705
*GASKET BEHAVIOR....706
*GASKET CONTACT AREA....707
*GASKET ELASTICITY....708
*GASKET SECTION....710
*GASKET THICKNESS BEHAVIOR....712
*GAS SPECIFIC HEAT....716
*GEL....717
*GEOSTATIC....718
*GLOBAL DAMPING....720
*GLOBAL RESPONSE....722
H....723
*HEADING....724
*HEATCAP....725
*HEAT GENERATION....726
*HEAT TRANSFER....727
*HOURGLASS STIFFNESS....729
*HYPERELASTIC....730
*HYPERFOAM....738
*HYPOELASTIC....740
*HYSTERESIS....741
I....742
*IMPEDANCE....743
*IMPEDANCE PROPERTY....745
*IMPERFECTION....747
*IMPORT....750
*IMPORT CONTROLS....754
*IMPORT ELSET....755
*IMPORT NSET....756
*IMPORT SURFACE....757
*INCIDENT WAVE....759
*INCIDENT WAVE FLUID PROPERTY....761
*INCIDENT WAVE INTERACTION....762
*INCIDENT WAVE INTERACTION PROPERTY....764
*INCIDENT WAVE PROPERTY....766
*INCIDENT WAVE REFLECTION....768
*INCLUDE....769
*INCREMENTATION OUTPUT....770

*INELASTIC HEAT FRACTION....771
*INERTIA RELIEF....772
*INITIAL CONDITIONS....773
*INSTANCE....796
*INTEGRATED OUTPUT....799
*INTEGRATED OUTPUT SECTION....801
*INTERFACE....803
*INTERFACE REACTION....805
*INTERNAL FLUX GENERATION....806
*ISOTROPIZATION PARAMETERS....807
*ITS....808
J....809
*JOINT....810
*JOINT ELASTICITY....811
*JOINT PLASTICITY....814
*JOINTED MATERIAL....816
*JOULE HEAT FRACTION....818
K....819
*KAPPA....820
*KINEMATIC....822
*KINEMATIC COUPLING....823
L....824
*LATENT HEAT....825
*LOAD CASE....826
*LOADING DATA....827
Defining the loading response data for uniaxial behavior in connectors....827
Defining the loading response data from uniaxial tests of fabric materials....831
*LOW DENSITY FOAM....834
M....836
*M1....838
*M2....840
*MAGNETIC PERMEABILITY....842
*MAGNETOSTATIC....846
*MANIFEST....847
*MAP SOLUTION....848
*MASS....850
*MASS ADJUST....852
*MASS DIFFUSION....854
*MASS FLOW RATE....855
*MATERIAL....857
*MATRIX....859
*MATRIX ASSEMBLE....860
*MATRIX CHECK....864
*MATRIX EXPORT....865
*MATRIX GENERATE....866
*MATRIX INPUT....868
*MATRIX OUTPUT....870
*MEAN FIELD DAMAGE....871

*MEAN FIELD HOMOGENIZATION....872
*MEDIA TRANSPORT....875
*MEMBRANE SECTION....876
*MODAL DAMPING....878
*MODAL DYNAMIC....881
*MODAL FILE....882
*MODAL OUTPUT....883
*MODAL PRINT....884
*MODEL CHANGE....885
*MOHR COULOMB....887
*MOHR COULOMB HARDENING....889
*MOISTURE SWELLING....890
*MOLECULAR WEIGHT....891
*MONITOR....892
*MOTION....893
Specifying motion during cavity radiation heat transfer analysis, steady-state transport analysis, or static analysis....893
Specifying motion during an eddy current analysis....895
*MPC....897
*MULLINS EFFECT....898
*MULTIPHYSICS LOAD....901
N....902
*NCOPY....903
*NETWORK STIFFNESS RATIO....908
*NFILL....910
*NGEN....911
*NMAP....913
*NO COMPRESSION....918
*NODAL ENERGY RATE....919
*NODAL THICKNESS....920
*NODE....922
*NODE FILE....924
*NODE OUTPUT....926
*NODE PRINT....929
*NODE RESPONSE....931
*NONLINEAR BH....933
*NONSTRUCTURAL MASS....934
*NORMAL....936
*NO TENSION....937
*NSET....938
O....940
*ONE-STEP INVERSE....941
*OPERATOR OUTPUT....942
*ORIENTATION....944
*ORNL....952
*OUTPUT....953
Defining output requests in an Abaqus/Standard analysis....953
Defining output requests in an Abaqus/Explicit analysis....955
P, Q....958

*PAPERBOARD HARDENING....960
*PAPERBOARD PLASTICITY....962
*PAPERBOARD THICKNESS COMPRESSION ELASTIC....963
*PAPERBOARD THICKNESS COMPRESSION PLASTICITY....964
*PAPERBOARD TRANSVERSE SHEAR PLASTICITY....965
*PARAMETER....966
*PARAMETER DEPENDENCE....967
*PARAMETER SHAPE VARIATION....968
*PARAMETER TABLE....971
*PARAMETER TABLE TYPE....972
*PART....973
*PARTICLE GENERATOR....974
*PARTICLE GENERATOR FLOW....976
*PARTICLE GENERATOR INLET....978
*PARTICLE GENERATOR MIXTURE....979
*PARTICLE OUTLET....980
*PARTICLE OUTLET FLOW....981
*PERFECTLY MATCHED LAYER....982
*PERIODIC....983
*PERIODIC MEDIA....986
*PERMANENT MAGNETIZATION....987
*PERMEABILITY....988
Defining permeability in Abaqus/Standard analyses....988
*PHYSICAL CONSTANTS....991
*PIEZOELECTRIC....993
*PIEZOELECTRIC DAMPING....996
*PIEZORESISTIVITY....998
*PIPE-SOIL INTERACTION....1001
*PIPE-SOIL STIFFNESS....1002
*PLANAR TEST DATA....1007
Using planar test data to define a hyperelastic material....1007
Using planar test data to define an elastomeric foam....1008
Using planar test data to define the Mullins effect material model....1009
*PLASTIC....1010
*PLASTIC AXIAL....1015
*PLASTICITY CORRECTION....1016
*PLASTIC M1....1018
*PLASTIC M2....1019
*PLASTIC TORQUE....1020
*PLY FABRIC FAILURE....1021
*PLY FABRIC HARDENING....1022
*PML COEFFICIENT....1024
*PORE FLUID PRESSURE....1025
*POROUS BULK MODULI....1026
*POROUS ELASTIC....1028
*POROUS ELECTRODE THEORY....1030
*POROUS FAILURE CRITERIA....1031
*POROUS METAL PLASTICITY....1032

*POST OUTPUT....1033
*POTENTIAL....1035
*PREPRINT....1038
*PRESSURE PENETRATION....1039
*PRESSURE STRESS....1041
*PRESTRESS HOLD....1043
*PRE-TENSION SECTION....1044
*PRINT....1046
*PROBABILITY DENSITY FUNCTION....1048
*PROPERTY TABLE....1050
*PROPERTY TABLE TYPE....1052
*PSD-DEFINITION....1053
R....1055
*RADIATE....1056
*RADIATION FILE....1058
*RADIATION OUTPUT....1059
*RADIATION PRINT....1060
*RADIATION SYMMETRY....1061
*RADIATION VIEW FACTOR....1062
*RANDOM RESPONSE....1064
*RATE DEPENDENT....1065
*RATIOS....1067
*REACTION RATE....1068
*REBAR....1069
*REBAR LAYER....1077
*REBAR LINE....1079
*REDUCED BASIS GENERATE....1080
*REFLECTION....1081
*RELEASE....1084
*RESPONSE SPECTRUM....1085
*RESTART....1088
Controlling the writing and reading of restart data in an Abaqus/Standard analysis....1088
Controlling the writing and reading of restart data in an Abaqus/Explicit analysis....1090
*RETAINED NODAL DOFS....1092
*RIGID BODY....1093
*RIGID SURFACE....1096
*ROTARY INERTIA....1100
S....1102
*SCALE MASS....1105
*SCALE STIFFNESS....1106
*SCALE STRESS DESIGN....1107
*SCALE THERMAL CONDUCTIVITY....1108
*SECTION CONTROLS....1109
*SECTION CURVATURE....1119
*SECTION FILE....1121
*SECTION INERTIA....1123
*SECTION ORIGIN....1124
*SECTION POINTS....1125

*SECTION PRINT....1127
*SECTION STIFFNESS....1129
*SECTION TANGENT....1131
*SELECT CYCLIC SYMMETRY MODES....1132
*SELECT EIGENMODES....1133
*SFILM....1135
*SFLOW....1137
*SHEAR CENTER....1139
*SHEAR FAILURE....1140
*SHEAR RETENTION....1142
*SHEAR TEST DATA....1143
Using shear test data to define a viscoelastic material....1143
*SHELL GENERAL SECTION....1144
*SHELL SECTION....1150
*SHELL TO SOLID COUPLING....1154
*SIMPEDANCE....1155
*SIMPLE SHEAR TEST DATA....1157
*SLIDE LINE....1158
*SLOAD....1160
*SOFT ROCK HARDENING....1161
*SOFT ROCK PLASTICITY....1163
*SOFTENING REGULARIZATION....1164
*SOILS....1165
*SOLID ELECTROLYTE THEORY....1168
*SOLID SECTION....1169
*SOLUBILITY....1172
*SOLUTION TECHNIQUE....1173
*SOLVER CONTROLS....1174
*SORPTION....1175
*SPECIFIC HEAT....1177
*SPECTRUM....1179
Defining a spectrum using values of S as a function of frequency and damping....1179
Creating a spectrum from a user-specified amplitude....1180
*SPH SURFACE BEHAVIOR....1182
*SPRING....1183
*SRADIATE....1187
*STATIC....1189
*STEADY STATE CRITERIA....1192
*STEADY STATE DETECTION....1193
*STEADY STATE DYNAMICS....1194
*STEADY STATE TRANSPORT....1198
*STEP....1201
Beginning a step in an Abaqus/Standard analysis....1201
Beginning a step in an Abaqus/Explicit analysis....1203
*STEP CONTROL....1205
*STEP CYCLING....1208
*STEP CYCLING CONTROL....1209
*SUBCYCLING....1211

*SUBMODEL....1212
*SUBMODEL CONDITIONS....1214
*SUBMODEL CUT....1215
*SUBSTRUCTURE CHANGE....1217
*SUBSTRUCTURE DAMPING....1218
*SUBSTRUCTURE DAMPING CONTROLS....1219
*SUBSTRUCTURE GENERATE....1221
*SUBSTRUCTURE LOAD CASE....1225
*SUBSTRUCTURE MATRIX OUTPUT....1226
*SUBSTRUCTURE MODAL DAMPING....1228
*SUBSTRUCTURE OUTPUT....1231
*SUBSTRUCTURE PATH....1232
*SUBSTRUCTURE PROPERTY....1233
*SUPERELASTIC....1237
*SUPERELASTIC HARDENING....1238
*SUPERELASTIC HARDENING MODIFICATIONS....1239
*SURFACE....1241
*SURFACE BEHAVIOR....1249
*SURFACE FLAW....1254
*SURFACE INTERACTION....1255
*SURFACE PROPERTY....1258
*SURFACE PROPERTY ASSIGNMENT....1259
*SURFACE SECTION....1266
*SURFACE SMOOTHING....1267
*SURFACE TENSION....1270
*SURFACE TENSION CONTACT ANGLE....1271
*SWELLING....1272
*SYMMETRIC MODEL GENERATION....1273
*SYMMETRIC RESULTS TRANSFER....1279
*SYSTEM....1281
T....1283
*TABLE COLLECTION....1284
*TEMPERATURE....1285
*TENSILE FAILURE....1290
*TENSION CUTOFF....1292
*TENSION STIFFENING....1293
*THERMAL EXPANSION....1295
*TIE....1296
*TIME POINTS....1299
*TORQUE....1301
*TORQUE PRINT....1303
*TRACER PARTICLE....1304
*TRANSFORM....1305
*TRANSPORT VELOCITY....1307
*TRANSVERSE SHEAR....1308
*TRANSVERSE SHEAR STIFFNESS....1310
*TRIAXIAL TEST DATA....1312
*TRS....1313

U....1316
*UEL PROPERTY....1317
*UNDEX CHARGE PROPERTY....1319
*UNIAXIAL....1321
*UNIAXIAL TEST DATA....1322
Using uniaxial test data to define a hyperelastic material....1322
Using uniaxial test data to define an elastomeric foam....1323
Using uniaxial test data to define a low-density foam material....1324
Using uniaxial test data to define the Mullins effect material model....1325
*UNIFORM....1326
*UNIT SYSTEM....1327
*UNLOADING DATA....1329
Defining the unloading response for uniaxial behavior in connectors....1329
Defining the unloading response data from uniaxial tests of fabric materials....1332
*USER DEFINED FIELD....1334
*USER ELEMENT....1336
Introducing a linear user-defined element (Abaqus/Standard only)....1336
Introducing a general user-defined element....1338
*USER MATERIAL....1340
*USER OUTPUT VARIABLES....1342
V....1343
*VARIABLE MASS SCALING....1344
*VIEW FACTOR OUTPUT....1346
*VISCO....1347
*VISCOELASTIC....1349
*VISCOSITY....1359
*VISCOUS....1366
*VOID NUCLEATION....1370
*VOLUMETRIC TEST DATA....1371
Hyperelastic material model....1371
Hyperfoam material model....1372
Viscoelastic material model....1373
W, X, Y, Z....1374
*WAVE....1375
*WEAR SURFACE PROPERTIES....1378
*WIND....1380
Product Index....1381
Abaqus/Standard....1382
Abaqus/Explicit....1398
Abaqus/CAE....1409
Abaqus/AMS....1421
Abaqus/Aqua....1422
Abaqus/Design....1423

Trademarks and Legal Notices

Trademarks

Abaqus, **3D**EXPERIENCE®, the 3DS logo, the Compass icon, IFWE, 3DEXCITE, 3DVIA, BIOVIA, CATIA, CENTRIC PLM, DELMIA, ENOVIA, GEOVIA, MEDIDATA, NETVIBES, OUTSCALE, SIMULIA and SOLIDWORKS are commercial trademarks or registered trademarks of Dassault Systèmes, a European company (Societas Europaea) incorporated under French law, and registered with the Versailles trade and companies registry under number 322 306 440, or its subsidiaries in the United States and/or other countries. All other trademarks are owned by their respective owners. Use of any Dassault Systèmes or its subsidiaries trademarks is subject to their express written approval.

Legal Notices

Abaqus and this documentation may be used or reproduced only in accordance with the terms of the software license agreement signed by the customer, or, absent such agreement, the then current software license agreement to which the documentation relates.

This documentation and the software described in this documentation are subject to change without prior notice.

Dassault Systèmes or its Affiliates shall not be responsible for the consequences of any errors or omissions that may appear in this documentation.

© Dassault Systèmes Americas Corp., 2025.

For a full list of the third-party software contained in this release, please go to the Legal Notices in the Abaqus 2025 HTML documentation, which can be obtained from a documentation installation, or in the SIMULIA Established Products 2025 Program Directory, which is available on *www.3ds.com*.

Abaqus Keywords Guide

The Abaqus Keywords Guide contains a complete description of all the input options that are available in Abaqus/Standard and Abaqus/Explicit.

This guide is a part of the Abaqus® documentation collection, which describes all the capabilities of the Abaqus finite element analysis technology used in SIMULIA® applications.

A brief description of the intended use of the keyword is listed at the top of each keyword section.

The **Products** field lists each of the products that support the keyword. Keywords that are at least partially supported in Abaqus/CAE include Abaqus/CAE in the list of products. The user interface in Abaqus/CAE does not necessarily support all optional parameters for each supported keyword.

The **Type** field indicates whether the keyword appears in the model or history data portion of the input file. For more information, see *Abaqus Model Definition*.

The **Level** field indicates the level(s) at which the keyword can appear within the input file if the model is defined in terms of an assembly of part instances. For more information, see *Assembly Definition*.

The Abaqus/CAE field indicates where within Abaqus/CAE you can locate the user interface related to the keyword. You can also refer to *Abaqus keyword browser table*, which lists all Abaqus keywords and their support within the user interface or from the input file reader.

To find examples of the usage of a particular keyword in an input file, you can use the **findkeyword** utility (defined in *Querying the Keyword/Problem Database*) to search the sample input files included with the Abaqus release. The **abaqus fetch** utility is used to extract these input files for use. For example, to fetch input file `boltpipeflange_3d_cyclsym.inp`, type

```
abaqus fetch job=boltpipeflange_3d_cyclsym.inp
```

The **abaqus fetch** utility is explained in detail in *Fetching Sample Input Files*.

What's New

This page describes recent changes in Abaqus Keywords.

2025 FD04

*COUPLED CONSTITUTIVE RESPONSE

You can use the new PROPERTIES parameter to set the number of real (floating point) property values needed as data in user subroutine `UCOUPLEDCONSTRESP`.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *COUPLED CONSTITUTIVE RESPONSE*.

*INTERNAL FLUX GENERATION

You can use the new PROPERTIES parameter to set the number of real (floating point) property values needed as data in user subroutine `UINTFLUXGEN`.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *INTERNAL FLUX GENERATION*.

*MEAN FIELD HOMOGENIZATION

You can use the new ODF ADJUST parameter to specify if the orientation distribution function (ODF) reconstructed from the orientation tensor is adjusted to remove any negative values.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *MEAN FIELD HOMOGENIZATION*.

*SYMMETRIC RESULTS TRANSFER

You can use the new LIBRARY parameter to specify the name of the previous axisymmetric or partial three-dimensional analysis from which to transfer results to a three-dimensional model.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *SYMMETRIC RESULTS TRANSFER*.

2025 FD03

*COUPLED CONSTITUTIVE RESPONSE (new)

You can use the new option to define a user-defined constitutive behavior across multiple physics fields.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *COUPLED CONSTITUTIVE RESPONSE*.

*CURE KINETICS

You can use the new NUMBER TERMS parameter to define the number of terms in the Kamal model. In addition, you can now define the Kamal model with temperature and/or field variables dependent coefficients.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *Curing Processes in Polymers* and *CURE KINETICS*.

*DECURRENT

You can use the new `DECURRENT` user subroutine together with table collections to specify electrical current density as a function of position, time, element number, field variables, etc. for coupled thermal-electrical, coupled temperature-displacement electrical, coupled thermal-electrochemical, and coupled temperature-displacement electrochemical analyses.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *DECURRENT*.

*DSECURRENT

You can use the new `DECURRENT` user subroutine together with table collections to specify electrical current density as a function of position, time, element number, field variables, etc. for coupled thermal-electrical, coupled temperature-displacement electrical, coupled thermal-electrochemical, and coupled temperature-displacement electrochemical analyses.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *DSECURRENT*.

*IMPORT

You can input a blank line as data to specify import of all element sets in the previous analysis with a section definition that is supported for import.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

This new format provides a simple way to import all elements in a model once per previous analysis without the need to list the element sets to be imported explicitly.

You can use the blank line input format with repositioning.

For more information, see *IMPORT*.

*SURFACE

You can now use the PROPERTY parameter to define named surface properties for surface tension loads.

Benefits: This feature expands the capabilities of Abaqus/Explicit.

For more information, see *SURFACE*.

*SURFACE PROPERTY

You can now use the PROPERTY parameter to define named surface properties for surface tension loads.

Benefits: This feature expands the capabilities of Abaqus/Explicit.

For more information, see *SURFACE PROPERTY*.

*SURFACE TENSION (new)

You can use this option to specify surface tension properties on element-based surfaces.

Benefits: This feature expands the capabilities of Abaqus/Explicit.

For more information, see *SURFACE TENSION*.

*SURFACE TENSION CONTACT ANGLE (new)

You can use this option to specify a surface tension contact angle as part of a surface interaction definition.

Benefits: This feature expands the capabilities of in Abaqus/Explicit.

For more information, see *SURFACE TENSION CONTACT ANGLE*.

2025 FD02

*DAMPING

You can use the new FREQUENCY DEPENDENT parameter to define frequency-dependent structural damping in Abaqus/Standard, and you can now use the DEPENDENCIES parameter to define variable alpha, beta, band-limited, and structural damping in Abaqus/Standard.

Benefits: This feature expands the capabilities of Abaqus/Standard.
In addition, the ability to define tabular dependence of multiple damping coefficients in a single damping option was removed to simplify the keyword interface. Instead, you can use multiple *DAMPING* options to define the tabular dependence of individual material damping coefficients independently of each other.

For more information, see *DAMPING* and *Material Damping*.

*ELEMENT RESPONSE

You can use the new FATIGUE parameter to specify fatigue design responses for adjoint fatigue design sensitivity analysis using a combined Abaqus-fe-safe workflow.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *ELEMENT RESPONSE* and *Adjoint Design Sensitivity Analysis*.

*INITIAL CONDITIONS

For any of the coupled electrochemical analyses, you can now set the TYPE parameter to BINDERVF and SOLIDVF to give the initial volume fraction of the binder and solid phases, respectively and TYPE=PARTICLE SPECIES CONCENTRATION to give initial species concentrations in the particles present in the active electrodes.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *INITIAL CONDITIONS*.

*MATERIAL

You can use the new USER TYPE parameter to define a material using one of a set of special-purpose user material types.

Benefits: You can now use any name for these special-purpose user materials when the appropriate USER TYPE is specified.

The new USER TYPE parameter can be used to model metallurgical phase transformations and to model cardiac tissue in the Living Heart Human Model. In earlier releases, you had to use a material name starting with a specific “ABQ_” prefix.

For more information, see *MATERIAL* and *About Special-Purpose Material Modeling Techniques*.

*MULLINS EFFECT

You can now use the new DEFINITION parameter to specify the exponential and the slow decay forms to define the evolution of the damage variable for the Mullins effect model. In addition, you must now use the DEFINITION parameter to specify the Mullins effect models, which were available in the earlier releases.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *MULLINS EFFECT*.

*SRADIATE

You can use the new TRANSITION parameter to control the emissivity dependence on contact status.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *SRADIATE*.

2025 FD01

*CLUSTER MASS INERTIA TABLE (new)

Specify mass and inertia for discrete particle clusters.

Benefits: This feature expands the capabilities of Abaqus/Explicit.

For more information, see *CLUSTER MASS INERTIA TABLE*.

*CONTACT CONTROLS ASSIGNMENT

You can use the new SHELL REPRESENTATION parameter to specify how to represent shells for general contact in Abaqus/Standard. In addition, the default value of the BEAM CROSS SECTION parameter is now EXACT in Abaqus/Standard.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *CONTACT CONTROLS ASSIGNMENT*.

*CONTACT FORMULATION

You now set the TYPE parameter equal to MAIN SECONDARY ROLES to control main-secondary roles for specific interactions in Abaqus/Explicit. In earlier releases, this value was supported only in Abaqus/Standard.

Benefits: This improvement unifies the user interface between Abaqus/Explicit and Abaqus/Standard.

For more information, see *CONTACT FORMULATION*.

*CONTACT INITIALIZATION DATA

For general contact in Abaqus/Standard, you can use ADJUST=NO for interactions that are permanently bonded with cohesive contact.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *CONTACT INITIALIZATION DATA*.

*DIAGNOSTICS

For Abaqus/Explicit analyses, the default value of the CONTACT INITIAL OVERCLOSURE parameter is now SUMMARY. In earlier releases, the default value was DETAIL.

Benefits: This feature expands the capabilities of Abaqus/Explicit.

For more information, see *DIAGNOSTICS*.

*DISTRIBUTING

You can now use the ROTATIONAL COUPLING parameter in both Abaqus/Standard and Abaqus/Explicit to specify whether the rotational constraint of the reference node should include the rotations of the cloud nodes in addition to the displacements of the cloud nodes. The parameter can take the new default value of AUTOMATIC to allow Abaqus to couple the rotation of the reference node to the displacement and the rotation of the cloud nodes within the influence radius.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *DISTRIBUTING*.

*ENERGY OUTPUT

You can now use the NSET parameter with direct-solution steady-state dynamic analyses. Alternatively, you can use the new SURFACE parameter to specify the name of the element-based surface representing the boundary for energy flow, power flow, radiated acoustic energy, and radiated acoustic power computations.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *ENERGY OUTPUT*.

*EQUIVALENT RADIATED SURFACE PROPERTIES (new)

Specify radiated surface properties.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *EQUIVALENT RADIATED SURFACE PROPERTIES*.

*FASTENER

You can now use the ROTATIONAL COUPLING parameter in both Abaqus/Standard and Abaqus/Explicit to specify whether the rotational constraint of the fastening point should include the rotations of the surface nodes in addition to the displacements of the surface nodes. The parameter can take the new default value of AUTOMATIC to allow Abaqus to couple the rotation of the fastening point to the displacement and the rotation of the surface nodes within the influence radius.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *FASTENER*.

*FRACTURE CRITERION

You can set the TYPE parameter equal to PRX-FATIGUE to use the PRX-Fatigue criterion as the fatigue crack propagation criterion.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *FRACTURE CRITERION*.

*HYPERELASTIC

You can now use the HENCKY, PROPERTIES, USER, and VALANIS-LANDEL parameters in both Abaqus/Standard and Abaqus/Explicit. In earlier releases, you could use these parameters only in Abaqus/Standard.

Benefits: This improvement unifies the user interface between Abaqus/Explicit and Abaqus/Standard.

For more information, see *HYPERELASTIC*.

*NORMAL

You can use the new BEAM NORMAL parameter to define nodal normals.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *NORMAL*.

*PAPERBOARD HARDENING (new)

Specify the hardening data of the in-plane paperboard plasticity model.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *PAPERBOARD HARDENING*.

*PAPERBOARD PLASTICITY (new)

Specify the in-plane paperboard plasticity model.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *PAPERBOARD PLASTICITY*.

*PAPERBOARD THICKNESS COMPRESSION ELASTIC (new)

Define the out-of-plane compression elasticity for the paperboard model.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *PAPERBOARD THICKNESS COMPRESSION ELASTIC*.

*PAPERBOARD THICKNESS COMPRESSION PLASTICITY (new)

Define the out-of-plane compression yield surface for the paperboard model.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *PAPERBOARD THICKNESS COMPRESSION PLASTICITY*.

*PAPERBOARD TRANSVERSE SHEAR PLASTICITY (new)

Define the transverse shear yield surface for the paperboard model.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *PAPERBOARD TRANSVERSE SHEAR PLASTICITY*.

*SFILM

You can use the new TRANSITION parameter to control the film coefficient's dependence on the contact status.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *SFILM*.

*STEP CONTROL

You can use the new TOLERANCE parameter to determine when the sensor has reached the threshold value.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *STEP CONTROL*.

*SURFACE BEHAVIOR

When used to modify the default surface behavior, you cannot use the PRESSURE-OVERCLOSURE parameter with the NO SEPARATION parameter in a general contact Abaqus/Explicit analysis, except for PRESSURE-OVERCLOSURE=LINEAR.

Benefits: This feature expands the capabilities of Abaqus/Explicit.

For more information, see *SURFACE BEHAVIOR*.

*SURFACE SECTION

You can now use this option to define a constant thickness surface element (independent of using the *REBAR LAYER option).

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *SURFACE SECTION*.

PDF Guides Available

You can download PDF versions of the Abaqus guides from each guide overview.

Benefits: You can easily access the PDF versions of the guides.
In earlier releases, PDF versions of the guides were updated only for the GA (General Availability) release and could be downloaded from the Dassault Systèmes Knowledge Base. As of 2025 FD01, the PDF guides will be updated with each release of the HTML versions of the guides.

For more information, see *Abaqus Keywords Guide*.

2025 GA

*FLUID EXCHANGE ACTIVATION

The Interaction module in Abaqus/CAE now supports this option.

Benefits: This feature extends the capabilities of Abaqus/CAE.

For more information, see *FLUID EXCHANGE ACTIVATION*.

*STEP CYCLING (new)

Specify a step or series of steps to be repeated over a set number of cycles.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *STEP CYCLING*.

*STEP CYCLING CONTROL (new)

Specify a sensor-based or a contact wear-based criterion to trigger early termination of step cycling before a set number of step cycles are complete.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *STEP CYCLING CONTROL*.

2024 FD04

*BEAM GENERAL SECTION

Use the new optional CONTROLS parameter to specify bulk viscosity scale factors or to ramp up the initial stress in an Abaqus/Explicit analysis.

Benefits: This feature expands the capabilities of Abaqus/Explicit.

For more information, see *BEAM GENERAL SECTION*.

*BEAM SECTION

Use the new optional CONTROLS parameter to specify bulk viscosity scale factors or to ramp up the initial stress in an Abaqus/Explicit analysis.

Benefits: This feature expands the capabilities of Abaqus/Explicit.

For more information, see *BEAM SECTION*.

*PLASTICITY CORRECTION

The Property module in Abaqus/CAE now supports this option.

Benefits: This feature extends the capabilities of Abaqus/CAE.

For more information, see *PLASTICITY CORRECTION*.

2024 FD03

*CONTROLS

You can use the new CONVERGENCE CHECK parameter to specify which convergence conditions Abaqus/Standard must satisfy before it accepts the incremental solution as converged.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *CONTROLS*.

*ELECTRICAL CONDUCTIVITY

You can now use this option to define electrical conductivity for elements that support the electric potential degree of freedom in a coupled thermal-electrochemical, coupled thermal-electrochemical-structural, or coupled thermal-electrochemical-structural-pore pressure analysis.

Benefits: This feature expands the capabilities of Abaqus/Standard.
In earlier releases, you could use *ELECTRICAL CONDUCTIVITY* only in coupled thermal-electrical and coupled thermal-electrical-structural analyses.

For more information, see *ELECTRICAL CONDUCTIVITY*.

*ELECTRICAL RESISTIVITY (new)

Define electrical resistivity for elements that support the electric potential degree of freedom in a coupled thermal-electrical, coupled thermal-electrical-structural, coupled thermal-electrochemical, coupled thermal-electrochemical-structural, or coupled thermal-electrochemical-structural-pore pressure analysis.

Benefits: This feature expands the capabilities of Abaqus/Standard.
You can use the new option to define electrical resistivity (as opposed to conductivity). This method of specifying electrical resistivity is required for piezoresistive materials.

For more information, see *ELECTRICAL RESISTIVITY* and *Piezoresistive Effect*.

*ELECTRIC MACHINE LOAD (new)

Define electromagnetic force loads for an electric machine.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *ELECTRIC MACHINE LOAD*.

*ELECTRIC MACHINE PROPERTY (new)

Define electric machine properties (or parameters) that are needed to apply electric machine loads.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *ELECTRIC MACHINE PROPERTY*.

*ELEMENT USER OUTPUT VARIABLES (new)

Specify the number of whole element and integration point user-defined variables defined for each element in user subroutine `UVAREL`.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *ELEMENT USER OUTPUT VARIABLES*.

*EXPANSION

You can use the new DEFINITION parameter to specify secant or tangent thermal expansion coefficients.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *EXPANSION*.

*MODAL DYNAMIC

You can use the new SOLVER parameter to specify if you want to use the state space solver or the conventional solver.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *MODAL DYNAMIC*.

*PIEZORESISTIVITY (new)

Define the piezoresistivity coefficients for a piezoresistive material.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *PIEZORESISTIVITY*.

*SECTION CONTROLS

You can now use this option to control the activation of bending stabilization for full-integration linear elements modeled with hyperelastic materials and to specify the scale factor.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *SECTION CONTROLS*.

*UNIT SYSTEM

You can use the new SYSTEM parameter to specify the unit system used in the model.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *UNIT SYSTEM*.

*VISCOELASTIC

You now set the LAW parameter equal to MODIFIED DARVEAUX to choose a modified Darveaux law for a nonlinear viscoelastic material definition.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *VISCOELASTIC*.

2024 FD02

*CONTOUR INTEGRAL

You can use the LENGTH SCALING FACTOR parameter to indicate that Abaqus/Standard will not adjust the locations or the radii of all nodes along the crack front when evaluating the line integral.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *CONTOUR INTEGRAL*.

*CO-SIMULATION

The CONTROLS parameter and the PROGRAM=ABAQUS value are deprecated. You can now set PROGRAM=MULTIPHYSICS for exchange of data between Abaqus and the SIMULIA Co-Simulation Engine, which in turn can exchange data with third-party analysis programs.

Benefits: All co-simulation control parameters are defined in the Co-Simulation Engine Configuration File.

For more information, see *CO-SIMULATION*.

*CO-SIMULATION CONTROLS (removed)

This option is removed. You now define all co-simulation control parameters in the Co-Simulation Engine Configuration File.

Benefits: Defining a co-simulation process is easier.

For more information, see *CO-SIMULATION*.

*CURE GLASS TRANSITION TEMPERATURE

You can now use this option with all cure kinetics models. In earlier releases, you could use it only with the Grindling model.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *CURE GLASS TRANSITION TEMPERATURE*.

*DENSITY

Abaqus/Explicit now supports the PORE FLUID parameter for undrained pore fluid flow and stress analysis.

Benefits: This feature expands the capabilities of Abaqus/Explicit.

For more information, see *DENSITY*.

*DYNAMIC

You can now set the PORE PRESSURE parameter equal to UNDRAINED to specify that an undrained pore fluid flow and stress analysis is to be performed in Abaqus/Explicit.

Benefits: This feature expands the capabilities of Abaqus/Explicit.

For more information, see *DYNAMIC*.

*EXPANSION

Abaqus/Explicit now supports the PORE FLUID parameter for undrained pore fluid flow and stress analysis.

Benefits: This feature expands the capabilities of Abaqus/Explicit.

For more information, see *EXPANSION*.

*HYPERELASTIC

You can use the new HENCKY parameter to use the Hencky model in Abaqus/Standard.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *HYPERELASTIC*.

*IMPORT

You can use the full-model input format to specify that all import model data, including elements and nodes, element sets and node sets, sections and materials, etc., are given in the input.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *IMPORT*.

*IMPORT CONTROLS

You can set the new MODEL parameter equal to YES to indicate that you want to use the full-model input format. In an Abaqus/Explicit analysis, you can use the new TIE CONFIG parameter to indicate whether to use the original or updated configuration in the tie constraint formation for all surfaces of the imported model.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *IMPORT CONTROLS*.

*INITIAL CONDITIONS

You can now set the TYPE parameter equal to PORE PRESSURE to define the initial pore pressure for undrained pore fluid flow and stress analysis in Abaqus/Explicit. Similarly, you can set the TYPE parameter equal to POROSITY to define the initial porosity values and to RATIO to define the initial void ratio values.

Benefits: This feature expands the capabilities of Abaqus/Explicit.

For more information, see *INITIAL CONDITIONS*.

*POROUS BULK MODULI

You can now use this option in Abaqus/Explicit to define the porous bulk moduli for undrained pore fluid flow and stress analysis.

Benefits: This feature expands the capabilities of Abaqus/Explicit.

For more information, see *POROUS BULK MODULI*.

*SURFACE PROPERTY

You can now use this option in Abaqus/Explicit to define named surface properties for contact-based wear modeling.

Benefits: This feature expands the capabilities of Abaqus/Explicit.

For more information, see *SURFACE PROPERTY*.

*VISCOELASTIC

You now set the LAW parameter equal to DARVEAUX to choose a Darveaux law.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *VISCOELASTIC*.

*WEAR SURFACE PROPERTIES

You can use this option in Abaqus/Explicit to specify wear properties to activate wear distance accumulation at contact surface nodes.

Benefits: This feature expands the capabilities of Abaqus/Explicit.

For more information, see *WEAR SURFACE PROPERTIES*.

2024 FD01

*ALLOWABLE STRESS (new)

Specify the allowable stress of a composite at the initiation of damage in a multiscale material.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *ALLOWABLE STRESS*.

*CFILM

You can use the new SINK AMPLITUDE parameter to define the variation of the sink temperature with time. The SINK AMPLITUDE parameter supersedes the AMPLITUDE parameter used in earlier versions.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *CFILM* and *Thermal Loads*.

*CONNECTOR SECTION

For SLIPRING connections, you can now request that Abaqus/Standard computes the contact angle automatically during the analysis.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *CONNECTOR SECTION*.

*CONTOUR INTEGRAL

You can now use the SURFACE NORMAL parameter when tetrahedral elements are used for fracture mechanics studies based on the conventional finite element method.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *CONTOUR INTEGRAL*.

*CRADIATE

You can use the new AMBIENT AMPLITUDE parameter to define the variation of the ambient temperature with time. The AMBIENT AMPLITUDE parameter supersedes the AMPLITUDE parameter used in earlier versions. You can also use the new RADIATION AMPLITUDE parameter to define a variation of the radiation flux with time.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *CRADIATE* and *Thermal Loads*.

*DAMAGE EVOLUTION

You can now set the TYPE parameter equal to DISCONTINUOUS to define the residual stiffness scaling factor after the initiation of damage.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *DAMAGE EVOLUTION*.

*DAMAGE INITIATION

You can now define general stress/strain-based damage initiation as a material property for continuum damage modeling in anisotropic materials. You can also now define damage initiation as a material property for continuum damage modeling in fiber-reinforced composites using the multiscale material model in Abaqus/Standard.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *DAMAGE INITIATION*.

*DAMPING

Abaqus/Standard now supports the BAND LIMITED, HIGH FREQUENCY CUTOFF, and LOW FREQUENCY CUTOFF parameters.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *DAMPING*.

*DISTRIBUTING

You can use the new SINGLE FACET parameter to specify whether or not to automatically convert distributing couplings where all cloud nodes are on a single shell facet to a constraint in Abaqus/Explicit.

Benefits: This feature expands the capabilities of Abaqus/Explicit.

For more information, see *DISTRIBUTING*.

*EXPANSION

In an Abaqus/Standard analysis, you can now use the PORE FLUID parameter to define field expansion of the pore fluid in a porous medium. Similarly, the USER and ZERO parameters now take field expansion into account.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *EXPANSION*.

*FILM

You can use the new SINK AMPLITUDE parameter to define the variation of the sink temperature with time. The SINK AMPLITUDE parameter supersedes the AMPLITUDE parameter used in earlier versions.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *FILM* and *Thermal Loads*.

*NSET

You can now use the new SURFACE parameter to generate a node set from a previously defined surface.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.
This enhancement makes it easier to create a node set from a surface definition. All the nodes included in the surface are assigned to the specified node set.

For more information, see *NSET*.

*RADIATE

You can use the new AMBIENT AMPLITUDE parameter to define the variation of the ambient temperature with time. The AMBIENT AMPLITUDE parameter supersedes the AMPLITUDE parameter used in earlier versions. You can also use the new RADIATION AMPLITUDE parameter to define a variation of the radiation flux with time.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *RADIATE* and *Thermal Loads*.

*REDUCED BASIS GENERATE (new)

Indicate that the step should be analyzed as a reduced basis generation step.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *REDUCED BASIS GENERATE*.

*SFILM

You can use the new SINK AMPLITUDE parameter to define the variation of the sink temperature with time. The SINK AMPLITUDE parameter supersedes the AMPLITUDE parameter used in earlier versions.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *SFILM* and *Thermal Loads*.

*SRADIATE

You can use the new AMBIENT AMPLITUDE parameter to define the variation of the ambient temperature with time. The AMBIENT AMPLITUDE parameter supersedes the AMPLITUDE parameter used in earlier versions. You can also use the new RADIATION AMPLITUDE parameter to define a variation of the radiation flux with time.

Benefits: This feature expands the capabilities of Abaqus/Standard and Abaqus/Explicit.

For more information, see *SRADIATE* and *Thermal Loads*.

*STEADY STATE DYNAMICS

You can use the new MODE BASED parameter to define the conventional modal solver or the spectral modal solver that can perform better than the conventional solver in analyses with many modes, frequency points, and load cases for finite element models with special, but commonly used, damping properties.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *STEADY STATE DYNAMICS* and *Mode-Based Steady-State Dynamic Analysis*.

*SUBMODEL CUT (new)

Identify a submodel cut on which to impose the submodel conditions and specify the submodel control settings specific to a submodel cut.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *SUBMODEL CUT*.

*SURFACE PROPERTY

You can now use this option to define named surface properties for contact-based wear modeling.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *SURFACE PROPERTY*.

*WEAR SURFACE PROPERTIES (new)

Specify wear properties to activate wear distance accumulation at contact surface nodes.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *WEAR SURFACE PROPERTIES*.

2024 GA

*BEAM SECTION

The Property module in Abaqus/CAE now supports this option.

Benefits: This feature extends the capabilities of Abaqus/CAE.

For more information, see *BEAM SECTION*, *Using a Beam Section Integrated during the Analysis to Define the Section Behavior*, and *BeamSection object*.

*CONTACT MASS SCALING

The Interaction module in Abaqus/CAE now supports this option.

Benefits: This feature extends the capabilities of Abaqus/CAE.

For more information, see *CONTACT MASS SCALING* and *Defining contact mass scaling*.

*IMPERFECTION

The Interaction module in Abaqus/CAE now supports this option.

Benefits: This feature extends the capabilities of Abaqus/CAE.

For more information, see *IMPERFECTION*.

*NODE OUTPUT

You can use the new SURFACE parameter to specify the name of the surface for which to request nodal output.

Benefits: This feature makes it easier to request nodal output on a surface in Abaqus/Standard and Abaqus/Explicit. The SURFACE parameter is available when the *NODE OUTPUT option is used in conjunction with the *OUTPUT, HISTORY option for Abaqus/Standard and Abaqus/Explicit or when the *NODE OUTPUT option is used in conjunction with the *OUTPUT, FIELD option for Abaqus/Explicit.

For more information, see *NODE OUTPUT* and *OUTPUT*.

*NODE RESPONSE

You can use the new RANGE OPERATOR parameter to define the operator used for aggregating the sensitivities and design response values over the specified frequency range for the node responses in direct steady-state dynamic adjoint sensitivity analysis.

Benefits: This parameter allows you to control the calculation of the design response over a frequency range for the newly supported steady-state dynamic adjoint sensitivity analysis technique.
By default, Abaqus/Standard outputs the maximum of the design response values as the design response over the frequency range specified for the node response in direct steady-state dynamic analysis when you request adjoint sensitivity computation. Alternatively, you can request output of the sum or average of the node response values over the specified frequency range for the response.

For more information, see *NODE RESPONSE* and *Adjoint Design Sensitivity Analysis*.

*OPERATOR OUTPUT

You can now use the *OPERATOR OUTPUT option in an uncoupled heat transfer analysis. In addition, the LOAD parameter can now take the value NET to output the net flux vector in an uncoupled heat transfer analysis.

Benefits: This feature expands the capabilities of Abaqus/Standard.

For more information, see *OPERATOR OUTPUT*.

A

This section describes each Abaqus input option that begins with A, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **ACOUSTIC CONTRIBUTION*
- **ACOUSTIC FLOW VELOCITY*
- **ACOUSTIC MEDIUM*
- **ACOUSTIC WAVE FORMULATION*
- **ACTIVATE ELEMENTS*
- **ADAPTIVE MESH*
- **ADAPTIVE MESH CONSTRAINT*
- **ADAPTIVE MESH CONTROLS*
- **ADAPTIVE MESH REFINEMENT*
- **ADJUST*
- **ALLOWABLE STRESS*
- **AMPLITUDE*
- **ANISOTROPIC HYPERELASTIC*
- **ANNEAL*
- **ANNEAL TEMPERATURE*
- **AQUA*
- **ASSEMBLY*
- **ASYMMETRIC-AXISYMMETRIC*
- **AXIAL*

*ACOUSTIC CONTRIBUTION: Request the computation of the acoustic contribution factors for the linear, eigenmode-based, steady-state dynamic procedure.

This option is relevant only during linear steady-state dynamics analyses that use the natural modes of the system (*STEADY STATE DYNAMICS without the DIRECT parameter). To enable computation of the acoustic contribution factors, the preceding *FREQUENCY step must activate the SIM architecture, and the ACOUSTIC COUPLING parameter should be set to PROJECTION (uncoupled modes).

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Acoustic Contribution Factors in Mode-Based and Subspace-Based Steady-State Dynamic Analyses*
- *Natural Frequency Extraction*
- *Mode-Based Steady-State Dynamic Analysis*
- *Subspace-Based Steady-State Dynamic Analysis*

Required parameters:

ACOUSTIC NODES

Set this parameter equal to the name of a node set containing the nodes in the acoustic domain. This set defines the locations of the response nodes—the nodes at which the acoustic contribution factors are determined.

NAME

Set this parameter equal to the name that will be used to refer to the acoustic contribution factor. This name will be used to define/access the SIM data set for the acoustic contribution factor.

TYPE

Set TYPE=MODAL ACOUSTIC to specify acoustic modal contribution factors.

Set TYPE=MODAL STRUCTURAL to specify acoustic structural modal contribution factors.

Set TYPE=MODAL LOAD to specify acoustic load modal contribution factors.

Set TYPE=LOAD to specify acoustic load contribution factors.

Set TYPE=PANEL to specify panel contribution factors.

Set TYPE=GRID to specify grid contribution factors.

Optional parameters:

OUTPUT

Set OUTPUT=FIELD to request field output of the computed acoustic contribution factors to the output database (`.odb`) file. This value is relevant only for TYPE=GRID. If you specify OUTPUT=FIELD, you cannot request output of the grid acoustic contribution factors to the SIM database (`.sim`) file.

Set OUTPUT=HISTORY to request history output of the computed acoustic contribution factors to the output database (`.odb`) file.

STRUCTURAL NODES

This parameter is relevant only for TYPE=PANEL or TYPE=GRID.

Set this parameter equal to the name of the node set containing the nodes on the acoustic-structural interface. The node set specifies the source nodes for panel and grid contribution factors.

If both the SURFACE and STRUCTURAL NODES parameters are included, the resulting node set is defined as the union of the node set specified with the STRUCTURAL NODES parameter and the node set obtained from the surface specified by the SURFACE parameter.

If both the STRUCTURAL NODES and SURFACE parameters are omitted, all structural nodes on the acoustic-structural interface are used to compute the acoustic contribution factors.

SURFACE

This parameter is relevant only for TYPE=PANEL or TYPE=GRID.

Set this parameter equal to the name of the surface containing the structural nodes on the acoustic-structural interface. The surface specifies the source nodes for panel and grid contribution factors.

If both the SURFACE and STRUCTURAL NODES parameters are included, the resulting node set is defined as the union of the node set specified with the STRUCTURAL NODES parameter and the node set obtained from the surface specified by the SURFACE parameter.

If both the STRUCTURAL NODES and SURFACE parameters are omitted, all structural nodes on the acoustic-structural interface are used to compute the acoustic contribution factors.

Data line to select the natural eigenmodes for the modal contribution factors:

First (and only) data line:

1. f_{min}, the lower bound of the frequency range. This data item is ignored if TYPE=LOAD, TYPE=PANEL, or TYPE=GRID.
2. f_{max}, the upper bound of the frequency range. This data item is ignored if TYPE=LOAD, TYPE=PANEL, or TYPE=GRID.

These bounds will be applied to the active eigenmodes (the eigenmodes selected with the **SELECT EIGENMODES* option for the current **STEADY STATE DYNAMICS* step).

*ACOUSTIC FLOW VELOCITY: Specify flow velocities as a predefined field for acoustic elements.

This option is used to specify the fluid flow velocity of node sets or individual nodes for acoustic analysis. This option defines an underlying flow, about which the acoustic analysis is a linear perturbation.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Coupled Acoustic-Structural Analysis*

Required, mutually exclusive parameters:

ROTATION

Include this parameter to define a flow velocity field due to a rigid body rotation about an axis.

TRANSLATION

Include this parameter to give the *x*-, *y*-, and *z*-components of translational flow velocity in the global coordinate system or in the local coordinate system if **TRANSFORM* was used at these nodes. Translational flow velocity is the default.

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve (defined in the **AMPLITUDE* option) that gives the time variation of the flow velocity throughout the step (*Amplitude Curves*).

If this parameter is omitted, the default is a STEP function.

Data lines to define translational flow velocity (TRANSLATION):

First line:

1. Node set label or node number.
2. First translational component of flow velocity prescribed (only degrees of freedom 1, 2, or 3 can be entered). See *Conventions* for a definition of the numbering of degrees of freedom in Abaqus.
3. Last translational component of flow velocity prescribed (only degrees of freedom 1, 2, or 3 can be entered). This field can be left blank if flow velocity for only one component is being prescribed.
4. Magnitude of the translational displacement or velocity. This magnitude will be modified by the **AMPLITUDE* specification if the AMPLITUDE parameter is used.

Repeat this data line as often as necessary to define translational flow velocity for different nodes and degrees of freedom.

Data lines to define rotational flow velocity (ROTATION):

First line:

1. Node set label or node number.

2. Magnitude of the rotation (in radians) or rotational velocity (in radians/time). This magnitude will be modified by the **AMPLITUDE* specification if the AMPLITUDE parameter is used.

   The rotation is about the axis defined from point **a** to point **b**, where the coordinates of **a** and **b** are given next. In steady-state transport analysis the position and orientation of the rotation axis are applied at the beginning of the step and remain fixed during the step.
3. Global *x*-component of point **a** on the axis of rotation.
4. Global *y*-component of point **a** on the axis of rotation.

The following data are required only for three-dimensional cases:

5. Global *z*-component of point **a** on the axis of rotation.
6. Global *x*-component of point **b** on the axis of rotation.
7. Global *y*-component of point **b** on the axis of rotation.
8. Global *z*-component of point **b** on the axis of rotation.

Repeat this data line as often as necessary to define rotational flow velocity for different nodes.

*ACOUSTIC MEDIUM: Specify an acoustic medium.

This option is used to define the properties of an acoustic medium used with acoustic or coupled poroelastic acoustic elements. The *ACOUSTIC MEDIUM option must be used in conjunction with the *MATERIAL option. The *ACOUSTIC MEDIUM option can be used multiple times to specify all the properties of an acoustic medium.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Acoustic Medium*
- *Poroelastic Acoustic Medium*

Optional, mutually exclusive parameters:

BULK MODULUS

Include this parameter to define the bulk modulus for the acoustic medium (default).

CAVITATION LIMIT

This parameter applies only to Abaqus/Explicit analyses.

Include this parameter to define the cavitation pressure limit for the acoustic medium. When the fluid absolute pressure drops to this limit, the acoustic medium undergoes free volume expansion or cavitation without a further decrease in the pressure. A negative cavitation limit value represents an acoustic medium that is capable of sustaining a negative absolute pressure up to the specified limit value. Any nonzero initial acoustic static pressure values such as those due to the atmospheric pressure and/or the hydrostatic loading can be specified using the *INITIAL CONDITIONS, TYPE=ACOUSTIC STATIC PRESSURE option.

If this parameter is omitted, the fluid is assumed not to cavitate even under arbitrarily large negative pressure conditions.

COMPLEX BULK MODULUS

Include this parameter to define the complex bulk modulus for the acoustic medium.

COMPLEX DENSITY

Include this parameter to define the complex density for the acoustic medium.

POROUS MODEL

This parameter applies only to Abaqus/Standard analyses.

Set POROUS MODEL=DELANY BAZLEY or DELANY-BAZLEY (default) to use the Delany-Bazley model to compute the frequency-dependent complex density and the complex bulk modulus.

Set POROUS MODEL=MIKI to use the Delany-Bazley-Miki model to compute the frequency-dependent complex density and the complex bulk modulus.

Set POROUS MODEL=BIOT-JOHNSON to use the model of Biot and Johnson et al. to compute the frequency-dependent properties of the coupled poroelastic acoustic material.

Set POROUS MODEL=BIOT-ATALLA to use the coupled poroelastic-acoustic material model of Biot and Atalla et al. together with the COMPLEX BULK MODULUS and COMPLEX DENSITY parameters specified as separate options.

VOLUMETRIC DRAG

Include this parameter to define the volumetric drag coefficient for the acoustic medium.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the acoustic medium, in addition to temperature. If this parameter is omitted, it is assumed that the acoustic medium property is constant or depends only on temperature. See *Material Data Definition* for more information.

Data lines to define the bulk modulus of an acoustic material:

First line:

1. Bulk modulus. (Units of FL^{-2}.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the bulk modulus as a function of temperature and other predefined field variables.

Data lines to define the cavitation pressure limit of an acoustic material:

First line:

1. Cavitation pressure limit. (Units of FL^{-2}.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the cavitation pressure limit as a function of temperature and other predefined field variables.

Data line to define the complex bulk modulus of an acoustic material:

First line:

1. Real part of the bulk modulus. (Units of FL^{-2}.)
2. Imaginary part of the bulk modulus. (Units of FL^{-2}.)
3. Frequency. (Units of T^{-1}.)

Repeat this data line as often as necessary to define the complex bulk modulus as a function of frequency.

Data line to define the complex density of an acoustic material:

First line:

1. Real part of the density. (Units of ML^{-3}.)
2. Imaginary part of the density. (Units of ML^{-3}.)
3. Frequency. (Units of T^{-1}.)

Repeat this data line as often as necessary to define the complex density as a function of frequency.

Data lines to define the volumetric drag of an acoustic material:

First line:

1. Volumetric drag coefficient. (Units of FTL^{-4}.)
2. Frequency. (Cycles/time.) Frequency dependence is active only during frequency domain procedures in Abaqus/Standard.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the volumetric drag as a function of frequency, temperature, and other predefined field variables.

Data line when POROUS MODEL=DELANY BAZLEY, DELANY-BAZLEY, or MIKI:

First (and only) line:

1. Flow resistivity. (Units of FTL^{-4}.)

Data line when POROUS MODEL=BIOT-ATALLA:

First (and only) line:

1. Fluid density. (Units of ML^{-3}.)
2. Tortuosity. (Dimensionless.)
3. Structural material bulk modulus. (Units of FL^{-2}.)
4. Fluid-structure coupling factor. (Dimensionless, default=1.)

Data lines when POROUS MODEL=BIOT-JOHNSON:

First line:

1. Fluid density. (Units of ML^{-3}.)
2. Tortuosity. (Dimensionless.)
3. Structural material bulk modulus. (Units of FL^{-2}.)
4. Fluid-structure coupling factor. (Dimensionless, default=1.)

Second line:

1. Static flow resistivity. (Units of FTL^{-4}.)

2. Viscous characteristic length. (Units of L.)
3. Thermal characteristic length. (Units of L.)
4. Fluid dynamic shear viscosity. (Units of $MT^{-1}L^{-1}$.)
5. Ambient fluid standard pressure. (Units of FL^{-2}.)
6. Ambient fluid heat capacity ratio of the specific heat per unit mass at constant pressure over the specific heat per unit mass at constant volume. (Dimensionless.)
7. Prandtl number, the ratio of viscous diffusion rate over the thermal diffusion rate. (Dimensionless.)

*ACOUSTIC WAVE FORMULATION: Specify the type of formulation in acoustic problems with incident wave loading.

This option is used to identify the type of incident wave loading formulation in acoustic problems.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Model attribute

References:

• *Acoustic and Shock Loads*

Optional parameters:

TYPE

Set TYPE=SCATTERED WAVE (default) to obtain the scattered wave field solution that will be produced by incident wave loading.

Set TYPE=TOTAL WAVE to obtain the total acoustic pressure wave solution.

There are no data lines associated with this option.

*ACTIVATE ELEMENTS: Activate elements within a step.



Warning: Development of this capability and user interface is expected to evolve in subsequent releases. Therefore, models may not be upward compatible without modification.

This option is used to turn on a progressive element activation feature within a step definition.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Progressive Element Activation*
- *UEPACTIVATIONVOL*
- **ELEMENT PROGRESSIVE ACTIVATION*

Required parameters:

ACTIVATION

Set this parameter equal to the name assigned to the progressive element activation feature on the **ELEMENT PROGRESSIVE ACTIVATION* option.

Optional parameters:

EIGEN TIME CONSTANT

Set this parameter equal to the time constant used to ramp up eigenstrains at element activation. The default is 0.0.

EXPANSION TIME CONSTANT

Set this parameter equal to the time constant used to ramp up thermal strains at element activation. The default value is 2 times the initial time increment.

Data lines to define progressive element activation:

First (and only) line:

1. Table collection name.

*ADAPTIVE MESH: Define an adaptive mesh domain.

This option is used to define an adaptive mesh domain and to specify the frequency and intensity of adaptive meshing for that domain.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Supported in the *Step module*; only one adaptive mesh domain can be defined per step.

References:

- *Defining ALE Adaptive Mesh Domains in Abaqus/Explicit*
- *ALE Adaptive Meshing and Remapping in Abaqus/Explicit*
- *Defining ALE Adaptive Mesh Domains in Abaqus/Standard*
- *ALE Adaptive Meshing and Remapping in Abaqus/Standard*
- **ADAPTIVE MESH CONTROLS*
- **ADAPTIVE MESH CONSTRAINT*

At least one of the following parameters is required:

ELSET

Set this parameter equal to the name of the element set that contains all the solid elements in the adaptive mesh domain.

OP

Set OP=MOD (default) to modify the frequency and intensity of adaptive meshing for an existing adaptive mesh domain (with the same element set name) or to define a new adaptive mesh domain.

Set OP=NEW if all adaptive mesh domains that are currently in effect should be removed. To remove only selected adaptive mesh domains, use OP=NEW and respecify all adaptive mesh domains that are to be retained.

The OP parameter must be the same for all uses of the **ADAPTIVE MESH* option within a single step.

Optional parameters:

CONTROLS

Set this parameter equal to the name of the **ADAPTIVE MESH CONTROLS* option associated with this adaptive mesh domain. Adaptive mesh controls can be used to control the adaptive meshing in explicit dynamic analysis and in implicit acoustic analysis and to control the advection algorithms applied to the adaptive mesh domain in explicit dynamic analysis.

FREQUENCY

Set this parameter equal to the frequency in increments at which adaptive meshing is to be performed. When the option is used in a spatial mesh constraint or an Eulerian boundary region is defined on the adaptive mesh domain in explicit dynamic analysis, the default frequency is 1. A default frequency of 1 is also used for acoustic domains. In all other cases the default frequency is 10.

INITIAL MESH SWEEPS

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the number of mesh sweeps to be performed at the beginning of the first step in which this adaptive mesh definition is active. The default number of initial mesh sweeps is 5 if **ADAPTIVE MESH CONTROLS*, SMOOTHING OBJECTIVE=UNIFORM is used. The default number of initial mesh sweeps is 2 if **ADAPTIVE MESH CONTROLS*, SMOOTHING OBJECTIVE=GRADED is used.

MESH SWEEPS

Set this parameter equal to the number of mesh sweeps to be performed in each adaptive mesh increment. The default number of mesh sweeps is 5.

There are no data lines associated with this option.

*ADAPTIVE MESH CONSTRAINT: Specify constraints on the motion of the mesh for an adaptive mesh domain.

Warning: Abaqus/Explicit does not admit jumps in mesh displacement. If no amplitude is specified, Abaqus/Explicit will ignore the user-supplied displacement value and enforce a zero mesh motion constraint.

This option is used to prescribe independent mesh motion for nodes in an adaptive mesh domain or to define nodes that must follow the material. It can be used only in conjunction with the **ADAPTIVE MESH* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Displacement and velocity adaptive mesh constraints are supported in the *Step module*.

References:

- *Defining ALE Adaptive Mesh Domains in Abaqus/Explicit*
- *Defining ALE Adaptive Mesh Domains in Abaqus/Standard*
- *UMESHMOTION*
- **ADAPTIVE MESH*

Optional parameters:

AMPLITUDE

This parameter is relevant only when some of the variables being prescribed have nonzero magnitudes. Set this parameter equal to the name of the amplitude curve defining the magnitude of the prescribed mesh motion (*Amplitude Curves*).

CONSTRAINT TYPE

Set CONSTRAINT TYPE=SPATIAL (default) to prescribe mesh motions that are independent of the underlying material.

Set CONSTRAINT TYPE=LAGRANGIAN to define nodes that must follow the material.

OP

Set OP=MOD (default) to modify existing mesh constraints or to add mesh constraints to degrees of freedom that were previously unconstrained.

Set OP=NEW if all mesh constraints that are currently in effect should be removed. To remove only selected mesh constraints, use OP=NEW and respecify all mesh constraints that are to be retained.

The OP parameter must be the same for all uses of the **ADAPTIVE MESH CONSTRAINT* option within a single step.

TYPE

Set TYPE=DISPLACEMENT (default) to prescribe mesh displacement.

Set TYPE=VELOCITY to prescribe mesh velocity.

USER

This parameter applies only to Abaqus/Standard analyses.

Include this parameter if the mesh motion is to be defined in user subroutine *UMESHMOTION*. This parameter cannot be used when CONSTRAINT TYPE=LAGRANGIAN.

Data lines to prescribe mesh motions that are independent of the material (CONSTRAINT TYPE=SPATIAL):

First line:

1. Node number or node set label.
2. First degree of freedom constrained. This value is ignored when the USER parameter is specified.
3. Last degree of freedom constrained. This field can be left blank if the mesh must be constrained only in one direction. This value is ignored when the USER parameter is specified.
4. Actual magnitude of the mesh motion (displacement or velocity). This magnitude will be modified by an amplitude specification if the AMPLITUDE parameter is used. This value will be ignored in an Abaqus/Explicit analysis if TYPE=DISPLACEMENT, no AMPLITUDE specification is provided, and this value is nonzero.

Repeat this data line as often as necessary to specify mesh constraints at different nodes and degrees of freedom.

Data lines to define nodes that must follow the material (CONSTRAINT TYPE=LAGRANGIAN):

First line:

1. Node number or node set label.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

*ADAPTIVE MESH CONTROLS: Specify controls for the adaptive meshing and advection algorithms.

This option is used to control various aspects of the adaptive meshing and advection algorithms applied to an adaptive mesh domain. It can be used only in conjunction with the *ADAPTIVE MESH* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Defining ALE Adaptive Mesh Domains in Abaqus/Explicit*
- *ALE Adaptive Meshing and Remapping in Abaqus/Explicit*
- *Defining ALE Adaptive Mesh Domains in Abaqus/Standard*
- *ALE Adaptive Meshing and Remapping in Abaqus/Standard*
- *ADAPTIVE MESH*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to this adaptive mesh controls definition. Adaptive mesh control names in the same input file must be unique.

Optional parameters:

ADVECTION

This parameter applies only to Abaqus/Explicit analyses.

Set ADVECTION=SECOND ORDER (default) to use a second-order algorithm to remap solution variables after adaptive meshing has been performed.

Set ADVECTION=FIRST ORDER to use a first-order algorithm to remap solution variables after adaptive meshing has been performed.

CURVATURE REFINEMENT

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the solution dependence weight, α_C. The default value is $\alpha_C = 1$.

GEOMETRIC ENHANCEMENT

Set GEOMETRIC ENHANCEMENT=YES (default in Abaqus/Explicit analyses) to use smoothing algorithms that are enhanced based on evolving element geometry.

Set GEOMETRIC ENHANCEMENT=NO (default in Abaqus/Standard analyses) to use the conventional form of the smoothing algorithms.

INITIAL FEATURE ANGLE

Set this parameter equal to the initial geometric feature angle, θ_I, in degrees ($0° \le \theta_I \le 180°$). This angle is used to detect geometric edges and corners. The default value is $\theta_I = 30°$. Setting $\theta_I = 180°$ will ensure that no geometric edges or corners are detected or enforced.

MESH CONSTRAINT ANGLE

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the mesh constraint angle, θ_C, in degrees ($5° \le \theta_C \le 85°$). The default value is $\theta_C = 60°$.

When adaptive mesh constraints are applied to nodes on Lagrangian or sliding boundary regions, the analysis will terminate if the angle between the normal to the boundary region and the direction of the prescribed constraint becomes less than θ_C. When adaptive mesh constraints are applied to nodes that are part of a Lagrangian or active geometric edge, the analysis will terminate if the angle between the prescribed constraint and the plane perpendicular to the edge becomes less than θ_C.

MESHING PREDICTOR

This parameter is interpreted differently in Abaqus/Explicit and Abaqus/Standard analyses.

In an Abaqus/Explicit analysis, set MESHING PREDICTOR=CURRENT (default if the adaptive mesh domain has no Eulerian boundary regions) to perform adaptive meshing based on current nodal positions; this method is recommended for all Lagrangian-like problems and for problems with very large distortions. Set MESHING PREDICTOR=PREVIOUS (default if the adaptive mesh domain has one or more Eulerian boundary regions) to perform adaptive meshing based on the positions of the nodes at the end of the previous adaptive mesh increment; this technique is recommended for Eulerian-like problems where material flow is significant compared to the overall deformation.

In an Abaqus/Standard analysis, set MESHING PREDICTOR=CURRENT to perform adaptive meshing based on the positions of the nodes at the start of the current adaptive mesh increment. Set MESHING PREDICTOR=PREVIOUS (default) to perform adaptive meshing based on the nodal positions in the original mesh.

MOMENTUM ADVECTION

This parameter applies only to Abaqus/Explicit analyses.

Set MOMENTUM ADVECTION=ELEMENT CENTER PROJECTION (default) to use the element center projection method for advecting momentum. This method is less expensive than the half-index shift method.

Set MOMENTUM ADVECTION=HALF INDEX SHIFT to use the half-index shift method for momentum advection. This algorithm is more expensive computationally but may demonstrate better dispersion properties than the element center projection method.

RESET

Include this parameter to reset all adaptive mesh controls to their default values. Controls that are specified with other parameters on the same **ADAPTIVE MESH CONTROLS* option are retained. If this parameter is omitted, only the specified controls will be changed in the current step; the others will remain at their settings from previous steps.

SMOOTHING OBJECTIVE

This parameter applies only to Abaqus/Explicit analyses.

Set SMOOTHING OBJECTIVE=UNIFORM (default if the adaptive mesh domain has no Eulerian boundary regions in explicit dynamic analysis) to perform adaptive meshing that minimizes element

distortion and improves element aspect ratios at the expense of diffusing initial mesh gradation. This objective is recommended for problems with moderate to large overall deformation.

Set SMOOTHING OBJECTIVE=GRADED (default if the adaptive mesh domain has one or more Eulerian boundary regions in explicit dynamic analysis) to perform adaptive meshing that attempts to preserve initial mesh gradation while reducing distortions as the analysis evolves. This objective is recommended only for adaptive mesh domains with reasonably structured graded meshes undergoing low to moderate overall deformation.

TRANSITION FEATURE ANGLE

Set this parameter equal to the transition geometric feature angle, θ_T, in degrees ($0° \leq \theta_T \leq 180°$). This angle is used to determine when geometric edges and corners should be deactivated to allow remeshing across them. The default value is $\theta_T = 30°$. Setting $\theta_T = 0°$ will ensure that no geometric edges or corners are deactivated.

Data line to define weights for combining the mesh smoothing methods in Abaqus/Explicit analyses:

First (and only) line:

1. The weight for the volumetric smoothing method. The default is 1.0.
2. The weight for the Laplacian smoothing method. The default is 0.0.
3. The weight for the equipotential smoothing method. The default is 0.0.

Each of the weights must be zero or positive and their sum should typically be 1.0. If the sum of the weights is less than 1.0, the mesh smoothing algorithm will be less aggressive at each adaptive mesh increment. If the sum of the weights is greater than 1.0, their values are normalized so that their sum is 1.0.

Data line to define weights for combining the mesh smoothing methods in Abaqus/Standard analyses:

First (and only) line:

1. The weight for the original configuration projection method. The default is 1.0.
2. The weight for the volumetric smoothing method. The default is 0.0.

Each of the weights must be zero or positive and their sum must be nonzero. The weights are significant only in a relative sense; their values are normalized so that their sum is 1.0.

*ADAPTIVE MESH REFINEMENT: Activate adaptive mesh refinement in an Eulerian domain.

This option is used to activate adaptive mesh refinement in an Eulerian domain and to specify the refinement criteria in that domain.

Products: Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Eulerian Analysis*
- *Defining Adaptive Mesh Refinement in the Eulerian Domain*
- **EULERIAN SECTION*

Required parameters:

ELSET

Set this parameter equal to the name of the element set where adaptive mesh refinement is applied.

Optional parameters:

COARSENING

This parameter is valid only when ENHANCED CONTACT=NO.

Set COARSENING=YES (default) to specify that refinement can be removed once the refinement criteria are no longer met.

Set COARSENING=NO to specify that refinement cannot be removed even when the refinement criteria are no longer met.

ENHANCED CONTACT

Set ENHANCED CONTACT=YES to activate the enhanced contact formulation that acts to prevent the potential leakage of liquid or gas through the Lagrangian surfaces.

Set ENHANCED CONTACT=NO (default) to deactivate the enhanced contact formulation for the Eulerian-Lagrangian interfaces.

LEVEL

This parameter is valid only when ENHANCED CONTACT=NO.

Set this parameter equal to the maximum number of levels of refinement. The default value is 1.

RATIO

Set this parameter equal to the ratio of the maximum increase in the number of elements during mesh refinement compared to the original number of elements in the specified element set. The default value is 8.0 when ENHANCED CONTACT=NO and is 1.0 when ENHANCED CONTACT=YES.

Data lines to define the criteria used in the mesh refinement when ENHANCED CONTACT=NO:

First line:

1. Refinement criteria label (see *Defining Adaptive Mesh Refinement in the Eulerian Domain*).

2. Value of the refinement criteria if applicable.

Repeat this data line as often as necessary to define multiple criteria for mesh refinement.

There are no data lines if ENHANCED CONTACT=YES

*ADJUST: Adjust user-specified nodal coordinates to lie on a given surface or to make adjustments based on a user-specified distribution.

This option is used to adjust user-specified nodal coordinates so that the nodes lie on a given surface. It is also used to make adjustments based on a user-defined distribution.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Interaction module

References:

• *Adjusting Nodal Coordinates*

Adjusting nodes so that the nodes lie on a given surface

Required parameters:

NODE SET

Set this parameter equal to the name of the node set containing the nodes to be adjusted.

SURFACE

Set this parameter equal to the name of surface to which the nodes are to be adjusted.

Optional parameters:

ORIENTATION

Set this parameter equal to the name of an orientation definition (see *Orientations*) that defines the direction for adjusting nodes. If this parameter is omitted, the nodes are adjusted normal to the specified surface. Only rectangular, cylindrical, and spherical orientation definitions are supported. Additional rotations defined as part of the orientation definition are ignored.

There are no data lines associated with this option.

Adjusting nodes using a distribution of adjustments

Required parameters:

DISTRIBUTION

Set this parameter to a distribution name to specify nodes and adjustments (displacements) to the position of these nodes.

NODE SET

Set this parameter equal to the name of the node set containing the nodes to be adjusted.

There are no data lines associated with this option.

*ALLOWABLE STRESS: Specify the allowable stress of a composite at the initiation of damage in a multiscale material.

This option is used to specify the allowable stress of a composite at the initiation of damage in a multiscale material.

This option can be used only in conjunction with the **MEAN FIELD HOMOGENIZATION* option.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

• *About Damage and Failure for Fiber-Reinforced Composites Using Multiscale Modeling*

Required parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the allowable stress. If this parameter is omitted, it is assumed that the allowable stresses are constant or depend only on temperature.

TYPE

Set TYPE=UD to specify allowable stress for a unidirectional fiber-reinforced composite.

Set TYPE=WOVEN to specify allowable stress for a plain weave composite.

Data lines to specify allowable stress for TYPE=UD:

First line:

1. Longitudinal tensile strength of the lamina.
2. Longitudinal compressive strength of the lamina.
3. Transverse tensile strength of the lamina.
4. Transverse compressive strength of the lamina.
5. Longitudinal shear strength of the lamina.
6. Transverse shear strength of the lamina.
7. Temperature, if temperature dependent.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the strengths on temperature and other predefined field variables.

Data lines to specify allowable stress for TYPE=WOVEN:

First line:

1. Tensile strength of the lamina along the 1-direction.
2. Compressive strength of the lamina along the 1-direction.

3. Longitudinal shear strength of the lamina.
4. Tensile stress of the lamina along the 1-direction at the initiation of the matrix failure.
5. Compressive stress of the lamina along the 1-direction at the initiation of the matrix failure.
6. Longitudinal shear stress of the lamina at the initiation of the matrix failure.
7. Temperature, if temperature dependent.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the strengths on temperature and other predefined field variables.

*AMPLITUDE: Define an amplitude curve.

This option allows arbitrary time (or frequency in an Abaqus/Standard analysis) variations of load, displacement, and other prescribed variable magnitudes to be given throughout a step.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Model, Step

Abaqus/CAE: *Amplitude toolset*; bubble loading is not supported. Similar functionality is available in the *Interaction module*.

References:

• *Amplitude Curves*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the amplitude curve.

Optional parameters:

DEFINITION

Set DEFINITION=TABULAR (default) to give the amplitude-time (or amplitude-frequency) definition in tabular form.

Set DEFINITION=EQUALLY SPACED, PERIODIC, MODULATED, DECAY, SMOOTH STEP, SOLUTION DEPENDENT, or BUBBLE to define the amplitude according to the definitions given in *Amplitude Curves*.

Set DEFINITION=USER to define the amplitude via user subroutines *UAMP* and *VUAMP*.

Set DEFINITION=ACTUATOR to define the amplitude via co-simulation with a logical modeling program.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

This parameter cannot be used if DEFINITION=USER or DEFINITION=ACTUATOR.

NORMALIZED

Set NORMALIZED=YES to define the amplitude time as normalized time (values must be in the range of 0–1). The normalized time is multiplied automatically by the current step time during the analysis.

Set NORMALIZED=NO (default) to indicate that the amplitude time values are not normalized values. In this case the amplitude time is not multiplied by the current step time during the analysis.

This parameter can be used only with DEFINITION=TABULAR or DEFINITION=SMOOTH STEP.

SCALEX

Set this parameter equal to the value by which the time values are to be scaled. The default is 1.

This parameter cannot be used if DEFINITION=SOLUTION DEPENDENT, BUBBLE, USER, or ACTUATOR.

SCALEY

Set this parameter equal to the value by which the amplitude values are to be scaled. The default is 1.

This parameter cannot be used if DEFINITION=SOLUTION DEPENDENT, BUBBLE, or USER.

SHIFTX

Set this parameter equal to the value by which the time values are to be shifted. The default is 0.

This parameter cannot be used if DEFINITION=SOLUTION DEPENDENT, BUBBLE, USER, or ACTUATOR.

SHIFTY

Set this parameter equal to the value by which the amplitude values are to be shifted. The default is 0.

This parameter cannot be used if DEFINITION=SOLUTION DEPENDENT, BUBBLE, or USER.

TIME

Set TIME=STEP TIME (default) for step time. If the step in which the amplitude is referenced is in the frequency domain, STEP TIME corresponds to frequency.

Set TIME=TOTAL TIME for total time accumulated over all non-perturbation analysis steps.

See *Conventions* for a discussion of these time measures.

VALUE

Set VALUE=RELATIVE (default) for relative magnitude definition.

Set VALUE=ABSOLUTE for direct input of absolute magnitudes. In this case the data line values in the load option are ignored. Because the values given in the field definition are ignored, the absolute amplitude value will be used to define both the temperature and the gradient. For this reason, VALUE=ABSOLUTE should not be used when temperatures or predefined field variables are specified for nodes connected to beam and shell elements whose section definition includes TEMPERATURE=GRADIENTS (default).

Required parameter for DEFINITION=EQUALLY SPACED:

FIXED INTERVAL

Set this parameter equal to the fixed time (or frequency) interval at which the amplitude data will be given.

Optional parameter for DEFINITION=EQUALLY SPACED:

BEGIN

Set this parameter equal to the time (or lowest frequency) at which the first amplitude is given. The default is BEGIN=0.0.

Optional parameter for DEFINITION=TABULAR or DEFINITION=EQUALLY SPACED:

SMOOTH

Set this parameter equal to the fraction of the time interval before and after each time point during which the piecewise linear time variation is to be replaced by a smooth quadratic time variation in

any case when time derivatives of the amplitude definition are required. The defaults are SMOOTH=0.25 in Abaqus/Standard and SMOOTH=0.0 in Abaqus/Explicit. The allowable range is $0.0 < \text{SMOOTH} \leq 0.5$. A value of 0.05 is suggested for amplitude definitions that contain large time intervals to avoid severe deviation from the specified definition. This parameter is applicable only when time derivatives are needed (for displacement or velocity boundary conditions in a direct integration dynamic analysis) and is ignored for all other uses of this option.

Optional parameters for DEFINITION=USER:

PROPERTIES

Set this parameter equal to the number of properties being entered. The properties are available for use in user subroutines *UAMP* and *VUAMP*. They can be defined on the data lines or directly within the user subroutine. The default is PROPERTIES=0.

VARIABLES

Set this parameter equal to the number of solution-dependent state variables that must be stored with this amplitude definition. Its value must be greater than 0. The default is VARIABLES=1.

Data lines for DEFINITION=TABULAR or DEFINITION=SMOOTH STEP with four data points (eight entries) per line:

First line:

1. Time or frequency. For NORMALIZED=YES, time values must be in the range of 0–1.
2. Amplitude value (relative or absolute) at the first point.
3. Time or frequency.
4. Amplitude value (relative or absolute) at the second point.
5. Etc., up to four pairs per line.

Repeat this data line as often as necessary. Each line (except the last one) must have exactly four time/magnitude or frequency/magnitude data pairs.

Data lines for DEFINITION=TABULAR or DEFINITION=SMOOTH STEP with one data pair (two entries) per line:

First line:

1. Time or frequency. For NORMALIZED=YES, time values must be in the range of 0–1.
2. Amplitude value (relative or absolute) at the first point.

Repeat this data line as often as necessary. Each line must have exactly one time/magnitude or frequency/magnitude data pair.

Data lines for DEFINITION=EQUALLY SPACED with eight values per line:

First line:

1. Amplitude value at the time or frequency given on the BEGIN parameter.
2. Amplitude value at the next point.
3. Etc., up to eight values per line.

Repeat this data line as often as necessary. Each line (except the last one) must have exactly eight amplitude values.

Data lines for DEFINITION=EQUALLY SPACED with one value per line:

First line:

1. Amplitude value at the time or frequency given on the BEGIN parameter.

Repeat this data line as often as necessary. Each line must have exactly one amplitude value.

Data lines to define periodic data (DEFINITION=PERIODIC):

First line:

1. N, the number of terms in the Fourier series.
2. ω, the circular frequency, in radians per time.
3. t_0, the starting time.
4. A_0, the constant term in the Fourier series.

Second line:

1. A_1, the first coefficient of the cosine terms.
2. B_1, the first coefficient of the sine terms.
3. A_2, the second coefficient of the cosine terms.
4. B_2, the second coefficient of the sine terms.
5. Etc., up to eight values per line.

Repeat this data line as often as necessary. Each line (except the last one) must have exactly eight entries, to a total of $2N$ entries.

Data line to define modulated data (DEFINITION=MODULATED):

First (and only) line:

1. A_0.
2. A.
3. t_0.
4. ω_1.
5. ω_2.

Data line to define exponential decay (DEFINITION=DECAY):

First (and only) line:

1. A_0, the constant term.
2. A, the coefficient of the exponential function.
3. t_0, the start time of the exponential function.
4. t_d, the decay time of the exponential function.

Data line to define a solution-dependent amplitude (DEFINITION=SOLUTION DEPENDENT):

First (and only) line:

1. Initial amplitude value (default = 1.0).

2. Minimum amplitude value (default = 0.1).
3. Maximum amplitude value (default = 1000.).

Data lines to define bubble loading (DEFINITION=BUBBLE):

First line:

1. Charge material constant, K.
2. Charge material constant, k.
3. Charge material constant, A.
4. Charge material constant, B.
5. Adiabatic charge constant, K_c.
6. Ratio of specific heats for gas, γ.
7. Density of charge material, ρ_c.
8. Mass of charge material, m_c.
9. Depth magnitude of charge material, d_I.

Second line:

1. Fluid mass density, ρ_f.
2. Sound speed in fluid, c_f.
3. X-direction cosine of fluid surface normal.
4. Y-direction cosine of fluid surface normal.
5. Z-direction cosine of fluid surface normal.

Third line:

1. Acceleration due to gravity, g.
2. Atmospheric pressure, p_{atm}.
3. Wave effect parameter, η. Set to 1.0 for wave effects in the fluid and gas; set to 0.0 to neglect these effects. The default is 1.0.
4. Flow drag coefficient, C_D. The default is 0.0.
5. Flow drag exponent, E_D ($E_D \geq 0$). The default is 2.0.

Fourth line:

1. Time duration, T_{final}.
2. Maximum number of time steps for the bubble simulation, N_{steps}. The bubble amplitude simulation ceases when the number of steps reaches N_{steps} or the time duration, T_{final}, is reached. The default is 1500.
3. Relative step size control parameter, Ω_{rel}. The default is 1×10^{-11}.
4. Absolute step size control parameter, X_{abs}. The default is 1×10^{-11}.
5. Step size control exponent, β. The step size, Δt, is decreased or increased according to the error estimate: $(\Omega_{\text{rel}} |x| + X_{\text{abs}})^{\beta} \leq \Delta t \left| \frac{dx}{dt} \right|^{\beta}$. The default is 0.2.

Data lines to define user amplitude properties when PROPERTIES is specified with DEFINITION=USER:

First line:

1. Enter the amplitude properties, eight per line.

Repeat this data line as often as necessary to define all amplitude properties.

There are no data lines if DEFINITION=ACTUATOR

*ANISOTROPIC HYPERELASTIC: Specify anisotropic hyperelastic properties for approximately incompressible materials.

This option is used to define material constants for a general anisotropic hyperelastic material.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Anisotropic Hyperelastic Behavior*
- *UANISOHYPER_STRAIN*
- *UANISOHYPER_INV*
- *VUANISOHYPER_STRAIN*
- *VUANISOHYPER_INV*

Required parameters:

DEFINITION

Set DEFINITION=FUNG-ANISOTROPIC to use the generalized Fung anisotropic strain energy potential.

Set DEFINITION=FUNG-ORTHOTROPIC to use the generalized Fung orthotropic strain energy potential.

Set DEFINITION=HOLZAPFEL-GASSER-OGDEN to use the Holzapfel-Gasser-Ogden strain energy potential.

Set DEFINITION=HOLZAPFEL-OGDEN to use the Holzapfel-Ogden strain energy potential.

Set DEFINITION=KALISKE-SCHMIDT to use the Kaliske-Schmidt strain energy potential.

Set DEFINITION=USER if the strain energy potential and its derivatives are defined in a user subroutine (user subroutines *UANISOHYPER_INV* and *UANISOHYPER_STRAIN* in Abaqus/Standard or *VUANISOHYPER_INV* and *VUANISOHYPER_STRAIN* in Abaqus/Explicit).

Required parameters if DEFINITION=USER is specified:

FORMULATION

Set FORMULATION=STRAIN to indicate that the anisotropic hyperelastic energy potential is formulated in terms of the components of the Green strain tensor and is defined by either *UANISOHYPER_STRAIN* in Abaqus/Standard or *VUANISOHYPER_STRAIN* in Abaqus/Explicit.

Set FORMULATION=INVARIANT to indicate that the anisotropic hyperelastic energy potential is formulated in terms of pseudo-invariants and is defined by either *UANISOHYPER_INV* in Abaqus/Standard or *VUANISOHYPER_INV* in Abaqus/Explicit.

TYPE

This parameter applies only to Abaqus/Standard analyses.

Set TYPE=INCOMPRESSIBLE to indicate that the anisotropic hyperelastic material defined by *UANISOHYPER_INV* or *UANISOHYPER_STRAIN* is incompressible.

Set TYPE=COMPRESSIBLE to indicate that the hyperelastic material defined by *UANISOHYPER_INV* or *UANISOHYPER_STRAIN* is compressible.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the anisotropic hyperelastic material properties. If this parameter is omitted, it is assumed that the material properties are constant or depend only on temperature. See *Specifying Field Variable Dependence* for more information.

LINEARIZATION STRETCH

This parameter applies only if DEFINITION=HOLZAPFEL-OGDEN.

Set LINEARIZATION STRETCH=$\boldsymbol{\lambda_{fL}}$ to indicate that the second and third exponential terms in the Holzapfel-Ogden strain energy potential are linearized/approximated by polynomial functions when fiber stretches reach $\boldsymbol{\lambda_{fL}}$. The value of $\boldsymbol{\lambda_{fL}}$ should be greater than or equal to 1.

If this parameter is omitted, the exponential terms in the strain energy potential are not linearized.

LOCAL DIRECTIONS

This parameter can be used only in combination with invariant-based strain energy potentials, such as DEFINITION=HOLZAPFEL-GASSER-OGDEN; DEFINITION=HOLZAPFEL-OGDEN; DEFINITION=KALISKE-SCHMIDT; and DEFINITION=USER, FORMULATION=INVARIANT.

Set this parameter equal to the number of preferred local directions (or fiber directions) in the material. The default is LOCAL DIRECTIONS=0.

When LOCAL DIRECTIONS=*N*, the definitions of the *N* local direction vectors in the reference configuration are specified using the **ORIENTATION*, LOCAL DIRECTIONS=*M* option, with *M* *N*. If *M* > *N*, the first *N* directions are used.

If the Holzapfel-Gasser-Ogden, Holzapfel-Ogden, or Kaliske-Schmidt strain energy potential is used, you must specify at least one local direction.

MODULI

This parameter is applicable only when the **ANISOTROPIC HYPERELASTIC* option is used in conjunction with the **VISCOELASTIC* option.

Set MODULI=INSTANTANEOUS to indicate that the anisotropic hyperelastic material constants define the instantaneous behavior. This parameter value is not available for frequency domain viscoelasticity in an Abaqus/Standard analysis. This is the only option available if the anisotropic hyperelastic potential is defined in a user subroutine.

Set MODULI=LONG TERM to indicate that the hyperelastic material constants define the long-term behavior. This option is not available when a user subroutine is used to define the anisotropic hyperelastic potential. It is the default for all other anisotropic hyperelastic models.

PROPERTIES

This parameter applies only if DEFINITION=USER is specified.

Set this parameter equal to the number of property values needed as data in user subroutines *UANISOHYPER_INV* and *UANISOHYPER_STRAIN* in Abaqus/Standard or *VUANISOHYPER_INV* and *VUANISOHYPER_STRAIN* in Abaqus/Explicit. The default value is 0.

Data lines to define the material constants for the FUNG-ANISOTROPIC model:

First line:

1. b_{1111}.
2. b_{1122}.
3. b_{2222}.
4. b_{1133}.
5. b_{2233}.
6. b_{3333}.
7. b_{1112}.
8. b_{2212}.

Second line:

1. b_{3312}.
2. b_{1212}.
3. b_{1113}.
4. b_{2213}.
5. b_{3313}.
6. b_{1213}.
7. b_{1313}.
8. b_{1123}.

Third line:

1. b_{2223}.
2. b_{3323}.
3. b_{1223}.
4. b_{1323}.
5. b_{2323}.
6. $\boldsymbol{c}$. (Units of FL^{-2}.)
7. $\boldsymbol{D}$. (Units of $\mathrm{F}^{-1}\mathrm{L}^{2}$.)
8. Temperature.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than zero):

1. First field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the material constants as a function of temperature and other predefined field variables.

Data lines to define the material constants for the FUNG-ORTHOTROPIC model:

First line:

1. b_{1111}.
2. b_{1122}.
3. b_{2222}.

4. b_{1133}.
5. b_{2233}.
6. b_{3333}.
7. b_{1212}.
8. b_{1313}.

Second line:

1. b_{2323}.
2. c. (Units of FL^{-2}.)
3. D. (Units of $F^{-1}L^2$.)
4. Temperature.
5. First field variable.
6. Etc., up to four field variables per line.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the material constants as a function of temperature and other predefined field variables.

Data lines to define the material constants for the HOLZAPFEL-GASSER-OGDEN model:

First line:

1. C_{10}. (Units of FL^{-2}.)
2. D. (Units of $F^{-1}L^2$.)
3. k_1. (Units of FL^{-2}.)
4. k_2.
5. Fiber dispersion parameter κ ($0 \le \kappa \le 1/3$).
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the material constants as a function of temperature and other predefined field variables.

Data lines to define the material constants for the HOLZAPFEL-OGDEN model:

First line:

1. a. (Units of FL^{-2}.)
2. b.
3. a_f. (Units of FL^{-2}.)
4. b_f.
5. a_s. (Units of FL^{-2}.)

6. b_s.
7. a_{fs}. (Units of FL^{-2}.)
8. b_{fs}.

Second line:

1. D. (Units of $F^{-1}L^2$.)
2. Temperature.
3. First field variable.
4. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the material constants as a function of temperature and other predefined field variables.

Data lines to define the material constants for the KALISKE-SCHMIDT model:

First line:

1. a_1. (Units of FL^{-2}.)
2. a_2. (Units of FL^{-2}.)
3. a_3. (Units of FL^{-2}.)
4. b_1. (Units of FL^{-2}.)
5. b_2. (Units of FL^{-2}.)
6. b_3. (Units of FL^{-2}.)
7. c_2. (Units of FL^{-2}.)
8. c_3. (Units of FL^{-2}.)

Second line:

1. c_4. (Units of FL^{-2}.)
2. c_5. (Units of FL^{-2}.)
3. c_6. (Units of FL^{-2}.)
4. d_2. (Units of FL^{-2}.)
5. d_3. (Units of FL^{-2}.)
6. d_4. (Units of FL^{-2}.)
7. d_5. (Units of FL^{-2}.)
8. d_6. (Units of FL^{-2}.)

Third line:

1. e_2. (Units of FL^{-2}.)
2. e_3. (Units of FL^{-2}.)
3. e_4. (Units of FL^{-2}.)
4. e_5. (Units of FL^{-2}.)
5. e_6. (Units of FL^{-2}.)
6. f_2. (Units of FL^{-2}.)

7. f_3. (Units of FL^{-2}.)
8. f_4. (Units of FL^{-2}.)

Fourth line:

1. f_5. (Units of FL^{-2}.)
2. f_6. (Units of FL^{-2}.)
3. g_2. (Units of FL^{-2}.)
4. g_3. (Units of FL^{-2}.)
5. g_4. (Units of FL^{-2}.)
6. g_5. (Units of FL^{-2}.)
7. g_6. (Units of FL^{-2}.)
8. D. (Units of $F^{-1}L^2$.)

Fifth line (leave blank if no temperature or field variables are specified):

1. Temperature.
2. First field variable.
3. Etc., up to seven field variables per line.

Repeat this set of data lines as often as necessary to define the material constants as a function of temperature and other predefined field variables.

Data lines to define the material properties for the USER anisotropic hyperelasticity model:

No data lines are needed if the PROPERTIES parameter is omitted or set to 0. Otherwise, first line:

1. Give the material properties, eight per line. If this option is used in conjunction with the **VISCOELASTIC* option, the material properties must define the instantaneous behavior. If this option is used in conjunction with the **MULLINS EFFECT* option, the material properties must define the primary response.

Repeat this data line as often as necessary to define the material properties.

*ANNEAL: Anneal the structure.

This option is used to anneal a structure by setting the velocities and all appropriate state variables to zero.

Products: Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

• *Annealing*

Optional parameters:

TEMPERATURE

Set this parameter equal to the temperature, θ, to which all nodes in the model will be set after the annealing has been completed. The default is to maintain the current temperature at all nodes in the model after the annealing has been completed.

There are no data lines associated with this option.

*ANNEAL TEMPERATURE: Specify material properties for modeling annealing or melting.

This option is used to define the annealing temperature of elastic-plastic materials. It must be used in conjunction with the *PLASTIC option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Annealing or Melting*
- *PLASTIC

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the annealing temperature. If this parameter is omitted, it is assumed that the annealing temperature is a constant. See *Material Data Definition* for more information.

Data lines to define the annealing temperature:

First line:

1. Value of the annealing temperature, θ.
2. First field variable.
3. Etc., up to seven field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than seven):

1. Eighth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the material parameter θ on field variables.

*AQUA: Define fluid variables for use in loading immersed beam-type structures.

This option is used to define the fluid properties and steady-current velocity.

Products: Abaqus/Aqua

Type: Model data

Level: Model

References:

• *Abaqus/Aqua Analysis*

Optional parameters:

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

Data lines to define fluid properties and a steady current:

First line:

1. Elevation of the seabed.
2. Elevation of the still fluid surface.
3. Gravitational constant.
4. Mass density of the fluid.

Second line:

1. Steady velocity of the fluid in the *X*-direction.
2. Steady velocity of the fluid in the *Y*-direction.
3. Steady velocity of the fluid in the *Z*-direction. Only relevant for three-dimensional cases.
4. Elevation.
5. *X*-coordinate defining the location where the velocity applies. If this value is omitted, the velocity is assumed to be independent of position in the *X*-direction.
6. *Y*-coordinate defining the location where the velocity applies. Only relevant for three-dimensional cases. If this value is omitted in a three-dimensional analysis, the velocity is assumed to be independent of position in the *Y*-direction.

Repeat the second data line as often as necessary to define the steady current velocity as a function of elevation and spatial coordinates. See *Input Syntax Rules* for a description of how to define a property as a function of multiple independent variables.

*ASSEMBLY: Begin an assembly definition.

This option is used to begin an assembly definition. It must be used in conjunction with the **END ASSEMBLY*, **INSTANCE*, and **PART* options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Assembly module

References:

- **END ASSEMBLY*
- *Assembly Definition*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the assembly.

There are no data lines associated with this option.

*ASYMMETRIC-AXISYMMETRIC: Define areas of integration for contact elements used with CAXAn or SAXAn elements.

This option is used to allow Abaqus/Standard to calculate appropriate areas of integration for ISL- and IRS-type contact elements used in conjunction with CAXA*n* or SAXA*n* elements. The **ASYMMETRIC-AXISYMMETRIC* option must be used in conjunction with the **INTERFACE* option.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance, Assembly

References:

- *About Contact Interactions*
- *Contact Modeling If Asymmetric-Axisymmetric Elements Are Present*
- **INTERFACE*

Required parameters:

ANGLE

Set this parameter equal to the angular position (measured in degrees) of the circumferential plane in which the contact elements exists. Valid values are $\theta = 0°$, 180° for $n = 1$; $\theta = 0°$, 90°, 180° for $n = 2$; $\theta = 0°$, 60°, 120°, 180° for $n = 3$; and $\theta = 0°$, 45°, 90°, 135°, 180° for $n = 4$. Abaqus/Standard does not model contact correctly on other circumferential planes.

MODE

Set this parameter equal to the number of Fourier modes used with the CAXA*n* or SAXA*n* elements that share nodes with the contact elements.

There are no data lines associated with this option.

*AXIAL: Used to define the axial behavior of beams.

This option can be used only in conjunction with the *BEAM GENERAL SECTION, SECTION=NONLINEAR GENERAL option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- *BEAM GENERAL SECTION
- *Using a General Beam Section to Define the Section Behavior*

Optional parameters (if neither ELASTIC nor LINEAR is included, elastic-plastic response is assumed):

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the axial force–axial strain relationship, in addition to temperature. If this parameter is omitted, it is assumed that the axial force–axial strain relationship is constant or depends only on temperature. See *Material Data Definition* for more information.

ELASTIC

Include this parameter if the axial force–axial strain relationship is nonlinear but elastic.

LINEAR

Include this parameter if the axial force–axial strain relationship is linear.

Data lines if the LINEAR parameter is included:

First line:

1. Axial stiffness of the section.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the axial stiffness as a function of temperature and other predefined field variables.

Data lines if the LINEAR parameter is omitted:

First line:

1. Axial force.
2. Axial strain.
3. Temperature.
4. First field variable.

5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the axial force–axial strain relationship as a function of temperature and other predefined field variables.

B

This section describes each Abaqus input option that begins with B, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **BASELINE CORRECTION*
- **BASE MOTION*
- **BEAM ADDED INERTIA*
- **BEAM FLUID INERTIA*
- **BEAM GENERAL SECTION*
- **BEAM SECTION*
- **BEAM SECTION GENERATE*
- **BEAM SECTION OFFSET*
- **BIAXIAL TEST DATA*
- **BLOCKAGE*
- **BOUNDARY*
- **BRITTLE CRACKING*
- **BRITTLE FAILURE*
- **BRITTLE SHEAR*
- **BUCKLE*
- **BUCKLING ENVELOPE*
- **BUCKLING LENGTH*
- **BUCKLING REDUCTION FACTORS*
- **BULK VISCOSITY*

*BASELINE CORRECTION: Include baseline correction.

This option is used to modify an acceleration history to minimize the overall drift of the displacement obtained from the time integration of the given acceleration. It must appear immediately after the data lines of the *AMPLITUDE option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model or history data

Level: Model, Step

Abaqus/CAE: Amplitude toolset

References:

- *AMPLITUDE
- *Amplitude Curves*

There are no parameters associated with this option.

Data lines to define the correction intervals (optional; if no data lines are given, the baseline correction treats the entire time of the amplitude definition as a single correction interval):

First line:

1. Time point defining the end of the first correction interval and the beginning of the second correction interval.
2. Time point defining the end of the second correction interval and the beginning of the third correction interval.
3. Etc., up to eight values per line.

Repeat this data line as often as necessary. Each line (except for the last one) must have exactly eight time points.

*BASE MOTION: Define the base motion for linear, eigenmode-based, dynamic procedures.

This option is relevant only during linear dynamics procedures that use the natural modes of the system (*STEADY STATE DYNAMICS without the DIRECT parameter, *MODAL DYNAMIC, and *RANDOM RESPONSE).

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Natural Frequency Extraction*
- *Transient Modal Dynamic Analysis*
- *Mode-Based Steady-State Dynamic Analysis*
- *Random Response Analysis*
- *Transformed Coordinate Systems*

Required parameters:

DOF

Set this parameter equal to the direction (1–6, including rotations) for which the base motion is being defined. For primary base motion, this direction is always specified in the global coordinate system. For secondary base motion, you can specify the degree of freedom in either the global (default) or local direction at a node with applied coordinate transformations, depending on the value of the DIRECTION parameter.

Set this parameter equal to 8 for the acoustic degree of freedom, which is for use for secondary base motions only in the high-performance implementation of the modal analysis procedures.

Required parameter for *MODAL DYNAMIC and *STEADY STATE DYNAMICS analyses:

AMPLITUDE

Set this parameter equal to the name of the *AMPLITUDE option that defines the time history (*MODAL DYNAMIC) or frequency spectrum (*STEADY STATE DYNAMICS) of the motion. This parameter is irrelevant for the *RANDOM RESPONSE procedure. The parameter DEFINITION=SOLUTION DEPENDENT cannot be used in an *AMPLITUDE referenced by this option.

Optional parameters:

BASE NAME

Set this parameter equal to the name of the base if this base motion is to be applied to a secondary base. The base name is defined with the BASE NAME parameter on the *BOUNDARY option in the *FREQUENCY step.

DIRECTION

For primary base motion, the base motion is always applied in the direction of the global coordinate axes. For secondary base motion, use this parameter to define the coordinate system in which mechanical degrees of freedom (1–6) are applied.

Set DIRECTION=GLOBAL (default) to apply the base motion in the direction of the global coordinate axes. To specify secondary base motion in a global direction at a node at which a coordinate transformation is applied, you must specify all three translational degrees of freedom (1–3) or all three rotational degrees of freedom (4–6) in the named boundary condition in the frequency extraction procedure if the base motion is translational or rotational, respectively.

Set DIRECTION=LOCAL to apply the secondary base motion in the local direction along one of the coordinate axes of the transformed coordinate systems at the base nodes as defined by the **TRANSFORM* option.

LOAD CASE

Set this parameter equal to the load case number. This parameter is used in **RANDOM RESPONSE* analysis, where it is the cross-reference for the load case on the **CORRELATION* option.

SCALE

Set this parameter equal to the scale factor for the amplitude curve. The default is SCALE=1.0. This parameter applies during **MODAL DYNAMIC* and **STEADY STATE DYNAMICS* procedures.

TYPE

Set TYPE=ACCELERATION (default), VELOCITY, or DISPLACEMENT.

Optional, mutually exclusive parameters for steady-state dynamics analysis:

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the base motion record given by the amplitude definition.

REAL

Include this parameter (default) to define the real (in-phase) part of the base motion record given by the amplitude definition.

There are no data lines associated with this option unless a primary base motion defines rotation about a point that is not the origin of the coordinate system

Data line to define the center of rotation for a prescribed rotation:

First (and only) line:

1. *X*-coordinate of the point about which the rotation is applied.
2. *Y*-coordinate of the point about which the rotation is applied.
3. *Z*-coordinate of the point about which the rotation is applied.

This data line is relevant only for a primary base motion defined in the **MODAL DYNAMIC* and **STEADY STATE DYNAMICS* procedures.

*BEAM ADDED INERTIA: Define additional beam inertia.

This option is used in conjunction with the **BEAM SECTION* or **BEAM GENERAL SECTION* option to define additional mass and rotary inertia per unit length in shear flexible Timoshenko beam elements. This option is also used to define mass proportional damping (for direct-integration dynamic analysis) and in Abaqus/Standard composite damping (for modal dynamic analysis) associated with the added inertia.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance, Assembly

References:

- *Choosing a Beam Element*
- *Beam Section Behavior*

Optional parameters:

ALPHA

Set this parameter equal to the α_R factor to create inertia proportional damping for added inertia associated with this option when used in direct-integration dynamics. This value is ignored in modal dynamics. The default is ALPHA=0.0. (Units of T^{-1}.)

COMPOSITE

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the fraction of critical damping to be used with the beam elements when calculating composite damping factors for the modes when used in modal dynamics. The default is COMPOSITE=0.0.

This value is ignored in direct-integration dynamics. It is also ignored in mode-based analyses based on the SIM architecture, where the **COMPOSITE MODAL DAMPING* option should be used instead.

Data line to define additional beam inertia:

First line:

1. Mass per unit length.
2. Local 1-coordinate of the center of mass within the beam cross-section, x_1.
3. Local 2-coordinate of the center of mass within the beam cross-section, x_2.
4. Orientation angle for the first axis of the oriented system relative to the first beam cross-sectional direction in which the rotary inertia is given, α (in degrees). Only relevant for beams in space; otherwise, leave blank.
5. Rotary inertia around the center of mass about the 1-axis in the local inertia system, I_{11}.
6. Rotary inertia around the center of mass about the 2-axis in the local inertia system, I_{22}. Only relevant for beams in space; otherwise, leave blank.
7. Product of inertia, I_{12}. Only relevant for beams in space; otherwise, leave blank.

The rotary inertia should be given in units of ML. Abaqus does not use any specific physical units, so the user's choice must be consistent.

Repeat this set of data lines as often as necessary to define the additional beam inertia.

*BEAM FLUID INERTIA: Define additional beam inertia due to immersion in a fluid.

This option is used in conjunction with the *BEAM SECTION or *BEAM GENERAL SECTION option to include added inertia effects in Timoshenko beam elements due to immersion in an inviscid fluid.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Property module

References:

- *Beam Section Behavior*
- *Coupled Acoustic-Structural Analysis*
- *Loading due to an incident dilatational wave field*

Optional, mutually exclusive parameters:

FULL

Use this parameter to specify a fully submerged beam (default).

HALF

Use this parameter to specify a half-submerged beam.

Data line to define beam fluid inertia:

First (and only) line:

1. Mass density of fluid.
2. Local 1-coordinate of the center of the cylindrical cross-section with respect to the beam cross-section, *x*.
3. Local 2-coordinate of the center of the cylindrical cross-section with respect to the beam cross-section, *y*.
4. Radius of the cylindrical cross-section, *r*.
5. Added mass coefficient, C_A (default = 1.0), for lateral motions of the beam.
6. Added mass coefficient, C_{A-E} (default = 0.0), for motions along the axis of the beam. This coefficient affects only the term added to the free end(s) of the beam.

*BEAM GENERAL SECTION: Specify a beam section when numerical integration over the section is not required.

This option is used to define linear or nonlinear beam section response when numerical integration over the section is not required.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: General beam sections with linear response are supported in the *Property module*.

References:

- *Using a General Beam Section to Define the Section Behavior*
- *About Beam Modeling*

Required parameters:

ELSET

Set this parameter equal to the name of the element set for which the section is defined.

Required parameter in Abaqus/Explicit, optional parameter in Abaqus/Standard:

DENSITY

Set this parameter equal to the mass density (mass per unit volume) of the beam material. In an Abaqus/Standard analysis this parameter is needed only when the mass of the elements is required, such as in dynamic analysis or gravity loading. This parameter cannot be used when SECTION=MESHED.

If the MATERIAL parameter is omitted, this density accounts for the mass of the beam because no material definition is provided.

If the MATERIAL parameter is included, the mass density from the material definition is ignored.

If the DENSITY parameter is omitted, the mass density from the material definition is used.

Optional parameters:

CONTROLS

In an Abaqus/Explicit analysis, set this parameter equal to the name of a section controls definition (see **SECTION CONTROLS*) to be used to specify bulk viscosity scale factors or to an amplitude name (see **AMPLITUDE*) to ramp up the initial stress.

In an Abaqus/Standard analysis, set this parameter equal to the name of a section controls definition (see **SECTION CONTROLS*) to be used in a subsequent Abaqus/Explicit import analysis.

DEPENDENCIES

This parameter cannot be used when SECTION=NONLINEAR GENERAL, when SECTION=MESHED, or when the MATERIAL parameter is included.

Set this parameter equal to the number of field variable dependencies included in the definition of material moduli, in addition to temperature. If this parameter is omitted, it is assumed that the moduli are constant or depend only on temperature.

LUMPED

This parameter is relevant only for linear Timoshenko beam elements in Abaqus/Standard.

Set LUMPED=YES (default) to use a lumped mass matrix in frequency extraction and modal analysis procedures.

Set LUMPED=NO to use a mass matrix based on a cubic interpolation of deflection and quadratic interpolation of the rotation fields in frequency extraction and modal analysis procedures.

MATERIAL

This parameter cannot be used when SECTION=NONLINEAR GENERAL or SECTION=MESHED. It also cannot be used when the DEPENDENCIES or ZERO parameters are included.

Set this parameter equal to the name of the material to be used with this beam section definition.

POISSON

Set this parameter equal to the effective Poisson's ratio for the section to provide uniform strain in the section due to strain of the beam axis (so that the cross-sectional area changes when the beam is stretched). The value of the effective Poisson's ratio must be between −1.0 and 0.5. The default is POISSON=0. A value of 0.5 enforces incompressible behavior of the element.

For PIPE elements with SECTION=PIPE, this parameter is also used along with the Young's modulus given on the third data line to compute the axial strain due to hoop strain.

This parameter is used only in large-displacement analysis. It is not used with element types B23, B33, or the equivalent “hybrid” elements (which are available only in Abaqus/Standard).

ROTARY INERTIA

This parameter is relevant only for three-dimensional Timoshenko beam elements.

Set ROTARY INERTIA=EXACT (default) to use the exact rotary inertia corresponding to the beam cross-section geometry in dynamic and eigenfrequency extraction procedures.

Set ROTARY INERTIA=ISOTROPIC to use an approximate rotary inertia for the cross-section. In Abaqus/Standard the rotary inertia associated with the torsional mode of deformation is used for all rotational degrees of freedom. In Abaqus/Explicit the rotary inertia for all rotational degrees of freedom is equal to a scaled flexural inertia with a scaling factor chosen to maximize the stable time increment. ROTARY INERTIA=ISOTROPIC is not relevant and cannot be used when SECTION=MESHED; the default value of EXACT always applies for meshed sections.

SECTION

Set SECTION=GENERAL (default) to define a general beam section with linear response.

Set SECTION=NONLINEAR GENERAL to define general nonlinear behavior of the cross-section.

Set SECTION=MESHED to define an arbitrarily shaped solid cross-section meshed with warping elements.

Set this parameter equal to the name of a library section to choose a standard library section (see *Beam Cross-Section Library*). The following cross-sections are available:

- ARBITRARY, for an arbitrary section.
- BOX, for a rectangular, hollow box section.
- CHANNEL, for a channel section.
- CIRC, for a solid circular section.
- HAT, for a hat section.
- HEX, for a hollow hexagonal section.

- I, for an I-beam section.
- L, for an L-beam section.
- PIPE, for a hollow, circular section.
- RECT, for a solid, rectangular section.
- TRAPEZOID, for a trapezoidal section.

TAPER

This parameter is relevant only for Abaqus/Standard analyses.

It cannot be used when SECTION=MESHED.

Include this parameter to define a general beam section with a tapered cross-section.

ZERO

This parameter cannot be used when SECTION=MESHED or when the MATERIAL parameter is included.

Set this parameter equal to the reference temperature for thermal expansion (θ^0), if required. The default is ZERO=0.

Data lines for SECTION=GENERAL:

First line:

1. Area, *A*.
2. Moment of inertia for bending about the 1-axis, I_{11}.
3. Moment of inertia for cross bending, I_{12}.
4. Moment of inertia for bending about the 2-axis, I_{22}.
5. Torsional constant, *J*.
6. Sectorial moment, Γ_0. (Only needed in Abaqus/Standard when the section is associated with open-section beam elements.)
7. Warping constant, Γ_W. (Only needed in Abaqus/Standard when the section is associated with open-section beam elements.)

Second line (optional; enter a blank line if the default values are to be used):

1. First direction cosine of the first beam section axis.
2. Second direction cosine of the first beam section axis.
3. Third direction cosine of the first beam section axis.

The entries on this line must be (0, 0, -1) for planar beams. The default for beams in space is (0, 0, -1) if the first beam section axis is not defined by an additional node in the element's connectivity. See *Beam Element Cross-Section Orientation* for details.

Third line:

1. Young's modulus, *E*.
2. Shear modulus, *G*.
3. Coefficient of thermal expansion.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.

2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the properties as a function of temperature and other predefined field variables.

Data lines for SECTION=GENERAL if the TAPER parameter is included:

First line (properties of the first beam node):

1. Area, A.
2. Moment of inertia for bending about the 1-axis, I_{11}.
3. Moment of inertia for cross bending, I_{12}.
4. Moment of inertia for bending about the 2-axis, I_{22}.
5. Torsional constant, J.
6. Sectorial moment, Γ_0. (Only needed in Abaqus/Standard when the section is associated with open-section beam elements.)
7. Warping constant, Γ_W. (Only needed in Abaqus/Standard when the section is associated with open-section beam elements.)

Second line (properties of the second beam node):

1. Area, A.
2. Moment of inertia for bending about the 1-axis, I_{11}.
3. Moment of inertia for cross bending, I_{12}.
4. Moment of inertia for bending about the 2-axis, I_{22}.
5. Torsional constant, J.
6. Sectorial moment, Γ_0. (Only needed in Abaqus/Standard when the section is associated with open-section beam elements.)
7. Warping constant, Γ_W. (Only needed in Abaqus/Standard when the section is associated with open-section beam elements.)

Third line (optional; enter a blank line if the default values are to be used):

1. First direction cosine of the first beam section axis.
2. Second direction cosine of the first beam section axis.
3. Third direction cosine of the first beam section axis.

The entries on this line must be $(0, 0, -1)$ for planar beams. The default for beams in space is $(0, 0, -1)$ if the first beam section axis is not defined by an additional node in the element's connectivity. See *Beam Element Cross-Section Orientation* for details.

Fourth line:

1. Young's modulus, E.
2. Shear modulus, G.
3. Coefficient of thermal expansion.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the properties as a function of temperature and other predefined field variables.

Data lines for SECTION=NONLINEAR GENERAL:

First line:

1. Area, A.
2. Moment of inertia for bending about the 1-axis, I_{11}.
3. Moment of inertia for cross bending, I_{12}.
4. Moment of inertia for bending about the 2-axis, I_{22}.
5. Torsional constant, J.

The axial and bending behaviors of the section are defined by using the **AXIAL*, **M1*, **M2*, **TORQUE*, and **THERMAL EXPANSION* options.

Second line (optional):

1. First direction cosine of the first beam section axis.
2. Second direction cosine of the first beam section axis.
3. Third direction cosine of the first beam section axis.

The entries on this line must be $(0, 0, -1)$ for planar beams. The default for beams in space is $(0, 0, -1)$ if the first beam section axis is not defined by an additional node in the element's connectivity. See *Beam Element Cross-Section Orientation* for details.

Data lines for SECTION=MESHED:

First line:

1. First direction cosine of the first beam section axis.
2. Second direction cosine of the first beam section axis.
3. Third direction cosine of the first beam section axis.

The entries on this line must be $(0, 0, -1)$ for planar beams. The default for beams in space is $(0, 0, -1)$ if the first beam section axis is not defined by an additional node in the element's connectivity. See *Beam Element Cross-Section Orientation* for details. The elastic behavior and the mass and inertia properties are defined using the **SECTION STIFFNESS*, **SECTION INERTIA*, **TRANSVERSE SHEAR STIFFNESS*, **CENTROID*, and **SHEAR CENTER* options. The entries on the data lines for these options result from the two-dimensional meshed cross-section generation procedure. The options are written to the *jobname*`.bsp` file during the cross-section generation and are typically read into a subsequent beam analysis using the **INCLUDE* option. See *Meshed Beam Cross-Sections* for details.

Second line (optional):

You can define the elastic behavior on this line instead of using the **SECTION STIFFNESS* option, unless full coupling is defined.

1. Axial stiffness of the section, (EA).
2. Bending stiffness about the 1-axis of the section, $(EI)_{11}$.
3. Stiffness for cross-bending, $(EI)_{12}$.
4. Bending stiffness about the 2-axis of the section, $(EI)_{22}$.
5. Torsional constant, (GJ).

Third line (optional):

You can define the mass and inertia properties on this line instead of using the **SECTION INERTIA* option.

1. Total mass of the section per unit length, (ρA).
2. Rotary inertia about the 1-axis of the section, $(\rho I)_{11}$.
3. Rotary product of inertia, $(\rho I)_{12}$.
4. Rotary inertia about the 2-axis of the section, $(\rho I)_{22}$.
5. Local 1-coordinate of the center of mass, x_{1cm}.
6. Local 2-coordinate of the center of mass, x_{2cm}.

Data lines for BOX, CHANNEL, CIRC, HAT, HEX, I, L, PIPE, RECT, and TRAPEZOID sections:

First line:

1. Beam section geometric data. Values should be given as specified in *Beam Cross-Section Library* for the chosen section type.
2. Etc.

Second line (optional; enter a blank line if the default values are to be used):

1. First direction cosine of the first beam section axis.
2. Second direction cosine of the first beam section axis.
3. Third direction cosine of the first beam section axis.

The entries on this line must be (0, 0, -1) for planar beams. The default for beams in space is (0, 0, -1) if the first beam section axis is not defined by an additional node in the element's connectivity. For more information , see *Beam Element Cross-Section Orientation*.

Third line (only needed if the MATERIAL parameter is omitted):

1. Young's modulus, *E*.
2. Shear modulus, *G*.
3. Coefficient of thermal expansion.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the MATERIAL parameter is omitted and the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the properties as a function of temperature and other predefined field variables.

Data lines for BOX, CHANNEL, CIRC, HAT, HEX, I, L, PIPE, RECT, and TRAPEZOID sections if the TAPER parameter is included:

First line (properties of the first beam node):

1. Beam section geometric data. Values should be given as specified in *Beam Cross-Section Library* for the chosen section type.
2. Etc.

Second line (properties of the second beam node):

1. Beam section geometric data. Values should be given as specified in *Beam Cross-Section Library* for the chosen section type.
2. Etc.

Third line (optional; enter a blank line if the default values are to be used):

1. First direction cosine of the first beam section axis.
2. Second direction cosine of the first beam section axis.
3. Third direction cosine of the first beam section axis.

The entries on this line must be (0, 0, **−1**) for planar beams. The default for beams in space is (0, 0, **−1**) if the first beam section axis is not defined by an additional node in the element's connectivity. See *Beam Element Cross-Section Orientation* for details.

Fourth line (only needed if the MATERIAL parameter is omitted):

1. Young's modulus, *E*.
2. Shear modulus, *G*.
3. Coefficient of thermal expansion.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the MATERIAL parameter is omitted and the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the properties as a function of temperature and other predefined field variables.

Data lines for SECTION=ARBITRARY:

First line:

1. Number of segments making up the section.
2. Local 1-coordinate of first point defining the section.
3. Local 2-coordinate of first point defining the section.
4. Local 1-coordinate of second point defining the section.
5. Local 2-coordinate of second point defining the section.
6. Thickness of first segment.

Second line:

1. Local 1-coordinate of next section point.
2. Local 2-coordinate of next section point.
3. Thickness of segment ending at this point.

Repeat the second data line as often as necessary to define the ARBITRARY section.

Third line (optional; enter a blank line if the default values are to be used):

1. First direction cosine of the first beam section axis.
2. Second direction cosine of the first beam section axis.
3. Third direction cosine of the first beam section axis.

The entries on this line must be (0, 0, **−1**) for planar beams. The default for beams in space is (0, 0, **−1**) if the first beam section axis is not defined by an additional node in the element's connectivity. See *Beam Element Cross-Section Orientation* for details.

Fourth line (only needed if the MATERIAL parameter is omitted):

1. Young's modulus, *E*.
2. Shear modulus, *G*.

3. Coefficient of thermal expansion.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the MATERIAL parameter is omitted and the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the properties as a function of temperature and other predefined field variables.

*BEAM SECTION: Specify a beam section when numerical integration over the section is required.

This option is used to define the cross-section for beam elements when numerical integration over the section is required (usually because of nonlinear material response in the section).

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- *Using a Beam Section Integrated during the Analysis to Define the Section Behavior*
- *About Beam Modeling*
- *Pipes and Pipebends with Deforming Cross-Sections: Elbow Elements*

Required parameters:

ELSET

Set this parameter equal to the name of the element set for which this section is defined.

MATERIAL

Set this parameter equal to the name of the material to be used with this beam section definition.

SECTION

Set this parameter equal to the name of the section type (see *Beam Cross-Section Library*). The following cross-sections are available for beam elements:

- ARBITRARY, for an arbitrary section.
- BOX, for a rectangular, hollow box section.
- CIRC, for a solid circular section.
- HEX, for a hollow hexagonal section.
- I, for an I-beam section.
- L, for an L-beam section.
- PIPE, for a thin-walled circular section.
- RECT, for a solid, rectangular section.
- THICK PIPE, for a thick-walled circular section (Abaqus/Standard only).
- TRAPEZOID, for a trapezoidal section.

Set SECTION=ELBOW for elbow elements, which are available only in Abaqus/Standard.

Optional parameters:

CONTROLS

In an Abaqus/Explicit analysis, set this parameter equal to the name of a section controls definition (see **SECTION CONTROLS*) to be used to specify bulk viscosity scale factors or to an amplitude name (see **AMPLITUDE*) to ramp up the initial stress.

In an Abaqus/Standard analysis, set this parameter equal to the name of a section controls definition (see **SECTION CONTROLS*) to be used in a subsequent Abaqus/Explicit import analysis.

LUMPED

This parameter is relevant only for linear Timoshenko beam elements in Abaqus/Standard.

Set LUMPED=YES (default) to use a lumped mass matrix in frequency extraction and modal analysis procedures.

Set LUMPED=NO to use a mass matrix based on a cubic interpolation of deflection and quadratic interpolation of the rotation fields in frequency extraction and modal analysis procedures.

POISSON

Set this parameter equal to the effective Poisson's ratio for the section to provide uniform strain in the section because of strain of the beam axis (so that the beam changes cross-sectional area when it is stretched). The value of the effective Poisson's ratio must be between −1.0 and 0.5. The default is POISSON=0. A value of 0.5 will enforce incompressible behavior of the element.

This parameter is used only in large-displacement analyses. It is not used with elbow elements or with element types B23, B33, PIPE21, PIPE22, and the equivalent “hybrid” elements (which are available only in Abaqus/Standard).

ROTARY INERTIA

This parameter is relevant only for three-dimensional Timoshenko beam elements.

Set ROTARY INERTIA=EXACT (default) to use the exact rotary inertia corresponding to the beam cross-section geometry in dynamic and eigenfrequency extraction procedures.

Set ROTARY INERTIA=ISOTROPIC to use an approximate rotary inertia for the cross-section. In Abaqus/Standard the rotary inertia associated with the torsional mode of deformation is used for all rotational degrees of freedom. In Abaqus/Explicit the rotary inertia for all rotational degrees of freedom is equal to a scaled flexural inertia with a scaling factor chosen to maximize the stable time increment.

TEMPERATURE

Use this parameter to select the mode of temperature and field variable input used on the **FIELD*, the **INITIAL CONDITIONS*, or the **TEMPERATURE* options.

For beam elements set TEMPERATURE=GRADIENTS (default) to specify temperatures and field variables as values at the origin of the cross-section, together with gradients with respect to the 2-direction and, for beams in space, the 1-direction of the section. Set TEMPERATURE=VALUES to give temperatures and field variables as values at the points shown in the beam section descriptions (see *Beam Cross-Section Library*).

For elbow elements set TEMPERATURE=GRADIENTS (default) to specify temperatures and field variables at the middle of the pipe wall and the gradient through the pipe thickness. Set TEMPERATURE=VALUES to give temperatures and field variables as values at points through the section, as shown in *Pipes and Pipebends with Deforming Cross-Sections: Elbow Elements*.

Data lines for BOX, CIRC, HEX, I, L, PIPE, RECT, THICK PIPE, and TRAPEZOID sections:

First line:

1. Beam section geometric data. Values should be given as specified in *Beam Cross-Section Library* for the chosen section type. For solid circular sections only, you can specify a distribution name to define a spatial distribution for the beam radius.
2. Etc.

Second line (optional; enter a blank line if the default values are to be used):

1. First direction cosine of the first beam section axis.

2. Second direction cosine of the first beam section axis.
3. Third direction cosine of the first beam section axis.

The entries on this line must be (0, 0, −1) for planar beams. The default for beams in space is (0, 0, −1) if the first beam section axis is not defined by an additional node in the element's connectivity. See *Beam Element Cross-Section Orientation* for details.

Third line (optional):

1. Number of integration points in the first direction or branch. This number must be an odd number (for Simpson's integration), unless noted otherwise in *Beam Cross-Section Library*.
2. Number of integration points in the second direction or branch. This number must be an odd number (for Simpson's integration), unless noted otherwise in *Beam Cross-Section Library*. This entry is needed for the THICK PIPE section, as well as for beams in space.
3. Number of integration points in the third direction or branch. This number must be an odd number (for Simpson's integration), unless noted otherwise in *Beam Cross-Section Library*. This entry is needed only for I-beams.

Data lines for ARBITRARY sections:

First line:

1. Number of segments making up the section.
2. Local 1-coordinate of first point defining the section.
3. Local 2-coordinate of first point defining the section.
4. Local 1-coordinate of second point defining the section.
5. Local 2-coordinate of second point defining the section.
6. Thickness of first segment.

Second line:

1. Local 1-coordinate of next section point.
2. Local 2-coordinate of next section point.
3. Thickness of segment ending at this point.

Repeat the second data line as often as necessary to define the ARBITRARY section.

Third line (optional):

1. First direction cosine of the first beam section axis.
2. Second direction cosine of the first beam section axis.
3. Third direction cosine of the first beam section axis.

The entries on this line must be (0, 0, −1) for planar beams. The default for beams in space is (0, 0, −1) if the first beam section axis is not defined by an additional node in the element's connectivity. See *Beam Element Cross-Section Orientation* for details.

Data lines for ELBOW sections:

First line:

1. Outside radius of the pipe, r.
2. Pipe wall thickness, t.
3. Elbow torus radius, R, measured to the pipe axis. For a straight pipe, set $\boldsymbol{R = 0}$.

Second line:

Enter the coordinates of the point of intersection of the tangents to the straight pipe segments adjoining the elbow, or, if this section is associated with straight pipes, the coordinates of a point off the pipe axis. The second cross-sectional axis will lie in the plane thus defined, with its positive direction pointing toward this off-axis point.

1. First coordinate of the point.

2. Second coordinate of the point.
3. Third coordinate of the point.

Third line:

1. Number of integration points through the pipe wall thickness. This number must be an odd number. (The default is 5.)
2. Number of integration points around the pipe. (The default is 20.)
3. Number of ovalization modes around the pipe (maximum 6). The section can be used with 0 (zero) ovalization modes, in which case uniform radial expansion only is included.

*BEAM SECTION GENERATE: Generate beam section properties for a meshed cross-section.

This option is used to calculate the cross-section warping function, to define the centroid and shear center, and to generate the stiffness and inertia properties for a meshed cross-section. These properties are written to the file *jobname*`.bsp` for use in a subsequent beam analysis using the **BEAM GENERAL SECTION*, SECTION=MESHED option.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Meshed Beam Cross-Sections*
- *Beam Section Behavior*
- *Using a General Beam Section to Define the Section Behavior*

There are no parameters or data lines associated with this option.

*BEAM SECTION OFFSET: Define an offset for the beam cross-section origin.

This option can be used in conjunction with the **BEAM GENERAL SECTION* or **BEAM SECTION* option for the standard cross-sections available in the beam cross-section library (see *Beam Element Cross-Section Orientation* for details).

It is used to define an offset for the cross-section origin from the beam axis with respect to the default (that is, without an offset) local (1, 2) axis system.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- *Using a General Beam Section to Define the Section Behavior*
- *Using a Beam Section Integrated during the Analysis to Define the Section Behavior*
- **BEAM GENERAL SECTION*
- **BEAM SECTION*

There are no parameters associated with this option.

Data line to define the offset of the section origin:

First (and only) line:

1. Local x_1-offset of the origin, x_{1o}. The default is 0.
2. Local x_2-offset of the origin, x_{2o}. The default is 0.

*BIAXIAL TEST DATA: Used to provide biaxial test data (compression and/or tension).

This option is used to provide biaxial test data. It can be used only in conjunction with the *HYPERELASTIC option, the *HYPERFOAM option, and the *MULLINS EFFECT option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Hyperelastic Behavior of Rubberlike Materials*
- *Hyperelastic Behavior in Elastomeric Foams*
- *Stress Softening in Rubberlike Materials*
- *Stress Softening in Elastomeric Foams*
- *HYPERELASTIC
- *HYPERFOAM
- *MULLINS EFFECT

Using biaxial test data to define a hyperelastic material

Optional parameters:

SMOOTH

Include this parameter to apply a smoothing filter to the stress-strain data. If the parameter is omitted, no smoothing is performed.

Set this parameter equal to the number n such that $2n + 1$ is equal to the total number of data points in the moving window through which a cubic polynomial is fit using the least-squares method. n should be larger than 1. The default is SMOOTH=3.

Optional parameter when the *BIAXIAL TEST DATA option is used in conjunction with the *HYPERELASTIC, MARLOW option or the *HYPERELASTIC, VALANIS-LANDEL option:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the test data. If this parameter is omitted, it is assumed that the test data depend only on temperature.

Data lines to specify biaxial test data for hyperelasticity other than the Marlow model or the Valanis-Landel model (the nominal strains must be arranged in either ascending or descending order if the SMOOTH parameter is used):

First line:

1. Nominal stress, T_B.

2. Nominal strain, ϵ_B.

Repeat this data line as often as necessary to give the stress-strain data.

Data lines to specify biaxial test data for the Marlow model or the Valanis-Landel model (the nominal strains must be arranged in ascending order if the SMOOTH parameter is used):

First line:

1. Nominal stress, T_B.
2. Nominal strain, ϵ_B.
3. Nominal lateral strain, ϵ_3. Available only for the Marlow model. Leave blank for Valanis-Landel. Not needed if the POISSON parameter is specified on the **HYPERELASTIC* option or if the **VOLUMETRIC TEST DATA* option is used (these are the only options available to define compressible behavior for the Valanis-Landel model with biaxial test data).
4. Temperature, θ.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the test data as a function of temperature and other predefined field variables. Nominal strains and nominal stresses must be given in ascending order.

Using biaxial test data to define an elastomeric foam

There are no parameters associated with this option.

Data lines to specify biaxial test data for a hyperfoam:

First line:

1. Nominal stress, T_L.
2. Nominal strain, ϵ_B.
3. Nominal transverse strain, ϵ_3. Default is zero. Not needed if the POISSON parameter is specified on the **HYPERFOAM* option.

Repeat this data line as often as necessary to give the stress-strain data.

Using biaxial test data to define the Mullins effect material model

There are no parameters associated with this option.

Data lines to specify biaxial test data for defining the unloading-reloading response of the Mullins effect material model:

First line:

1. Nominal stress, T_L.
2. Nominal strain, ϵ_B.

Repeat this data line as often as necessary to give the stress-strain data.

*BLOCKAGE: Control contacting surfaces for blockage.

This option is used to control the combination of surfaces that can cause blockage of flow out of a surface-based fluid cavity. It must be used in conjunction with the *SURFACE INTERACTION option.

Products: Abaqus/Explicit

Type: Model or history data

Level: Model, Step

References:

- *Fluid Exchange Definition*
- *About Mechanical Contact Properties*
- *Contact Blockage*
- *SURFACE INTERACTION
- *FLUID EXCHANGE ACTIVATION

There are no parameters or data lines associated with this option.

*BOUNDARY: Specify boundary conditions.

This option is used to prescribe boundary conditions at nodes or to specify the driven nodes in a submodeling analysis. In Abaqus/Standard it is also used to define primary and secondary bases for modal superposition procedures and to prescribe boundary conditions at phantom nodes for enriched elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Model, Step

Abaqus/CAE: *Load module*; fluid cavity pressure and generalized plane strain boundary conditions are not supported.

References:

- *Abaqus Model Definition*
- *Boundary Conditions*
- *DISP*
- *VDISP*
- *Natural Frequency Extraction*
- *Node-Based Submodeling*
- *Modeling Discontinuities as an Enriched Feature Using the Extended Finite Element Method*
- *Defining ALE Adaptive Mesh Domains in Abaqus/Explicit*

Prescribing boundary conditions at nodes

No parameters are used when fixed boundary conditions are specified as model data:

Optional parameters (history data only):

AMPLITUDE

This parameter is ignored in a one-step inverse procedure.

This parameter is relevant only when some of the variables being prescribed have nonzero magnitudes. Set this parameter equal to the name of the amplitude curve defining the magnitude of the prescribed boundary conditions (*Amplitude Curves*).

If this parameter is omitted in an Abaqus/Standard analysis, either the reference magnitude is applied linearly over the step (a RAMP function) or it is applied immediately at the beginning of the step and subsequently held constant (a STEP function). The choice of RAMP or STEP function depends on the value assigned to the AMPLITUDE parameter on the **STEP* option (*Defining an Analysis*). There are two exceptions. The first is when displacement or rotation components are given with TYPE=DISPLACEMENT, for which the default is always a RAMP function. The second is when displacement or rotation components in a static step or in a dynamic step with APPLICATION=QUASI-STATIC are given with TYPE=VELOCITY, for which the default is always a STEP function.

If this parameter is omitted in an Abaqus/Explicit analysis, the reference magnitude is applied immediately at the beginning of the step and subsequently held constant (a STEP function).

In an Abaqus/Standard dynamic or modal dynamic procedure, amplitude curves specified for TYPE=DISPLACEMENT or TYPE=VELOCITY will be smoothed automatically. In an Abaqus/Explicit analysis, the user must request that such amplitude curves are smoothed. For more information, see *Amplitude Curves*.

BLOCKING

This parameter applies only to Abaqus/Explicit analyses when the USER parameter is specified.

Set BLOCKING=YES (default) to enable blocking for a given node set. The blocking size will be set to a predefined value in Abaqus/Explicit.

Set BLOCKING=NO to disable blocking.

FIXED

This parameter applies only to Abaqus/Standard analyses and cannot be used with the TYPE and USER parameters.

Include this parameter to indicate that the values of the variables being prescribed with this **BOUNDARY* option remain fixed throughout the step using the values at the start of the step. If this parameter is used, any magnitudes given on the data lines are ignored.

This parameter should be used only with the "direct" format to specify the degrees of freedom to fix (see *Boundary Conditions*).

Except for a one-step inverse procedure, this parameter is ignored if it is used in the first step of an analysis. If this parameter is used for a one-step inverse procedure, all the degrees of freedom of the nodes given on the data lines are constrained based on their initial reference configuration.

LOAD CASE

This parameter applies only to Abaqus/Standard analyses. It is ignored in all procedures except **BUCKLE*.

Set this parameter equal to 1 (default) or 2. LOAD CASE=1 can be used to define boundary conditions for the applied loads, and LOAD CASE=2 can be used to define antisymmetry boundary conditions for the buckling modes.

NAME

This parameter applies only to Abaqus/Explicit analyses when the USER parameter is specified.

Set this parameter equal to the name that will be used to reference the boundary condition in user subroutine *VDISP*. Boundary names that appear in an Abaqus/Explicit analysis must be unique. They cannot begin with a number, and they must adhere to the naming convention for labels. See *Input Syntax Rules* for the syntax of such names.

OP

This parameter is ignored in a one-step inverse procedure.

Set OP=MOD (default) to modify existing boundary conditions or to add boundary conditions to degrees of freedom that were previously unconstrained.

Set OP=NEW if all boundary conditions that are currently in effect should be removed. To remove only selected boundary conditions, use OP=NEW and respecify all boundary conditions that are to be retained.

If a boundary condition is removed in a stress/displacement analysis in Abaqus/Standard, it will be replaced by a concentrated force equal to the reaction force calculated at the restrained degree of freedom at the end of the previous step. If the step is a general nonlinear analysis step, this

concentrated force will then be removed according to the AMPLITUDE parameter on the **STEP* option. Therefore, if the default amplitudes are used, the concentrated force will be reduced linearly to zero over the period of the step in a static analysis and immediately in a dynamic analysis.

The OP parameter must be the same for all uses of the **BOUNDARY* option within a single step except in a **BUCKLE* step, where OP=NEW can be used with LOAD CASE=2 even when OP=MOD is used with LOAD CASE=1.

PHANTOM

This parameter applies only to enriched elements in Abaqus/Standard.

Set PHANTOM=NODE to apply boundary conditions to a phantom node that is originally located coincident with the specified real node in an enriched element.

Set PHANTOM=EDGE to apply boundary conditions to a phantom node located at an element edge between the two specified real corner nodes in an enriched element. This setting applies only to nodes with pore pressure degrees of freedom.

Set PHANTOM=INCLUDED to indicate that the boundary conditions applied to a phantom node located at an element edge will be interpolated automatically from the specified real corner nodes when the enriched element is cracked. This setting applies only to nodes with pore pressure degrees of freedom.

REGION TYPE

This parameter applies only to Abaqus/Explicit analyses.

This parameter is relevant only for boundary conditions applied to nodes on the boundary of an adaptive mesh domain. If boundary conditions are applied to nodes in the interior of an adaptive mesh domain, these nodes will always follow the material. Abaqus/Explicit will create a Lagrangian boundary region automatically for surface-type constraints (symmetry planes, moving boundary planes, and fully clamped boundaries).

Set REGION TYPE=LAGRANGIAN (default) to apply the boundary conditions to a Lagrangian boundary region. The edge of a Lagrangian boundary region will follow the material while allowing adaptive meshing along the edge and in the interior of the region.

Set REGION TYPE=SLIDING to define a sliding boundary region. The edge of a sliding boundary region will slide over the material. Adaptive meshing will occur on the edge and in the interior of the region. Mesh constraints are typically applied on the edge of a sliding boundary region to fix it spatially.

Set REGION TYPE=EULERIAN to apply the boundary conditions to an Eulerian boundary region. This option is used to create a boundary region across which material can flow and is typically used with velocity boundary conditions. Mesh constraints must be used normal to an Eulerian boundary region to allow material to flow through the region. If no mesh constraints are applied, an Eulerian boundary region will behave in the same way as a sliding boundary region.

TYPE

This parameter cannot be used with the FIXED parameter.

This parameter is used in a stress/displacement analysis to specify whether the magnitude is in the form of a displacement history, a velocity history, or an acceleration history. In an Abaqus/Standard analysis TYPE=VELOCITY should normally be used to specify finite rotations.

Set TYPE=DISPLACEMENT (default) to give a displacement history. Abaqus/Explicit does not admit jumps in displacement. If no amplitude is specified, Abaqus/Explicit will ignore the user-supplied displacement value and enforce a zero displacement boundary condition. See *Boundary Conditions* for details.

Set TYPE=VELOCITY to give a velocity history. Velocity histories can be specified in static analyses in Abaqus/Standard, as discussed in "Prescribing large rotations" in *Boundary Conditions*. In this case the default variation is STEP.

Set TYPE=ACCELERATION to give an acceleration history. Acceleration histories should not be used in static analysis steps in Abaqus/Standard.

If amplitude functions are specified as piecewise linear functions in Abaqus/Explicit and a displacement history is used, there will be a jump in the velocity and a spike in the acceleration at points on the curve where the curve changes slope. This will result in a "noisy" solution. If possible, use **AMPLITUDE*, DEFINITION=SMOOTH STEP; **AMPLITUDE*, SMOOTH; or **BOUNDARY*, TYPE=VELOCITY or TYPE=ACCELERATION. For TYPE=ACCELERATION the value of the initial velocity (given in **INITIAL CONDITIONS*, TYPE=VELOCITY) must be specified to obtain the correct displacement history.

USER

This parameter cannot be used with the FIXED parameter.

For Abaqus/Standard include this parameter to indicate that any nonzero magnitudes associated with variables prescribed through this option can be redefined in user subroutine *DISP*. Any magnitudes defined on the data lines of the option (and possibly modified by the AMPLITUDE parameter) will be passed into user subroutine *DISP* and can be redefined in subroutine *DISP*. The value of the TYPE parameter is ignored when this option is used.

For Abaqus/Explicit include this parameter to indicate that the boundary value associated with variables prescribed through this option are to be defined in user subroutine *VDISP*. Any magnitudes defined on the data lines of the option are ignored and the amplitude, if the AMPLITUDE parameter is included, is passed into the *VDISP* routine for your usage. The type of user prescribed variable in subroutine *VDISP* is determined by the TYPE parameter. The NAME parameter can be used in user subroutine *VDISP* to distinguish multiple boundary conditions. Only translational and rotational degrees of freedom are supported for user-prescribed boundary conditions.

Optional, mutually exclusive parameters for matrix generation and direct-solution, steady-state dynamics analysis (history data only):

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the boundary condition.

REAL

Include this parameter (default) to define the real (in-phase) part of the part of the boundary condition.

Data lines to define zero-valued boundary conditions using the "type" format (model data only):

First line:

1. Node number or node set label.
2. Label specifying the type of boundary condition to be applied (see *Boundary Conditions*). Only one type specification can be used per line.

Repeat this data line as often as necessary to specify fixed boundary conditions at different nodes and degrees of freedom.

Data lines to prescribe boundary conditions using the "direct" format when the PHANTOM parameter is not used:

First line:

1. Node number or node set label.

2. First degree of freedom constrained. For a definition of the numbering of degrees of freedom in Abaqus, see *Conventions*.
3. Last degree of freedom constrained. This field can be left blank if only one degree of freedom is being constrained.

The following data item is necessary only when nonzero boundary conditions are specified as history data. Any magnitude given will be ignored when the boundary conditions are given as model data.

4. Actual magnitude of the variable (displacement, velocity, or acceleration, etc.). This magnitude will be modified by an amplitude specification if the AMPLITUDE parameter is used. If this magnitude is a rotation, it must be given in radians. If TYPE=DISPLACEMENT in an Abaqus/Explicit analysis and no AMPLITUDE specification is provided, this value will be ignored (see *Boundary Conditions*). In Abaqus/Standard the magnitude can be redefined in user subroutine *DISP* if the USER parameter is included. In Abaqus/Explicit the magnitude will be redefined in user subroutine *VDISP* if the USER parameter is included. In this case the input magnitude will be ignored.

Repeat this data line as often as necessary to specify boundary conditions at different nodes and degrees of freedom.

Data lines to prescribe boundary conditions using the "direct" format when PHANTOM=NODE:

First line:

1. Node number of a real node that is originally located coincident with the phantom node.
2. First degree of freedom constrained. For a definition of the numbering of degrees of freedom in Abaqus, see *Conventions*.
3. Last degree of freedom constrained. This field can be left blank if only one degree of freedom is being constrained.

The following data item is necessary only when nonzero boundary conditions are specified as history data. Any magnitude given will be ignored when the boundary conditions are given as model data.

4. Actual magnitude of the variable (displacement, velocity, or acceleration, etc.). This magnitude will be modified by an amplitude specification if the AMPLITUDE parameter is used. If this magnitude is a rotation, it must be given in radians. The magnitude can be redefined in user subroutine *DISP* if the USER parameter is included.

Repeat this data line as often as necessary to specify boundary conditions at different nodes and degrees of freedom.

Data lines to prescribe boundary conditions using the "direct" format when PHANTOM=EDGE:

First line:

1. Node number of one of the two real corner nodes to define an edge in an enriched element.
2. Node number of the other real corner node to define an edge in an enriched element.
3. First degree of freedom constrained. For a definition of the numbering of the degrees of freedom in Abaqus, see *Conventions*.
4. Last degree of freedom constrained. This field can be left blank if only one degree of freedom is being constrained.

The following data item is necessary only when nonzero boundary conditions are specified as history data. Any magnitude given will be ignored when the boundary conditions are given as model data.

5. Actual magnitude of the variable (displacement, velocity, or acceleration, etc.). This magnitude will be modified by an amplitude specification if the AMPLITUDE parameter is used. If this magnitude is a rotation, it must be given in radians. The magnitude can be redefined in user subroutine *DISP* if the USER parameter is included.

Repeat this data line as often as necessary to specify boundary conditions at different nodes and degrees of freedom.

Data lines to prescribe boundary conditions using the "direct" format when PHANTOM=INCLUDED:

First line:

1. Node number or node set label.
2. First degree of freedom constrained. For a definition of the numbering of degrees of freedom in Abaqus, see *Conventions*.
3. Last degree of freedom constrained. This field can be left blank if only one degree of freedom is being constrained.

The following data item is necessary only when nonzero boundary conditions are specified as history data. Any magnitude given will be ignored when the boundary conditions are given as model data.

4. Actual magnitude of the variable (displacement, velocity, or acceleration, etc.). This magnitude will be modified by an amplitude specification if the AMPLITUDE parameter is used. If this magnitude is a rotation, it must be given in radians. The magnitude can be redefined in user subroutine *DISP* if the USER parameter is included.

Repeat this data line as often as necessary to specify boundary conditions at different nodes and degrees of freedom.

Defining primary and secondary bases for modal superposition procedures

Optional parameters:

BASE NAME

This parameter is used to define a secondary base and can be used only in a frequency extraction step (*Natural Frequency Extraction*). Set this parameter equal to the name of a secondary base (*About Dynamic Analysis Procedures*). In subsequent modal superposition steps this base will be excited as specified by the **BASE MOTION* option that refers to the same base name. If this parameter is not used in a frequency extraction step, the nodes will be assigned to the primary base.

Data lines to define a primary or a secondary base within a *FREQUENCY procedure:

First line:

1. Node number or node set label.
2. First degree of freedom constrained. For a definition of the numbering of degrees of freedom in Abaqus/Standard, see *Conventions*.
3. Last degree of freedom constrained. This field can be left blank if only one degree of freedom is being constrained.

Repeat this data line as often as necessary to specify boundary conditions at different nodes and degrees of freedom.

Submodel boundary conditions

Required parameters:

STEP

Set this parameter equal to the step number in the global analysis for which the values of the driven variables will be read during this step of the submodel analysis.

SUBMODEL

Include this parameter to specify that the boundary conditions are the “driven variables” in a submodel analysis. Nodes used in this option must be listed in the **SUBMODEL* model definition option.

Optional parameters:

INC

This parameter can be used only in a static linear perturbation step (*General and Perturbation Procedures*). Set this parameter equal to the increment in the selected step of the global analysis at which the solution will be used to specify the values of the driven variables. By default, Abaqus/Standard will use the solution at the last increment of the selected step.

OP

Set OP=MOD (default) for existing **BOUNDARY* conditions to remain, with this option defining boundary conditions to be added or modified.

Set OP=NEW if all boundary conditions that are currently in effect should be removed. To remove only selected boundary conditions, use OP=NEW and respecify all boundary conditions that are to be retained.

If a boundary condition is removed in a stress/displacement analysis, it will be replaced by a concentrated force equal to the reaction force calculated at the restrained degree of freedom at the end of the previous step. If the step is a general nonlinear analysis step, this concentrated force will then be removed according to the AMPLITUDE parameter on the **STEP* option. Therefore, by default the concentrated force will be reduced linearly to zero over the period of the step in a static analysis and immediately in a dynamic analysis.

The OP parameter must be the same for all uses of the **BOUNDARY* option in a step.

SCALE

Set this parameter equal to the value by which the driven variables read from the global analysis are to be scaled. The default is SCALE=1.0.

TIMESCALE

If the submodel analysis step time is different from the global analysis step time, use the TIMESCALE parameter to adjust the time variable for the driven nodes' amplitude functions. The time variable of each driven node's amplitude function is scaled to match the submodel analysis step time. If this parameter is omitted, the time variable is not scaled.

Data lines for shell-to-shell or solid-to-solid submodeling:

First line:

1. Node number or node set label.
2. First degree of freedom constrained. For a definition of the numbering of degrees of freedom in Abaqus/Standard and Abaqus/Explicit, see *Conventions*.
3. Last degree of freedom constrained. This field can be left blank if only one degree of freedom is being constrained.

Repeat this data line as often as necessary to specify submodel boundary conditions at different nodes and degrees of freedom.

Data lines for shell-to-solid submodeling:

First line:

1. Node number or node set label.
2. Thickness of the center zone size around the shell midsurface (given in the units of the model). If this value is omitted, a default value of 10% of the shell thickness specified on the **SUBMODEL* option is used. If more than one **SUBMODEL* option is used, the default value is 10% of the maximum thickness specified on any of the **SUBMODEL* options.

Repeat this data line as often as necessary to specify submodel boundary conditions at different nodes.

Data lines for acoustic-to-structure submodeling:

First line:

1. Node number or node set label.
2. The pressure degree of freedom constrained (8).

Repeat this data line as often as necessary to specify submodel boundary conditions at different nodes.

*BRITTLE CRACKING: Define brittle cracking properties.

This option is used to define cracking and postcracking properties for the brittle cracking material model. The **BRITTLE CRACKING* option must be used in conjunction with the **BRITTLE SHEAR* option and must immediately precede it. The **BRITTLE CRACKING* option can be used in conjunction with the **BRITTLE FAILURE* option to specify a brittle failure criterion.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Cracking Model for Concrete*
- **BRITTLE FAILURE*
- **BRITTLE SHEAR*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the postcracking behavior, in addition to temperature. If this parameter is omitted, it is assumed that the postcracking behavior depends only on temperature. See *Material Data Definition* for more information.

TYPE

Set TYPE=STRAIN (default) to specify the postcracking behavior by entering the postfailure stress-strain relationship directly.

Set TYPE=DISPLACEMENT to define the postcracking behavior by entering the postfailure stress/displacement relationship directly.

Set TYPE=GFI to define the postcracking behavior by entering the failure stress, σ_{tu}^{I}, and the Mode I fracture energy, G_{f}^{I}.

Data lines if the TYPE=STRAIN parameter is included (default):

First line:

1. Remaining direct stress after cracking, σ_{t}^{I}. (Units of $\mathbf{FL^{-2}}$.)
2. Direct cracking strain, e_{nn}^{ck}.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

The first point at each value of temperature must have a cracking strain of 0.0 and gives the failure stress value.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.

2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the postcracking behavior on temperature and other predefined field variables.

Data lines if the TYPE=DISPLACEMENT parameter is included:

First line:

1. Remaining direct stress after cracking, σ_t^I. (Units of $\mathbf{FL^{-2}}$.)
2. Direct cracking displacement, u_n^{ck}. (Units of L.)
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

The first point at each value of temperature must have a cracking displacement of 0.0 and gives the failure stress value.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the postcracking behavior on temperature and other predefined field variables.

Data lines if the TYPE=GFI parameter is included:

First line:

1. Failure stress, σ_{tu}^I. (Units of $\mathbf{FL^{-2}}$.)
2. Mode I fracture energy, G_f^I. (Units of $\mathbf{FL^{-1}}$.)
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the postcracking behavior on temperature and other predefined field variables.

*BRITTLE FAILURE: Specify brittle failure criterion.

This option is used with the brittle cracking material model to specify brittle failure of the material. It must be used in conjunction with the *BRITTLE CRACKING and the *BRITTLE SHEAR options.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Cracking Model for Concrete*
- *BRITTLE CRACKING
- *BRITTLE SHEAR

Optional parameters:

CRACKS

Set CRACKS=1 (default) to indicate that an element will be removed when any local direct cracking strain (or displacement) component reaches the failure value.

Set CRACKS=2 to indicate that an element will be removed when any two direct cracking strain (or displacement) components reach the failure value.

Set CRACKS=3 to indicate that an element will be removed when all three possible direct cracking strain (or displacement) components reach the failure value.

The value for the CRACKS parameter can only be 1 for beam or truss elements. It cannot be greater than 2 for plane stress and shell elements, and it cannot be greater than 3 for three-dimensional, plane strain, and axisymmetric elements.

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the failure criterion, in addition to temperature. If this parameter is omitted, it is assumed that the failure criterion depends only on temperature. See *Material Data Definition* for more information.

Data lines if TYPE=STRAIN (default) is used on the *BRITTLE CRACKING option:

First line:

1. Direct cracking failure strain, $\left(\tilde{\varepsilon}_{nn}^{ck}\right)_f$.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the postcracking behavior on temperature and other predefined field variables.

Data lines if TYPE=DISPLACEMENT or GFI is included on the *BRITTLE CRACKING option:

First line:

1. Direct cracking failure displacement, $\left(u_n^{ck}\right)_f$. (Units of L.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the postcracking behavior on temperature and other predefined field variables.

*BRITTLE SHEAR: Define the postcracking shear behavior of a material used with the brittle cracking model.

This option is used to define the postcracking shear behavior of a material used in a brittle cracking model. The **BRITTLE SHEAR* option must be used with the **BRITTLE CRACKING* option and must immediately follow it. The **BRITTLE SHEAR* option can be used in conjunction with the **BRITTLE FAILURE* option to specify a brittle failure criterion.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Cracking Model for Concrete*
- **BRITTLE CRACKING*
- **BRITTLE FAILURE*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the cracked shear behavior, in addition to temperature. If this parameter is omitted, it is assumed that the parameters defining cracked shear behavior are constant or depend only on temperature. See *Material Data Definition* for more information.

TYPE

Set TYPE=RETENTION FACTOR (default) to specify the postcracking shear behavior by entering the shear retention factor–crack opening strain relationship directly.

Set TYPE=POWER LAW to specify the postcracking shear behavior by entering the material parameters p and $\bar{\epsilon}^{ck}_{max}$ for the power law shear retention model.

Data lines if the TYPE=RETENTION FACTOR parameter is included (default):

First line:

1. Shear retention factor, ρ.
2. Crack opening strain, $\bar{\epsilon}^{ck}_{nn}$.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

The first point at each value of temperature must have a retention factor of 1.0 and a cracking strain of 0.0.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the postcracking shear behavior on temperature and other predefined field variables.

Data lines if the TYPE=POWER LAW parameter is included:

First line:

1. $\bar{\varepsilon}^{ck}_{max}$.
2. p.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the postcracking shear behavior on temperature and other predefined field variables.

*BUCKLE: Obtain eigenvalue buckling estimates.

This option is used to control eigenvalue buckling estimation.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

• *Eigenvalue Buckling Prediction*

Optional parameters:

EIGENSOLVER

Use this parameter to choose the eigensolver.

Set EIGENSOLVER=SUBSPACE (default) to invoke the subspace iteration eigensolver.

Set EIGENSOLVER=LANCZOS to invoke the Lanczos eigensolver.

Data line for an eigenvalue buckling analysis when EIGENSOLVER=SUBSPACE:

First (and only) line:

1. Number of eigenvalues to be estimated.
2. Maximum eigenvalue of interest. If this field is left blank, no maximum is set.
3. Number of vectors used in the iteration. Abaqus/Standard usually determines this number, but you can change the number using this entry. In general, the convergence in solving the eigenproblem is more rapid if more vectors are carried in the iteration; therefore, use this data field if past experience suggests that the convergence is slow for a particular type of buckling problem. If the number of eigenvalues requested is n, the default number of vectors used is the minimum of ($2n$, n+8).
4. Maximum number of iterations. The default is 30.

Data line for an eigenvalue buckling analysis when EIGENSOLVER=LANCZOS:

First (and only) line:

1. Number of eigenvalues to be estimated. If the evaluation of all the eigenvalues in the given range is desired, enter the maximum number of expected eigenmodes.
2. Minimum eigenvalue of interest. Providing this value directs the Lanczos solution procedure to find eigenvalues near this value with minimal computational effort. Lower eigenvalues might also be found with no additional computational cost. If this field is left blank, no minimum is set.
3. Maximum eigenvalue of interest. Providing this value directs the Lanczos solution procedure to find eigenvalues near this value with minimal computational effort. Higher eigenvalues might also be found with no additional computational cost. If this field is left blank, no maximum is set.
4. Block size. If this entry is omitted, a default value, which is usually appropriate, is created.
5. Maximum number of block Lanczos steps within each Lanczos run. If this entry is omitted, a default value, which is usually appropriate, is created.

*BUCKLING ENVELOPE: Define a nondefault buckling envelope for buckling strut response of frame elements with PIPE sections.

This option is used to define the coefficients characterizing the buckling strut envelope for the buckling strut response of frame elements. It can be used in conjunction with the *FRAME SECTION, SECTION=PIPE, YIELD STRESS=σ^0 option with or without the PINNED parameter.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- *Frame Section Behavior*
- *FRAME SECTION

There are no parameters associated with this option.

Data line to define the buckling strut envelope:

First (and only) data line:

1. ξ, coefficient defining $P_y = \xi \sigma^0 A$ (default value 0.95).
2. γ, coefficient defining the isotropic hardening slope γEA (default value 0.02).
3. α_0, coefficient defining $\alpha = \alpha_0 + \alpha_1 \frac{L}{D}$ (default value 0.03).
4. α_1, coefficient defining $\alpha = \alpha_0 + \alpha_1 \frac{L}{D}$ (default value 0.004).
5. κ, coefficient defining compressive force for discontinuity in buckling envelope (default value 0.28).
6. β, buckling envelope slope coefficient (default value 0.02).
7. ζ, coefficient defining the force axis intercept point (default value $\min\left(1.0, \frac{5.8}{\xi}\left(\frac{t}{D}\right)^{0.7}\right)$).

In the above data line A is the cross-section area, σ^0 is a yield stress value, E is Young's modulus, L is the element length, D is the outer pipe diameter, and t is the pipe wall thickness.

*BUCKLING LENGTH: Define buckling length data for buckling strut response of frame elements with PIPE sections.

This option is used to define two sets of coefficients used in the ISO equation that predicts P_{cr} for frame elements with buckling strut response. For a user-defined buckling envelope it can be used only in conjunction with both the *FRAME SECTION, SECTION=PIPE, YIELD STRESS=σ^0 option and the *BUCKLING ENVELOPE option. For the default buckling envelope it can be used only in conjunction with the *FRAME SECTION, BUCKLING, SECTION=PIPE, YIELD STRESS=σ^0 option, with or without the PINNED parameter.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- *Frame Section Behavior*
- *BUCKLING ENVELOPE*
- *FRAME SECTION*

There are no parameters associated with this option.

Data line to define the buckling length coefficients:

First (and only) data line:

1. Effective length factor in the first cross-section direction.
2. Effective length factor in the second cross-section direction.
3. Added length in the first cross-section direction.
4. Added length in the second cross-section direction.

*BUCKLING REDUCTION FACTORS: Define buckling reduction factors for buckling strut response of frame elements with PIPE sections.

This option is used to define two coefficients used in the ISO equation, which predicts P_{cr}, the axial load at which the response switches to buckling only, for frame elements with buckling strut response. For a nondefault buckling envelope the *BUCKLING REDUCTION FACTORS option can be used only in conjunction with both the *FRAME SECTION, SECTION=PIPE, YIELD STRESS=σ^0 option and the *BUCKLING ENVELOPE option. For the default buckling envelope it can be used only in conjunction with the *FRAME SECTION, BUCKLING, SECTION=PIPE, YIELD STRESS=σ^0 option.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance, Model

References:

- *Frame Elements*
- *Frame Section Behavior*
- *Buckling strut response for frame elements*

Optional parameters:

AXIS1

Include this parameter to define the method for calculating the buckling reduction factor c_{m1} for bending about the first cross-section direction.

Set AXIS1=TYPE1 (default) to set c_{m1} to the constant value of 0.85.

Set AXIS1=TYPE2 for members with no distributed transverse loading. Then

c_{m1}=max$(0.6 - 0.4M_1/M_2, 0.4)$, where M_1/M_2 is the ratio of smaller to larger moments about the first cross-section axis at the element ends.

Set AXIS1=TYPE3 for members with distributed transverse loading. Then

c_{m1}=min$(1.0 - 0.4f_c/F_{e1}, 0.85)$, where f_c is the compressive axial stress and F_{e1} is the Euler buckling stress corresponding to the first cross-section direction.

AXIS2

Include this parameter to define the method for calculating the buckling reduction factor c_{m2} for bending about the second cross-section direction.

Set AXIS2=TYPE1 (default) to set c_{m2} to the constant value of 0.85.

Set AXIS2=TYPE2 for members with no distributed transverse loading. Then

c_{m2}=max$(0.6 - 0.4M_1/M_2, 0.4)$, where M_1/M_2 is the ratio of smaller to larger moments about the second cross-section axis at the element ends.

Set AXIS2=TYPE3 for members with distributed transverse loading. Then

c_{m2}=min$(1.0 - 0.4f_c/F_{e2}, 0.85)$, where f_c is the compressive axial stress and F_{e2} is the Euler buckling stress corresponding to the second cross-section direction.

Data line to define the buckling reduction coefficients:

First (and only) data line:

1. c_{m1}, buckling reduction factor in the first cross-section direction.
2. c_{m2}, buckling reduction factor in the second cross-section direction.

If a blank is given on the data line, it is interpreted as zero. If a blank or zero value is given on the data line and either the AXIS1 or AXIS2 parameter is included for this reduction factor, the parameter value will override the zero value given on the data line. If a nonzero value is given on the data line and the AXIS1 or AXIS2 parameter is specified for the same reduction coefficient, an error is issued.

*BULK VISCOSITY: Modify bulk viscosity parameters.

This option is used to redefine bulk viscosity parameters in a model.

Products: Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

• *Explicit Dynamic Analysis*

There are no parameters associated with this option.

Data line to define the bulk viscosity parameters:

First (and only) line:

1. Linear bulk viscosity parameter, b_1. If the **BULK VISCOSITY* option is omitted or is specified without the data line, the default value is 0.06. If the data line is given and the value of b_1 is omitted, the default value is 0.0.
2. Quadratic bulk viscosity parameter, b_2. If the **BULK VISCOSITY* option is omitted or is specified without the data line, the default value is 1.2. If the data line is given and the value of b_2 is omitted, the default value is 0.0.

C

This section describes each Abaqus input option that begins with C, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **C ADDED MASS*
- **CAPACITY*
- **CAP CREEP*
- **CAP HARDENING*
- **CAP PLASTICITY*
- **CAST IRON COMPRESSION HARDENING*
- **CAST IRON PLASTICITY*
- **CAST IRON TENSION HARDENING*
- **CAVITY DEFINITION*
- **CECHARGE*
- **CECURRENT*
- **CENTROID*
- **CFILM*
- **CFLOW*
- **CFLUX*
- **CHANGE FRICTION*
- **CHARACTERISTIC LENGTH*
- **CLAY HARDENING*
- **CLAY PLASTICITY*
- **CLEARANCE*
- **CLOAD*
- **CLUSTER MASS INERTIA TABLE*
- **COHESIVE BEHAVIOR*
- **COHESIVE SECTION*
- **COMBINATORIAL RULE*
- **COMBINED TEST DATA*
- **COMPLEX FREQUENCY*
- **COMPOSITE MODAL DAMPING*
- **CONCENTRATION TENSOR*
- **CONCRETE*
- **CONCRETE COMPRESSION DAMAGE*
- **CONCRETE COMPRESSION HARDENING*
- **CONCRETE DAMAGED PLASTICITY*
- **CONCRETE FAILURE*
- **CONCRETE TENSION DAMAGE*
- **CONCRETE TENSION STIFFENING*

- **CONDUCTIVITY*
- **CONNECTOR BEHAVIOR*
- **CONNECTOR CONSTITUTIVE REFERENCE*
- **CONNECTOR DAMAGE EVOLUTION*
- **CONNECTOR DAMAGE INITIATION*
- **CONNECTOR DAMPING*
- **CONNECTOR DERIVED COMPONENT*
- **CONNECTOR ELASTICITY*
- **CONNECTOR FAILURE*
- **CONNECTOR FRICTION*
- **CONNECTOR HARDENING*
- **CONNECTOR LOAD*
- **CONNECTOR LOCK*
- **CONNECTOR MOTION*
- **CONNECTOR PLASTICITY*
- **CONNECTOR POTENTIAL*
- **CONNECTOR SECTION*
- **CONNECTOR STOP*
- **CONNECTOR UNIAXIAL BEHAVIOR*
- **CONSTITUENT*
- **CONSTRAINT CONTROLS*
- **CONTACT*
- **CONTACT CLEARANCE*
- **CONTACT CLEARANCE ASSIGNMENT*
- **CONTACT CONTROLS*
- **CONTACT CONTROLS ASSIGNMENT*
- **CONTACT DAMPING*
- **CONTACT EXCLUSIONS*
- **CONTACT FILE*
- **CONTACT FORMULATION*
- **CONTACT INCLUSIONS*
- **CONTACT INITIALIZATION ASSIGNMENT*
- **CONTACT INITIALIZATION DATA*
- **CONTACT INTERFERENCE*
- **CONTACT MASS SCALING*
- **CONTACT OUTPUT*
- **CONTACT PAIR*
- **CONTACT PERMEABILITY*
- **CONTACT PRINT*
- **CONTACT PROPERTY ASSIGNMENT*
- **CONTACT RESPONSE*
- **CONTACT STABILIZATION*
- **CONTOUR INTEGRAL*
- **CONTROLS*
- **CONWEP CHARGE PROPERTY*
- **CORRELATION*

- **CO-SIMULATION*
- **CO-SIMULATION REGION*
- **COUPLED CONSTITUTIVE RESPONSE*
- **COUPLED TEMPERATURE-DISPLACEMENT*
- **COUPLED THERMAL-ELECTRICAL*
- **COUPLED THERMAL-ELECTROCHEMICAL*
- **COUPLING*
- **CRADIATE*
- **CREEP*
- **CREEP STRAIN RATE CONTROL*
- **CRUSHABLE FOAM*
- **CRUSHABLE FOAM HARDENING*
- **CRUSH STRESS*
- **CRUSH STRESS VELOCITY FACTOR*
- **CURE GLASS TRANSITION TEMPERATURE*
- **CURE HEAT GENERATION*
- **CURE KINETICS*
- **CURE MAX CONVERSION*
- **CURE SHRINKAGE*
- **CYCLED PLASTIC*
- **CYCLIC*
- **CYCLIC HARDENING*
- **CYCLIC SYMMETRY MODEL*

*C ADDED MASS: Specify concentrated added mass in a FREQUENCY step.

This option is used to include the “added mass” contributions due to concentrated fluid inertia loads in a *FREQUENCY step.

Products: Abaqus/Aqua

Type: History data

Level: Step

References:

• *Abaqus/Aqua Analysis*

There are no parameters associated with this option.

Data lines to define concentrated fluid added mass:

First line:

1. Node number or node set label.
2. Load type label TSI.
3. Tangential added-mass coefficient, L_{ts}.
4. Structural acceleration shape factor for the tangential inertia term, F_{2s}.

Second line:

1. *X*-direction cosine of the outward normal to the exposed transition section area, pointing into the fluid, in the initial configuration.
2. *Y*-direction cosine of the outward normal to the exposed transition section area, pointing into the fluid, in the initial configuration.
3. *Z*-direction cosine of the outward normal to the exposed transition section area, pointing into the fluid, in the initial configuration.

Repeat this pair of data lines as often as necessary to define concentrated fluid added mass at various nodes or node sets.

*CAPACITY: Define the molar heat capacity at constant pressure for an ideal gas species.

This option is used to define the molar heat capacity at constant pressure for an ideal gas species. It can be used only in conjunction with the **FLUID BEHAVIOR* option.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Fluid Cavity Definition*
- *Inflator Definition*
- **FLUID BEHAVIOR*
- **FLUID CAVITY*

Required parameters:

TYPE

Set TYPE=POLYNOMIAL to define the molar heat capacity in the form of a polynomial expression.

Set TYPE=TABULAR to define the molar heat capacity in tabular form.

Optional parameters:

DEPENDENCIES

This parameter is relevant only for TYPE=TABULAR. Set this parameter equal to the number of field variables included in the specification of the molar heat capacity at constant pressure. If this parameter is omitted, the molar heat capacity at constant pressure is assumed not to depend on any field variables but may still depend on temperature.

Data line for TYPE=POLYNOMIAL:

First (and only) line:

1. $\tilde{a}$, the first molar heat capacity coefficient. (Units of $\text{JMOLE}^{-1}\text{K}^{-1}$.)
2. $\tilde{b}$, the second molar heat capacity coefficient. (Units of $\text{JMOLE}^{-1}\text{K}^{-2}$.)
3. $\tilde{c}$, the third molar heat capacity coefficient. (Units of $\text{JMOLE}^{-1}\text{K}^{-3}$.)
4. $\tilde{d}$, the fourth molar heat capacity coefficient. (Units of $\text{JMOLE}^{-1}\text{K}^{-4}$.)
5. $\tilde{e}$, the fifth molar heat capacity coefficient. (Units of $\text{JMOLE}^{-1}\text{K}$.)

Data lines for TYPE=TABULAR:

First line:

1. Molar heat capacity, $\tilde{c}_p$, at constant pressure. (Units of $\text{JMOLE}^{-1}\text{K}^{-1}$.)
2. Temperature, if temperature dependent.
3. First field variable.

4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the heat capacity at constant pressure as a function of temperature and other predefined field variables.

*CAP CREEP: Specify a cap creep law and material properties.

This option is used to define a cap creep model and material properties. Creep behavior defined by this option is active only during **SOILS*, CONSOLIDATION; **COUPLED TEMPERATURE-DISPLACEMENT*; and **VISCO* procedures. It must be used in conjunction with the **CAP PLASTICITY* and the **CAP HARDENING* options.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Modified Drucker-Prager/Cap Model*
- **CAP PLASTICITY*
- **CAP HARDENING*
- *CREEP*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the creep constants, in addition to temperature. If this parameter is omitted, it is assumed that the creep constants depend only on temperature. See *Material Data Definition* for more information.

LAW

Set LAW=STRAIN (default) to choose a strain hardening law. This parameter setting is superseded by LAW=POWER LAW.

Set LAW=TIME to choose a time hardening power law. This parameter setting is superseded by LAW=TIME POWER LAW.

Set LAW=SINGHM to choose a Singh-Mitchell type law.

Set LAW=USER to input the creep law using user subroutine *CREEP*.

Set LAW=POWER LAW to choose a power law. It is recommended that you use this parameter setting rather than LAW=STRAIN.

Set LAW=TIME POWER LAW to choose a time power law. It is recommended that you use this parameter setting rather than LAW=TIME.

MECHANISM

Set MECHANISM=COHESION (default) to choose the cohesion creep mechanism, which is similar in behavior to Drucker-Prager creep.

Set MECHANISM=CONSOLIDATION to choose the consolidation creep mechanism, which is similar in behavior to the cap zone of plasticity.

TIME

This parameter is relevant only when LAW=TIME, LAW=SINGHM, or LAW=TIME POWER LAW is used.

Set TIME=CREEP to use creep time.

Set TIME=TOTAL (default) to use total time.

Data lines for LAW=TIME or LAW=STRAIN:

First line:

1. A. (Units of $F^{-n}L^{2n}T^{-1-m}$.)
2. n.
3. m.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the creep constants on temperature and other predefined field variables.

Data lines for LAW=SINGHM:

First line:

1. A. (Units of T^{-1}.)
2. α. (Units of $F^{-1}L^2$.)
3. m.
4. t_1. (Units of T.)
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the creep constants on temperature and other predefined field variables.

Data lines for LAW=POWER LAW:

First line:

1. q_0. (Units of FL^{-2}.)
2. n.
3. m.
4. Blank space.
5. $\dot{\varepsilon}_0$. The default is 1.0. (Units of T^{-1}.)
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the creep constants on temperature and other predefined field variables.

Data lines for LAW=TIME POWER LAW:

First line:

1. q_0. (Units of FL^{-2}.)
2. n.
3. m.
4. $\dot{\varepsilon}_0$. The default is 1.0. (Units of T^{-1}.)
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the creep constants on temperature and other predefined field variables.

*CAP HARDENING: Specify Drucker-Prager/Cap plasticity hardening.

This option is used to specify the hardening part of the material model for elastic-plastic materials that use the Drucker-Prager/Cap yield surface. It must be used in conjunction with the **CAP PLASTICITY* option and, if creep material behavior is included in an Abaqus/Standard analysis, with the **CAP CREEP* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Modified Drucker-Prager/Cap Model*
- **CAP PLASTICITY*
- **CAP CREEP*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the hydrostatic yield stress, in addition to temperature. If this parameter is omitted, it is assumed that the hydrostatic yield stress depends only on the volumetric plastic strain and, possibly, on the temperature. See *Material Data Definition* for more information.

SCALESTRESS

Set this parameter equal to the factor by which you want the yield stress to be scaled.

Data lines to define Drucker-Prager/Cap plasticity hardening:

First line:

1. Hydrostatic pressure yield stress. (The initial tabular value must be greater than zero, and values must increase with increasing volumetric inelastic strain.)
2. Absolute value of the corresponding volumetric inelastic strain.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of hydrostatic yield stress on volumetric inelastic strain (in Abaqus/Standard) or volumetric plastic strain (in Abaqus/Explicit) and, if needed, on temperature and other predefined field variables.

*CAP PLASTICITY: Specify the Modified Drucker-Prager/Cap plasticity model.

This option is used to define yield surface parameters for elastic-plastic materials that use the modified Drucker-Prager/Cap plasticity model. It must be used in conjunction with the **CAP HARDENING* option and, if creep material behavior is included in an Abaqus/Standard analysis, with the **CAP CREEP* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Modified Drucker-Prager/Cap Model*
- **CAP HARDENING*
- **CAP CREEP*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies, in addition to temperature, included in the definition of the Drucker-Prager/Cap parameters. If this parameter is omitted, it is assumed that the Drucker-Prager/Cap parameters are constant or depend only on temperature. See *Material Data Definition* for more information.

Data lines to define Drucker-Prager/Cap plasticity yield surface parameters:

First line:

1. Material cohesion, d, in the p–t plane (Abaqus/Standard) or in the p–q plane (Abaqus/Explicit). (Units of FL^{-2}.)
2. Material angle of friction, β, in the p–t plane (Abaqus/Standard) or in the p–q plane (Abaqus/Explicit). Give the value in degrees.
3. Cap eccentricity parameter, R. Its value must be greater than zero (typically $0.0001 \le R \le 1000.0$).
4. Initial cap yield surface position on the volumetric inelastic strain axis, $\varepsilon_{vol}^{in}|_0$.
5. Transition surface radius parameter, α. Its value should be a small number compared to unity. If this field is left blank, the default of 0.0 is used (i.e., no transition surface). If creep properties are included in the material model, α must be set to zero.
6. (Not used in Abaqus/Explicit) K, the ratio of the flow stress in triaxial tension to the flow stress in triaxial compression. The value of K should be such that $0.778 \le K \le 1.0$. If this field is left blank or a value of 0.0 is entered, the default of 1.0 is used. If creep properties are included in the material model, K should be set to 1.0.
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the Drucker-Prager/Cap parameters on temperature and other predefined field variables.

*CAST IRON COMPRESSION HARDENING: Specify hardening in compression for the gray cast iron plasticity model.

This option is used to specify the compression hardening data for gray cast iron. It must be used in conjunction with the *CAST IRON PLASTICITY and *CAST IRON TENSION HARDENING options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Cast Iron Plasticity*
- *CAST IRON PLASTICITY
- *CAST IRON TENSION HARDENING

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the compressive yield stress, in addition to temperature. If this parameter is omitted, it is assumed that the compressive yield stress depends only on plastic strain and, possibly, on temperature. See *Material Data Definition* for more information.

Data lines to define compression hardening:

First line:

1. Yield stress in compression, σ_c.
2. Absolute value of the corresponding plastic strain. (The first tabular value entered must always be zero.)
3. Not used.
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the yield stress on plastic strain and, if needed, on temperature and other predefined field variables.

*CAST IRON PLASTICITY: Specify plastic material properties for gray cast iron.

This option is used to define the plastic properties for gray cast iron. It must be used in conjunction with the *CAST IRON COMPRESSION HARDENING and *CAST IRON TENSION HARDENING options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Cast Iron Plasticity*
- *CAST IRON COMPRESSION HARDENING
- *CAST IRON TENSION HARDENING

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the material properties, in addition to temperature. If this parameter is omitted, it is assumed that the material properties depend only on temperature. See *Material Data Definition* for more information.

Data lines to define the plastic “Poisson's ratio”:

First line:

1. Value of the plastic “Poisson's ratio,” ν_{pl}, where $-1.0 < \nu_{pl} \leq 0.5$. (Dimensionless.) If no value is provided, a default value of 0.04 is assumed.
2. Temperature, θ.
3. First field variable.
4. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the material parameter ν_{pl} on temperature and field variables.

*CAST IRON TENSION HARDENING: Specify hardening in tension for the gray cast iron plasticity model.

This option is used to specify the tension hardening data for gray cast iron. It must be used in conjunction with the *CAST IRON PLASTICITY and *CAST IRON COMPRESSION HARDENING options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Cast Iron Plasticity*
- *CAST IRON COMPRESSION HARDENING
- *CAST IRON PLASTICITY

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the tensile yield stress, in addition to temperature. If this parameter is omitted, it is assumed that the tensile yield stress depends only on the plastic strain and, possibly, on temperature. See *Material Data Definition* for more information.

Data lines to define tension hardening:

First line:

1. Yield stress in uniaxial tension, σ_t.
2. Corresponding plastic strain. (The first tabular value entered must always be zero.)
3. Not used.
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the yield stress on plastic strain and, if needed, on temperature and other predefined field variables.

*CAVITY DEFINITION: Define a cavity for thermal radiation.

This option is used to define cavities for thermal radiation heat transfer. It can be used only in conjunction with the *SURFACE, TYPE=ELEMENT option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Cavity Radiation in Abaqus/Standard*
- *SURFACE
- *SURFACE PROPERTY

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the cavity.

Optional parameters:

AMBIENT TEMP

Set this parameter equal to the reference temperature of the external medium to which radiation takes place in the case of an open cavity. If this parameter is omitted, the cavity is assumed to be closed.

PARALLEL DECOMPOSITION

Set PARALLEL DECOMPOSITION=ON to enable parallel decomposition of a cavity during a cavity radiation analysis.

Set PARALLEL DECOMPOSITION=OFF (default) to disable parallel decomposition of a cavity during a cavity radiation analysis.

SET PROPERTY

Include this parameter to set, or to redefine, surface properties for the surfaces making up the cavity. If this parameter is omitted, the cavity is assumed to consist of surfaces for which surface properties have already been defined as part of the surface definitions.

Data lines to define a cavity for thermal radiation using surfaces with defined surface properties (default):

First line:

1. List of surfaces that compose this cavity.

Repeat this data line as often as necessary to define the cavity. Up to 16 entries are allowed per line.

Data lines to define a cavity when the SET PROPERTY parameter is included:

First line:

1. Surface name.
2. Surface property name.

Repeat this data line as often as necessary to define the cavity. Up to 16 entries are allowed per line.

*CECHARGE: Specify concentrated electric charges in piezoelectric analysis.

This option is used to apply electric charge to any node in a piezoelectric model.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

• *Piezoelectric Analysis*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the magnitude of the electric charge during the step. If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (*Defining an Analysis*).

OP

Set OP=MOD (default) for existing **CECHARGE*s to remain, with this option modifying existing electric charges or defining additional electric charges.

Set OP=NEW if all existing **CECHARGE*s applied to the model should be removed.

Optional, mutually exclusive parameters for matrix generation and direct-solution steady-state dynamics analysis:

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the concentrated electric charges.

REAL

Include this parameter (default) to define the real (in-phase) part of the concentrated electric charges.

Data lines to define concentrated electric charges:

First line:

1. Node number or node set label.
2. Leave blank.
3. Reference electric charge magnitude. (Units of C.)

Repeat this data line as often as necessary to define concentrated electric charges at various nodes or node sets.

*CECURRENT: Specify concentrated current in an electric conduction analysis.

This option is used to apply concentrated current to any node of a model in coupled thermal-electrical, coupled thermal-electrical-structural, coupled thermal-electrochemical, coupled thermal-electrochemical-structural, and coupled thermal-electrochemical-structural-pore pressure analyses.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- **DSECURRENT*
- *Coupled Thermal-Electrical Analysis*
- *Fully Coupled Thermal-Electrical-Structural Analysis*
- *Coupled Thermal-Electrochemical Analysis*
- *Fully Coupled Thermal-Electrochemical-Structural Analysis*
- *Fully Coupled Thermal-Electrochemical-Structural–Pore Pressure Analysis*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the magnitude of the current during the step (*Amplitude Curves*). If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (*Defining an Analysis*).

OP

Set OP=MOD (default) for existing **CECURRENT*s to remain, with this option modifying existing concentrated currents or defining additional concentrated currents.

Set OP=NEW if all existing **CECURRENT*s applied to the model should be removed.

Data lines to define concentrated current at nodes:

First line:

1. Node number or node set label.
2. Leave blank.
3. Reference magnitude for current. (Units of CT^{-1}.)

Repeat this data line as often as necessary to define current at various nodes or node sets.

*CENTROID: Define the position of the centroid of the beam section.

This option can be used only in conjunction with the *BEAM GENERAL SECTION, SECTION=GENERAL or the *BEAM GENERAL SECTION, SECTION=MESHED option. It cannot be used if you include the FULL COUPLING parameter for the *SECTION STIFFNESS option.

It is used to define the position of the centroid of the section with respect to the local (1, 2) axis system.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- *BEAM GENERAL SECTION
- *Using a General Beam Section to Define the Section Behavior*
- *Meshed Beam Cross-Sections*

There are no parameters associated with this option.

Data line to define the position of the centroid:

First (and only) line:

1. Local x_1-coordinate of centroid, x_{1c}. The default is 0.
2. Local x_2-coordinate of centroid, x_{2c}. The default is 0.

*CFILM: Define film coefficients and associated sink temperatures at one or more nodes or vertices.

This option is used to provide film coefficients and sink temperatures at any node in the model for all procedures that solve for nodal temperature degrees of freedom.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *Thermal Loads*
- *FILM*

Optional parameters:

FILM AMPLITUDE

Set this parameter equal to the name of the **AMPLITUDE* option that gives the variation of the film coefficient, h, with time.

If this parameter is omitted, the reference film coefficient is applied immediately at the beginning of the step and kept constant over the step.

The FILM AMPLITUDE parameter is ignored if a film coefficient is defined to be a function of temperature and field variables via the **FILM PROPERTY* option.

For nonuniform film coefficients, the film coefficient amplitude is defined in user subroutine *FILM* in Abaqus/Standard or user subroutine *VFILM* in Abaqus/Explicit; and FILM AMPLITUDE references are used only to modify the film coefficient passed into the user subroutine.

OP

Set OP=MOD (default) for existing **CFILM*s to remain, with this option modifying existing films or defining additional films.

Set OP=NEW if all existing **CFILM*s applied to the model should be removed.

PHANTOM

This parameter applies only to Abaqus/Standard analyses.

Set PHANTOM=NODE to define film coefficients and sink temperatures at a phantom node that is originally located coincident with the specified real node in an enriched element.

Set PHANTOM=EDGE to define film coefficients and sink temperatures at a phantom node located on an element edge between the two specified real corner nodes in an enriched element. This setting applies only to nodes with both pore pressure and temperature degrees of freedom.

REGION TYPE

This parameter applies only to Abaqus/Explicit analyses.

This parameter is relevant only for concentrated films applied on the boundary of an adaptive mesh domain. If concentrated films are applied to nodes in the interior of an adaptive mesh domain, these nodes will always follow the material.

Set REGION TYPE=LAGRANGIAN (default) to apply a concentrated film to a node that follows the material (nonadaptive).

Set REGION TYPE=SLIDING to apply a concentrated film to a node that can slide over the material. Mesh constraints are typically applied to the node to fix it spatially.

Set REGION TYPE=EULERIAN to apply a concentrated film to a node that can move independently of the material. This option is used only for boundary regions where the material can flow into or out of the adaptive mesh domain. Mesh constraints must be used normal to an Eulerian boundary region to allow material to flow through the region. If no mesh constraints are applied, an Eulerian boundary region will behave in the same way as a sliding boundary region.

SINK AMPLITUDE

Set this parameter equal to the name of the **AMPLITUDE* option that gives the variation of the sink temperature, θ^0, with time.

If this parameter is omitted in an Abaqus/Standard analysis, the reference sink temperature is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (*Defining an Analysis*). If this parameter is omitted in an Abaqus/Explicit analysis, the reference sink temperature given on the data lines is applied immediately at the beginning of the step.

For nonuniform film coefficients, the sink temperature amplitude is defined in user subroutine *FILM* in Abaqus/Standard or user subroutine *VFILM* in Abaqus/Explicit; and the SINK AMPLITUDE references are used only to modify the sink temperature passed into the user subroutine.

USER

Include this parameter to indicate that any nonzero film coefficients prescribed through this option will be defined in user subroutine *FILM* in Abaqus/Standard or user subroutine *VFILM* in Abaqus/Explicit.

If this parameter is used, any film coefficient and sink temperature values defined on the data lines of the option (and possibly modified by the AMPLITUDE and FILM AMPLITUDE parameters) are passed into user subroutines *FILM* and *VFILM* and can be modified within the user subroutines.

Data lines to define sink temperatures and film coefficients:

First line:

1. Node number or node set label.
2. Appropriate area associated with the node where the concentrated film condition is applied. The default is 1.0.
3. Reference sink temperature value, θ^0. (Units of θ.) For nonuniform film coefficients the sink temperature must be defined in user subroutine *FILM* in Abaqus/Standard or *VFILM* in Abaqus/Explicit. If given, this value will be modified by the SINK AMPLITUDE reference and passed into the user subroutine.
4. Reference film coefficient value, h (units of $JT^{-1}L^{-2}\theta^{-1}$), or name of the film property table defined with the **FILM PROPERTY* option. Nonuniform film coefficients must be defined in user subroutine *FILM* in Abaqus/Standard or *VFILM* in Abaqus/Explicit. If given, this value will be modified by the FILM AMPLITUDE reference and passed into the user subroutine.

Repeat this data line as often as necessary to define film conditions.

Data lines to define sink temperatures and film coefficients when PHANTOM=NODE:

First line:

1. Node number of a real node that is originally located coincident with the phantom node.
2. Appropriate area associated with the node where the concentrated film condition is applied. The default is 1.0.
3. Reference sink temperature value, θ^0. (Units of θ.) For nonuniform film coefficients, the sink temperature must be defined in user subroutine *FILM*. If given, this value is modified by the SINK AMPLITUDE reference and passed into the user subroutine.
4. Reference film coefficient value, h (units of $JT^{-1}L^{-2}\theta^{-1}$), or the name of the film property table defined with the **FILM PROPERTY* option. Nonuniform film coefficients must be defined in user subroutine *FILM*. If given, this value is modified by the FILM AMPLITUDE reference and passed into the user subroutine.

Repeat this data line as often as necessary to define film conditions.

Data lines to define sink temperatures and film coefficients when PHANTOM=EDGE:

First line:

1. Node number of one of the two real corner nodes to define an edge in an enriched element.
2. Node number of the other real corner node to define an edge in an enriched element.
3. Appropriate area associated with the node where the concentrated film condition is applied. The default is 1.0.
4. Reference sink temperature value, θ^0. (Units of θ.) For nonuniform film coefficients, the sink temperature must be defined in user subroutine *FILM*. If given, this value is modified by the SINK AMPLITUDE reference and passed into the user subroutine.
5. Reference film coefficient value, h (units of $JT^{-1}L^{-2}\theta^{-1}$), or the name of the film property table defined with the **FILM PROPERTY* option. Nonuniform film coefficients must be defined in user subroutine *FILM*. If given, this value is modified by the FILM AMPLITUDE reference and passed into the user subroutine.

Repeat this data line as often as necessary to define film conditions.

*CFLOW: Specify concentrated fluid flow.

This option is used to apply concentrated fluid flow to any node including phantom nodes in enriched elements in consolidation problems.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- *Coupled Pore Fluid Diffusion and Stress Analysis*
- *Geostatic Stress State*
- *Pore Fluid Flow*
- *Modeling Discontinuities as an Enriched Feature Using the Extended Finite Element Method*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the magnitude of the flow during the step. If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (see *Defining an Analysis*).

OP

Set OP=MOD (default) for existing **CFLOW*s to remain, with this option modifying existing concentrated flows or defining additional concentrated flows.

Set OP=NEW if all existing **CFLOW*s applied to the model should be removed.

PHANTOM

This parameter applies only to enriched elements in Abaqus/Standard.

Set PHANTOM=NODE to apply concentrated flows to a phantom node that is originally located coincident with the specified real node in an enriched element.

Set PHANTOM=EDGE to apply concentrated flows to a phantom node located at an element edge between the two specified real corner nodes in an enriched element.

Set PHANTOM=INCLUDED to indicate that the pore pressure applied to a phantom node located at an element edge will be interpolated automatically from the specified real corner nodes when the enriched element is cracked.

Data lines to define concentrated flow when the PHANTOM parameter is not used:

First line:

1. Node number or node set label.
2. (Not used.)
3. Reference concentrated flow magnitude.

Repeat this data line as often as necessary to define concentrated flows.

Data lines to define concentrated flow when PHANTOM=NODE:

First line:

1. Node number of a real node that is originally located coincident with the phantom node.
2. (Not used.)
3. Reference concentrated flow magnitude.

Repeat this data line as often as necessary to define concentrated flows.

Data lines to define concentrated flow when PHANTOM=EDGE:

First line:

1. Node number of one of the two real corner nodes to define an edge in an enriched element.
2. Node number of the other real corner node to define an edge in an enriched element.
3. Reference concentrated flow magnitude.

Repeat this data line as often as necessary to define concentrated flows.

Data lines to define concentrated flow when PHANTOM=INCLUDED:

First line:

1. Node number or node set label.
2. (Not used.)
3. Reference concentrated flow magnitude.

Repeat this data line as often as necessary to define concentrated flows.

*CFLUX: Specify concentrated fluxes in heat transfer or mass diffusion analyses.

This option is used to apply a flux to any node of the model in fully coupled thermal-stress analysis. In Abaqus/Standard it is also used for heat transfer, coupled thermal-electrical, coupled thermal-electrical-structural, and mass diffusion analyses.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

• *Thermal Loads*

Required parameter for reading concentrated nodal flux from an output database file:

FILE

Set this parameter equal to the name of the output database file from which the data are to be read. The file extension is optional.

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the magnitude of the flux during the step (*Amplitude Curves*).

If this parameter is omitted in an Abaqus/Standard analysis, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (see *Defining an Analysis*). If this parameter is omitted in an Abaqus/Explicit analysis, the reference magnitude is applied immediately at the beginnning of the step.

INC

This parameter is relevant only when the FILE parameter is used.

Set this parameter equal to the increment in the selected step of the previous analysis from which the concentrated nodal fluxes will be read. By default, the concentrated nodal fluxes will be read from the last increment of the step specified on the STEP parameter or from the last step if the STEP parameter is omitted.

OP

Set OP=MOD (default) for existing **CFLUX*s to remain, with this option modifying existing fluxes or defining additional fluxes.

Set OP=NEW if all existing **CFLUX*s applied to the model should be removed.

PHANTOM

This parameter applies only to Abaqus/Standard analyses.

Set PHANTOM=NODE to apply concentrated fluxes to a phantom node that is originally located coincident with the specified real node in an enriched element.

Set PHANTOM=EDGE to apply concentrated fluxes to a phantom node located on an element edge between the two specified real corner nodes in an enriched element. This setting applies only to nodes with both pore pressure and temperature degrees of freedom.

REGION TYPE

This parameter applies only to Abaqus/Explicit analyses.

This parameter is relevant only for concentrated fluxes applied on the boundary of an adaptive mesh domain. If concentrated fluxes are applied to nodes in the interior of an adaptive mesh domain, these nodes will always follow the material.

Set REGION TYPE=LAGRANGIAN (default) to apply a concentrated flux to a node that follows the material (nonadaptive).

Set REGION TYPE=SLIDING to apply a concentrated flux to a node that can slide over the material. Mesh constraints are typically applied to the node to fix it spatially.

Set REGION TYPE=EULERIAN to apply a concentrated flux to a node that can move independently of the material. This option is used only for boundary regions where the material can flow into or out of the adaptive mesh domain. Mesh constraints must be used normal to an Eulerian boundary region to allow material to flow through the region. If no mesh constraints are applied, an Eulerian boundary region will behave in the same way as a sliding boundary region.

STEP

This parameter is relevant only when the FILE parameter is used.

Set this parameter equal to the step number of the previous analysis from which the concentrated nodal fluxes will be read. By default, the concentrated nodal fluxes will be read from the last step of the previous analysis.

Data lines to define a concentrated flux:

First line:

1. Node number or node set label.
2. Degree of freedom. If a blank or 0 is entered, degree of freedom 11 is assumed. For shell heat transfer elements enter 11, 12, or 13, etc.
3. Reference magnitude for flux (units of JT^{-1} in heat transfer analysis and PL^3T^{-1} in mass diffusion analysis).

Repeat this data line as often as necessary to define concentrated fluxes at different nodes and degrees of freedom.

Data lines to define a concentrated flux when PHANTOM=NODE:

First line:

1. Node number of a real node that is originally located coincident with the phantom node.
2. Degree of freedom. If a blank or 0 is entered, degree of freedom 11 is assumed.
3. Reference magnitude for flux (units of JT^{-1}).

Repeat this data line as often as necessary to define concentrated fluxes at different nodes.

Data lines to define a concentrated flux when PHANTOM=EDGE:

First line:

1. Node number of one of the two real corner nodes to define an edge in an enriched element.
2. Node number of the other real corner node to define an edge in an enriched element.

3. Reference magnitude for flux (units of JT^{-1}).

Repeat this data line as often as necessary to define concentrated fluxes at different nodes.

*CHANGE FRICTION: Change friction properties.

Use this option in conjunction with the *FRICTION option to change the values of friction properties from step to step.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *About Mechanical Contact Properties*
- *Frictional Behavior*
- *Connector Behavior*
- *FRICTION

Required, mutually exclusive parameters:

ELSET

Use this parameter if the contact conditions have been modeled with contact elements or if friction is defined in connector elements. Set this parameter equal to the name of the element set containing the contact or connector elements for which the friction properties are being redefined.

INTERACTION

Use this parameter if the contact conditions have been modeled with surface-based contact pairs or general contact. Set this parameter equal to the name of the *SURFACE INTERACTION property definition for which the friction properties are being redefined.

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve (defined in the *AMPLITUDE option) that gives the time variation of any changes in friction coefficients and allowable elastic slip throughout the step (*Amplitude Curves*).

If this parameter is omitted, transitions in these friction properties occur according to the value assigned to the AMPLITUDE parameter on the *STEP option (see *Defining an Analysis*). Changes in friction properties other than the friction coefficient and the allowable elastic slip are always made immediately. Sudden changes in friction properties when the frictional stress is nonzero can cause convergence difficulties.

RESET

Include this parameter to reset the friction properties to their original values. When this parameter is used, no *FRICTION option is needed to redefine the friction properties.

There are no data lines associated with this option.

*CHARACTERISTIC LENGTH: Define characteristic element length at a material point.

This option is used to define the characteristic element length to be used by Abaqus for the regularization of models that exhibit strain softening or to be passed to user subroutines that are called at the material point. If used, it must appear within a *MATERIAL definition (*Material Data Definition*). If this option is not specified, Abaqus computes the characteristic element length using the geometric mean–based definition.

Products: Abaqus/Explicit

Type: Model data

Level: Model

References:

- *VUCHARLENGTH*

Optional parameters:

DEFINITION

Set DEFINITION=GEOMETRIC MEAN (default) to use the geometric mean–based definition of characteristic element length.

Set DEFINITION=USER to specify the characteristic element length in user subroutine *VUCHARLENGTH*.

Optional parameters for use with DEFINITION=USER:

COMPONENTS

Set this parameter equal to the number of components of characteristic element length.

PROPERTIES

Set this parameter equal to the number of properties being entered. The properties are available for use in user subroutine *VUCHARLENGTH*.

No data lines are needed for DEFINITION=GEOMETRIC MEAN

Data lines for DEFINITION=USER when the PROPERTIES parameter is specified:

First line:

1. Enter the material properties, eight per line.

Repeat this data line as often as necessary to define all material properties.

*CLAY HARDENING: Specify hardening for the clay plasticity model.

This option is used to define piecewise linear hardening/softening of the Cam-clay plasticity yield surface. It can be used only in conjunction with the *CLAY PLASTICITY option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Critical State (Clay) Plasticity Model*
- *CLAY PLASTICITY

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies, in addition to temperature, included in the definition of the hydrostatic pressure stress. If this parameter is omitted, the hydrostatic pressure stress may depend only on the volumetric plastic strain and, possibly, on the temperature. See *Material Data Definition* for more information.

SR

This parameter is relevant only when the *SOFTENING REGULARIZATION option is used.

Set SR=ON (default) to use softening regularization for the clay hardening data.

Set SR=OFF if softening regularization is not required.

TYPE

Set TYPE=COMPRESSION (default) to define the hardening behavior by providing the hydrostatic compression yield stress, p_c, as a function of volumetric plastic strain.

Set TYPE=TENSION to define the hardening behavior by providing the hydrostatic tension yield stress, p_t, as a function of volumetric plastic strain.

Compression hardening must always be defined. Tensile hardening is optional and can be specified, if needed, by repeating the *CLAY HARDENING option with TYPE=TENSION.

Data lines to define hardening for Cam-clay plasticity:

First line:

1. Value of the hydrostatic pressure stress at yield, p_c or p_t. p_c is given as a positive value and must increase with increasing plastic strain. p_t can be zero or a negative value and must decrease with increasing plastic strain.
2. Absolute value of the corresponding compressive volumetric plastic strain. The first tabular value must always be zero.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the yield surface size on volumetric plastic strain and, if needed, on temperature and other predefined field variables.

*CLAY PLASTICITY: Specify the extended Cam-clay plasticity model.

This option is used to specify the plastic part of the material behavior for elastic-plastic materials that use the extended Cam-clay plasticity model.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Critical State (Clay) Plasticity Model*
- **CLAY HARDENING*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies, in addition to temperature, included in the definition of the Cam-clay parameters. If this parameter is omitted, the Cam-clay parameters may depend only on temperature. See *Material Data Definition* for more information.

HARDENING

Set HARDENING=EXPONENTIAL (default for Abaqus/Standard) to specify an exponential hardening/softening law. This hardening law is not supported in Abaqus/Explicit.

Set HARDENING=TABULAR (default and only option for Abaqus/Explicit) to specify a piecewise linear hardening/softening relationship. The **CLAY HARDENING* option must be used in this case. HARDENING=TABULAR and the use of the INTERCEPT parameter are mutually exclusive.

INTERCEPT

This parameter applies only to Abaqus/Standard analyses.

It is used as an alternative to the direct specification of the initial yield surface size, a_0, when the exponential hardening law is specified. Set this parameter equal to e_1, the intercept of the virgin consolidation line with the void ratio axis in a plot of void ratio versus the logarithm of pressure stress. If this parameter is included, the value given for a_0 on the data line is ignored. This parameter cannot be used when the HARDENING=TABULAR parameter is used.

Data lines to define Cam-clay plasticity:

First line:

1. Logarithmic plastic bulk modulus, λ (dimensionless). This data item is ignored if HARDENING=TABULAR.
2. Stress ratio at critical state, M.
3. Enter the initial yield surface size, a_0 (units of FL^{-2}), if HARDENING=EXPONENTIAL. Enter the initial volumetric plastic strain, $\left.\varepsilon_{vol}^{pl}\right|_0$ corresponding to $\left.p_c\right|_0$ according to the **CLAY HARDENING* definition if HARDENING=TABULAR. A positive value must be entered. This data item is ignored if the INTERCEPT parameter is included.

4. β, the parameter defining the size of the yield surface on the "wet" side of critical state. If this value is omitted or set to zero, a value of 1.0 is assumed.
5. K, the ratio of the flow stress in triaxial tension to the flow stress in triaxial compression. $0.778 \leq K \leq 1.0$. If this value is left blank or set to zero, a value of 1.0 is assumed.
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the Cam-clay parameters on temperature and other predefined field variables.

*CLEARANCE: Specify a particular initial clearance value and a contact direction for the secondary nodes on a surface.

This option is used to define initial clearance values and/or contact directions precisely at contact secondary nodes. In an Abaqus/Standard analysis it can also be used to define overclosure values. The **CLEARANCE* option can be used with small-sliding contact only. In Abaqus/Explicit it can be used only in the first step of an analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data in Abaqus/Standard; History data in Abaqus/Explicit

Level: Model in Abaqus/Standard; Step in Abaqus/Explicit

Abaqus/CAE: Interaction module

References:

- *Common Difficulties Associated with Contact Modeling in Abaqus/Standard*
- *Common Difficulties Associated with Contact Modeling Using Contact Pairs in Abaqus/Explicit*
- *Contact Initialization for Contact Pairs in Abaqus/Standard*
- *Contact Initialization for Contact Pairs in Abaqus/Explicit*
- **CONTACT INITIALIZATION DATA*

Required parameter for Abaqus/Explicit analyses:

CPSET

Set this parameter equal to the name of the contact pair set name to associate these clearance data with the appropriate contact pairs.

Required parameter for defining initial clearances for general contact in Abaqus/Standard:

NAME

Set this parameter equal to a label that is used to refer to this clearance definition by the **CONTACT INITIALIZATION DATA* option for general contact interactions in portions of the model where the small-sliding tracking approach is active or for interactions in heat transfer and coupled electrical-thermal procedures that have no mechanical degrees of freedom.

Required parameters for defining initial clearances for contact pairs in Abaqus/Standard:

MAIN

Set this parameter equal to the name of the main surface of the contact pair.

SECONDARY

Set this parameter equal to the name of the secondary surface of the contact pair.

Required, mutually exclusive parameters:

TABULAR

Include this parameter to specify the secondary nodes or the node sets and their corresponding initial clearance/overclosure values (and, if required, contact directions) on the data lines of this option. In an Abaqus/Explicit analysis only initial clearances are allowed.

VALUE

Set this parameter equal to the initial clearance/overclosure for the entire set of secondary nodes. In Abaqus/Standard a positive value specifies an initial clearance, and a negative value specifies an initial overclosure. In an Abaqus/Explicit analysis this value must be positive since only initial clearances are allowed.

Optional parameters when the TABULAR parameter is included:

BOLT

Include this parameter to indicate that the appropriate contact normal directions for a threaded bolt connection will be generated automatically based on thread geometry data and two points used to define a direction vector on the axis of the bolt and bolt-hole assembly specified on the data lines. This parameter is valid only for single threaded bolts.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. The data lines in the alternate input file should be in the same format as that for the TABULAR parameter.

If this parameter is omitted and the TABULAR parameter is included, it is assumed that the data follow the keyword line.

Optional parameters when the TABULAR and BOLT parameter are included:

HANDEDNESS

Set this parameter equal to RIGHT (default) to specify right-handed reference threads.

Set this parameter equal to LEFT to specify left-handed reference threads.

NORMAL ADJUSTMENT

Set this parameter equal to UNIFORM AXIAL COMPONENT (default) if all secondary nodes should be associated with the same top or bottom thread face, according to the convention discussed in *Generating Contact Normal Directions Based on a Reference Thread Geometry*, such that the axial component (and small circumferential component) of the contact normal direction is consistent for all secondary nodes.

Set this parameter equal to LOCATION DEPENDENT if each secondary node should be associated with the top or bottom thread face based on its initial position with respect to the reference thread geometry, as discussed in *Generating Contact Normal Directions Based on a Reference Thread Geometry*.

Data lines if the TABULAR parameter is included with neither the INPUT parameter nor the BOLT parameter:

First line:

1. Node number or node set label.
2. Clearance value. (In an Abaqus/Standard analysis a positive value indicates an opening between the surfaces and a negative value indicates overclosure.) If this field is left blank, the clearance value automatically calculated will not be modified.
3. First component of the normal.
4. Second component of the normal.
5. Third component of the normal.

Repeat the above data line as often as necessary to define the clearance value and the direction in which Abaqus tests for contact between the secondary node and the corresponding closest point on

the main surface. The specification of the normal is optional. If the normal is given, it should be in the direction of the main surface's outward normal. If the normal is not given, Abaqus calculates it from the geometry of the main surface (see *Common Difficulties Associated with Contact Modeling in Abaqus/Standard* and *Common Difficulties Associated with Contact Modeling Using Contact Pairs in Abaqus/Explicit*).

Data lines if both the TABULAR parameter and the BOLT parameter are included without the INPUT parameter (see figures below):

First line:

1. Half-thread angle, , (in degrees).
2. Pitch (thread-to-thread distance), *p*.
3. Bolt major thread diameter, *d*. If the mean diameter is given, the major diameter is ignored.
4. Bolt mean thread diameter, *dm*. The default value is *d-0.649519p*.

Second line:

1. Node number or node set label.
2. Clearance value. (In an Abaqus/Standard analysis a positive value indicates an opening between the surfaces and a negative value indicates overclosure.) If this field is left blank, the clearance value calculated automatically will not be modified.
3. *X*-coordinate of point *a* along the axis of the bolt/bolt hole.
4. *Y*-coordinate of point *a* along the axis of the bolt/bolt hole.
5. *Z*-coordinate of point *a* along the axis of the bolt/bolt hole.
6. *X*-coordinate of point *b* along the axis of the bolt/bolt hole.
7. *Y*-coordinate of point *b* along the axis of the bolt/bolt hole.
8. *Z*-coordinate of point *b* along the axis of the bolt/bolt hole.

Repeat the second data line as often as necessary to define the clearance value and the direction vector on the axis of the bolt and bolt-hole assembly that Abaqus uses to calculate the contact normal directions based on the thread geometry (see *Additional Contact Initialization Options for Small-Sliding Contact in Abaqus/Standard* and *Specifying Initial Clearance Values Precisely*).

To define a clearance value by using the VALUE parameter:

No data lines are used with this option when the VALUE parameter is specified.

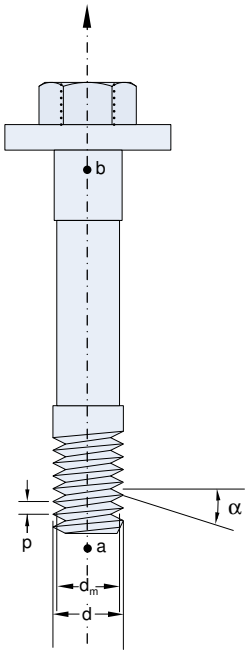


Figure 1: Thread geometry.

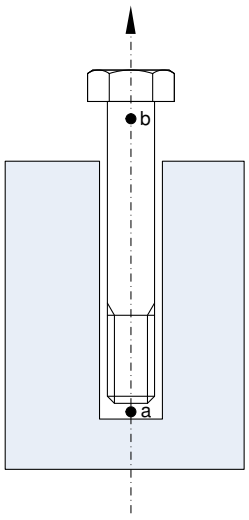


Figure 2: Points *a* and *b* on the centerline of the bolt and bolt-hole assembly.

*CLOAD: Specify concentrated forces and moments.

This option is used to apply concentrated forces and moments at any node in the model. The **CLOAD* option can also be used to specify concentrated buoyancy, drag, and inertia loads in an Abaqus/Aqua analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE Abaqus/Aqua

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- *Concentrated Loads*
- *Abaqus/Aqua Analysis*
- *Analysis of Models that Exhibit Cyclic Symmetry*
- *Defining ALE Adaptive Mesh Domains in Abaqus/Explicit*

Applying concentrated loads

Required parameter for reading concentrated nodal force from an output database file:

FILE

Set this parameter equal to the name of the output database file from which the data are to be read. The file extension is optional.

Required parameter for cyclic symmetry models in steady-state dynamics analyses:

CYCLIC MODE

Set this parameter equal to the cyclic symmetry mode number of loads that are applied in the current steady-state dynamics procedure.

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the magnitude of the load during the step.

If this parameter is omitted in an Abaqus/Standard analysis, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (see *Defining an Analysis*). If this parameter is omitted in an Abaqus/Explicit analysis, the reference magnitude is applied immediately at the beginning of the step.

FOLLOWER

Include this parameter if the direction of the load is assumed to rotate with the rotation at this node.

This parameter should be used only for large-displacement analysis and can be used only at nodes with active rotational degrees of freedom (such as the nodes of beam or shell elements).

Concentrated buoyancy, drag, and fluid inertia loads in Abaqus/Aqua analyses are automatically considered to be follower forces, so this parameter is not necessary in those cases.

In general, UNSYMM=YES should be used on the **STEP* option in conjunction with the FOLLOWER parameter in **DYNAMIC* and **STATIC* analyses in Abaqus/Standard. The UNSYMM parameter is ignored in eigenvalue analyses (such as **BUCKLE* or **FREQUENCY*) since Abaqus/Standard can perform an eigenvalue extraction only on symmetric matrices.

INC

This parameter is relevant only when the FILE parameter is used.

Set this parameter equal to the increment in the selected step of the previous analysis from which the concentrated nodal forces will be read. By default, the concentrated nodal forces will be read from the last increment of the step specified on the STEP parameter or from the last step if the STEP parameter is omitted.

LOAD CASE

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the load case number. This parameter is used in **RANDOM RESPONSE* analysis (*Random Response Analysis*), when it is the cross-reference for the load case on the **CORRELATION* option. The parameter's value is ignored in all other procedures.

OP

Set OP=MOD (default) for existing **CLOAD*s to remain, with this option modifying existing concentrated loads or defining additional concentrated loads.

Set OP=NEW if all existing **CLOAD*s applied to the model should be removed. New concentrated loads can be defined.

REGION TYPE

This parameter applies only to Abaqus/Explicit analyses.

This parameter is relevant only for concentrated loads applied on the boundary of an adaptive mesh domain. If concentrated loads are applied to nodes in the interior of an adaptive mesh domain, these nodes will always follow the material.

Set REGION TYPE=LAGRANGIAN (default) to apply a concentrated load to a node that follows the material (nonadaptive).

Set REGION TYPE=SLIDING to apply a concentrated load to a node that can slide over the material. Mesh constraints are typically applied to the node to fix it spatially.

Set REGION TYPE=EULERIAN to apply a concentrated load to a node that can move independently of the material. This option is used only for boundary regions where the material can flow into or out of the adaptive mesh domain. Mesh constraints must be used normal to an Eulerian boundary region to allow material to flow through the region. If no mesh constraints are applied, an Eulerian boundary region will behave in the same way as a sliding boundary region.

STEP

This parameter is relevant only when the FILE parameter is used.

Set this parameter equal to the step number of the previous analysis from which the concentrated nodal forces will be read. By default, the concentrated nodal forces will be read from the last step of the previous analysis.

Optional, mutually exclusive parameters for matrix generation and steady-state dynamics analysis:

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the loading.

REAL

Include this parameter (default) to define the real (in-phase) part of the loading.

Data lines to define concentrated loads for specific degrees of freedom:

First line:

1. Node number or node set label.
2. Degree of freedom.
3. Reference magnitude for load.

Repeat this data line as often as necessary to define concentrated loads.

Applying Abaqus/Aqua loads

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the magnitude of the load during the step. If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (see *Defining an Analysis*).

OP

Set OP=MOD (default) for existing **CLOAD*s to remain, with this option modifying existing concentrated loads or defining additional concentrated loads.

Set OP=NEW if all existing **CLOAD*s applied to the model should be removed.

Data lines to define concentrated buoyancy forces:

First line:

1. Node number or node set label.
2. Concentrated load type label, TSB.
3. Magnitude factor, *M*. The default value is 1.0. This factor will be scaled by any **AMPLITUDE* specification associated with this **CLOAD* option.
4. Exposed area.

Give the following direction cosines in the local coordinate system if the **TRANSFORM* option was used at this node:

11. *X*-direction cosine of the outward normal to the exposed area, pointing into the fluid, in the initial configuration.
13. *Y*-direction cosine of the outward normal to the exposed area, pointing into the fluid, in the initial configuration.
15. *Z*-direction cosine of the outward normal to the exposed area, pointing into the fluid, in the initial configuration.

The following data should be provided only if it is necessary to change the fluid properties specified under the *AQUA option, as described in *Buoyancy Loads*. Gravity waves do not affect the buoyancy loading when any external fluid property is overridden.

19. Density of the fluid outside the element.
21. Free surface elevation of the fluid outside the element.
23. Constant pressure, added to the hydrostatic pressure outside the element.

Repeat this data line as often as necessary to define concentrated buoyancy at various nodes or node sets.

Data lines to define concentrated fluid drag loading:

First line:

1. Node number or node set label.
2. Concentrated load type label, TFD (fluid) or TWD (wind).
3. Magnitude factor, M. The default value is 1.0. This factor will be scaled by any *AMPLITUDE specification associated with this *CLOAD option.
4. Exposed area, ΔA.
5. Drag coefficient, C_n.
6. Structural velocity factor, α_R. The default value is 1.0 if this entry is left blank or set equal to 0.0.
7. For load type TFD, name of the *AMPLITUDE curve used for scaling steady current velocities (A_c). For load type TWD, name of the *AMPLITUDE curve used for scaling the local x-direction wind velocity (A_x). If this data item is blank, the velocities are not scaled ($A_c = 1$ or $A_x = 1$).
8. For load type TFD, name of the *AMPLITUDE curve used for scaling wave velocities (A_w). For load type TWD, name of the *AMPLITUDE curve used for scaling the local y-direction wind velocity (A_y). If this data item is blank, the velocities are not scaled ($A_w = 1$ or $A_y = 1$).

Second line:

Give the following direction cosines in the local coordinate system if the *TRANSFORM option was used at this node:

1. X-direction cosine of the outward normal to the exposed transition section area, pointing into the fluid, in the initial configuration.
2. Y-direction cosine of the outward normal to the exposed transition section area, pointing into the fluid, in the initial configuration.
3. Z-direction cosine of the outward normal to the exposed transition section area, pointing into the fluid, in the initial configuration.

Repeat this pair of data lines as often as necessary to define concentrated fluid or wind drag loading at various nodes or node sets.

Data lines to define concentrated fluid inertia loading:

First line:

1. Node number or node set label.
2. Load type label, TSI.
3. Magnitude factor, M. The default value is 1.0. This factor will be scaled by any *AMPLITUDE specification associated with this *CLOAD option.
4. Tangential inertia coefficient, K_{ts}.

5. Fluid acceleration shape factor for the tangential inertia term, F_{1s}.
6. Tangential added-mass coefficient, L_{ts}.
7. Structural acceleration shape factor for the tangential inertia term, F_{2s}.
8. Name of the *AMPLITUDE curve to be used for scaling fluid particle accelerations (A_w). If this data item is blank, the fluid particle accelerations are not scaled ($A_w = 1$).

Second line:
Give the following direction cosines in the local coordinate system if the *TRANSFORM option was used at this node:

1. *X*-direction cosine of the outward normal to the exposed transition section area, pointing into the fluid, in the initial configuration.
2. *Y*-direction cosine of the outward normal to the exposed transition section area, pointing into the fluid, in the initial configuration.
3. *Z*-direction cosine of the outward normal to the exposed transition section area, pointing into the fluid, in the initial configuration.

Repeat this pair of data lines as often as necessary to define concentrated fluid inertia loading for various nodes or node sets.

*CLUSTER MASS INERTIA TABLE: Specify mass and inertia discrete particle clusters.

This option is used to define the mass and inertia for PD3D element clusters.

Products: Abaqus/Explicit

Type: Model data

Level: Model

Abaqus/CAE: Unsupported

References:

• *Discrete Element Method*

There are no parameters associated with this option.

Data lines to define the cluster mass inertia table:

Data lines to define the cluster mass and inertia as a function of the parent particle radius:

1. M.
2. I_{xx}.
3. I_{xy}.
4. I_{xz}.
5. I_{yy}.
6. I_{yz}.
7. I_{zz}.
8. R.

Repeat this data line as often as necessary to define the table of mass and inertia values of the cluster as a function of the parent particle radius.

*COHESIVE BEHAVIOR: Specify contact cohesive behavior properties.

This option is used to define contact cohesive behavior in a mechanical contact analysis. It must be used in conjunction with the **SURFACE INTERACTION* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Contact Cohesive Behavior*
- **SURFACE INTERACTION*

Optional, mutually exclusive parameters:

COHERE

Set COHERE=ORIGINAL CONTACTS (default) to restrict cohesive behavior to only those nodes of the secondary surface that are in contact with the main surface at the beginning of a simulation.

Set COHERE=FIRST CONTACTS to establish cohesive bonds for active contact constraints at the beginning of a simulation plus the first time an initially not-in-contact region comes into contact during a simulation. This parameter value is not allowed for general contact in Abaqus/Standard.

Set COHERE=REPEATED CONTACTS to allow cohesive rebonding each time contact is established, even for nodes previously involved in cohesive contact that have fully damaged and debonded.

ELIGIBILITY

Set ELIGIBILITY=ORIGINAL CONTACTS (default) to restrict cohesive behavior to only those nodes of the secondary surface that are in contact with the main surface at the beginning of a simulation.

Set ELIGIBILITY=CURRENT CONTACTS to define cohesive behavior not only for all nodes of the secondary surface that are in contact with the main surface at the start of a step, but also for secondary nodes that are not initially in contact but may come in contact during the course of a step. This setting must be used together with the REPEATED CONTACTS parameter for general contact in Abaqus/Standard.

Set ELIGIBILITY=SPECIFIED CONTACTS to restrict cohesive behavior to a subset of the secondary nodes defined using **INITIAL CONDITIONS*, TYPE=CONTACT. This parameter value is available only for contact pairs in Abaqus/Standard.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the moduli. If this parameter is omitted, it is assumed that the moduli are constant or depend only on temperature. See *Material Data Definition* for more information.

REPEATED CONTACTS

Include this parameter to modify the default post-failure behavior when progressive damage has been defined and the COHERE parameter is not specified. By default, cohesive behavior is not enforced for nodes on the secondary surface once ultimate failure has occurred at those nodes. Use the REPEATED CONTACTS parameter to enforce cohesive behavior for recurrent contacts at nodes on the secondary surface subsequent to ultimate failure.

TYPE

Set TYPE=UNCOUPLED (default) to define uncoupled traction behavior.

Set TYPE=COUPLED to define coupled traction behavior.

Data lines to define uncoupled traction separation behavior (TYPE=UNCOUPLED):

First line:

1. K_{nn}. (Units of FL^{-3}.)
2. K_{ss}.
3. K_{tt}.
4. Temperature.
5. First field variable.
6. Etc., up to four field variables per line.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four; relevant only for defining uncoupled traction behavior):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

Data lines to define coupled traction separation behavior (TYPE=COUPLED):

First line:

1. K_{nn}. (Units of FL^{-3}.)
2. K_{ss}.
3. K_{tt}.
4. K_{ns}.
5. K_{nt}.
6. K_{st}.
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

*COHESIVE SECTION: Specify element properties for cohesive elements.

This option is used to define the properties of cohesive elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- *About Cohesive Elements*
- *Defining the Constitutive Response of Cohesive Elements Using a Continuum Approach*
- *Defining the Constitutive Response of Cohesive Elements Using a Traction-Separation Description*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the elements for which the cohesive properties are being defined.

MATERIAL

Set this parameter equal to the name of the material to be used with these elements.

RESPONSE

This parameter specifies the geometric assumption that defines the constitutive behavior of the cohesive elements.

Set RESPONSE=TRACTION SEPARATION if the response is defined directly in terms of traction and separation.

Set RESPONSE=CONTINUUM to specify that the cohesive elements model a strain state involving one direct (opening strain) and two transverse shear components.

Set RESPONSE=GASKET to specify that the stress state in the cohesive elements is uniaxial.

When RESPONSE=CONTINUUM or GASKET, the constitutive behavior of the element must be defined in terms of continuum material properties using any available material model in Abaqus (subject to the limitation that certain models are not available for a one-dimensional stress state).

Optional parameters:

CONTROLS

Set this parameter equal to the name of a **SECTION CONTROLS* definition (see *Section Controls*). The **SECTION CONTROLS* option can be used to specify whether the cohesive elements should be deleted once they completely fail. This option may also be used to specify a maximum value of the scalar degradation (damage) parameter, D, and/or the viscosity coefficient, μ, for viscous regularization.

HALF

Include this parameter to enforce the symmetric modeling of cohesive elements in Abaqus/Standard.

This parameter is recommended if the cohesive elements are connected to pipe elements.

If cohesive elements without pore pressure degrees of freedom are used, including this parameter is the same as including the SYMMETRIC parameter.

If cohesive elements with pore pressure degrees of freedom are used to model tangential flow while **CFLOW* is used to define the fluid injected into the middle plane of the cohesive elements, only half of the flow magnitude as that for a corresponding full model should be used.

ORIENTATION

Set this parameter equal to the name given for the **ORIENTATION* option (*Orientations*) to be used to define a local coordinate system for integration point calculations in the cohesive elements in the specified element set.

STACK DIRECTION

Set this parameter equal to 1, 2, 3, or ORIENTATION to define the cohesive element stack or thickness direction. Specify one of the numerical values to select the corresponding isoparametric direction of the element as the stack or thickness direction. The default is STACK DIRECTION=3 for three-dimensional cohesive elements and STACK DIRECTION=2 for two-dimensional and axisymmetric elements.

If STACK DIRECTION=ORIENTATION, the ORIENTATION parameter is also required.

To obtain a desired thickness direction, the appropriate numerical value for the STACK DIRECTION parameter depends on the element connectivity. For a mesh-independent specification, use STACK DIRECTION=ORIENTATION.

This parameter cannot be used with pore pressure cohesive elements.

SYMMETRIC

Include this parameter to enforce the symmetric modeling of cohesive elements in Abaqus/Standard.

If cohesive elements without pore pressure degrees of freedom are used, including this parameter is the same as including the HALF parameter.

If cohesive elements with pore pressure degrees of freedom are used to model tangential flow while **CFLOW* is used to define the fluid injected into the middle plane of the cohesive elements, the same flow magnitude as that for a corresponding full model should be used.

THICKNESS

Set THICKNESS=GEOMETRY if the initial constitutive thickness of the cohesive layer is determined from the nodal coordinates of the elements.

Set THICKNESS=SPECIFIED to specify the initial constitutive thickness of the layer on the data line below. If the data field representing the initial constitutive thickness is left blank or set equal to zero, a unit thickness is assumed.

The default value of this parameter depends on the choice of the RESPONSE parameter. If RESPONSE=TRACTION SEPARATION, the default is THICKNESS=SPECIFIED. If RESPONSE=CONTINUUM, the default is THICKNESS=GEOMETRY. If RESPONSE=GASKET, there is no default; the THICKNESS parameter must be stated explicitly.

Data line to define the attributes of cohesive elements:

First (and only) line:

1. Initial constitutive thickness of the cohesive element.
2. Out-of-plane thickness for two-dimensional cohesive elements. The default is 1.0. The value is ignored for cohesive elements that do not require this input.

*COMBINATORIAL RULE: Define controls associated with combinatorial rules to derive interaction properties from surface properties.

This option is used to define controls associated with combinatorial rules that are used to determine interaction properties as functions of surface properties.

Products: Abaqus/Explicit

Type: Model data

Level: Step

References:

- *Deriving Friction Coefficients from Quantities Specified as Surface Properties*
- *Frictional Behavior*
- *Surface-Based Friction Coefficients*

Optional parameters:

ALPHA

Set this parameter equal to α in the equation $\mu_{AB} = \left[1 + \alpha\left(1 - \frac{\mu_A}{\mu_B}\right)\right]\mu_A$, where $\mu_A \leq \mu_B$, to compute a friction coefficient for contact, μ_{AB}, based on friction coefficients μ_A and μ_B specified as surface properties.

ANISOTROPY WEIGHTING

This parameter applies only if friction anisotropy is specified as a surface property.

Set ANISOTROPY WEIGHTING=BALANCED to specify a balanced weighting method for combining surface-based directional preferences for the anisotropic friction model.

Set ANISOTROPY WEIGHTING=PROPORTIONAL to specify an epsilon-proportional weighting method for combining surface-based directional preferences for the anisotropic friction model.

Set ANISOTROPY WEIGHTING=DOMINANT to specify the maximum-epsilon-dominant weighting method for combining surface-based directional preferences for the anisotropic friction model.

There are no data lines associated with this option.

*COMBINED TEST DATA: Specify simultaneously the normalized shear and bulk compliances or relaxation moduli as functions of time.

This option can be used only in conjunction with the *VISCOELASTIC option and cannot be used if the *SHEAR TEST DATA and *VOLUMETRIC TEST DATA options are used.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Time Domain Viscoelasticity*
- *VISCOELASTIC

Optional parameters:

SHRINF

To specify creep test data, set this parameter equal to the value of the long-term, normalized shear compliance $j_S(\infty)$.

To specify relaxation test data, set this parameter equal to the value of the long-term, normalized shear modulus $g_R(\infty)$.

The shear compliance and shear modulus are related by $j_S(\infty) = 1/g_R(\infty)$. The fitting procedure will use the specified value in the constraint $1 - \sum_{i=1}^{N} \overline{g}_i^P = g_R(\infty)$.

VOLINF

To specify creep test data, set this parameter equal to the value of the long-term, normalized volumetric compliance $j_K(\infty)$.

To specify relaxation test data, set this parameter equal to the value of the long-term normalized volumetric modulus $k_R(\infty)$. The volumetric compliance and volumetric modulus are related by $j_K(\infty) = 1/k_R(\infty)$. The fitting procedure will use this value in the constraint $1 - \sum_{i=1}^{N} \overline{k}_i^P = k_R(\infty)$.

Data lines to specify creep test data:

First line:

1. Normalized shear compliance $j_S(t), \quad (j_S(t) \geq 1)$.
2. Normalized volumetric (bulk) compliance $j_K(t), \quad (j_K(t) \geq 1)$.
3. Time t, $\quad (t > 0)$.

Repeat the above data line as often as necessary to give the compliance-time data.

Data lines to specify relaxation test data:

First line:

1. Normalized shear modulus $g_R(t), \quad (0 \le g_R(t) \le 1)$.
2. Normalized volumetric (bulk) modulus $k_R(t), \quad (0 \le k_R(t) \le 1)$.
3. Time t, $\quad (t > 0)$.

Repeat the above data line as often as necessary to give the modulus-time data.

*COMPLEX FREQUENCY: Extract complex eigenvalues and modal vectors.

This option is used to perform eigenvalue extraction to calculate the complex eigenvalues and corresponding complex mode shapes of a system.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

• *Complex Eigenvalue Extraction*

Optional parameters:

FRICTION DAMPING

Set FRICTION DAMPING=NO (default) to ignore friction-induced damping effects.

Set FRICTION DAMPING=YES to include friction-induced damping effects.

NORMALIZATION

Set NORMALIZATION=DISPLACEMENT (default for SIM-based analyses) to normalize the complex eigenvectors so that the real part of the largest value in each vector is unity and the imaginary part is zero.

Set NORMALIZATION=MODAL (the only option for analyses that are not based on the SIM architecture) to normalize only the complex eigenvectors of the projected system.

PROPERTY EVALUATION

Set this parameter equal to the frequency at which to evaluate frequency-dependent properties for viscoelasticity, springs, and dashpots during complex eigenvalue extraction. If this parameter is omitted, Abaqus/Standard will evaluate the material properties associated with frequency-dependent springs and dashpots at zero frequency and will not consider the contributions from frequency-domain viscoelasticity in the **COMPLEX FREQUENCY* step.

STIFFNESS PROJECTION

Set STIFFNESS PROJECTION=YES (default) to project the stiffness operator on the modal subspace.

Set STIFFNESS PROJECTION=NO to skip projection of the stiffness operator on the modal subspace. In this case the diagonal matrix of the real eigenvalues corresponding to the modal subspace basis vectors is used as a modal stiffness operator and some advanced mechanical effects (such as friction-induced effects, frequency-dependent properties, and other effects) are ignored. This option can significantly reduce the computational time in the stiffness projection phase of the analysis, but it should be used with caution.

Set STIFFNESS PROJECTION=CONTACT ONLY if advanced mechanical properties (for example, friction) are localized in the contact area only and the rest of the model has the same properties as in the preceding real eigenfrequency extraction analysis. In this case the modal stiffness operator is combined from the diagonal matrix of the real eigenvalues corresponding to the modal subspace basis vectors and the projection of the additional contact stiffness terms on the modal subspace.

This option can significantly reduce the computational time in the stiffness projection phase of the analysis, but it should be used with caution.

UNSTABLE MODES ONLY

Set this parameter equal to the cutoff value for complex modes. Only complex modes with the real part of the eigenvalue higher than the cutoff value are processed for output. The default value of this parameter is 0.0. If this parameter is omitted, all complex modes are output.

Optional, mutually exclusive parameters:

LEFT EIGENVECTORS

Include this parameter to request left complex eigenvectors. This parameter is relevant only in analyses that are based on the SIM architecture.

RIGHT EIGENVECTORS

Include this parameter (default) to request right complex eigenvectors.

Data line for complex eigenvalue extraction:

First (and only) line:

1. Number of complex eigenmodes to be extracted. If this entry is omitted, all the eigenmodes available in the projected subspace, formulated on the basis of all eigenmodes computed in the preceding **FREQUENCY* step and possibly reduced by using the **SELECT EIGENMODES* option, will be extracted.
2. Minimum frequency of interest, in cycles per time. If this field is left blank, no minimum is set.
3. Maximum frequency of interest, in cycles per time. If this field is left blank, no maximum is set.
4. Shift point, S, in cycles per time ($S \quad 0$). The eigenvalues with the imaginary part closest to this point are extracted. The default value is zero.

*COMPOSITE MODAL DAMPING: Specify composite modal damping for modal analyses based on the SIM architecture.

This option can be used only with the SIM architecture and can be used only with the **FREQUENCY* option. It specifies the composite modal damping data that enable calculation of the weighted mass and stiffness composite damping per each of the extracted eigenmodes during an eigenvalue extraction analysis. The damping data are requested for the mass proportional fraction of critical damping and for the stiffness proportional fraction of critical damping for specified element sets. This option can also be used to assign critical damping fractions to both mass and stiffness matrix input.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Material Damping*
- *About Dynamic Analysis Procedures*

Optional parameters:

MASS MATRIX INPUT

Set this parameter equal to the value of fraction of critical damping for all mass matrices included in the model by using the **MATRIX INPUT*, MATRIX=MASS option. If this parameter is omitted, the default is zero.

STIFFNESS MATRIX INPUT

Set this parameter equal to the value of fraction of critical damping for all stiffness matrices included in the model by using the **MATRIX INPUT*, MATRIX=STIFFNESS option. If this parameter is omitted, the default is zero.

Data lines to define a fraction of critical damping:

First line:

1. Element number or element set name.
2. Fraction of critical damping, ξ_M, for mass matrices included in this element or element set.
3. Fraction of critical damping, ξ_K, for stiffness matrices included in this element or element set.

Repeat this data line as often as necessary to define modal damping for different elements and/or element sets.

*CONCENTRATION TENSOR: Define the concentration tensor for an inclusion in the aggregate.

This option is used to define the concentration tensor, which is used to calculate the strain or temperature gradient in an inclusion (including the void) directly using the strain or temperature gradient of the aggregate. This option should appear immediately after the *CONSTITUENT* option. If this option is defined in one of the *CONSTITUENT* options, it must be defined in all nonmatrix type constituents.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- **CONSTITUENT*
- **MEAN FIELD HOMOGENIZATION*
- *Mean-Field Homogenization*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the concentration tensor. If this parameter is omitted, it is assumed that the concentration tensor is constant or depends only on the temperature.

TYPE

Set TYPE=CONDUCTIVITY to define the second-order conductivity concentration tensor used to calculate the temperature gradient in a constituent.

Set TYPE=STRAIN (default) to define the fourth-order strain concentration tensor used to calculate the strain in a constituent.

Data lines to define components of the second-order conductivity concentration tensor (TYPE=CONDUCTIVITY):

First line:

1. b_{11}.
2. b_{12}.
3. b_{13}.
4. b_{21}.
5. b_{22}.
6. b_{23}.
7. b_{31}.
8. b_{32}.

Second line:

1. b_{33}.
2. Temperature.
3. First field variable.

4. Second field variable.
5. Etc. up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the conductivity concentration tensor as a function of temperature and other predefined field variables.

Data lines to define components of the fourth-order strain concentration tensor (TYPE=STRAIN):

First line:

1. B_{1111}.
2. B_{1122}.
3. B_{1133}.
4. B_{1112}.
5. B_{1113}.
6. B_{1123}.
7. B_{2211}.
8. B_{2222}.

Second line:

1. B_{2233}.
2. B_{2212}.
3. B_{2213}.
4. B_{2223}.
5. B_{3311}.
6. B_{3322}.
7. B_{3333}.
8. B_{3312}.

Third line:

1. B_{3313}.
2. B_{3323}.
3. B_{1211}.
4. B_{1222}.
5. B_{1233}.
6. B_{1212}.
7. B_{1213}.
8. B_{1223}.

Fourth line:

1. B_{1311}.
2. B_{1322}.

3. B_{1333}.
4. B_{1312}.
5. B_{1313}.
6. B_{1323}.
7. B_{2311}.
8. B_{2322}.

Fifth line:

1. B_{2333}.
2. B_{2312}.
3. B_{2313}.
4. B_{2323}.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the strain concentration tensor as a function of temperature and other predefined field variables.

*CONCRETE: Define concrete properties beyond the elastic range.



Warning: Success in analyzing plain and reinforced concrete problems depends significantly on making sensible choices regarding the concrete material parameters described in this section as well as, in the case of reinforced concrete, the definition of rebar in the problem. It is important to be familiar with the issues relating to concrete modeling and rebar definition by referring to *Concrete Smeared Cracking*, *Defining Rebar as an Element Property*, and the appropriate sections in the *Abaqus Theory Guide* and the *Abaqus Example Problems Guide*.

The **CONCRETE* option is used to define the properties of plain concrete outside the elastic range in an Abaqus/Standard analysis. It must be used in conjunction with the **TENSION STIFFENING* option and may also appear with the **SHEAR RETENTION* and **FAILURE RATIOS* options. The properties and locations of reinforcement bars are given separately (*Defining Rebar as an Element Property*).

The **BRITTLE CRACKING* option is used to analyze concrete in an Abaqus/Explicit analysis (see *Cracking Model for Concrete*).

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Concrete Smeared Cracking*
- **TENSION STIFFENING*
- **SHEAR RETENTION*
- **FAILURE RATIOS*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the compressive yield stress, in addition to temperature. If this parameter is omitted, it is assumed that the compressive yield stress depends only on the plastic strain and, possibly, on temperature. See *Material Data Definition* for more information.

Data lines to define the concrete properties:

First line:

1. Absolute value of compressive stress. (Units of FL^{-2}.)
2. Absolute value of plastic strain. The first stress-strain point given at each value of temperature and field variable must be at zero plastic strain and will define the initial yield point for that temperature and field variable.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.

2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of compressive yield stress on plastic strain and, if needed, on temperature and other predefined field variables.

*CONCRETE COMPRESSION DAMAGE: Define compression damage properties for the concrete damaged plasticity model.

This option is used to define compression damage (or stiffness degradation) properties for the concrete damaged plasticity material model. The **CONCRETE COMPRESSION DAMAGE* option must be used in conjunction with the **CONCRETE DAMAGED PLASTICITY*, **CONCRETE TENSION STIFFENING*, and **CONCRETE COMPRESSION HARDENING* options. In addition, the option can be used with the **CONCRETE TENSION DAMAGE* option to specify tensile stiffness degradation damage and the **CONCRETE FAILURE* option to define material failure criteria.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Concrete Damaged Plasticity*
- **CONCRETE DAMAGED PLASTICITY*
- **CONCRETE TENSION STIFFENING*
- **CONCRETE COMPRESSION HARDENING*
- **CONCRETE TENSION DAMAGE*
- **CONCRETE FAILURE*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the compression damage, in addition to temperature. If this parameter is omitted, it is assumed that the compression damage behavior depends only on temperature. See *Material Data Definition* for more information.

TENSION RECOVERY

This parameter is used to define the stiffness recovery factor w_t, which determines the amount of tension stiffness that is recovered as the loading changes from compression to tension. If $w_t = 1$, the material fully recovers the tensile stiffness; if $w_t = 0$, there is no stiffness recovery. Intermediate values of $w_t (0 \le w_t \le 1)$ result in partial recovery of the tensile stiffness. The default value is 0.0.

Data lines to define compression damage:

First line:

1. Compressive damage variable, d_c.
2. Inelastic (crushing) strain, $\tilde{\varepsilon}_c^{in}$.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

The first point at each value of temperature must have a crushing strain of 0.0 and a compressive damage value of 0.0.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the compressive damage behavior on crushing strain, temperature, and other predefined field variables.

*CONCRETE COMPRESSION HARDENING: Define hardening in compression for the concrete damaged plasticity model.

This option is used to define the compression hardening data for the concrete damaged plasticity material model. It must be used in conjunction with the *CONCRETE DAMAGED PLASTICITY and *CONCRETE TENSION STIFFENING options. In addition, the option can be used with the *CONCRETE TENSION DAMAGE and/or *CONCRETE COMPRESSION DAMAGE options to specify tensile and/or compressive stiffness degradation damage and the *CONCRETE FAILURE option to define material failure criteria.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Concrete Damaged Plasticity*
- *CONCRETE DAMAGED PLASTICITY
- *CONCRETE TENSION STIFFENING
- *CONCRETE TENSION DAMAGE
- *CONCRETE COMPRESSION DAMAGE
- *CONCRETE FAILURE

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the compressive yield stress, in addition to temperature. If this parameter is omitted, the compressive yield stress depends only on the inelastic strain, the strain rate, and, possibly, on temperature. See *Material Data Definition* for more information.

Data lines to define compressive hardening:

First line:

1. Yield stress in compression, σ_c. (Units of $\mathbf{FL^{-2}}$.)
2. Inelastic (crushing) strain, $\tilde{\varepsilon}_c^{in}$.
3. Inelastic (crushing) strain rate, $\dot{\tilde{\varepsilon}}_c^{in}$. (Units of $\mathbf{T^{-1}}$.)
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

The first point at each value of temperature must have a crushing strain of 0.0 and gives the initial yield stress value, σ_{c0}.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the compressive yield stress on crushing strain, crushing strain rate, and other predefined field variables.

*CONCRETE DAMAGED PLASTICITY: Define flow potential, yield surface, and viscosity parameters for the concrete damaged plasticity model.

This option is used to define flow potential, yield surface, and viscosity parameters for the concrete damaged plasticity material model. The **CONCRETE DAMAGED PLASTICITY* option must be used in conjunction with the **CONCRETE TENSION STIFFENING* and the **CONCRETE COMPRESSION HARDENING* options. In addition, the option can be used with the **CONCRETE TENSION DAMAGE* and/or the **CONCRETE COMPRESSION DAMAGE* options to specify tensile and/or compressive stiffness degradation damage and the **CONCRETE FAILURE* option to define material failure criteria.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Concrete Damaged Plasticity*
- **CONCRETE TENSION STIFFENING*
- **CONCRETE COMPRESSION HARDENING*
- **CONCRETE TENSION DAMAGE*
- **CONCRETE COMPRESSION DAMAGE*
- **CONCRETE FAILURE*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the material parameters other than temperature. If this parameter is omitted, it is assumed that the material parameters depend only on temperature. See *Material Data Definition* for more information.

REF LENGTH

Set this parameter equal to the specimen length, l_0, used to properly account for the length units when converting the cracking displacement values to "plastic" displacement values. By default, the value is $l_0 = 1.0$, which means one unit length. (Units of L.)

Data lines to define concrete damaged plasticity flow potential, yield surface, and viscosity parameters:

First line:

1. Dilation angle, ψ, in the p–q plane. Give the value in degrees.
2. Flow potential eccentricity, ϵ. The eccentricity is a small positive number that defines the rate at which the hyperbolic flow potential approaches its asymptote. If this field is left blank or a value of 0.0 is entered, the default of $\epsilon = 0.1$ is used.
3. σ_{b0}/σ_{c0}, the ratio of initial equibiaxial compressive yield stress to initial uniaxial compressive yield stress. If this field is left blank or a value of 0.0 is entered, the default of **1.16** is used.

4. K_c, the ratio of the second stress invariant on the tensile meridian, $q_{(TM)}$, to that on the compressive meridian, $q_{(CM)}$, at initial yield for any given value of the pressure invariant p such that the maximum principal stress is negative, $\hat{\sigma}_{max} < 0$. It must satisfy the condition $0.5 < K_c \leq 1.0$. If this field is left blank or a value of 0.0 is entered, the default of $2/3$ is used.
5. Viscosity parameter, μ, used for the visco-plastic regularization of the concrete constitutive equations in Abaqus/Standard analyses. This parameter is ignored in Abaqus/Explicit. The default value is 0.0. (Units of T.)
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the material parameters on temperature and other predefined field variables.

*CONCRETE FAILURE: Specify material failure criteria and allow element deletion for the concrete damaged plasticity material model.

This option is used to define material failure criteria for the concrete damaged plasticity material model. The **CONCRETE FAILURE* option must be used in conjunction with the **CONCRETE DAMAGED PLASTICITY*, **CONCRETE TENSION STIFFENING*, and **CONCRETE COMPRESSION HARDENING* options. In addition, the option can be used with the **CONCRETE TENSION DAMAGE* and/or **CONCRETE COMPRESSION DAMAGE* option to specify material failure criteria based on tensile and/or compressive stiffness degradation damage, respectively.

Products: Abaqus/Explicit

Type: Model data

Level: Model

References:

- **CONCRETE DAMAGED PLASTICITY*
- **CONCRETE TENSION STIFFENING*
- **CONCRETE COMPRESSION HARDENING*
- **CONCRETE TENSION DAMAGE*
- **CONCRETE COMPRESSION DAMAGE*
- *Concrete Damaged Plasticity*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the failure criteria, in addition to temperature. If this parameter is omitted, it is assumed that the failure criteria are constant or depend only on temperature. See *Material Data Definition* for more information.

TYPE

Set TYPE=STRAIN (default) to specify the tensile failure based on a tensile cracking strain criterion.

Set TYPE=DISPLACEMENT to specify the tensile failure based on a tensile cracking displacement criterion.

Data lines if the tensile failure is defined based on a tensile cracking strain criterion (TYPE=STRAIN):

First line:

1. Tensile cracking strain at failure, $\left(\tilde{\varepsilon}_t^{ck}\right)_f$.
2. Compressive inelastic (crushing) strain at failure, $\left(\tilde{\varepsilon}_c^{in}\right)_f$.
3. Tensile damage variable at failure, $\left(d_t\right)_f$, if the **CONCRETE TENSION DAMAGE* option is specified for the material.
4. Compressive damage variable at failure, $\left(d_c\right)_f$, if the **CONCRETE COMPRESSION DAMAGE* option is specified for the material.
5. Temperature.

6. First field variable.
7. Second field variable.
8. Etc., up to three field variables.

A material failure criterion is considered only if the corresponding data line entry is positive.

Second line (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this data line as often as necessary to define the material failure criteria as a function of temperature and other predefined field variables.

Data lines if the tensile failure is defined based on a tensile cracking displacement criterion (TYPE=DISPLACEMENT):

First line:

1. Tensile cracking displacement at failure, $\left(u_t^{ck}\right)_f$. (Units of L.)
2. Compressive inelastic (crushing) strain at failure, $\left(\tilde{\varepsilon}_c^{in}\right)_f$.
3. Tensile damage variable at failure, $\left(d_t\right)_f$, if the **CONCRETE TENSION DAMAGE* option is specified for the material.
4. Compressive damage variable at failure, $\left(d_c\right)_f$, if the **CONCRETE COMPRESSION DAMAGE* option is specified for the material.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Etc., up to three field variables.

A material failure criterion is considered only if the corresponding data line entry is positive.

Second line (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this data line as often as necessary to define the material failure criteria as a function of temperature and other predefined field variables.

*CONCRETE TENSION DAMAGE: Define postcracking damage properties for the concrete damaged plasticity model.

This option is used to define postcracking damage (or stiffness degradation) properties for the concrete damaged plasticity material model. The **CONCRETE TENSION DAMAGE* option must be used in conjunction with the **CONCRETE DAMAGED PLASTICITY*, **CONCRETE TENSION STIFFENING*, and **CONCRETE COMPRESSION HARDENING* options. In addition, the option can be used with the **CONCRETE COMPRESSION DAMAGE* option to specify compressive stiffness degradation damage and the **CONCRETE FAILURE* option to define material failure criteria.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Concrete Damaged Plasticity*
- **CONCRETE DAMAGED PLASTICITY*
- **CONCRETE TENSION STIFFENING*
- **CONCRETE COMPRESSION HARDENING*
- **CONCRETE COMPRESSION DAMAGE*
- **CONCRETE FAILURE*

Optional parameters:

COMPRESSION RECOVERY

This parameter is used to define the stiffness recovery factor, w_c, which determines the amount of compression stiffness that is recovered as the loading changes from tension to compression. If $w_c = 1$, the material fully recovers the compressive stiffness; if $w_c = 0$, there is no stiffness recovery.

Intermediate values of $w_c (0 \le w_c \le 1)$ result in partial recovery of the compressive stiffness. The default value is **1.0**, which corresponds to the assumption that as cracks close the compressive stiffness is unaffected by tensile damage.

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the tension damage, in addition to temperature. If this parameter is omitted, it is assumed that the tension damage behavior depends only on temperature. See *Material Data Definition* for more information.

TYPE

Set TYPE=STRAIN (default) to specify the tensile damage variable as a function of cracking strain.

Set TYPE=DISPLACEMENT to specify the tensile damage variable as a function of cracking displacement.

Data lines if the tensile damage is specified as a function of cracking strain (TYPE=STRAIN):

First line:

1. Tensile damage variable, d_t.
2. Direct cracking strain, $\tilde{\varepsilon}_t^{ck}$.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

The first point at each value of temperature must have a cracking strain of 0.0 and a tensile damage value of 0.0.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the tensile damage behavior on cracking strain, temperature, and other predefined field variables.

Data lines if the tensile damage is specified as a function of cracking displacement (TYPE=DISPLACEMENT):

First line:

1. Tensile damage variable, d_t.
2. Direct cracking displacement, u_t^{ck}. (Units of L.)
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

The first point at each value of temperature must have a cracking displacement of 0.0 and a tensile damage value of 0.0.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the tensile damage behavior on cracking displacement, temperature, and other predefined field variables.

*CONCRETE TENSION STIFFENING: Define postcracking properties for the concrete damaged plasticity model.

This option is used to define cracking and postcracking properties for the concrete damaged plasticity material model. The **CONCRETE TENSION STIFFENING* option must be used in conjunction with the **CONCRETE DAMAGED PLASTICITY* and **CONCRETE COMPRESSION HARDENING* options. In addition, the option can be used with the **CONCRETE TENSION DAMAGE* and/or **CONCRETE COMPRESSION DAMAGE* options to specify tensile and/or compressive stiffness degradation damage and the **CONCRETE FAILURE* option to define material failure criteria.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Concrete Damaged Plasticity*
- **CONCRETE DAMAGED PLASTICITY*
- **CONCRETE COMPRESSION HARDENING*
- **CONCRETE TENSION DAMAGE*
- **CONCRETE COMPRESSION DAMAGE*
- **CONCRETE FAILURE*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the postcracking behavior, in addition to temperature. If this parameter is omitted, the postcracking stress depends only on the cracking strain, the strain rate, and, possibly, on temperature. See *Material Data Definition* for more information.

TYPE

Set TYPE=STRAIN (default) to specify the postcracking behavior by entering the postfailure stress/cracking-strain relationship.

Set TYPE=DISPLACEMENT to define the postcracking behavior by entering the postfailure stress/cracking-displacement relationship.

Set TYPE=GFI to define the postcracking behavior by entering the failure stress, σ_{t0}, and the fracture energy, G_f.

Data lines if the TYPE=STRAIN parameter is included (default):

First line:

1. Remaining direct stress after cracking, σ_t. (Units of $\mathbf{FL^{-2}}$.)
2. Direct cracking strain, $\tilde{\varepsilon}_t^{ck}$.
3. Direct cracking strain rate, $\dot{\tilde{\varepsilon}}_t^{ck}$. (Units of $\mathbf{T^{-1}}$.)

4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

The first point at each value of temperature must have a cracking strain of 0.0 and gives the failure stress value, σ_{t0}.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the postcracking behavior on temperature and other predefined field variables.

Data lines if the TYPE=DISPLACEMENT parameter is included:

First line:

1. Remaining direct stress after cracking, σ_t. (Units of $\mathbf{FL^{-2}}$.)
2. Direct cracking displacement, u_t^{ck}. (Units of L.)
3. Direct cracking displacement rate, $\dot{u}_t^{ck}$. (Units of $\mathbf{LT^{-1}}$.)
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

The first point at each value of temperature must have a cracking displacement of 0.0 and gives the failure stress value.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the postcracking behavior on temperature and other predefined field variables.

Data lines if the TYPE=GFI parameter is included:

First line:

1. Failure stress, σ_{t0}. (Units of $\mathbf{FL^{-2}}$.)
2. Fracture energy, G_f. (Units of $\mathbf{FL^{-1}}$.)
3. Direct cracking displacement rate, $\dot{u}_t^{ck}$. (Units of $\mathbf{LT^{-1}}$.)
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the postcracking behavior on temperature and other predefined field variables.

*CONDUCTIVITY: Specify thermal conductivity.

This option is used to specify a material's thermal conductivity.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Conductivity*

Optional parameters:

DEPENDENCIES

This parameter applies only to Abaqus/Standard and Abaqus/Explicit analyses.

Set this parameter equal to the number of field variables included in the definition of conductivity. If this parameter is omitted, it is assumed that the conductivity is constant or depends only on temperature. See *Material Data Definition* for more information.

PORE FLUID

This parameter applies only to Abaqus/Standard analyses.

Include this parameter if the conductivity of the pore fluid in a porous medium is being defined.

The conductivity of a fluid must be isotropic; therefore, TYPE=ORTHO, TYPE=TRANSVERSELY ISOTROPIC, and TYPE=ANISO cannot be used if this parameter is included.

SLURRY

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to define the conductivity of a slurry in a geotechnical application that models slurry transport and placement. You can use this parameter only if you included the PORE FLUID parameter.

The conductivity of the slurry flow must be isotropic. Therefore, TYPE=ORTHO, TYPE=TRANSVERSELY ISOTROPIC, and TYPE=ANISO cannot be used if this parameter is included.

TYPE

Set TYPE=ISO (default) to define isotropic conductivity.

Set TYPE=ORTHO to define orthotropic conductivity.

Set TYPE=ANISO to define fully anisotropic conductivity.

Set TYPE=TRANSVERSELY ISOTROPIC to define transversely isotropic conductivity.

Data lines to define isotropic thermal conductivity (TYPE=ISO):

First line:

1. Conductivity, k. (Units of $JT^{-1}L^{-1}\theta^{-1}$.)
2. Temperature, if temperature dependent.

3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the thermal conductivity as a function of temperature and other predefined field variables.

Data lines to define orthotropic thermal conductivity (TYPE=ORTHO):

First line:

1. k_{11}. (Units of $JT^{-1}L^{-1}\theta^{-1}$.)
2. k_{22}.
3. k_{33}.
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the thermal conductivity as a function of temperature and other predefined field variables.

Data lines to define anisotropic thermal conductivity (TYPE=ANISO):

First line:

1. k_{11}. (Units of $JT^{-1}L^{-1}\theta^{-1}$.)
2. k_{12}.
3. k_{22}.
4. k_{13}.
5. k_{23}.
6. k_{33}.
7. Temperature, if temperature dependent.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the thermal conductivity as a function of temperature and other predefined field variables.

Data lines to define transversely isotropic thermal conductivity (TYPE=TRANSVERSELY ISOTROPIC):

First line:

1. k_{11}. (Units of $JT^{-1}L^{-1}\theta^{-1}$.)
2. k_{22}.
3. Temperature, if temperature dependent.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the thermal conductivity as a function of temperature and other predefined field variables.

Data lines to define isotropic thermal conductivity with the SLURRY parameter:

First line:

1. Conductivity of the slurry, k. (Units of $JT^{-1}L^{-1}\theta^{-1}$.)
2. Slurry concentration.
3. Temperature, if temperature dependent.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the thermal conductivity as a function of slurry concentration, temperature, and other predefined field variables.

*CONNECTOR BEHAVIOR: Begin the specification of a connector behavior.

This option is used to indicate the start of a connector behavior definition.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Connector Behavior*
- **CONNECTOR SECTION*

Required parameters:

NAME

Set this parameter equal to the behavior name referred to on the **CONNECTOR SECTION* option. Connector behavior names in the same input file must be unique.

Optional parameters:

EXTRAPOLATION

The choice of extrapolation defined here applies to all suboptions of the connector behavior unless it is redefined on the suboption.

Set EXTRAPOLATION=CONSTANT (default) to use constant extrapolation of the dependent variables outside the specified range of the independent variables.

Set EXTRAPOLATION=LINEAR to use linear extrapolation of the dependent variables outside the specified range of the independent variables.

INTEGRATION

This parameter applies only to Abaqus/Explicit analyses.

Set INTEGRATION=IMPLICIT (default) to integrate the connector behavior with implicit time integration.

Set INTEGRATION=EXPLICIT to integrate the connector behavior with explicit time integration.

REGULARIZE

This parameter applies only to Abaqus/Explicit analyses. The choice of regularization defined here applies to all suboptions of the connector behavior unless it is redefined on the suboption.

Set REGULARIZE=ON (default) to regularize the user-defined tabular connector behavior data.

Set REGULARIZE=OFF to use the user-defined tabular connector behavior data directly without regularization.

RTOL

This parameter applies only to Abaqus/Explicit analyses. The regularization tolerance defined here applies to all suboptions of the connector behavior unless it is redefined on the suboption.

Set this parameter equal to the tolerance to be used to regularize the connector behavior data. The default is RTOL=0.03.

There are no data lines associated with this option.

*CONNECTOR CONSTITUTIVE REFERENCE: Define reference lengths and angles to be used in specifying connector constitutive behavior.

This option is used to define reference lengths and angles for constitutive response in connector elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Connector Behavior*
- **CONNECTOR BEHAVIOR*

There are no parameters associated with this option.

Data line to define reference lengths and angles:

First (and only) line:

Enter a blank to use the (default) reference length or angle calculated from the initial geometry.

1. Reference length associated with the connector's first component of relative motion.
2. Reference length associated with the connector's second component of relative motion.
3. Reference length associated with the connector's third component of relative motion. Only relevant for three-dimensional analyses.
4. Reference angle (in degrees) associated with the connector's fourth component of relative motion. Only relevant for three-dimensional analyses.
5. Reference angle (in degrees) associated with the connector's fifth component of relative motion. Only relevant for three-dimensional analyses.
6. Reference angle (in degrees) associated with the connector's sixth component of relative motion.

*CONNECTOR DAMAGE EVOLUTION: Specify connector damage evolution for connector elements.

This option is used to define connector damage evolution for connector elements that have available components of relative motion. It must be used in conjunction with the *CONNECTOR DAMAGE INITIATION option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Connection Types*
- *Connector Behavior*
- *Connector Damage Behavior*
- *CONNECTOR BEHAVIOR
- *CONNECTOR DAMAGE INITIATION
- *CONNECTOR POTENTIAL

Required parameters:

TYPE

Set TYPE=MOTION to use either connector constitutive relative motions (displacements/rotations) or plastic relative motions (displacement/rotations) to specify damage evolution.

Set TYPE=ENERGY to use post-damage initiation dissipation energies to specify damage evolution.

Optional parameters:

AFFECTED COMPONENTS

Include this parameter to identify on the data line the components of relative motion that will be damaged.

If this parameter is omitted and the COMPONENT parameter is included on the associated *CONNECTOR DAMAGE INITIATION option, only the specified component will undergo damage.

If both this parameter and the COMPONENT parameter on the associated *CONNECTOR DAMAGE INITIATION option are omitted, only the components of relative motion involved in the associated *CONNECTOR POTENTIAL definition will undergo damage.

DEGRADATION

Set DEGRADATION=MAXIMUM (default) to indicate that the damage value associated with this option will be first compared to damage values from other damage mechanisms (if defined) and that only the maximum value will be considered for the overall damage.

Set DEGRADATION=MULTIPLICATIVE to indicate that the damage value associated with this option will contribute multiplicatively to the overall damage.

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the connector damage evolution, in addition to temperature. If this parameter is omitted, it is assumed

that the connector damage evolution is independent of field variables. See *Material Data Definition* for more information.

EXTRAPOLATION

Set EXTRAPOLATION=CONSTANT (default unless **CONNECTOR BEHAVIOR*, EXTRAPOLATION=LINEAR is used) to use constant extrapolation of the dependent variables outside the specified range of the independent variables.

Set EXTRAPOLATION=LINEAR to use linear extrapolation of the dependent variables outside the specified range of the independent variables.

REGULARIZE

This parameter applies only to Abaqus/Explicit analyses.

Set REGULARIZE=ON (default unless **CONNECTOR BEHAVIOR*, REGULARIZE=OFF is used) to regularize the user-defined tabular connector damage data.

Set REGULARIZE=OFF to use the user-defined tabular connector damage data directly without regularization.

RTOL

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the tolerance to be used to regularize the connector damage data.

If this parameter is omitted, the default is RTOL=0.03 unless the tolerance is specified on the **CONNECTOR BEHAVIOR* option.

SOFTENING

This parameter can be used only in conjunction with TYPE=MOTION.

Set SOFTENING=LINEAR (default) to specify a linear damage evolution law.

Set SOFTENING=EXPONENTIAL to specify an exponential damage evolution law.

Set SOFTENING=TABULAR to specify a damage evolution law in tabular form.

Data lines to define the damage evolution for TYPE=MOTION, SOFTENING=LINEAR:

First line (needed only if the AFFECTED COMPONENTS parameter is included):

1. First component of relative motion number that will be damaged.
2. Second component of relative motion number that will be damaged.
3. Etc., up to six entries.

Second line if the AFFECTED COMPONENTS parameter is included; otherwise, first line:

1. Post-initiation equivalent relative plastic motion at ultimate failure if CRITERION=PLASTIC MOTION is specified on the associated **CONNECTOR DAMAGE INITIATION* option. Otherwise, post-initiation constitutive relative motion (displacement/rotation) at ultimate failure. See *Connector Behavior* for a description of the connector relative motions.
2. Mode mix ratio if CRITERION=PLASTIC MOTION and the COMPONENT parameter is omitted from the associated **CONNECTOR DAMAGE INITIATION* option. Leave blank otherwise.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Do not repeat the data line that specifies the affected components. Repeat the subsequent set of data lines as often as necessary to define connector damage evolution by specifying the connector relative plastic or constitutive motion at ultimate failure as a function of mode mix ratio, temperature, and other predefined field variables.

Data lines to define the damage evolution for TYPE=MOTION, SOFTENING=EXPONENTIAL:

First line (needed only if the AFFECTED COMPONENTS parameter is included):

1. First component of relative motion number that will be damaged.
2. Second component of relative motion number that will be damaged.
3. Etc., up to six entries.

Second line if the AFFECTED COMPONENTS parameter is included; otherwise, first line:

1. Post-initiation equivalent relative plastic motion at ultimate failure if CRITERION=PLASTIC MOTION is specified on the associated **CONNECTOR DAMAGE INITIATION* option. Otherwise, post-initiation constitutive relative motion (displacement/rotation) at ultimate failure. See *Connector Behavior* for a description of the connector relative motions.
2. Exponential law parameter, α (see *Connector Damage Behavior*).
3. Mode mix ratio if CRITERION=PLASTIC MOTION and the COMPONENT parameter is omitted from the associated **CONNECTOR DAMAGE INITIATION* option. Leave blank otherwise.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Do not repeat the data line that specifies the affected components. Repeat the subsequent set of data lines as often as necessary to define connector damage evolution by specifying the connector relative plastic or constitutive motion at ultimate failure and the exponential law parameter as functions of mode mix ratio, temperature, and other predefined field variables.

Data lines to define the damage evolution for TYPE=MOTION, SOFTENING=TABULAR:

First line (needed only if the AFFECTED COMPONENTS parameter is included):

1. First component of relative motion number that will be damaged.
2. Second component of relative motion number that will be damaged.
3. Etc., up to six entries.

Second line if the AFFECTED COMPONENTS parameter is included; otherwise, first line:

1. Damage variable.
2. Post-initiation equivalent relative plastic motion if CRITERION=PLASTIC MOTION on the associated **CONNECTOR DAMAGE INITIATION* option. Otherwise, post-initiation constitutive relative motion (displacement/rotation). See *Connector Behavior* for a description of the connector relative motions.
3. Mode mix ratio if CRITERION=PLASTIC MOTION and the COMPONENT parameter is omitted from the associated **CONNECTOR DAMAGE INITIATION* option. Leave blank otherwise.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Do not repeat the data line that specifies the affected components. Repeat the subsequent set of data lines as often as necessary to define connector damage evolution as a function of connector relative plastic or constitutive motion, mode mix ratio, temperature, and other predefined field variables.

Data lines to define the damage evolution for TYPE=ENERGY:

First line (needed only if the AFFECTED COMPONENTS parameter is included):

1. First component of relative motion number that will be damaged.
2. Second component of relative motion number that will be damaged.
3. Etc., up to six entries.

Second line if the AFFECTED COMPONENTS parameter is included; otherwise, first line:

1. Total energy dissipated by damage at ultimate failure.
2. Mode mix ratio if CRITERION=PLASTIC MOTION and the COMPONENT parameter is omitted from the associated **CONNECTOR DAMAGE INITIATION* option. Leave blank otherwise.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Do not repeat the data line that specifies the affected components. Repeat the subsequent set of data lines as often as necessary to define connector damage evolution by specifying the post-initiation dissipation energy as a function of mode mix ratio, temperature, and other predefined field variables.

*CONNECTOR DAMAGE INITIATION: Specify connector damage initiation criteria for connector elements.

This option is used to define connector damage initiation criteria for connector elements that have available components of relative motion. It is almost always used in conjunction with the *CONNECTOR DAMAGE EVOLUTION option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Connection Types*
- *Connector Behavior*
- *Connector Damage Behavior*
- **CONNECTOR BEHAVIOR*
- **CONNECTOR DAMAGE EVOLUTION*
- **CONNECTOR PLASTICITY*
- **CONNECTOR POTENTIAL*

Optional parameters:

COMPONENT

Set this parameter equal to the connector's component of relative motion for which a connector damage initiation criterion is specified. See *Connection Types* for components of relative motion definitions. If this parameter is used, the **CONNECTOR POTENTIAL* option cannot be used in conjunction with the **CONNECTOR DAMAGE INITIATION* option.

Omit this parameter and use the **CONNECTOR POTENTIAL* option in conjunction with the **CONNECTOR DAMAGE INITIATION* option to specify a connector damage initiation criterion involving several components of relative motion.

CRITERION

Set CRITERION=FORCE (default) to specify a damage initiation criterion based on total forces/moments in the connector.

Set CRITERION=MOTION to specify a damage initiation criterion based on relative displacements/rotations in the connector.

Set CRITERION=PLASTIC MOTION to specify a damage initiation criterion based on the equivalent plastic relative motion as defined by the associated plasticity definition. If CRITERION=PLASTIC MOTION, the **CONNECTOR POTENTIAL* option cannot be used in conjunction with the **CONNECTOR DAMAGE INITIATION* option.

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the connector damage initiation criterion, in addition to temperature. If this parameter is omitted, it is assumed that the connector damage initiation criterion is independent of field variables. See *Material Data Definition* for more information.

EXTRAPOLATION

Set EXTRAPOLATION=CONSTANT (default unless *CONNECTOR BEHAVIOR, EXTRAPOLATION=LINEAR is used) to use constant extrapolation of the dependent variables outside the specified range of the independent variables.

Set EXTRAPOLATION=LINEAR to use linear extrapolation of the dependent variables outside the specified range of the independent variables.

RATE FILTER FACTOR

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the factor to be used for filtering the equivalent relative plastic motion rate for the evaluation of rate-dependent connector damage initiation data. The default value is 0.9.

RATE INTERPOLATION

This parameter applies only to Abaqus/Explicit analyses and is used only to interpolate rate-dependent connector damage initiation data.

Set RATE INTERPOLATION=LINEAR (default) to use linear intervals for the equivalent relative plastic motion rate while interpolating rate-dependent damage initiation data.

Set RATE INTERPOLATION=LOGARITHMIC to use logarithmic intervals for the equivalent relative plastic motion rate while interpolating rate-dependent damage initiation data.

REGULARIZE

This parameter applies only to Abaqus/Explicit analyses.

Set REGULARIZE=ON (default unless *CONNECTOR BEHAVIOR, REGULARIZE=OFF is used) to regularize the user-defined tabular connector damage initiation data.

Set REGULARIZE=OFF to use the user-defined tabular connector damage initiation data directly without regularization.

RTOL

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the tolerance to be used to regularize the connector damage initiation data.

If this parameter is omitted, the default is RTOL=0.03 unless the tolerance is specified on the *CONNECTOR BEHAVIOR option.

Data lines for CRITERION=FORCE:

First line:

1. Lower (compression) limiting force or moment. If not specified, no lower limit is used.
2. Upper (tension) limiting force or moment. If not specified, no upper limit is used.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the connector damage initiation limiting values as a function of temperature and other predefined field variables.

Data lines for CRITERION=MOTION:

First line:

1. Lower (compression) limiting connector constitutive relative displacement or rotation. If not specified, no lower limit is used.
2. Upper (tension) limiting connector constitutive relative displacement or rotation. If not specified, no upper limit is used.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the connector damage initiation limiting values as a function of temperature and other predefined field variables.

Data lines for CRITERION=PLASTIC MOTION:

First line:

1. Relative equivalent plastic displacement/rotation at which damage will be initiated.
2. Leave blank if the COMPONENT parameter is specified.

   Otherwise, mode mix ratio. See *Mode Mix Ratio* for information on how this quantity is defined.
3. Relative equivalent plastic displacement/rotation rate.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the connector damage initiation criterion as a function of mode mix ratio, equivalent plastic motion rate, temperature, and other predefined field variables.

*CONNECTOR DAMPING: Define connector damping behavior.

This option is used to define the damping behavior for connector elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Connector Behavior*
- *Connector Damping Behavior*
- **CONNECTOR BEHAVIOR*

Optional parameters:

COMPONENT

Set this parameter equal to the connector's component of relative motion for which damping behavior is specified. For this component of relative motion the connector will act as a dashpot for TYPE=VISCOUS. Omit this parameter to define coupled behavior.

TYPE

Set this parameter equal to VISCOUS (default) to specify velocity proportional damping.

Set this parameter equal to STRUCTURAL to specify displacement proportional damping. This setting applies to the relevant analyses listed below.

In mode-based analyses both types apply to analyses that support nondiagonal damping in Abaqus/Standard.

Optional parameters for TYPE=VISCOUS:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the connector damping data, in addition to temperature. If this parameter is omitted, it is assumed that the connector damping is independent of field variables. See *Material Data Definition* for more information.

EXTRAPOLATION

Set EXTRAPOLATION=CONSTANT (default unless **CONNECTOR BEHAVIOR*, EXTRAPOLATION=LINEAR is used) to use constant extrapolation of the dependent variables outside the specified range of the independent variables.

Set EXTRAPOLATION=LINEAR to use linear extrapolation of the dependent variables outside the specified range of the independent variables.

FREQUENCY DEPENDENCE

This parameter is relevant only for coupled linear viscous damping definitions in an Abaqus/Standard analysis. Use this parameter to define viscous damping terms with frequency dependence.

Set FREQUENCY DEPENDENCE=OFF (default) if frequency dependence of the damping terms is not defined.

Set FREQUENCY DEPENDENCE=ON if frequency dependence of the damping terms is defined.

INDEPENDENT COMPONENTS

This parameter can be used only if the COMPONENT and NONLINEAR parameters are included.

Set INDEPENDENT COMPONENTS=POSITION (default) to specify dependencies on components of relative position included in the damping definition.

Set INDEPENDENT COMPONENTS=CONSTITUTIVE MOTION to specify dependencies on components of constitutive relative motion included in the damping definition.

If damping is dependent on only the relative velocity in the component specified with the COMPONENT parameter, the INDEPENDENT COMPONENTS parameter should not be used.

NONLINEAR

This parameter can be used only if the COMPONENT parameter is included.

Include this parameter to define nonlinear damping behavior. Omit this parameter to define linear damping behavior.

REGULARIZE

This parameter applies only to Abaqus/Explicit analyses.

Set REGULARIZE=ON (default unless **CONNECTOR BEHAVIOR*, REGULARIZE=OFF is used) to regularize the user-defined tabular connector damping data.

Set REGULARIZE=OFF to use the user-defined tabular connector damping data directly without regularization.

RTOL

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the tolerance to be used to regularize the connector damping data.

If this parameter is omitted, the default is RTOL=0.03 unless the tolerance is specified on the **CONNECTOR BEHAVIOR* option.

UNSYMM

This parameter is relevant only for linear coupled viscous damping definitions in an Abaqus/Standard analysis.

Include this parameter if the linear coupled viscous damping matrices are not symmetric.

Data lines to define linear uncoupled viscous damping behavior (TYPE=VISCOUS, COMPONENT with the NONLINEAR parameter omitted):

First line:

1. Damping coefficient (force or moment per relative velocity).
2. Leave blank in an Abaqus/Explicit analysis. In an Abaqus/Standard analysis this field corresponds to frequency (in cycles per time). Applicable for **STEADY STATE DYNAMICS*, DIRECT; **STEADY STATE DYNAMICS*, SUBSPACE PROJECTION; and **STEADY STATE DYNAMICS* and **MODAL DYNAMIC* analyses that support nondiagonal damping.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the damping coefficient as a function of frequency, temperature, and other predefined field variables.

Data lines to define linear coupled viscous damping behavior (TYPE=VISCOUS with the COMPONENT, NONLINEAR, and UNSYMM parameters omitted; all 21 damping constants must be specified, regardless of whether temperature or field variable dependencies are included):

First line:

1. C_{11}. (Units of FTL^{-1}.)
2. C_{12}. (Units of FTL^{-1}.)
3. C_{22}. (Units of FTL^{-1}.)
4. C_{13}. (Units of FTL^{-1}.)
5. C_{23}. (Units of FTL^{-1}.)
6. C_{33}. (Units of FTL^{-1}.)
7. C_{14}. (Units of FT.)
8. C_{24}. (Units of FT.)

Second line:

1. C_{34}. (Units of FT.)
2. C_{44}. (Units of FTL.)
3. C_{15}. (Units of FT.)
4. C_{25}. (Units of FT.)
5. C_{35}. (Units of FT.)
6. C_{45}. (Units of FTL.)
7. C_{55}. (Units of FTL.)
8. C_{16}. (Units of FT.)

Third line:

1. C_{26}. (Units of FT.)
2. C_{36}. (Units of FT.)
3. C_{46}. (Units of FTL.)
4. C_{56}. (Units of FTL.)
5. C_{66}. (Units of FTL.)
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the connector damping behavior as a function of temperature and other predefined field variables.

Data lines to define linear coupled viscous damping behavior with frequency dependence (TYPE=VISCOUS with the COMPONENT, NONLINEAR, and UNSYMM parameters omitted, and

FREQUENCY DEPENDENCE=ON; all 21 damping constants must be specified, regardless of whether frequency, temperature, or field variable dependencies are included):

First line:

1. C_{11}. (Units of FTL^{-1}.)
2. C_{12}. (Units of FTL^{-1}.)
3. C_{22}. (Units of FTL^{-1}.)
4. C_{13}. (Units of FTL^{-1}.)
5. C_{23}. (Units of FTL^{-1}.)
6. C_{33}. (Units of FTL^{-1}.)
7. C_{14}. (Units of FT.)
8. C_{24}. (Units of FT.)

Second line:

1. C_{34}. (Units of FT.)
2. C_{44}. (Units of FTL.)
3. C_{15}. (Units of FT.)
4. C_{25}. (Units of FT.)
5. C_{35}. (Units of FT.)
6. C_{45}. (Units of FTL.)
7. C_{55}. (Units of FTL.)
8. C_{16}. (Units of FT.)

Third line:

1. C_{26}. (Units of FT.)
2. C_{36}. (Units of FT.)
3. C_{46}. (Units of FTL.)
4. C_{56}. (Units of FTL.)
5. C_{66}. (Units of FTL.)
6. Leave blank in an Abaqus/Explicit analysis. In an Abaqus/Standard analysis this field corresponds to frequency (in cycles per time). Applicable for **STEADY STATE DYNAMICS*, DIRECT; **STEADY STATE DYNAMICS*, SUBSPACE PROJECTION; and **STEADY STATE DYNAMICS* and **MODAL DYNAMIC* analyses that support nondiagonal damping.
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the connector damping behavior as a function of frequency, temperature, and other predefined field variables.

Data lines to define linear coupled viscous damping behavior using unsymmetric storage (the COMPONENT and NONLINEAR parameters are omitted and UNSYMM is included; all 36 damping

constants must be specified, regardless of whether temperature or field variable dependencies are included):

First line:

1. C_{11}. (Units of FTL^{-1}.)
2. C_{21}. (Units of FTL^{-1}.)
3. C_{31}. (Units of FTL^{-1}.)
4. C_{41}. (Units of FT.)
5. C_{51}. (Units of FT.)
6. C_{61}. (Units of FT.)
7. C_{12}. (Units of FTL^{-1}.)
8. C_{22}. (Units of FTL^{-1}.)

Second line:

1. C_{32}. (Units of FTL^{-1}.)
2. C_{42}. (Units of FT.)
3. C_{52}. (Units of FT.)
4. C_{62}. (Units of FT.)
5. C_{13}. (Units of FTL^{-1}.)
6. C_{23}. (Units of FTL^{-1}.)
7. C_{33}. (Units of FTL^{-1}.)
8. C_{43}. (Units of FT.)

Third line:

1. C_{53}. (Units of FT.)
2. C_{63}. (Units of FT.)
3. C_{14}. (Units of FT.)
4. C_{24}. (Units of FT.)
5. C_{34}. (Units of FT.)
6. C_{44}. (Units of FTL.)
7. C_{54}. (Units of FTL.)
8. C_{64}. (Units of FTL.)

Fourth line:

1. C_{15}. (Units of FT.)
2. C_{25}. (Units of FT.)
3. C_{35}. (Units of FT.)
4. C_{45}. (Units of FTL.)
5. C_{55}. (Units of FTL.)
6. C_{65}. (Units of FTL.)
7. C_{16}. (Units of FT.)
8. C_{26}. (Units of FT.)

Fifth line:

1. C_{36}. (Units of FT.)
2. C_{46}. (Units of FTL.)
3. C_{56}. (Units of FTL.)
4. C_{66}. (Units of FTL.)
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the unsymmetric connector damping behavior as a function of temperature and other predefined field variables.

Data lines to define linear coupled viscous damping behavior using unsymmetric storage and frequency dependence (the COMPONENT and NONLINEAR parameters are omitted, the UNSYMM parameter is included, and FREQUENCY DEPENDENCE=ON; all 36 damping constants must be specified, regardless of whether frequency, temperature, or field variable dependencies are included):

First line:

1. C_{11}. (Units of FTL^{-1}.)
2. C_{21}. (Units of FTL^{-1}.)
3. C_{31}. (Units of FTL^{-1}.)
4. C_{41}. (Units of FT.)
5. C_{51}. (Units of FT.)
6. C_{61}. (Units of FT.)
7. C_{12}. (Units of FTL^{-1}.)
8. C_{22}. (Units of FTL^{-1}.)

Second line:

1. C_{32}. (Units of FTL^{-1}.)
2. C_{42}. (Units of FT.)
3. C_{52}. (Units of FT.)
4. C_{62}. (Units of FT.)
5. C_{13}. (Units of FTL^{-1}.)
6. C_{23}. (Units of FTL^{-1}.)
7. C_{33}. (Units of FTL^{-1}.)
8. C_{43}. (Units of FT.)

Third line:

1. C_{53}. (Units of FT.)
2. C_{63}. (Units of FT.)

3. C_{14}. (Units of FT.)
4. C_{24}. (Units of FT.)
5. C_{34}. (Units of FT.)
6. C_{44}. (Units of FTL.)
7. C_{54}. (Units of FTL.)
8. C_{64}. (Units of FTL.)

Fourth line:

1. C_{15}. (Units of FT.)
2. C_{25}. (Units of FT.)
3. C_{35}. (Units of FT.)
4. C_{45}. (Units of FTL.)
5. C_{55}. (Units of FTL.)
6. C_{65}. (Units of FTL.)
7. C_{16}. (Units of FT.)
8. C_{26}. (Units of FT.)

Fifth line:

1. C_{36}. (Units of FT.)
2. C_{46}. (Units of FTL.)
3. C_{56}. (Units of FTL.)
4. C_{66}. (Units of FTL.)
5. Leave blank in an Abaqus/Explicit analysis. In an Abaqus/Standard analysis this field corresponds to frequency (in cycles per time). Applicable for **STEADY STATE DYNAMICS*, DIRECT; **STEADY STATE DYNAMICS*, SUBSPACE PROJECTION; and **STEADY STATE DYNAMICS* and **MODAL DYNAMIC* analyses that support nondiagonal damping.
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the unsymmetric connector damping behavior as a function of frequency, temperature, and other predefined field variables.

Data lines to define nonlinear viscous damping behavior that depends on the velocity in the direction of the specified component of relative motion (TYPE=VISCOUS, COMPONENT, NONLINEAR with the INDEPENDENT COMPONENTS parameter omitted):

First line:

1. Force or moment.
2. Relative velocity.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the connector damping behavior as a function of temperature and other predefined field variables.

Data lines to define linear viscous damping behavior that depends on the relative displacement, positions, or motions in several component directions (TYPE=VISCOUS, COMPONENT, NONLINEAR, INDEPENDENT COMPONENTS):

First line:

1. First independent component number (1–6).
2. Second independent component number (1–6).
3. Etc., up to N_i entries (maximum six).

Subsequent lines:

1. Force or moment in the direction specified by the COMPONENT parameter.
2. Relative velocity in the direction specified by the COMPONENT parameter.
3. Connector relative position or constitutive relative motion in the first independent component identified on the first data line.
4. Connector relative position or constitutive relative motion in the second independent component identified on the first data line.
5. Etc., up to N_i entries as identified on the first data line. If six independent components are used and no temperature or field variable dependencies are specified, a blank data line must be placed after this line.
6. Temperature.
7. First field variable.
8. Second field variable.

If the number of data entries exceeds the limit of eight entries per line, continue the input on the next data line.

Continuation line (if needed):

1. Third field variable.
2. Etc., up to eight entries per line.

Do not repeat the first data line. Repeat the subsequent data lines as often as necessary to define the damping behavior as a function of connector relative (angular) velocity, position, or motion; temperature; and other predefined field variables.

Data lines to define linear, uncoupled structural damping behavior (TYPE=STRUCTURAL, COMPONENT):

First line:

1. Damping coefficient.
2. Frequency (in cycles per time). Applicable for **STEADY STATE DYNAMICS*, DIRECT; **STEADY STATE DYNAMICS*, SUBSPACE PROJECTION; and **STEADY STATE DYNAMICS* and **MODAL DYNAMIC* analyses that support nondiagonal damping.

Repeat this data line as often as necessary to define the damping coefficient as a function of frequency.

Data lines to define linear, coupled structural damping behavior (TYPE=STRUCTURAL with the COMPONENT parameter omitted):

First line:

1. s_{11}.
2. s_{12}.
3. s_{22}.
4. s_{13}.
5. s_{23}.
6. s_{33}.
7. s_{14}.
8. s_{24}.

Second line:

1. s_{34}.
2. s_{44}.
3. s_{15}.
4. s_{25}.
5. s_{35}.
6. s_{45}.
7. s_{55}.
8. s_{16}.

Third line:

1. s_{26}.
2. s_{36}.
3. s_{46}.
4. s_{56}.
5. s_{66}.

*CONNECTOR DERIVED COMPONENT: Specify user-defined components in connector elements.

This option is used as many times as necessary in conjunction with the **CONNECTOR FRICTION* and **CONNECTOR POTENTIAL* options to define user-customized components from numbered components.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Connector Behavior*
- *Connector Functions for Coupled Behavior*
- **CONNECTOR BEHAVIOR*
- **CONNECTOR FRICTION*
- **CONNECTOR POTENTIAL*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the derived component.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the connector derived component, in addition to temperature. If this parameter is omitted, it is assumed that the connector derived components are independent of field variables. See *Material Data Definition* for more information.

EXTRAPOLATION

Set EXTRAPOLATION=CONSTANT (default unless **CONNECTOR BEHAVIOR*, EXTRAPOLATION=LINEAR is used) to use constant extrapolation of the dependent variables outside the specified range of the independent variables.

Set EXTRAPOLATION=LINEAR to use linear extrapolation of the dependent variables outside the specified range of the independent variables.

INDEPENDENT COMPONENTS

Set INDEPENDENT COMPONENTS=POSITION (default) to specify dependencies on components of relative position included in the derived component definition.

Set INDEPENDENT COMPONENTS=CONSTITUTIVE MOTION to specify dependencies on components of constitutive relative motion included in the derived component definition.

OPERATOR

Set OPERATOR=NORM (default) to use a square root of a sum of the squares function of the contributing components.

Set OPERATOR=MACAULEY SUM to sum the contributing components with a Macauley bracket function applied to each contribution.

Set OPERATOR=SUM to sum the contributing components directly.

REGULARIZE

This parameter applies only to Abaqus/Explicit analyses.

Set REGULARIZE=ON (default unless *CONNECTOR BEHAVIOR, REGULARIZE=OFF is used) to regularize the user-defined tabular connector derived component data.

Set REGULARIZE=OFF to use the user-defined tabular connector derived component data directly without regularization.

RTOL

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the tolerance to be used to regularize the connector derived component data.

If this parameter is omitted, the default is RTOL=0.03 unless the tolerance is specified on the *CONNECTOR BEHAVIOR option.

SIGN

Set SIGN=POSITIVE (default) to provide an overall positive sign to the derived component definition.

Set SIGN=NEGATIVE to provide an overall negative sign to the derived component definition.

Data lines to define the derived component if the INDEPENDENT COMPONENTS parameter is omitted:

First line:

1. First component number (1–6) to be used in the definition of the derived component.
2. Second component number (1–6) to be used in the definition of the derived component.
3. Etc., up to N_c entries (maximum six).

Subsequent lines:

1. Scaling constant (α_1) that multiplies the first component identified on the first data line.
2. Scaling constant (α_2) that multiplies the second component identified on the first data line.
3. Etc., up to N_c entries as identified on the first data line.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to eight entries per line.

If the number of data entries exceeds the limit of eight entries per line, continue the input on the next data line.

Do not repeat the first data line. Repeat the subsequent data lines as often as necessary to define the contributions to the derived component as a function of temperature and field variables.

Data lines to define the derived component if the INDEPENDENT COMPONENTS parameter is included:

First line:

1. First independent component number (1–6).
2. Second independent component number (1–6).

3. Etc., up to N_i entries (maximum six).

Second line:

1. First component number (1–6) to be used in the definition of the derived component.
2. Second component number (1–6) to be used in the definition of the derived component.
3. Etc., up to N_c entries (maximum six).

Third line:

1. Scaling constant (α_1) that multiplies the first component identified on the second data line.
2. Scaling constant (α_2) that multiplies the second component identified on the second data line.
3. Etc., up to N_c entries as identified on the second data line.
4. Connector relative position or constitutive relative motion in the first independent component identified on the first data line.
5. Connector relative position or constitutive relative motion in the second independent component identified on the first data line.
6. Etc., up to N_i entries as identified on the first data line.
7. Temperature.
8. First field variable.

If the number of data entries exceeds the limit of eight entries per line, continue the input on the next data line.

Continuation line (if needed):

1. Second field variable.
2. Etc., up to eight entries per line.

Do not repeat the first two data lines. Repeat the subsequent data lines as often as necessary to define the contributions to the derived component as a function of connector relative position or constitutive relative motion, temperature, and field variables.

*CONNECTOR ELASTICITY: Define connector elastic behavior.

This option is used to define the elastic behavior for connector elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Connector Behavior*
- *Connector Elastic Behavior*
- **CONNECTOR BEHAVIOR*

Optional parameters:

COMPONENT

Set this parameter equal to the connector's component of relative motion for which elastic behavior is specified. For this component of relative motion the connector will act as a spring. Omit this parameter if linear coupled behavior is to be defined.

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the connector elasticity data, in addition to temperature. If this parameter is omitted, it is assumed that the connector elasticity is independent of field variables. See *Material Data Definition* for more information.

EXTRAPOLATION

Set EXTRAPOLATION=CONSTANT (default unless **CONNECTOR BEHAVIOR*, EXTRAPOLATION=LINEAR is used) to use constant extrapolation of the dependent variables outside the specified range of the independent variables.

Set EXTRAPOLATION=LINEAR to use linear extrapolation of the dependent variables outside the specified range of the independent variables.

FREQUENCY DEPENDENCE

This parameter is relevant only for coupled linear spring stiffness definitions in an Abaqus/Standard analysis. Use this parameter to define spring stiffness terms with frequency dependence.

Set FREQUENCY DEPENDENCE=OFF (default) if frequency dependence of the spring stiffness terms is not defined.

Set FREQUENCY DEPENDENCE=ON if frequency dependence of the spring stiffness terms is defined.

INDEPENDENT COMPONENTS

This parameter can be used only if the COMPONENT and NONLINEAR parameters are included.

Set INDEPENDENT COMPONENTS=POSITION (default) to specify dependencies on components of relative position included in the elasticity definition.

Set INDEPENDENT COMPONENTS=CONSTITUTIVE MOTION to specify dependencies on components of constitutive relative motion included in the elasticity definition.

If elasticity is dependent on only the component of constitutive relative motion specified with the COMPONENT parameter (uncoupled behavior), the INDEPENDENT COMPONENTS parameter should not be used.

NONLINEAR

This parameter can be used only if the COMPONENT parameter is included.

Include this parameter to define nonlinear elastic behavior. Omit this parameter to define linear elastic behavior.

REGULARIZE

This parameter applies only to Abaqus/Explicit analyses.

Set REGULARIZE=ON (default unless **CONNECTOR BEHAVIOR*, REGULARIZE=OFF is used) to regularize the user-defined tabular connector elastic data.

Set REGULARIZE=OFF to use the user-defined tabular connector elastic data directly without regularization.

RTOL

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the tolerance to be used to regularize the connector elastic data.

If this parameter is omitted, the default is RTOL=0.03 unless the tolerance is specified on the **CONNECTOR BEHAVIOR* option.

RIGID

Include this parameter to indicate that rigid elastic behavior is defined.

UNSYMM

This parameter is relevant only for coupled linear spring stiffness definitions in an Abaqus/Standard analysis.

Include this parameter if the coupled linear spring stiffness matrices are not symmetric.

Data lines to define linear uncoupled elastic behavior (the COMPONENT parameter is included and the NONLINEAR parameter is omitted):

First line:

1. Elastic stiffness (force or moment per relative displacement or rotation; force for SLIPRING).
2. Leave blank in an Abaqus/Explicit analysis. In an Abaqus/Standard analysis this field corresponds to frequency (in cycles per time, for **STEADY STATE DYNAMICS*, DIRECT and **STEADY STATE DYNAMICS*, SUBSPACE PROJECTION analyses only).
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic stiffness as a function of frequency, temperature, and other predefined field variables.

Data lines to define linear coupled elastic behavior (the COMPONENT, NONLINEAR, and UNSYMM parameters are omitted; all 21 elasticity constants must be specified, regardless of whether temperature or field variable dependencies are included):

First line:

1. D_{11}. (Units of FL^{-1}; F for SLIPRING.)
2. D_{12}. (Units of FL^{-1}; F for SLIPRING.)
3. D_{22}. (Units of FL^{-1}; F for SLIPRING.)
4. D_{13}. (Units of FL^{-1}; F for SLIPRING.)
5. D_{23}. (Units of FL^{-1}; F for SLIPRING.)
6. D_{33}. (Units of FL^{-1}; F for SLIPRING.)
7. D_{14}. (Units of F.)
8. D_{24}. (Units of F.)

Second line:

1. D_{34}. (Units of F.)
2. D_{44}. (Units of FL.)
3. D_{15}. (Units of F.)
4. D_{25}. (Units of F.)
5. D_{35}. (Units of F.)
6. D_{45}. (Units of FL.)
7. D_{55}. (Units of FL.)
8. D_{16}. (Units of F.)

Third line:

1. D_{26}. (Units of F.)
2. D_{36}. (Units of F.)
3. D_{46}. (Units of FL.)
4. D_{56}. (Units of FL.)
5. D_{66}. (Units of FL.)
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the connector elastic behavior as a function of temperature and other predefined field variables

Data lines to define linear coupled elastic behavior with frequency dependence (the COMPONENT, NONLINEAR, and UNSYMM parameters are omitted, and FREQUENCY

DEPENDENCE=ON; all 21 stiffness constants must be specified, regardless of whether frequency, temperature, or field variable dependencies are included):

First line:

1. D_{11}. (Units of FL^{-1}; F for SLIPRING.)
2. D_{12}. (Units of FL^{-1}; F for SLIPRING.)
3. D_{22}. (Units of FL^{-1}; F for SLIPRING.)
4. D_{13}. (Units of FL^{-1}; F for SLIPRING.)
5. D_{23}. (Units of FL^{-1}; F for SLIPRING.)
6. D_{33}. (Units of FL^{-1}; F for SLIPRING.)
7. D_{14}. (Units of F.)
8. D_{24}. (Units of F.)

Second line:

1. D_{34}. (Units of F.)
2. D_{44}. (Units of FL.)
3. D_{15}. (Units of F.)
4. D_{25}. (Units of F.)
5. D_{35}. (Units of F.)
6. D_{45}. (Units of FL.)
7. D_{55}. (Units of FL.)
8. D_{16}. (Units of F.)

Third line:

1. D_{26}. (Units of F.)
2. D_{36}. (Units of F.)
3. D_{46}. (Units of FL.)
4. D_{56}. (Units of FL.)
5. D_{66}. (Units of FL.)
6. Leave blank in an Abaqus/Explicit analysis. In an Abaqus/Standard analysis this field corresponds to frequency (in cycles per time, for **STEADY STATE DYNAMICS*, DIRECT and **STEADY STATE DYNAMICS*, SUBSPACE PROJECTION analyses only).
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the connector elastic behavior as a function of frequency, temperature, and other predefined field variables.

Data lines to define the linear coupled stiffness matrix using unsymmetric storage (the COMPONENT and NONLINEAR parameters are omitted and the UNSYMM parameter is included;

all 36 stiffness constants must be specified, regardless of whether temperature or field variable dependencies are included):

First line:

1. D_{11}. (Units of FL^{-1}.)
2. D_{21}. (Units of FL^{-1}.)
3. D_{31}. (Units of FL^{-1}.)
4. D_{41}. (Units of F.)
5. D_{51}. (Units of F.)
6. D_{61}. (Units of F.)
7. D_{12}. (Units of FL^{-1}.)
8. D_{22}. (Units of FL^{-1}.)

Second line:

1. D_{32}. (Units of FL^{-1}.)
2. D_{42}. (Units of F.)
3. D_{52}. (Units of F.)
4. D_{62}. (Units of F.)
5. D_{13}. (Units of FL^{-1}.)
6. D_{23}. (Units of FL^{-1}.)
7. D_{33}. (Units of FL^{-1}.)
8. D_{43}. (Units of F.)

Third line:

1. D_{53}. (Units of F.)
2. D_{63}. (Units of F.)
3. D_{14}. (Units of F.)
4. D_{24}. (Units of F.)
5. D_{34}. (Units of F.)
6. D_{44}. (Units of FL.)
7. D_{54}. (Units of FL.)
8. D_{64}. (Units of FL.)

Fourth line:

1. D_{15}. (Units of F.)
2. D_{25}. (Units of F.)
3. D_{35}. (Units of F.)
4. D_{45}. (Units of FL.)
5. D_{55}. (Units of FL.)
6. D_{65}. (Units of FL.)
7. D_{16}. (Units of F.)
8. D_{26}. (Units of F.)

Fifth line:

1. D_{36}. (Units of F.)
2. D_{46}. (Units of FL.)
3. D_{56}. (Units of FL.)
4. D_{66}. (Units of FL.)
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the unsymmetric connector stiffness behavior as a function of temperature and other predefined field variables.

Data lines to define the linear coupled stiffness matrix using unsymmetric storage and frequency dependence (the COMPONENT and NONLINEAR parameters are omitted, the UNSYMM parameter is included, and FREQUENCY DEPENDENCE=ON; all 36 stiffness constants must be specified, regardless of whether frequency, temperature, or field variable dependencies are included):

First line:

1. D_{11}. (Units of FL^{-1}.)
2. D_{21}. (Units of FL^{-1}.)
3. D_{31}. (Units of FL^{-1}.)
4. D_{41}. (Units of F.)
5. D_{51}. (Units of F.)
6. D_{61}. (Units of F.)
7. D_{12}. (Units of FL^{-1}.)
8. D_{22}. (Units of FL^{-1}.)

Second line:

1. D_{32}. (Units of FL^{-1}.)
2. D_{42}. (Units of F.)
3. D_{52}. (Units of F.)
4. D_{62}. (Units of F.)
5. D_{13}. (Units of FL^{-1}.)
6. D_{23}. (Units of FL^{-1}.)
7. D_{33}. (Units of FL^{-1}.)
8. D_{43}. (Units of F.)

Third line:

1. D_{53}. (Units of F.)
2. D_{63}. (Units of F.)
3. D_{14}. (Units of F.)

4. D_{24}. (Units of F.)
5. D_{34}. (Units of F.)
6. D_{44}. (Units of FL.)
7. D_{54}. (Units of FL.)
8. D_{64}. (Units of FL.)

Fourth line:

1. D_{15}. (Units of F.)
2. D_{25}. (Units of F.)
3. D_{35}. (Units of F.)
4. D_{45}. (Units of FL.)
5. D_{55}. (Units of FL.)
6. D_{65}. (Units of FL.)
7. D_{16}. (Units of F.)
8. D_{26}. (Units of F.)

Fifth line:

1. D_{36}. (Units of F.)
2. D_{46}. (Units of FL.)
3. D_{56}. (Units of FL.)
4. D_{66}. (Units of FL.)
5. Leave blank in an Abaqus/Explicit analysis. In an Abaqus/Standard analysis this field corresponds to frequency (in cycles per time, for **STEADY STATE DYNAMICS*, DIRECT and **STEADY STATE DYNAMICS*, SUBSPACE PROJECTION analyses only).
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the unsymmetric connector stiffness behavior as a function of frequency, temperature, and other predefined field variables.

Data lines to define nonlinear elastic behavior that depends on the displacement/rotation in the direction of the specified component of relative motion (the COMPONENT and NONLINEAR parameters are included and the INDEPENDENT COMPONENTS parameter is omitted):

First line:

1. Force or moment.
2. Constitutive relative displacement or rotation.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.

2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the connector elastic behavior as a function of temperature and other predefined field variables.

Data lines to define nonlinear elastic behavior that depends on the relative positions or motions in several component directions (the COMPONENT, NONLINEAR, and INDEPENDENT COMPONENTS parameters are included):

First line:

1. First independent component number (1–6).
2. Second independent component number (1–6).
3. Etc., up to N_i entries (maximum six).

Subsequent lines:

1. Force or moment in the direction specified by the COMPONENT parameter.
2. Connector relative position or constitutive relative motion in the first independent component identified on the first data line.
3. Connector relative position or constitutive relative motion in the second independent component identified on the first data line.
4. Etc., up to N_i entries as identified on the first data line.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Etc., up to eight entries per line.

If the number of data entries exceeds the limit of eight entries per line, continue the input on the next data line.

Do not repeat the first data line. Repeat the subsequent data lines as often as necessary to define the elastic stiffness as a function of connector relative position or constitutive relative motion, temperature, and other predefined field variables.

Data lines to define rigid-like elastic behavior if the COMPONENT parameter is omitted:

First (and only) line:

1. First available component of relative motion for which rigid-like elastic behavior is defined.
2. Second available component of relative motion for which rigid-like elastic behavior is defined.
3. Etc., up to as many available components of relative motion as exist for the connection type.

Omit this data line if rigid-like elastic behavior is defined for all available components of relative motion.

*CONNECTOR FAILURE: Define a failure criterion for connector elements.

This option is used to define a failure criterion for connector elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Connector Behavior*
- *Connection Types*
- **CONNECTOR BEHAVIOR*

Required parameters:

COMPONENT

Set this parameter equal to the connector's component number for which a failure criterion is defined in Abaqus/Standard; only an available component of relative motion can be chosen. In Abaqus/Explicit any connector component number can be specified. See *Connection Types* for connector components of relative motion definitions.

Optional parameters:

RELEASE

In Abaqus/Standard set this parameter equal to ALL (default) to release all available components of relative motion when the failure criterion is satisfied. In Abaqus/Explicit set this parameter equal to ALL (default) to release all components (available or constrained) when the failure criterion is satisfied.

In Abaqus/Standard set this parameter equal to an available component of relative motion number to release only that component when the failure criterion is satisfied. In Abaqus/Explicit set this parameter equal to a component number to release only that component when the failure criterion is satisfied.

Data line to define the failure criterion:

First (and only) line:

1. Lower bound on the connector's component of relative position specified by the COMPONENT parameter. If not specified, no lower bound is used for the selected component.
2. Upper bound on the connector's component of relative position specified by the COMPONENT parameter. If not specified, no upper bound is used for the selected component.
3. Lower bound on the force or moment in the direction of the component of relative motion indicated by the COMPONENT parameter. If not specified, no lower bound is used for the selected force or moment.
4. Upper bound on the force or moment in the direction of the component of relative motion indicated by the COMPONENT parameter. If not specified, no upper bound is used for the selected force or moment.

*CONNECTOR FRICTION: Define friction forces and moments in connector elements.

This option is used to define friction forces and moments in connector elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Connection Types*
- *Connector Behavior*
- *Connector Friction Behavior*
- **CONNECTOR BEHAVIOR*
- **CONNECTOR DERIVED COMPONENT*
- **CONNECTOR POTENTIAL*
- **FRICTION*

Optional parameters:

PREDEFINED

Include this parameter to specify predefined friction behavior (if available for the connection type). Abaqus defines the contact forces and the magnitude of the tangential tractions automatically, as illustrated in *Connection Types*.

STICK STIFFNESS

Set this parameter equal to the stick stiffness associated with frictional behavior. If this parameter is omitted, a default value (which usually is appropriate) is chosen.

Optional parameters used to specify user-defined friction (mutually exclusive with the PREDEFINED parameter):

COMPONENT

Set this parameter equal to the connector's component of relative motion for which user-defined frictional behavior is specified.

Omit this parameter and use the **CONNECTOR POTENTIAL* option in conjunction with the **CONNECTOR FRICTION* option to specify coupled user-defined frictional behavior.

CONTACT FORCE

Set this parameter equal to the name of the associated **CONNECTOR DERIVED COMPONENT* option or the number of the connector component of relative motion that defines the friction-generating contact force.

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the connector friction data, in addition to temperature. If this parameter is omitted, it is assumed that the friction forces and moments or the contact normal force contributions are independent of field variables. See *Material Data Definition* for more information.

EXTRAPOLATION

Set EXTRAPOLATION=CONSTANT (default unless *CONNECTOR BEHAVIOR, EXTRAPOLATION=LINEAR is used) to use constant extrapolation of the dependent variables outside the specified range of the independent variables.

Set EXTRAPOLATION=LINEAR to use linear extrapolation of the dependent variables outside the specified range of the independent variables.

INDEPENDENT COMPONENTS

Set INDEPENDENT COMPONENTS=POSITION (default) to specify dependencies on components of relative position included in the frictional behavior definition.

Set INDEPENDENT COMPONENTS=CONSTITUTIVE MOTION to specify dependencies on components of constitutive relative motion included in the frictional behavior definition.

REGULARIZE

This parameter applies only to Abaqus/Explicit analyses.

Set REGULARIZE=ON (default unless *CONNECTOR BEHAVIOR, REGULARIZE=OFF is used) to regularize the user-defined tabular connector friction data.

Set REGULARIZE=OFF to use the user-defined tabular connector friction data directly without regularization.

RTOL

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the tolerance to be used to regularize the connector friction data.

If this parameter is omitted, the default is RTOL=0.03 unless the tolerance is specified on the *CONNECTOR BEHAVIOR option.

Data line to define the parameters (geometric constants and internal contact forces) for predefined frictional behavior (the PREDEFINED parameter is included):

First (and only) line:

1. First parameter used to specify predefined friction, as illustrated in *Connection Types*.
2. Second friction parameter.
3. Etc., up to as many friction parameters discussed in *Connection Types*.

No data line is required for connection type SLIPRING.

Data lines to define the internal contact forces for user-defined friction that does not depend on the relative positions or motions in one or more component directions (both the PREDEFINED and INDEPENDENT COMPONENTS parameters are omitted):

First line:

1. Internal contact force/moment generating friction.
2. Accumulated slip.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the internal contact force as a function of accumulated slip, temperature, and field variables. Omit these data lines if internal contact forces do not need to be specified.

Data lines to define the internal contact forces for user-defined friction that depends on the relative positions or motions in one or more component directions (the PREDEFINED parameter is omitted and the INDEPENDENT COMPONENTS parameter is included):

First line:

1. First independent component number (1–6).
2. Second independent component number (1–6).
3. Etc., up to N_i entries (maximum six).

Subsequent lines:

1. Internal contact force/moment generating friction.
2. Connector relative position or constitutive relative motion in the first independent component identified on the first data line.
3. Connector relative position or constitutive relative motion in the second independent component identified on the first data line.
4. Etc., up to N_i entries as identified on the first data line.
5. Accumulated slip.
6. Temperature.
7. First field variable.
8. Second field variable.

If the number of data entries exceeds the limit of eight entries per line, continue the input on the next data line.

Continuation line (if needed):

1. Third field variable.
2. Etc., up to eight entries per line.

Do not repeat the first data line. Repeat the subsequent data lines as often as necessary to define the internal contact force as a function of connector relative position or constitutive relative motion, accumulated slip, temperature, and other predefined field variables.

*CONNECTOR HARDENING: Define the plasticity initial yield value and hardening behavior in connector elements.

This option is used to specify the initial yield surface size and, optionally, the post-yield hardening behavior in connector available components of relative motion. It must be used in conjunction with the **CONNECTOR PLASTICITY* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Connector Behavior*
- *Connector Plastic Behavior*
- *Models for Metals Subjected to Cyclic Loading*
- **CONNECTOR BEHAVIOR*
- **CONNECTOR ELASTICITY*
- **CONNECTOR HARDENING*
- **CONNECTOR PLASTICITY*
- **CONNECTOR POTENTIAL*

Optional parameters:

DEFINITION

Set DEFINITION=EXPONENTIAL LAW to specify the isotropic hardening parameters $Q_{\inf}$ and b directly. This parameter is valid only for TYPE=ISOTROPIC.

Set DEFINITION=HALF CYCLE (default for TYPE=KINEMATIC) to provide force/moment versus plastic motion data of a first half-cycle. This parameter is valid only for TYPE=KINEMATIC.

Set DEFINITION=PARAMETERS to specify the kinematic hardening parameters C and γ directly. This parameter is valid only for TYPE=KINEMATIC.

Set DEFINITION=STABILIZED to provide force/moment versus plastic motion data of a stabilized cycle. This parameter is valid only for TYPE=KINEMATIC.

Set DEFINITION=TABULAR (default for TYPE=ISOTROPIC) to provide force/moment versus plastic motion values. Either uniaxial test data or processed data (as explained in *Connector Behavior*) from cyclic experiments can be used. This parameter is valid only for TYPE=ISOTROPIC.

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the connector hardening data, in addition to temperature. If this parameter is omitted, it is assumed that the connector hardening is independent of field variables. See *Material Data Definition* for more information.

EXTRAPOLATION

Set EXTRAPOLATION=CONSTANT (default unless *CONNECTOR BEHAVIOR*, EXTRAPOLATION=LINEAR is used) to use constant extrapolation of the dependent variables outside the specified range of the independent variables.

Set EXTRAPOLATION=LINEAR to use linear extrapolation of the dependent variables outside the specified range of the independent variables.

MODE MIX DEPENDENT

Include this parameter to indicate that the equivalent yield force or moment is a tabular function of the mode mix ratio (in addition to other dependencies).

RATE FILTER FACTOR

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the factor to be used for filtering the equivalent relative plastic motion rate for the evaluation of rate-dependent connector hardening data. The default value is 0.9.

RATE INTERPOLATION

This parameter applies only to Abaqus/Explicit analyses and is used only to interpolate rate-dependent connector hardening data.

Set RATE INTERPOLATION=LINEAR (default) to use linear intervals for the equivalent relative plastic motion rate while interpolating rate-dependent hardening data.

Set RATE INTERPOLATION=LOGARITHMIC to use logarithmic intervals for the equivalent relative plastic motion rate while interpolating rate-dependent hardening data.

REGULARIZE

This parameter applies only to Abaqus/Explicit analyses.

Set REGULARIZE=ON (default unless *CONNECTOR BEHAVIOR*, REGULARIZE=OFF is used) to regularize the user-defined tabular connector hardening data.

Set REGULARIZE=OFF to use the user-defined tabular connector hardening data directly without regularization.

RTOL

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the tolerance to be used to regularize the connector hardening data.

If this parameter is omitted, the default is RTOL=0.03 unless the tolerance is specified on the *CONNECTOR BEHAVIOR* option.

TYPE

Set TYPE=ISOTROPIC (default) to specify the initial yield surface size and, optionally, isotropic hardening data.

Set TYPE=KINEMATIC to specify kinematic hardening data.

Data lines for TYPE=ISOTROPIC, DEFINITION=TABULAR:

First line:

1. Equivalent yield force or moment defining the size of the elastic range.
2. Equivalent relative plastic motion.
3. Equivalent relative plastic motion rate.
4. Temperature.
5. First field variable.
6. Second field variable.

7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the size of the elastic range as a function of connector equivalent relative plastic motion, equivalent relative plastic motion rate, temperature, and field variables.

Data lines for TYPE=ISOTROPIC, DEFINITION=TABULAR, MODE MIX DEPENDENT:

First line:

1. Equivalent yield force or moment defining the size of the elastic range.
2. Equivalent relative plastic motion.
3. Mode mix ratio.
4. Equivalent relative plastic motion rate.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the size of the elastic range as a function of connector equivalent relative plastic motion, mode mix ratio, equivalent relative plastic motion rate, temperature, and field variables.

Data lines for TYPE=ISOTROPIC, DEFINITION=EXPONENTIAL LAW:

First line:

1. Equivalent force or moment defining the size of the elastic range at zero plastic motion.
2. Isotropic hardening parameter $Q_{\inf}$.
3. Isotropic hardening parameter b.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the size of the elastic range and the isotropic hardening parameters as functions of temperature and field variables.

Data lines for TYPE=KINEMATIC, DEFINITION=HALF CYCLE:

First line:

1. Yield force or moment.
2. Connector relative plastic motion.
3. Temperature.
4. First field variable.

5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define yield forces/moments as a function of connector relative plastic motion, temperature, and field variables.

Data lines for TYPE=KINEMATIC, DEFINITION=STABILIZED:

First line:

1. Yield force or moment.
2. Connector relative plastic motion.
3. Connector relative constitutive motion range.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define yield forces/moments as a function of connector relative plastic motion, constitutive motion range, temperature, and field variables.

Data lines for TYPE=KINEMATIC, DEFINITION=PARAMETERS:

First line:

1. Yield force or moment at zero relative plastic motion.
2. Kinematic hardening parameter C.
3. Kinematic hardening parameter γ. Set γ=0 to specify linear Ziegler kinematic hardening.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the yield force/moment at zero relative plastic motion and the kinematic hardening parameters as functions of temperature and field variables.

*CONNECTOR LOAD: Specify loads for available components of relative motion in connector elements.

This option is used to apply concentrated forces and moments to the available components of relative motion in connector elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- *Connector Actuation*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the magnitude of the load during the step.

If this parameter is omitted in an Abaqus/Standard analysis, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (see *Defining an Analysis*). If this parameter is omitted in an Abaqus/Explicit analysis, the reference magnitude is applied immediately at the beginning of the step.

LOAD CASE

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the load case number. This parameter is used in **RANDOM RESPONSE* analysis (*Random Response Analysis*), when it is the cross-reference for the load case on the **CORRELATION* option. The parameter's value is ignored in all other procedures.

OP

Set OP=MOD (default) for existing **CONNECTOR LOAD*s to remain, with this option modifying existing connector loads or defining additional connector loads.

Set OP=NEW if all existing **CONNECTOR LOAD*s applied to the model should be removed. New connector loads can be defined.

Optional, mutually exclusive parameters for matrix generation and steady-state dynamics analysis (direct, modal, or subspace):

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the loading.

REAL

Include this parameter (default) to define the real (in-phase) part of the loading.

Data lines to define connector loads for specific components of relative motion:

First line:

1. Connector element number or element set label.
2. Available component of relative motion number.
3. Reference magnitude for the load.

Repeat this data line as often as necessary to define connector loads.

*CONNECTOR LOCK: Define a locking criterion for connector elements.

This option is used to define a locking criterion for connector elements that have available components of relative motion.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Connector Behavior*
- *Connection Types*
- **CONNECTOR BEHAVIOR*

Required parameters:

COMPONENT

Set this parameter equal to the component number on which a locking criterion is based. See *Connection Types* for components of relative motion definitions.

Optional parameters:

DEPENDENCIES

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the number of field variable dependencies included in the definition of the connector lock data, in addition to temperature. If this parameter is omitted, it is assumed that the connector lock is independent of field variables. See *Material Data Definition* for more information.

EXTRAPOLATION

Set EXTRAPOLATION=CONSTANT (default unless **CONNECTOR BEHAVIOR*, EXTRAPOLATION=LINEAR is used) to use constant extrapolation of the dependent variables outside the specified range of the independent variables.

Set EXTRAPOLATION=LINEAR to use linear extrapolation of the dependent variables outside the specified range of the independent variables.

LOCK

Set this parameter equal to ALL (default) to lock all components of relative motion when the locking criterion is satisfied.

Set this parameter equal to an available component number to lock only that component of relative motion when the locking criterion is satisfied.

REGULARIZE

This parameter applies only to Abaqus/Explicit analyses.

Set REGULARIZE=ON (default unless **CONNECTOR BEHAVIOR*, REGULARIZE=OFF is used) to regularize the user-defined tabular connector lock data.

Set REGULARIZE=OFF to use the user-defined tabular connector lock data directly without regularization.

RTOL

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the tolerance to be used to regularize the connector lock data.

If this parameter is omitted, the default is RTOL=0.03 unless the tolerance is specified on the **CONNECTOR BEHAVIOR* option.

Data line to define the locking criterion:

First (and only) line for Abaqus/Standard:

1. Lower bound on the connector's relative position specified by the COMPONENT parameter. By default, no lower bound is used for the selected component.
2. Upper bound on the connector's relative position specified by the COMPONENT parameter. By default, no upper bound is used for the selected component.
3. Lower bound on the force or moment in the direction indicated by the COMPONENT parameter. By default, no lower bound is used for the selected force or moment.
4. Upper bound on the force or moment in the direction indicated by the COMPONENT parameter. By default, no upper bound is used for the selected force or moment.

Data lines for Abaqus/Explicit:

1. Lower bound on the connector's relative position specified by the COMPONENT parameter. By default, no lower bound is used for the selected component.
2. Upper bound on the connector's relative position specified by the COMPONENT parameter. By default, no upper bound is used for the selected component.
3. Lower bound on the force or moment in the direction indicated by the COMPONENT parameter. By default, no lower bound is used for the selected force or moment.
4. Upper bound on the force or moment in the direction indicated by the COMPONENT parameter. By default, no upper bound is used for the selected force or moment.
5. Lower bound on velocity in the direction specified by the COMPONENT parameter. By default, no lower bound is used for the selected velocity.
6. Upper bound on velocity in the direction specified by the COMPONENT parameter. By default, no upper bound is used for the selected velocity.
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the lock criterion as a function of temperature, and other predefined field variables.

*CONNECTOR MOTION: Specify the motion of available components of relative motion in connector elements.

This option is used to prescribe the motion of available components of relative motion in connector elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Model, Step

Abaqus/CAE: Load module

References:

- *Connection Types*
- *DISP*

Optional parameters (history data only):

AMPLITUDE

This parameter is relevant only when some of the variables being prescribed have nonzero magnitudes. Set this parameter equal to the name of the amplitude curve defining the magnitude of the prescribed connector motions (*Amplitude Curves*).

If this parameter is omitted in an Abaqus/Standard analysis, either the reference magnitude is applied linearly over the step (a RAMP function) or it is applied immediately at the beginning of the step and subsequently held constant (a STEP function). The choice of RAMP or STEP function depends on the value assigned to the AMPLITUDE parameter on the **STEP* option (*Defining an Analysis*). Two exceptions are displacement or rotation components given with TYPE=DISPLACEMENT, for which the default is always a RAMP function, and displacement or rotation components in a static step given with TYPE=VELOCITY, for which the default is always a STEP function.

If this parameter is omitted in an Abaqus/Explicit analysis, the reference magnitude is applied immediately at the beginning of the step and subsequently held constant (a STEP function).

In an Abaqus/Standard dynamic procedure, amplitude curves specified for TYPE=DISPLACEMENT or TYPE=VELOCITY will be smoothed automatically. In an explicit dynamic analysis using Abaqus/Explicit, the user must request that such amplitude curves are smoothed. For more information, see *Amplitude Curves*.

LOAD CASE

This parameter applies only to Abaqus/Standard analyses.

This parameter is ignored in all procedures except **BUCKLE*. The parameter can be set equal to 1 (default) or 2.

LOAD CASE=1 can be used to define the connector motion for the applied loads and LOAD CASE=2 can be used to define antisymmetry connector motion for the buckling modes.

OP

Set OP=MOD (default) to modify existing connector motions or to add connector motions to available components of relative motion that were previously unconstrained.

Set OP=NEW if all connector motions that are currently in effect should be removed. To remove only selected connector motions, use OP=NEW and respecify all connector motions that are to be retained.

If a connector motion is removed in a stress/displacement analysis, it will be replaced by a concentrated force equal to the reaction force calculated at the restrained degree of freedom at the end of the previous step. If the step is a general nonlinear analysis step, this concentrated force will then be removed according to the AMPLITUDE parameter on the **STEP* option. Therefore, by default the concentrated force will be reduced linearly to zero over the period of the step in a static analysis and immediately in a dynamic analysis.

Optional, mutually exclusive parameters (history data only):

FIXED

Include this parameter to indicate that the values of the variables being prescribed with this **CONNECTOR MOTION* option should remain fixed at their current values at the start of the step. If this parameter is used, any magnitudes given on the data lines are ignored.

TYPE

This parameter is used in a stress/displacement analysis to specify whether the magnitude is in the form of a displacement history, a velocity history, or an acceleration history.

Set TYPE=DISPLACEMENT (default) to give a displacement history.

Set TYPE=VELOCITY to give a velocity history. Velocity histories can be specified in static analyses. In this case the default variation is STEP.

Set TYPE=ACCELERATION to give an acceleration history. Acceleration histories should not be used in static analysis steps.

USER

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to indicate that any nonzero magnitudes associated with variables prescribed through this option will be defined in user subroutine *DISP*. If this parameter is used, any magnitudes defined by the data lines of the option (and possibly modified by the AMPLITUDE parameter) can be redefined in subroutine *DISP*. The value of the TYPE parameter is ignored when this option is used.

Optional, mutually exclusive parameters for matrix generation and direct-solution steady-state dynamics analysis (history data only):

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the connector motion.

REAL

Include this parameter (default) to define the real (in-phase) part of the connector motion.

Data lines to prescribe connector motion:

First line:

1. Connector element number or element set label.
2. Available component of relative motion number for which the motion is specified. See *Connection Types* for definitions of the available components of relative motion.

The following data item is necessary only when nonzero connector motion is specified as history data. Any magnitude given will be ignored when the connector motion is given as model data.

3. Actual magnitude of the variable (displacement, velocity, or acceleration). This magnitude will be modified by an amplitude specification if the AMPLITUDE parameter is used. If this magnitude is a rotation, it must be given in radians. The magnitude can be redefined in user subroutine *DISP* if the USER parameter is included.

Repeat this data line as often as necessary to specify connector motion for different connector elements and available components of relative motion.

*CONNECTOR PLASTICITY: Define plasticity behavior in connector elements.

This option is used to define plasticity behavior in connector elements. It must be used in conjunction with the *CONNECTOR HARDENING option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Connector Behavior*
- *Connector Plastic Behavior*
- *CONNECTOR BEHAVIOR
- *CONNECTOR HARDENING
- *CONNECTOR POTENTIAL

Optional parameters:

COMPONENT

Set this parameter equal to the connector's component of relative motion for which plasticity behavior is specified.

If this parameter is omitted, the *CONNECTOR POTENTIAL option must be used in conjunction with the *CONNECTOR PLASTICITY option to specify coupled plasticity behavior.

There are no data lines associated with this option.

*CONNECTOR POTENTIAL: Specify user-defined potentials in connector elements.

This option is used to define a restricted set of mathematical functions to represent yield or limiting surfaces in the space spanned by connector available components. It can be used only in conjunction with the following options: **CONNECTOR DAMAGE EVOLUTION*, **CONNECTOR DAMAGE INITIATION*, **CONNECTOR FRICTION*, or **CONNECTOR PLASTICITY*.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Connection Types*
- *Connector Behavior*
- *Connector Functions for Coupled Behavior*
- **CONNECTOR BEHAVIOR*
- **CONNECTOR DAMAGE EVOLUTION*
- **CONNECTOR DAMAGE INITIATION*
- **CONNECTOR DERIVED COMPONENT*
- **CONNECTOR FRICTION*
- **CONNECTOR PLASTICITY*

Optional parameters:

EXPONENT

This parameter can be used only if OPERATOR=SUM.

Set this parameter equal to the inverse of the overall exponent in the potential definition, β. β must be a positive number. The default value is $\beta = 2.0$.

OPERATOR

Set OPERATOR=SUM (default) to define the potential as the sum of the contributions defined on each data line.

Set OPERATOR=MAX to define the potential as the contribution coming from the data line that yields the maximum value. The EXPONENT parameter is ignored in this case.

Data lines to define the potential:

First line:

1. Connector component number (1–6) or connector derived component name that is used in the contribution.
2. Nonzero scaling factor R. The default value is $R = 1.0$.
3. Positive exponent α. The default value is that of the EXPONENT parameter, $\alpha = \beta$. The exponent is ignored if OPERATOR=MAX.

4. The function H to be used to generate the contribution. H can be ABS (absolute value), MACAULEY (Macauley bracket), or NONE (the identity function). If OPERATOR=SUM, the function NONE can be used only if $\alpha = \beta = 1.0$. The default value is ABS.
5. Shift factor a. The default value is $a = 0.0$.
6. Sign of this contribution s. The only admissible values are $s = 1.0$ (default) and $s = -1.0$.

Repeat this data line as often as necessary to define the potential.

*CONNECTOR SECTION: Specify connector attributes for connector elements.

This option is used to define the attributes of connector elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Interaction module

References:

- *Connector Elements*
- *Connection Types*
- **CONNECTOR BEHAVIOR*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the connector elements for which the connection attributes are being defined.

Optional parameters:

BEHAVIOR

Set this parameter equal to the name of the connector behavior that defines these connector elements. If this parameter is omitted, the connector element's behavior is determined by kinematic constraints only.

CONTROLS

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the name of a section controls definition (see *Section Controls*) to be used for the connector elements.

Section controls can be used to specify whether the connector elements should be deleted once they fail completely. If this parameter is omitted, the failed elements will not be deleted. Section controls can also be used to specify a maximum value of the scalar degradation (damage) parameter, $D_{\max}$, and to specify the viscosity coefficient, μ, for viscous damping or regularization.

ELIMINATION

This parameter applies only to Abaqus/Explicit analyses.

Set ELIMINATION=NO (default) if the constraint or kinetic forces/moments of the associated connector elements are to be solved for directly in the implicit constraint solver in Abaqus/Explicit.

Set ELIMINATION=YES if the constraint or kinetic forces/moments of the associated connector elements are to be solved for using a condensation technique.

Data lines to define the connection attributes:

First line:

1. Basic translational connection type, basic rotational connection type, assembled connection type, or complex connection type from *Connection Types*. If an assembled or a complex connection is selected, no additional data can be entered on this data line.
2. Basic rotational or basic translational connection component. If the first entry of this data line is a basic translational connection component, the second entry (if provided) must be a basic rotational connection component. Similarly, if the first entry of this data line is a basic rotational connection component, the second entry (if provided) must be a basic translational connection component.

Second line (optional):

1. Orientation name specifying the local directions at the first node (or ground node) of the connector element.
2. Orientation name specifying the local directions at the second node (or ground node) of the connector element. If an orientation name is not specified, the local directions at the first node are used.

Omit the second line if neither of the two orientations is specified and the third line is omitted. Leave blank if neither of the two orientations is specified and the third line is included.

Third line (optional) for SLIPRING connection type:

1. Mass per unit reference length of belt material.
2. Contact angle (in radians) made by belt wrapping around node b (optional). The value must be zero or positive.

   In Abaqus/Standard the default value is 0.0. However, you can leave this data item blank and set the 5th data item to UPDATED ANGLE to have Abaqus/Standard update the contact angle automatically during the analysis. In Abaqus/Explicit if you do not specify the contact angle, Abaqus/Explicit computes it automatically.
3. Leave this data item blank only if you request the contact angle to be computed automatically in Abaqus/Standard. Omit it otherwise.
4. Leave this data item blank only if you request the contact angle to be computed automatically in Abaqus/Standard. Omit it otherwise.
5. Optionally, specify UPDATED ANGLE to request that Abaqus/Standard computes the contact angle automatically during the analysis. If you specify the contact angle directly as the 2nd data item on this line, Abaqus/Standard ignores the UPDATED ANGLE setting.

Omit the third line if no data are specified.

Third line (optional) for RETRACTOR or FLOW-CONVERTER connection types:

1. Scaling factor for material flow at node b, β_s (default value is 1.0).

Omit the third line if no data are specified.

*CONNECTOR STOP: Specify connector stops for connector elements.

This option is used to define connector stops for connector elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Connector Behavior*
- **CONNECTOR BEHAVIOR*

Required parameters:

COMPONENT

Set this parameter equal to the connector's available component of relative motion number for which connector stops are defined.

Data line to define connector stops:

First (and only) line:

1. Lower limit for the connector's relative position specified by the COMPONENT parameter. If not specified, no lower limit is used.
2. Upper limit for the connector's relative position specified by the COMPONENT parameter. If not specified, no upper limit is used.

*CONNECTOR UNIAXIAL BEHAVIOR: Define uniaxial behavior in connector elements.

This option is used to define uniaxial behavior in connector elements by specifying the loading and unloading response for the component of relative motion.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Connector Behavior*
- *Connector Uniaxial Behavior*
- **CONNECTOR BEHAVIOR*
- **LOADING DATA*
- **UNLOADING DATA*

Required parameters:

COMPONENT

Set this parameter equal to the connector's component of relative motion for which the uniaxial behavior is specified.

There are no data lines associated with this option.

*CONSTITUENT: Define a constituent in the multiscale material.

This option is used to define the properties of a constituent in the multiscale material. It must be used in conjunction with the *MEAN FIELD HOMOGENIZATION option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- *MEAN FIELD HOMOGENIZATION
- *CONCENTRATION TENSOR
- *Mean-Field Homogenization*

Required parameters:

TYPE

Set TYPE=INCLUSION to define inclusion properties.

Set TYPE=MATRIX to define matrix properties. There can only be one matrix-typed constituent in the aggregate.

Set TYPE=VOID to define void inclusion properties. Void inclusions are supported only in Abaqus/Standard.

Optional parameters:

DIRECTION

This parameter is not relevant for spherical inclusion.

Set DIRECTION=FIXED to define a fixed inclusion direction.

Set DIRECTION=ORIENTATION TENSOR to define the orientation distribution using a second-order orientation tensor.

Set DIRECTION=RANDOM3D to define a random orientation distribution in 3D.

If this parameter is omitted, the inclusions are assumed to be aligned. The direction vector is (1, 0, 0) in the local coordinate system.

ISOTROPIZATION COEFFICIENT

This parameter is meaningful only when the constituent is modeled with an elastic-plastic material and the plasticity is defined with isotropic hardening.

Set this parameter equal to the factor used for scaling the plastic strain of the constituent when calculating the isotropic part of the tangent. The value must be greater than or equal to one. If this parameter is omitted, the general method is used for the isotropization.

MATERIAL

Set this parameter equal to the name of the material to be used with this constituent.

This parameter is not relevant if the constituent is void (TYPE=VOID).

NAME

Set this parameter equal to the name of the constituent.

The constituent name must be unique. This parameter is not relevant if the constituent is void (TYPE=VOID).

RESPONSE

This parameter applies only to constituents with TYPE=INCLUSION and DIRECTION=ORIENTATION TENSOR or RANDOM3D.

Set RESPONSE=GRANULAR (default) to compute the response of the inclusion in each subdomain ("grain") individually. See *Specifying a Second-Order Orientation Tensor in Abaqus/Standard* for details.

Set RESPONSE=AVERAGE to compute the response of the inclusion using the averaged strain/temperature gradient over all subdomains. This value can be used only if UNIFORM MATRIX STRAIN=YES on the **MEAN FIELD HOMOGENIZATION* option.

In Abaqus/Explicit only the RESPONSE=AVERAGE setting is supported.

SHAPE

This parameter is not relevant for the matrix constituent (TYPE=MATRIX) or when the concentration tensor is defined directly.

Set SHAPE=CYLINDER to define cylinder-shaped inclusions with a circular cross-section. The aspect ratio is infinite.

Set SHAPE=ELLIPTIC CYLINDER to define cylinder-shaped inclusions with an elliptical

cross-section. The aspect ratio is a_2/a_3. See *Specifying the Shape, Volume Fraction, and Aspect Ratio* for more details.

Set SHAPE=OBLATE to define oblate-shaped inclusions. The aspect ratio must be less than one.

Set SHAPE=PROLATE to define prolate-shaped inclusions. The aspect ratio must be greater than one.

Set SHAPE=PENNY to define penny-shaped inclusions. The aspect ratio must be less than one.

Set SHAPE=SPHERE to define spherical inclusions. The aspect ratio is one.

Data line to define constituent properties with aligned directions (DIRECTION=FIXED):

First (and only) line:

1. A positive scalar value defining the volume fraction of the constituent. In Abaqus/Standard the value can also be the name of a distribution (*Distribution Definition*) that defines a spatially varying volume fraction.
2. A positive scalar value defining the aspect ratio of the constituent. Leave blank if SHAPE=CYLINDER, SHAPE=SPHERE, or if the concentration tensor is defined directly. In Abaqus/Standard the value can also be the name of a distribution (*Distribution Definition*) that defines a spatially varying aspect ratio.
3. Components of the direction vector defined in the local coordinate system. In Abaqus/Explicit the direction vector must be aligned with the 1-direction of the local coordinate system.

Data line to define constituent properties with specified second-order orientation tensor (DIRECTION=ORIENTATION TENSOR):

First (and only) line:

1. A positive scalar value defining the volume fraction of the constituent. In Abaqus/Standard the value can also be the name of a distribution (*Distribution Definition*) that defines a spatially varying volume fraction.

2. A positive scalar value defining the aspect ratio of the constituent. Leave blank if SHAPE=CYLINDER, SHAPE=SPHERE, or if the concentration tensor is defined directly. In Abaqus/Standard the value can also be the name of a distribution (*Distribution Definition*) that defines a spatially varying aspect ratio.
3. Components of the second-order orientation tensor in the local coordinate system. In Abaqus/Standard the value can also be the name of a distribution (*Distribution Definition*) that defines a spatially varying orientation tensor. In Abaqus/Explicit a spatially varying orientation tensor is specified in the orientation definition of the section.

Data line to define constituent properties with random orientation (DIRECTION=RANDOM3D):

First (and only) line:

1. Volume fraction.
2. Aspect ratio. Leave blank if SHAPE=CYLINDER, SHAPE=SPHERE, or if the concentration tensor is defined directly.

*CONSTRAINT CONTROLS: Reset overconstraint checking controls.



Warning: Use this option to specify the technique to be used to enforce constraints associated with connector elements. Otherwise, this option should not be used unless the user is certain that the model is free of overconstraints. An overconstraint means applying multiple consistent or inconsistent kinematic constraints. Many models have nodal degrees of freedom that are overconstrained, and such overconstraints may lead to inaccurate solutions or nonconvergence. By default, the model will be checked for overconstraints. The consistent overconstraints will be removed whenever possible, while an error message is issued if an inconsistent overconstraint is detected.

Products: Abaqus/Standard

Type: Model or history data

Level: Model, Step

References:

- *Overconstraint Checks*
- *About Connectors*
- *Mesh Tie Constraints*
- *Common Difficulties Associated with Contact Modeling in Abaqus/Standard*

Optional and mutually exclusive parameters (model data only):

DELETE SECONDARY

Include this parameter to delete contact elements associated with tied secondary nodes.

NO CHANGES

Include this parameter to perform overconstraint checks but to prevent Abaqus from changing the model to remove redundant constraints. Detailed messages regarding overconstraints are generated. If this parameter is omitted, Abaqus will attempt to change the model automatically.

NO CHECKS

Include this parameter to suppress overconstraint checks for this model. If this parameter is omitted, overconstraint checks are performed.

PRINT

Set PRINT=YES to print the constraint chains to the message file. If you set PRINT=NO (default), the constraint chains will not be printed.

Optional parameters (history data only):

CHECK FREQUENCY

Set this parameter equal to the desired overconstraint check frequency, in increments. Overconstraint checks are always performed at the beginning of the first increment of the step unless overconstraint checks are suppressed. The default value is CHECK FREQUENCY=1 such that overconstraint checks are performed every increment. Set CHECK FREQUENCY=0 to suppress overconstraint checks in this step.

TERMINATE ANALYSIS

Set TERMINATE ANALYSIS=NO (default) to allow an analysis to continue when an overconstraint is encountered. Detailed messages regarding the overconstraints are issued.

Set TERMINATE ANALYSIS=FIRST OCCURRENCE if the analysis is to be terminated the first time an overconstraint is encountered in a nonlinear general step.

Set TERMINATE ANALYSIS=CONVERGED if the analysis is to be terminated when convergence is achieved in an increment in a nonlinear general step and an overconstraint exists.

If either FIRST OCCURRENCE or CONVERGED is used in a linear perturbation step (where iterations are not necessary), the analysis will be stopped in the first increment when an overconstraint is encountered.

There are no data lines associated with this option.

*CONTACT: Begin the definition of general contact.

This option is used to indicate the start of a general contact definition. The various aspects of a general contact definition are specified by using other options in conjunction with the **CONTACT* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Model, Step

Abaqus/CAE: Interaction module

References:

- *About General Contact in Abaqus/Standard*
- *About General Contact in Abaqus/Explicit*

Optional parameters:

OP

This parameter applies only to Abaqus/Explicit.

Set OP=MOD (default) to modify an existing general contact definition relative to the previous step.

Set OP=NEW to delete any previously specified general contact definition and specify a new one. OP=NEW is ignored when the general contact definition is specified as model data.

There are no data lines associated with this option.

*CONTACT CLEARANCE: Define contact clearance properties.

This option is used to create a contact clearance property definition. The contact clearance properties will govern any contact interactions that are assigned these properties via the **CONTACT CLEARANCE ASSIGNMENT* option.

Products: Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Contact Initialization for General Contact in Abaqus/Explicit*
- **CONTACT*
- **CONTACT CLEARANCE ASSIGNMENT*
- **DISTRIBUTION*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to this contact clearance property.

Optional parameters:

ADJUST

Set ADJUST=YES (default) to resolve clearances by adjusting the nodal coordinates without creating strain in the model. ADJUST=YES can be used only for clearances defined in the first step of an analysis.

Set ADJUST=NO to store contact offsets so that the clearances can be satisfied without adjusting the nodal coordinates.

CLEARANCE

Set this parameter equal to the value of the initial clearance for the entire set of secondary nodes or to the name of a nodal distribution (see *Distribution Definition*). The clearance values must be non-negative for secondary nodes on solid element surfaces. The default value is 0.0.

SEARCH ABOVE

Set this parameter equal to the distance above the surfaces that will be searched for secondary nodes to be included in the clearance specification. The default for solid elements is approximately one-tenth of the element size of the elements attached to a secondary node. The default for structural elements (e.g., shell elements) is the thickness associated with the secondary node.

This parameter cannot be used if the SEARCH NSET parameter has been used.

SEARCH BELOW

Set this parameter equal to the distance below the surfaces that will be searched for secondary nodes to be included in the clearance specification. The default for solid elements is approximately one-tenth of the element size of the elements attached to a secondary node. The default for structural elements (e.g., shell elements) is the thickness associated with the secondary node.

This parameter cannot be used if the SEARCH NSET parameter has been used.

SEARCH NSET

Set this parameter equal to the name of the node set containing the secondary nodes to be included in the clearance specification. The specified clearance will be enforced at all secondary nodes in this node set irrespective of whether they are above or below their respective main surfaces. This parameter can also be used to identify initially bonded nodes in a VCCT analysis.

This parameter cannot be used if either the SEARCH ABOVE or SEARCH BELOW parameter has been used.

There are no data lines associated with this option.

*CONTACT CLEARANCE ASSIGNMENT: Assign contact clearances between surfaces in the general contact domain.

This option is used to define initial contact clearances between contact surfaces and to control how initial contact overclosures are resolved in the general contact algorithm.

Products: Abaqus/Explicit

Type: Model or history data

Level: Model, Step

References:

- *Contact Initialization for General Contact in Abaqus/Explicit*
- **CONTACT*
- **CONTACT CLEARANCE*

There are no parameters associated with this option.

Data lines to define nondefault contact clearances:

First line:

1. The name of the first (single-sided) surface.
2. The name of the second (single-sided) surface.
3. The name of the model data **CONTACT CLEARANCE* definition to be used.

Optional data item when a **CONTACT CLEARANCE* definition specified with ADJUST=YES is referenced:

9. Blank, the “word” MAIN, or the “word” SECONDARY to indicate how the surfaces will be treated while adjusting the surface nodes to resolve contact clearance violations. A blank entry indicates that the interaction will be treated as balanced main-secondary. A setting of MAIN or SECONDARY specifies the behavior of the first surface in a pure main-secondary interaction.

Repeat this data line as often as necessary. If the contact clearance assignments overlap, the last assignment applies in the overlap region.

*CONTACT CONTROLS: Specify additional controls for contact.

This option is used to provide additional optional solution controls for contact pairs in Abaqus/Standard and Abaqus/Explicit and global solution controls for general contact in Abaqus/Standard.

The standard solution controls are usually sufficient, but additional controls are helpful to obtain cost-effective solutions for models involving complicated geometries and numerous contact interfaces, as well as for models in which rigid body motions are initially not constrained.

The **CONTACT CONTROLS* option can be repeated to set different control values for different contact pairs. It must be used in conjunction with the **CONTACT PAIR* option in Abaqus/Explicit analyses.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *Generally Applicable Contact Controls in Abaqus/Standard*
- *About Contact Pairs in Abaqus/Explicit*
- *Contact Controls for Contact Pairs in Abaqus/Explicit*
- **CONTACT PAIR*

Specifying additional controls for contact in an Abaqus/Standard analysis

Optional, mutually exclusive parameters applicable to augmented Lagrangian constraint enforcement:

ABSOLUTE PENETRATION TOLERANCE

Set this parameter equal to the allowable penetration. Only contact constraints defined with augmented Lagrangian surface behavior will be affected by this parameter.

RELATIVE PENETRATION TOLERANCE

Set this parameter equal to the ratio of the allowable penetration to the characteristic contact surface face dimension. Only contact constraints defined with augmented Lagrangian surface behavior will be affected by this parameter. By default, the RELATIVE PENETRATION TOLERANCE parameter is set to 0.1% except for finite-sliding, surface-to-surface contact, in which case the default setting is 5%.

Optional parameters:

MAIN

Set this parameter equal to the main surface name to apply the controls to a specific contact pair. This parameter must be used in conjunction with the SECONDARY parameter to specify a contact pair.

PERTURBATION TANGENT SCALE FACTOR

Set this parameter equal to the factor by which Abaqus/Standard will scale the default tangential stiffness used for the contact pairs in a particular linear perturbation step. Only contact constraints enforced with penalty methods will be affected by this parameter. This tangential scale factor is activated when a nonzero friction is specified on the data line of the **FRICTION* option.

PRESSURE DEPENDENT PERTURBATION

Set this parameter equal to the value of the p_1 coefficient controlling the base state contact pressure–dependent enforcement of contact constraints in a particular linear perturbation step. This parameter allows you to relax or remove both normal and tangential contact constraints with low pressure.

RESET

Include this parameter to reset all contact controls to their default values. This parameter cannot be used with any other parameters, except for the SECONDARY and MAIN parameters. When this parameter is used in conjunction with the SECONDARY and MAIN parameters, the controls applied to the specific contact pair are removed.

SECONDARY

Set this parameter equal to the secondary surface name to apply the controls to a specific contact pair. This parameter must be used in conjunction with the MAIN parameter to specify a contact pair.

STABILIZE

Include this parameter to address situations where rigid body modes exist as long as contact is not fully established. This parameter activates damping in the normal and tangential directions based on the stiffness of the underlying mesh and the time step size. If no value is assigned to this parameter, Abaqus calculates the damping coefficient automatically. If a numerical value is assigned to this parameter, Abaqus multiplies the automatically calculated damping coefficient by this value. If the damping coefficient is defined directly on the data line, any numerical value assigned to this parameter is ignored.

Set STABILIZE =USER ADAPTIVE to scale the automatically calculated damping coefficient or the damping coefficient specified on the data line by a factor that decreases over iterations within one increment, according to the pattern specified on the second data line.

The STABILIZE parameter can be used to specify damping for the whole model or for an individual contact pair by using the SECONDARY and MAIN parameters. Values specified for a specific contact pair override the values for the whole model, if given.

STIFFNESS SCALE FACTOR

Set this parameter equal to the factor by which Abaqus/Standard will scale the default penalty stiffness to obtain the stiffnesses used for the contact pairs.

Set STIFFNESS SCALE FACTOR=USER ADAPTIVE to scale the default penalty stiffness by a factor that increases over iterations of the first increment and remains constant (equal to the last specified value) for subsequent increments.

This scale factor acts as an additional multiplier on any scale factor specified on the data line of the **SURFACE BEHAVIOR* option.

TANGENT FRACTION

Set this parameter equal to a fraction of the damping in the normal direction as specified with the STABILIZE parameter. By default, the tangential and normal stabilization are the same.

Optional data line if the PRESSURE DEPENDENT PERTURBATION parameter is included:

First (and only) line:

1. The p_0 coefficient. Default is equal to p_1.

Optional data lines if the STABILIZE parameter is included:

First line:

1. Damping coefficient to be used in the contact interface. The value entered overrides the damping coefficient calculated by Abaqus.
2. Fraction of the damping that remains at the end of the step. The default is zero. Set to one to keep the damping constant over the step. If a nonzero value is specified, convergence problems may occur in a subsequent step if stabilization is not used in that step.
3. Clearance at which the damping becomes zero. By default, the clearance is calculated by Abaqus based on the facet size associated with the contact pair. Set to a large value to obtain damping independent of the opening distance.

Second line (required if STABILIZE=USER ADAPTIVE):

1. Up to eight values in descending order. For example, 1.0, 0.0.

For STABILIZE without the USER ADAPTIVE setting, omitting the first data line can have different effects than specifying a blank data line. A blank data line causes parameters of the data line to be set to default values; whereas prior values of these parameters will persist if the data line is omitted and RESET is not in effect.

Optional data line if STIFFNESS SCALE FACTOR=USER ADAPTIVE:

Third line if STABILIZE=USER ADAPTIVE; otherwise, first line:

1. Up to eight scale values in ascending order. By default, the stiffness scale factors are $10^{-7}, 10^{-5}, 10^{-3}, 10^{-1}, 1$.

Specifying additional controls for contact in an Abaqus/Explicit analysis



Warning: These controls are intended for experienced analysts and should be used with care. Using nondefault values of these controls may greatly increase the computational time of the analysis or produce inaccurate results.

Required parameters:

CPSET

Set this parameter equal to the name of the contact pair set associated with this contact controls definition. The contact controls defined with this option will be applied to all contact pairs having this contact pair set name.

Optional parameters:

FASTLOCALTRK

Set FASTLOCALTRK=NO if contact is not being enforced appropriately. A more conservative local tracking method will be used that may resolve the error. The default is FASTLOCALTRK=YES, which uses a more computationally efficient local tracking method.

GLOBTRKINC

Set this parameter equal to the maximum number of increments between global contact searches. The default is 100 increments for two-surface contact and 4 increments for self-contact.

RESET

Include this parameter to reset all of the optional controls to their default values. Those controls that are explicitly specified with other parameters on the same **CONTACT CONTROLS* option are not reset. If this parameter is omitted, only the explicitly specified controls will be changed in the current step; the others will remain at their previous settings.

SCALE PENALTY

Set this parameter equal to the factor by which Abaqus/Explicit will scale the default penalty stiffnesses to obtain the stiffnesses used for the penalty contact pairs within the contact pair set specified with the CPSET parameter. Penalty contact constraints defined with softened surface behavior and kinematic contact constraints is not affected by this parameter. By default, the SCALE PENALTY parameter is set to unity.

WARP CHECK PERIOD

Set this parameter equal to the number of increments between checks for highly warped facets on main surfaces. By default, this check is performed every 20 increments. More frequent checks will cause a slight increase in computational time.

WARP CUT OFF

Set this parameter equal to the out-of-plane warping angle, measured in degrees, at which a facet will be considered to be highly warped. The out-of-plane warping angle is defined as the amount of variation of the surface normal over a facet. The default is WARP CUT OFF=20.

There are no data lines associated with this option.

*CONTACT CONTROLS ASSIGNMENT: Assign contact controls for the general contact algorithm.

This option is used to modify contact controls for specific contact interactions within the domain considered by the general contact algorithm in Abaqus/Explicit and Abaqus/Standard. It must be used in conjunction with the **CONTACT* option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model or history data

Level: Model, Step

References:

- *Generally Applicable Contact Controls in Abaqus/Standard*
- *Contact Controls for General Contact in Abaqus/Explicit*
- **CONTACT*

Required, mutually exclusive parameters:

AUTOMATIC OVERCLOSURE RESOLUTION

This parameter applies only to Abaqus/Explicit analyses.

Include this parameter to store offsets instead of adjusting nodes during initial overclosure resolution between surface pairs in the general contact domain as specified in the data lines.

BEAM CROSS SECTION

Set BEAM CROSS SECTION=EXACT (default) to activate contact calculations based on the actual cross-sectional shape of the beam element for the entire model.

Set BEAM CROSS SECTION=CIRCUMSCRIBED CIRCLE to activate contact calculations based on a circumferential approximation of the cross-sectional shape of the beam element for the entire model.

CONTACT THICKNESS REDUCTION

This parameter applies only to Abaqus/Explicit analyses.

Set CONTACT THICKNESS REDUCTION=SELF to limit automatic contact thickness reductions to only regions of potential self-contact and the perimeters of shell surfaces for the entire model.

Set CONTACT THICKNESS REDUCTION=NOPERIMSELF to limit automatic contact thickness reductions to only regions of potential self-contact for the entire model.

NODAL EROSION

This parameter applies only to Abaqus/Explicit analyses.

Set NODAL EROSION=NO (default) to keep a node of an element-based surface in the general contact domain as a point mass after all contact faces and edges to which it is attached have eroded.

Set NODAL EROSION=YES to delete a node of an element-based surface from the general contact domain once all contact faces and edges to which it is attached have eroded.

ROTATIONAL TERMS

This parameter applies only to Abaqus/Explicit analyses.

Set ROTATIONAL TERMS=NONE (default) to ignore the effects of shell and beam thickness offsets for contact.

Set ROTATIONAL TERMS=STRUCTURAL to account for the effects of shell and beam thickness offsets for contact.

SHELL REPRESENTATION

This parameter applies only to general contact in Abaqus/Standard analyses.

Set SHELL REPRESENTATION=REFERENCE SURFACE (default) to represent shells in general contact by the shell reference surface.

Set SHELL REPRESENTATION=EXTRA PERIMETER FACES to represent shell perimeter regions in general contact with internally generated surface elements near the exterior of the shell, with non-perimeter regions represented by the shell reference surface.

TYPE

This parameter applies only to Abaqus/Explicit analyses.

Set TYPE=FOLD INVERSION CHECK to activate the fold inversion check as specified in the data lines.

Set TYPE=SCALE PENALTY to assign a scale factor to the default penalty stiffnesses as specified in the data lines.

Optional parameters:

SEEDING

This parameter applies only to Abaqus/Explicit analyses. It controls how the contact seeds are created on Lagrangian surfaces during a coupled Eulerian-Lagrangian analysis.

Set SEEDING=GLOBAL (default if adaptive mesh refinement is not activated) to create seeds on Lagrangian surfaces based on the smallest Eulerian element size in the entire Eulerian mesh; the seeding is performed once at the beginning of the analysis.

Set SEEDING=LOCAL to create seeds on Lagrangian surfaces based on the smallest Eulerian element size in the vicinity of the Lagrangian faces; the seeding is performed once for each face, as soon as a nearby Eulerian element is detected.

Set SEEDING=DYNAMIC (default if adaptive mesh refinement is activated) to create seeds on Lagrangian surfaces based on the smallest Eulerian element size in the vicinity of the Lagrangian faces; the seed density is updated during the analysis.

Data lines for AUTOMATIC OVERCLOSURE RESOLUTION:

First line:

1. The name of the first surface. If the first surface name is omitted, a default surface that encompasses the entire general contact domain (including all nodes and facets) is assumed.
2. The name of the second surface. If the second surface name is omitted or is the same as the first surface name, the specified contact controls are assigned to contact interactions between the first surface and itself.
3. The overclosure resolution method. The “words” ADJUST NODES (default) or STORE OFFSETS.

Repeat this data line as often as necessary. If the contact controls assignments overlap, the last assignment applies in the overlap region.

Data lines for TYPE=FOLD INVERSION CHECK:

First line:

1. The name of the surface for which the fold inversion check should be activated. If the surface name is omitted, a default surface that encompasses the entire general contact domain (including all nodes and facets) is assumed.

Repeat this data line as often as necessary.

Data lines for TYPE=SCALE PENALTY:

First line:

1. The name of the first surface. If the first surface name is omitted, a default surface that encompasses the entire general contact domain (including all nodes and facets) is assumed.
2. The name of the second surface. If the second surface name is omitted or is the same as the first surface name, the specified contact controls are assigned to contact interactions between the first surface and itself.
3. The factor by which Abaqus/Explicit will scale the default penalty stiffnesses for the specified contact pairings.

Repeat this data line as often as necessary. If the contact controls assignments overlap, the last assignment applies in the overlap region.

*CONTACT DAMPING: Define viscous damping between contacting surfaces.

This option is used to define viscous damping between two interacting surfaces. It must be used in conjunction with the *SURFACE INTERACTION, the *GAP, or the *INTERFACE option. In Abaqus/Standard this option is primarily used to damp relative motions of the surfaces during approach or separation. In Abaqus/Explicit this option is used to damp oscillations when using penalty or softened contact. This option is not applicable if user subroutine *VUINTER* or *VUINTERACTION* is specified for the surface interaction.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data in Abaqus/Standard; Model or history data in Abaqus/Explicit

Level: Part, Part instance, Assembly, Model in Abaqus/Standard; Model or Step in Abaqus/Explicit

Abaqus/CAE: Interaction module

References:

- *About Mechanical Contact Properties*
- *Contact Damping*

Required parameters:

DEFINITION

Use this parameter to choose the dimensionality of the damping coefficient that is specified on the data line. The only option that is available in an Abaqus/Standard analysis is DEFINITION=DAMPING COEFFICIENT.

Set DEFINITION=CRITICAL DAMPING FRACTION to use a unitless damping coefficient, B.

The damping forces are calculated with $f_{vd} = B\sqrt{4mk_c}v_{rel}^{el}$, where m is the nodal mass, k_c is the nodal contact stiffness (in units of FL^{-1}), and v_{rel}^{el} is the rate of relative elastic slip between the surfaces. A default value of B=0.03 is used for kinematic contact with softened behavior and penalty contact.

Set DEFINITION=DAMPING COEFFICIENT to specify damping in terms of a damping coefficient, C, with units of pressure per relative velocity such that the damping forces will be calculated with $f_{vd} = CAv_{rel}^{el}$, where A is the nodal area and v_{rel}^{el} is the rate of relative elastic slip between the surfaces. If a contact area is not defined, such as may occur for node-based surfaces or for GAP- or ITT-type contact elements, coefficient units are force per relative velocity. For contact with three-dimensional beams or trusses, coefficient units are force per unit length per unit velocity.

Optional parameters:

TANGENT FRACTION

Set this parameter equal to the tangential damping coefficient divided by the normal damping coefficient. This parameter affects only the tangential damping; the normal direction damping coefficient is defined on the data line below. Set this parameter equal to zero if no tangential damping is desired. The default is 0.0 in Abaqus/Standard and 1.0 in Abaqus/Explicit.

Data line to define viscous damping in the normal direction between the contacting surfaces:

First (and only) line:

1. Damping coefficient.

The remaining data items are used only in Abaqus/Standard analyses. For Abaqus/Explicit damping is applied only when the surfaces are in contact, whereas for Abaqus/Standard damping is applied independent of the open/close state.

5. Clearance at which the damping coefficient is zero, c_0.
7. Fraction of the clearance interval between zero clearance and c_0 over which the damping coefficient is constant, $\eta \quad (0 \le \eta \le 1)$. The default is $\eta = 0.0$.

*CONTACT EXCLUSIONS: Specify self-contact surfaces or surface pairings to exclude from the general contact domain.

This option is used to exclude self-contact surfaces and surface pairings from consideration by the general contact algorithm. It should be used in conjunction with the *CONTACT option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Model, Step

Abaqus/CAE: Interaction module

References:

- *About General Contact in Abaqus/Standard*
- *About General Contact in Abaqus/Explicit*
- *CONTACT

There are no parameters associated with this option.

Data lines to specify contact exclusions:

First line:

1. The name of the first surface. If the first surface name is omitted, the default all-inclusive, element-based surface defined by Abaqus is assumed.
2. The name of the second surface. If the second surface name is omitted or is the same as the first surface name, Abaqus assumes that self-contact is being excluded. Self-contact means contact of a surface with itself, without consideration of whether a surface contains disconnected regions. If different names are specified for the first and second surfaces, self-contact is not excluded except in any overlap between the two surfaces.

Repeat this data line as often as necessary.

*CONTACT FILE: Define results file requests for contact variables.

This option is used to control writing contact variables (for contact surface pairs) to the Abaqus/Standard results (`.fil`) file.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Unsupported; Abaqus/CAE reads output from the output database file only.

References:

• *Output to the Data and Results Files*

Optional parameters:

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be written at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

MAIN

Set this parameter equal to the name of the main surface for which this output request is being made.

NSET

Set this parameter equal to the name of the node set for which this output request is being made.

SECONDARY

Set this parameter equal to the name of the secondary surface for which this output request is being made.

Data lines to request contact variable output in the results file:

First line:

1. Give the identifying keys for the variables to be written to the results file for this contact pair. The keys are defined in *Abaqus/Standard Output Variable Identifiers*.

Repeat this data line as often as necessary to define the list of variables to be written. If this line is omitted, the default variables will be output.

*CONTACT FORMULATION: Specify a nondefault contact formulation for the general contact algorithm.

This option is used to modify the contact formulation for specific contact interactions within the domain considered by general contact. It must be used in conjunction with the **CONTACT* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data in Abaqus/Standard; Model or history data in Abaqus/Explicit

Level: Model in Abaqus/Standard; Model or Step in Abaqus/Explicit

Abaqus/CAE: Interaction module

References:

- *Contact Controls Specific to General Contact in Abaqus/Standard*
- *Contact Formulation for General Contact in Abaqus/Explicit*
- **CONTACT*

Required parameters:

TYPE

Set TYPE=EDGE TO EDGE to control the edge-to-edge (beam-to-beam) contact formulations used in Abaqus/Standard. This setting does not apply for Abaqus/Explicit.

Set TYPE=MAIN SECONDARY ROLES to control main-secondary roles for specific interactions.

Set TYPE=PURE MAIN-SECONDARY to specify that a contact interaction should use pure main-secondary weighting for specific node-to-face contact surface pairs in Abaqus/Explicit. This setting does not apply for Abaqus/Standard.

Set TYPE=POLARITY to choose which sides of double-sided elements will be considered for node-to-face or Eulerian-Lagrangian contact with another surface in Abaqus/Explicit. This setting does not apply for Abaqus/Standard.

Set TYPE=SLIDING TRANSITION to control the smoothness of the surface-to-surface formulation upon sliding for specific interactions in Abaqus/Standard. This setting does not apply for Abaqus/Explicit.

Set TYPE=SLIDING FORMULATION to activate the small-sliding tracking approach for specific interactions in Abaqus/Standard. This setting does not apply for Abaqus/Explicit.

Optional parameters:

FORMULATION

This parameter applies only if TYPE=EDGE TO EDGE. It is used to activate the contact formulations to be used globally for edge-to-edge (beam-to-beam) contact.

Set FORMULATION=CROSS to activate an edge-to-edge contact formulation applicable to nonparallel beams that bases the contact normal direction on the cross product of the respective beam axial directions.

Set FORMULATION=RADIAL to activate an edge-to-edge contact formulation applicable to nearly parallel beams that bases the contact normal direction on a beam radial direction.

Set FORMULATION=BOTH to activate both edge-to-edge contact formulations.

Set FORMULATION=NO (default) to deactivate edge-to-edge contact formulations.

Data lines to control main-secondary roles for contact interactions:

First line:

1. The name of the first surface. If the first surface name is omitted, a default surface that encompasses the entire contact domain is assumed.
2. The name of the second surface.
3. The “word”BALANCED, the “word”SECONDARY, or the “word”MAIN. A balanced main-secondary formulation is used if BALANCED is specified; otherwise, a pure main-secondary formulation is used with SECONDARY or MAIN indicating the desired behavior of the first surface.

Repeat this data line as often as necessary. Specifying named XFEM-based crack surfaces on the data lines is not currently supported.

Data lines to assign pure main-secondary roles to contact interactions in Abaqus/Explicit:

First line:

1. The name of the first surface. If the first surface name is omitted, a default surface that encompasses the entire contact domain is assumed.
2. The name of the second surface.
3. The “word”SECONDARY (default) or the “word”MAIN. This entry refers to the desired behavior of the first surface.

Repeat this data line as often as necessary.

Data lines to assign polarity to contact interactions in Abaqus/Explicit:

First line:

1. The name of the first surface. If the first surface name is omitted, a default surface that encompasses the entire contact domain is assumed.
2. The name of the second surface.
3. The label SPOS , the label SNEG, the label TWO SIDED, or blank (the polarity of each face in the second surface will be defined according to the side label given in the surface definition). This entry refers to the sides of the (double-sided) elements in the second surface that will be considered for node-to-face or Eulerian-Lagrangian contact with the first surface.

Repeat this data line as often as necessary.

Data lines to control the smoothness of the surface-to-surface formulation upon sliding for contact interactions in Abaqus/Standard:

First line:

1. The name of the first surface. If the first surface name is omitted, a default surface that encompasses the entire contact domain is assumed.
2. The name of the second surface.
3. The “words”ELEMENT ORDER SMOOTHING (default), the “words”LINEAR SMOOTHING, or the “words”QUADRATIC SMOOTHING.

Repeat this data line as often as necessary.

Data lines to specify the small-sliding tracking approach for contact interactions in Abaqus/Standard:

First line:

1. The name of the first surface. If the first surface name is omitted, a default surface that encompasses the entire contact domain is assumed.
2. The name of the second surface.
3. The “words”SMALL SLIDING to specify the small-sliding tracking approach or a blank entry to specify the default behavior, which is the finite-sliding tracking approach. Edges and vertices are deactivated for interactions with the small-sliding tracking approach.

Repeat this data line as often as necessary.

There are no data lines to control the edge-to-edge (beam-to-beam) contact formulations in Abaqus/Standard

*CONTACT INCLUSIONS: Specify self-contact surfaces or surface pairings to include in the general contact domain.

This option is used to specify the self-contact surfaces and surface pairings that should be considered by the general contact algorithm. It should be used in conjunction with the **CONTACT* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Model, Step

Abaqus/CAE: Interaction module

References:

- *About General Contact in Abaqus/Standard*
- *About General Contact in Abaqus/Explicit*
- **CONTACT*

Optional parameters:

ALL EXTERIOR

Include this parameter to specify self-contact for a default unnamed, all-inclusive surface that includes all element-based surface facets and all analytical rigid surfaces. This is the simplest way to define the contact domain. The option should have no data lines when this parameter is used.

If this parameter is omitted, the contact surfaces must be specified on the data lines.

Data lines to specify contact inclusions if the ALL EXTERIOR parameter is omitted:

First line:

1. The name of the first surface. If the first surface name is omitted, the default all-inclusive, surface defined by Abaqus is assumed.
2. The name of the second surface. If the second surface name is omitted or is the same as the first surface name, Abaqus assumes that self-contact is defined. Self-contact means contact of a surface with itself, without consideration of whether a surface contains disconnected regions. If different names are specified for the first and second surfaces, self-contact is not considered except in any overlap between the two surfaces.

Repeat this data line as often as necessary.

*CONTACT INITIALIZATION ASSIGNMENT: Assign contact initialization methods for general contact.

This option is used to modify contact initialization methods for specific contact interactions within the domain considered by general contact. It must be used in conjunction with the **CONTACT* and **CONTACT INITIALIZATION DATA* options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Model, Step

Abaqus/CAE: Interaction module

References:

- *Contact Initialization for General Contact in Abaqus/Standard*
- *Contact Initialization for General Contact in Abaqus/Explicit*
- **CONTACT*
- **CONTACT INITIALIZATION DATA*

There are no parameters associated with this option.

Data lines to assign nondefault contact initialization methods:

First line:

1. The name of the first surface. If the first surface name is omitted, a default surface that encompasses the entire general contact domain is assumed.
2. The name of the second surface. If the second surface name is omitted or is the same as the first surface name, the specified contact initialization method definition is assigned to contact interactions between the first surface and itself.
3. The name of the **CONTACT INITIALIZATION DATA* definition to be assigned.

Optional data item for Abaqus/Explicit when a **CONTACT INITIALIZATION DATA*, INITIAL CLEARANCE definition specified with ADJUST=YES is referenced:

9. Blank, the "words" FIRST SURFACE SFC, or the "words" SECOND SURFACE SFC to indicate how the surfaces will be treated while adjusting the surface nodes to resolve contact clearance violations. A blank entry indicates that the interaction will be treated as balanced main-secondary. A setting of FIRST SURFACE SFC or SECOND SURFACE SFC specifies the behavior as a secondary surface of the first or second surface, respectively, in a pure main-secondary interaction.

Repeat this data line as often as necessary. If the contact initialization method assignments overlap, the last assignment applies in the overlap region.

*CONTACT INITIALIZATION DATA: Define contact initialization methods for general contact.

This option is used to define contact initialization methods. The contact initialization method is applied to a contact interaction using the **CONTACT INITIALIZATION ASSIGNMENT* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Contact Initialization for General Contact in Abaqus/Standard*
- *Common Difficulties Associated with Contact Modeling in Abaqus/Standard*
- *Contact Initialization for General Contact in Abaqus/Explicit*
- **CLEARANCE*
- **CONTACT*
- **CONTACT INITIALIZATION ASSIGNMENT*

Required parameters:

NAME

Set this parameter equal to a label that is used to refer to this contact initialization method.

Optional, mutually exclusive parameters:

INITIAL CLEARANCE

Set this parameter equal to a positive value to specify an initial clearance distance.

In Abaqus/Standard set this parameter equal to the name of a **CLEARANCE* option to specify an initial clearance distance. The named clearance definition can be used for general contact interactions in portions of the model where the small-sliding tracking approach is active, for general contact interactions that use the finite-sliding tracking approach, or for interactions in procedures that have no mechanical degrees of freedom, such as heat transfer, coupled thermal-electrical, or coupled thermal-electrochemical.

INTERFERENCE FIT

Include this parameter without setting it to a value to treat initial overclosures as interference fits.

Set this parameter equal to a positive value to specify an interference distance.

If this parameter is omitted, initial overclosures are resolved with strain-free adjustments.

Optional parameters:

ADJUST

Set ADJUST=YES (default) to resolve clearances/overclosures by adjusting the nodal coordinates without creating strain in the model. In Abaqus/ExplicitADJUST=YES can be used only for clearances/overclosures defined in the first step of an analysis. In Abaqus/StandardADJUST=YES can be used only for clearances/overclosures defined in model data (before the first step).

Set ADJUST=NO to store contact offsets so that the clearances/overclosures can be satisfied without adjusting the nodal coordinates. In Abaqus/Standard this setting can be used for general contact interactions in portions of the model where the small-sliding tracking approach is active, for interactions that are permanently bonded with cohesive contact, and for interactions in heat transfer and coupled electrical-thermal procedures that have no mechanical degrees of freedom. The setting is ignored when the INTERFERENCE FIT parameter or the INITIAL CLEARANCE parameter with a value is specified.

MINIMUM DISTANCE

This parameter applies only to Abaqus/Standard analyses.

Set MINIMUM DISTANCE=YES (default) to automatically activate localized contact damping when nearby surfaces are touching at only a single point.

Set MINIMUM DISTANCE=NO to forgo this automatic localized damping.

SEARCH ABOVE

Set this parameter equal to a positive value to ensure that the search zone for contact initialization includes gaps at least as large as the specified value.

SEARCH BELOW

Set this parameter equal to a positive value to ensure that the search zone for contact initialization includes overclosures at least as large as the specified value.

SEARCH NSET

This parameter applies only to Abaqus/Explicit analyses using the INITIAL CLEARANCE parameter. It cannot be used if either the SEARCH ABOVE or SEARCH BELOW parameter is used.

Set this parameter equal to the name of the node set containing the secondary nodes to be included in the initial clearance specification. The specified clearance is enforced at all secondary nodes in this node set irrespective of whether they are above or below their respective main surfaces. This parameter can also be used to identify initially bonded nodes in a VCCT analysis.

STEP FRACTION

This parameter applies only to Abaqus/Explicit analyses using the INTERFERENCE FIT parameter.

Fraction of the step time (between 0.0 and 1.0) in which the interference fit has to be solved. The default value is 1.0.

There are no data lines associated with this option.

*CONTACT INTERFERENCE: Prescribe time-dependent allowable interferences of contact pairs and contact elements.

This option is used to prescribe time-dependent allowable interferences for contact pairs and contact elements. It is useful for solving problems where there are large initial overclosures of the contacting bodies.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *Modeling Contact Interference Fits in Abaqus/Standard*
- *Contact Initialization for Contact Pairs in Abaqus/Standard*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the magnitude of the prescribed interference during the step. If this parameter is omitted, the prescribed interference is applied immediately at the beginning of the step and ramped down to zero linearly over the step.

OP

Set OP=MOD (default) for existing **CONTACT INTERFERENCE* definitions to remain, with this option defining a contact interference to be added or modified. Set OP=NEW if all **CONTACT INTERFERENCE* definitions defined in previous steps should be removed.

SHRINK

Include this parameter to invoke the automatic shrink fit capability. This capability can be used only in the first step of an analysis. When this parameter is included, no data are required other than the contact pairs or elements to which the option is applied. In addition, any AMPLITUDE reference specified will be ignored.

TYPE

Use this parameter to specify whether the prescribed interference will be applied to contact pairs or contact elements. Set TYPE=CONTACT PAIR (default) to specify a contact interference for contact pairs. Set TYPE=ELEMENT to specify a contact interference for contact elements.

Data lines to define an allowable contact interference for a contact pair (TYPE=CONTACT PAIR):

First line:

1. Secondary surface name.
2. Main surface name. It must be distinct from the secondary surface name; self-contact is not allowed with this option.

If the SHRINK parameter is included, no additional data are required. Otherwise:

7. Reference allowable interference, v.
9. X-direction cosine of the shift direction vector (optional).
11. Y-direction cosine of the shift direction vector (optional).
13. Z-direction cosine of the shift direction vector (optional).

Repeat this data line as often as necessary to specify additional contact pairs. Each line defines a distinct contact interference between one contact pair.

Data lines to define an allowable contact interference for contact elements (TYPE=ELEMENT):

First line:

1. Name of the element set containing the contact elements.

If the SHRINK parameter is included, no additional data are required. Otherwise:

5. Reference allowable interference, v.
7. X-direction cosine of the shift direction vector (optional).
9. Y-direction cosine of the shift direction vector (optional).
11. Z-direction cosine of the shift direction vector (optional).

Repeat this data line as often as necessary to specify additional element sets containing contact elements.

*CONTACT MASS SCALING: Specify contact mass scaling during the step.

This option is used to specify contact mass scaling during the step for part or all of the model.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Model, Step

Abaqus/CAE: Interaction module

References:

• *Mass Scaling*

Optional parameters:

LOCATION

Set LOCATION=NONE for no contact mass scaling.

Set LOCATION=ELEMENT MASS SCALING (default) to apply contact mass scaling only to surface nodes potentially involved in element mass scaling.

Set LOCATION=ALL CONTACT SURFACES to apply contact mass scaling to all the surface nodes involved in general contact.

Set LOCATION=SPECIFIED SURFACES to apply contact mass scaling only to the contact surfaces listed in the data lines.

Data lines to define contact mass scaling when LOCATION=SPECIFIED SURFACES:

First line:

1. List of contact surfaces, eight per line.

Repeat this data line as often as necessary.

*CONTACT OUTPUT: Specify contact variables to be written to the output database.

This option is used to write contact variables to the output database. It must be used in conjunction with the *OUTPUT option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Output to the Output Database*
- *OUTPUT

One of the following mutually exclusive parameters is required when the *CONTACT OUTPUT option is used in conjunction with the *OUTPUT, HISTORY option in an Abaqus/Explicit analysis:

CPSET

Set this parameter equal to the name of the contact pair set for which this output request is being made.

SURFACE

Set this parameter equal to the name of the surface in the general contact domain for which this output request is being made.

Optional parameter when the SURFACE parameter is included:

SECOND SURFACE

This parameter is used to write contact output limited to a pair of contact surfaces. Set this parameter equal to the name of the second surface in the general contact domain that along with the first surface specified by the SURFACE parameter identifies the pair of contact surfaces.

Optional parameters when the *CONTACT OUTPUT option is used in conjunction with the *OUTPUT, FIELD option in an Abaqus/Explicit analysis:

CPSET

Set this parameter equal to the name of the contact pair set for which this output request is being made. If this parameter and the GENERAL CONTACT parameter are omitted, the output will be written for all of the contact pairs in the model and the general contact domain (if it has been defined).

GENERAL CONTACT

Include this parameter to request output for the general contact domain. If this parameter and the CPSET parameter are omitted, the output will be written for all of the contact pairs in the model and the general contact domain (if it has been defined).

SURFACE

Set this parameter equal to the name of the surface in the general contact domain for which this output request is being made.

Optional parameter when the SURFACE parameter is included:

SECOND SURFACE

This parameter is used to write contact output limited to a pair of contact surfaces. Set this parameter equal to the name of the second surface in the general contact domain that along with the first surface specified by the SURFACE parameter identifies the pair of contact surfaces.

Optional parameter in Abaqus/Explicit analyses:

VARIABLE

Set VARIABLE=ALL to indicate that all contact variables applicable to this procedure should be written to the output database.

Set VARIABLE=PRESELECT to indicate that the default contact output variables for the current procedure type should be written to the output database. Additional output variables can be requested on the data lines.

If this parameter is omitted, the contact variables requested for output must be specified on the data lines.

Optional parameters in Abaqus/Standard analyses:

MAIN

Set this parameter equal to the name of a main surface of one or more contact pairs for which this request is being made. Specifying this parameter eliminates output for general contact associated with this request, regardless of whether or not this surface participates in general contact.

NSET

Set this parameter equal to the name of the node set for which this output request is being made.

SECONDARY

Set this parameter equal to the name of a secondary surface of one or more contact pairs for which this request is being made. Specifying this parameter eliminates output for general contact associated with this request, regardless of whether or not this surface participates in general contact.

SURFACE

Set this parameter equal to the name of the surface containing facets of cracked enriched elements for which this request is being made. This parameter is valid only for small sliding contact between cracked surfaces in enriched elements.

VARIABLE

Set VARIABLE=ALL to indicate that all contact variables applicable to this procedure should be written to the output database.

Set VARIABLE=PRESELECT to indicate that the default contact output variables for the current procedure type should be written to the output database. Additional output variables can be requested on the data lines.

If this parameter is omitted, the contact variables requested for output must be specified on the data lines.

Data lines to request contact output:

First line:

1. Specify the identifying keys for the output variables to be written to the output database. The keys are defined in *Abaqus/Standard Output Variable Identifiers* and *Abaqus/Explicit Output Variable Identifiers*.

Repeat this data line as often as necessary to define the list of variables to be written to the output database.

*CONTACT PAIR: Define surfaces that contact each other.

This option is used to define pairs of surfaces or pairs of node sets and surfaces that may contact or interact with each other during the analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data in Abaqus/Standard; History data in Abaqus/Explicit

Level: Model in Abaqus/Standard; Step in Abaqus/Explicit

Abaqus/CAE: Interaction module

References:

- *About Contact Pairs in Abaqus/Standard*
- *Contact Initialization for Contact Pairs in Abaqus/Standard*
- *Defining Tied Contact in Abaqus/Standard*
- *Generally Applicable Contact Controls in Abaqus/Standard*
- *Contact Formulations in Abaqus/Standard*
- *Smoothing Contact Surfaces in Abaqus/Standard*
- *Common Difficulties Associated with Contact Modeling in Abaqus/Standard*
- *About Contact Pairs in Abaqus/Explicit*
- *Contact Formulations for Contact Pairs in Abaqus/Explicit*
- *Contact Initialization for Contact Pairs in Abaqus/Explicit*

Defining contacting surfaces in an Abaqus/Standard analysis

Required parameters:

INTERACTION

Set this parameter equal to the name of the **SURFACE INTERACTION* property definition associated with the contact pair being defined.

Optional parameters:

ADJUST

Set this parameter equal to a node set label or a value to adjust the initial positions of the surfaces specified in this option. These adjustments are made at the start of the analysis and do not create any strain.

Set ADJUST=YES and use the POSITION TOLERANCE parameter to adjust the initial positions of the surfaces specified in this option. These adjustments are made at the start of the analysis and do not create any strain.

Alternatively, for SMALL SLIDING contact or TIED contact, set ADJUST=NO and use the POSITION TOLERANCE parameter to control which secondary nodes are considered initially in contact or tied. In these cases, initial positions of the surfaces are not adjusted, and offset distances are considered for penetration calculations of the respective secondary nodes such that the initial penetration is zero.

This parameter is required for TIED contact.

If this parameter is omitted, overclosures are treated as interference fits.

EXTENSION ZONE

Set this parameter equal to a fraction of the end segment or facet edge length by which the main surface is to be extended to avoid numerical roundoff errors associated with contact modeling. The value given must lie between 0.0 and 0.2. The default is 0.1. This parameter affects only node-to-surface contact.

GEOMETRIC CORRECTION

Set this parameter equal to the name of the surface smoothing property defined by **SURFACE SMOOTHING*. This parameter affects only surface-to-surface contact.

HCRIT

Set this parameter equal to the distance by which a point on the secondary surface must penetrate the main surface before Abaqus/Standard abandons the current increment and tries again with a smaller increment. The default value of HCRIT is half of the length of a characteristic element face on the secondary surface. This parameter does not apply to contact pairs that use the finite-sliding, surface-to-surface contact formulation.

MIDFACE NODES

Set MIDFACE NODES=YES to automatically convert most three-dimensional second-order element types with no midface node (serendipity elements) that form a secondary surface of a surface-to-surface contact pair into elements with a midface node.

Set MIDFACE NODES=NO (default) to avoid adding midface nodes to elements underlying the secondary surface of a surface-to-surface contact pair.

This parameter can be used only with surface-to-surface contact pairs. Abaqus/Standard automatically converts most serendipity elements that form a secondary surface of a node-to-surface contact pair into elements with a midface node.

MINIMUM DISTANCE

Set MINIMUM DISTANCE=YES (default) to automatically activate localized contact damping when nearby surfaces are initially touching at only a single point.

Set MINIMUM DISTANCE=NO to forgo this automatic localized damping.

This parameter can be used only with the finite-sliding, surface-to-surface contact formulation.

NO THICKNESS

Include this parameter to ignore surface thickness effects in the contact calculations. This parameter affects only contact formulations that account for surface thickness by default (it does not affect finite-sliding, node-to-surface contact).

POSITION TOLERANCE

This parameter is applicable only if ADJUST=YES or ADJUST=NO.

Set this parameter equal to the upper bound of the initial gap distance for which secondary nodes are to be adjusted if ADJUST=YES. If this parameter is omitted, Abaqus/Standard adjusts only those secondary nodes that are penetrating the main surface.

Set this parameter equal to the upper bound of the initial gap distance for which secondary nodes are considered initially in contact or tied if ADJUST=NO. Initial positions of these secondary nodes are not adjusted automatically. Instead, offset distances for use in penetration calculations for these nodes are generated internally such that the initial penetration is zero with consideration of the offset distance. These offsets remain in effect throughout the simulation.

SMALL SLIDING

Include this parameter to indicate that the small-sliding contact formulation, rather than the finite-sliding contact formulation, should be used. This parameter is not allowed with self-contact.

SMOOTH

Set this parameter equal to the degree of smoothing used for element-based main surfaces in the finite-sliding, node-to-surface contact formulation. The value given must lie between 0.0 and 0.5. The default is 0.2. This parameter does not affect contact pairs with analytical rigid surfaces or contact formulations other than the finite-sliding, node-to-surface contact formulation.

SLIDING TRANSITION

Set SLIDING TRANSITION=ELEMENT ORDER SMOOTHING to have smoothing of the nodal force redistribution upon sliding be of the same order as the elements underlying the secondary surface.

Set SLIDING TRANSITION=LINEAR SMOOTHING to have linear smoothing of the nodal force redistribution upon sliding.

Set SLIDING TRANSITION=QUADRATIC SMOOTHING to have quadratic smoothing of the nodal force redistribution upon sliding.

This parameter can be used only with the surface-to-surface contact formulation.

SUPPLEMENTARY CONSTRAINTS

Set SUPPLEMENTARY CONSTRAINTS=SELECTIVE (default) to use a selective scheme of supplementary constraints.

Set SUPPLEMENTARY CONSTRAINTS=YES to add the supplementary contact constraints when applicable.

Set SUPPLEMENTARY CONSTRAINTS=NO to forgo the supplementary contact constraints.

TIED

Include this parameter to indicate that the surfaces of this **CONTACT PAIR* are to be “tied” together for the duration of the simulation. The ADJUST parameter is required when the TIED parameter is used. This parameter is not allowed with self-contact.

TRACKING

This parameter controls which contact tracking algorithm is used for finite-sliding, surface-to-surface contact; it has no effect on contact pairs that use other formulations.

Set TRACKING=PATH (default) to invoke a path-based contact tracking algorithm for finite-sliding, surface-to-surface contact.

Set TRACKING=STATE to invoke a state-based contact tracking algorithm for finite-sliding, surface-to-surface contact.

TYPE

Set TYPE=NODE TO SURFACE (default) to have the contact constraint coefficients generated according to the interpolation functions at the point where the secondary node projects onto the main surface.

Set TYPE=SURFACE TO SURFACE to have the contact constraint coefficients generated such that stress accuracy is optimized for the specified surface type pairings. This parameter setting will be ignored for contact pairs that include a node-based surface.

Data lines to define the surfaces and node sets forming the contact pairs:

First line:

1. The secondary surface name.
2. The main surface name. If the main surface name is omitted or is the same as the secondary surface name, Abaqus/Standard assumes that self-contact is defined.
3. Optional orientation name to specify the local tangent directions on the secondary surface.
4. Optional orientation name to specify the local tangent directions on the main surface.

Repeat this data line as often as necessary to define all of the surfaces or node sets forming the contact pairs. Each data line defines a pair of surfaces or a node set and a surface that may interact with one another.

Defining contacting surfaces in an Abaqus/Explicit analysis

Optional parameters:

CPSET

Set this parameter equal to the name of the contact pair set to which the contact pairs being defined should be added. The CPSET name can be used to associate contact pairs with a **CLEARANCE* option or with a **CONTACT CONTROLS* option, which can be used to adjust algorithmic control parameters. It can also be used with the **CONTACT OUTPUT* option to specify the contact pairs for which output database results are desired.

INTERACTION

Set this parameter equal to the name of the **SURFACE INTERACTION* property definition associated with the contact pair being defined.

MECHANICAL CONSTRAINT

Set this parameter equal to the name of the method used to enforce the contact constraints.

Set MECHANICAL CONSTRAINT=KINEMATIC (default) to choose the kinematic contact method.

Set MECHANICAL CONSTRAINT=PENALTY to choose the penalty contact method.

OP

Set OP=ADD (default) to add new contact pairs to the existing set of contact pairs. Set OP=DELETE to remove the contact pairs given in this use of the option from the active set of contact pairs.

SMALL SLIDING

Include this parameter to indicate that the small-sliding contact formulation, rather than the finite-sliding contact formulation, should be used. This parameter can be used only for contact pairs that are defined in the first step of the simulation and use the kinematic constraint method.

WEIGHT

Set this parameter equal to the weighting factor for the contact surfaces.

Data lines to define the surfaces and node sets forming contact pairs:

First line:

1. The name of the first surface.

2. The name of the second surface. If the second surface name is omitted or is the same as the first surface name, Abaqus/Explicit assumes that self-contact is defined.

Repeat this data line as often as necessary define all of the surfaces or node sets forming contact pairs. Each data line defines a pair of surfaces or a node set and a surface that may interact with one another.

*CONTACT PERMEABILITY: Specify fluid permeability contact property.

This option is used to modify pore fluid permeability in a surface interaction model. It must be used in conjunction with the **SURFACE INTERACTION* option. If this option is omitted, the intended idealized behavior is no resistance to fluid flow (infinite permeability) across an interface while contact is active and for clearances up to a default tolerance distance, which is approximated with a penalty method (large permeability) for general contact and directly enforced for contact pairs.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Pore Fluid Contact Properties*
- *Coupled Pore Fluid Diffusion and Stress Analysis*
- **SURFACE INTERACTION*
- **SOILS*

Optional parameters:

CONSTRAINT METHOD

Set this parameter equal to the constraint enforcement method. The default setting and usage of this parameter depends on whether the contact permeability is specified on the data lines.

Set CONSTRAINT METHOD=DIRECT to directly enforce the intended physical model for contact permeability without approximation.

- If contact permeability is specified on the data lines, this setting is the default and must be in effect.
- If contact permeability is not specified on the data lines, this setting is the default for contact pairs and can optionally be used for general contact. Matching pore pressures across the interface are strictly enforced (corresponding to infinite contact permeability).

Set CONSTRAINT METHOD=PENALTY to numerically approximate the enforcement of matching pore pressures across the interface.

- If contact permeability is specified on the data lines, this setting is not applicable and not allowed.
- If contact permeability is not specified on the data lines, this setting is the default for general contact and can optionally be used for contact pairs.

CUTOFF FLOW ACROSS

Set this parameter equal to a cutoff clearance distance above which no fluid flow occurs across a contact interface.

CUTOFF GAP FILL

Set this parameter equal to a cutoff clearance distance above which no fluid flow occurs into or out of a contact interface due to changes in clearance distance.

DEPENDENCIES

Set this parameter equal to the number of field variables on which the contact permeability, $\boldsymbol{k}$, depends.

Data lines to define the contact permeability, k, directly:

First line:

1. Contact permeability, k. (Units of L^2TM^{-1}.)
2. Contact pressure, $p_{contact}$.
3. Average pore pressure, p_{pore}.
4. Average temperature, $\overline{\theta}$.
5. Average value of the first field variable, $\overline{f_1}$.
6. Average value of the second field variable, $\overline{f_2}$.
7. Etc.

Repeat this data line as often as necessary to define the dependence of contact permeability on the contact pressure, average surface pore pressure, average surface temperature, and the average of any predefined field variables on the surfaces.

*CONTACT PRINT: Define print requests for contact variables.

This option is used to provide tabular printed output of contact variables for contact surface pairs.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

• *Output to the Data and Results Files*

Optional parameters:

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

MAIN

Set this parameter equal to the name of the main surface for which this output request is being made.

NSET

Set this parameter equal to the name of the node set for which this output request is being made.

SECONDARY

Set this parameter equal to the name of the secondary surface for which this output request is being made.

SUMMARY

Set SUMMARY=YES (default) to obtain a summary of the maximum and minimum values in each column of the table and their locations. Set SUMMARY=NO to suppress this summary.

TOTALS

Set TOTALS=YES to print the total of each column in the table. The default is TOTALS=NO.

Data lines to request contact variable output in the data file:

First line:

1. Give the identifying keys for the variables to be written to the data file for this contact pair. The keys are defined in *Abaqus/Standard Output Variable Identifiers*.

Repeat this data line as often as necessary: each line defines a table. If this line is omitted, the default variables will be output.

*CONTACT PROPERTY ASSIGNMENT: Assign contact properties for the general contact algorithm.

This option is used to modify contact properties for specific contact interactions within the domain considered by general contact. It must be used in conjunction with the *CONTACT and *SURFACE INTERACTION options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data in Abaqus/Standard; Model or history data in Abaqus/Explicit

Level: Model in Abaqus/Standard; Model or Step in Abaqus/Explicit

Abaqus/CAE: Interaction module

References:

- *Contact Properties for General Contact in Abaqus/Standard*
- *Assigning Contact Properties for General Contact in Abaqus/Explicit*
- *CONTACT
- *SURFACE INTERACTION

There are no parameters associated with this option.

Data lines to assign nondefault contact properties:

First line:

1. The name of the first surface or material. If this entry is omitted, a default surface that encompasses the entire general contact domain is assumed.
2. The name of the second surface or material. If this entry is omitted or is the same as the first entry, the specified contact property definition is assigned to contact interactions between the first surface and itself.
3. The name of the model data *SURFACE INTERACTION property definition to be assigned.
4. Specify SURFACE (default) or omit if the first entry is a surface name. Specify MATERIAL if the first entry is a material name.
5. Specify SURFACE (default) or omit if the second entry is a surface name. Specify MATERIAL if the second entry is a material name.

Repeat this data line as often as necessary. If the contact property assignments overlap, the last assignment applies in the overlap region.

*CONTACT RESPONSE: Define contact responses for design sensitivity analysis.

This option is used to write contact response sensitivities to the output database. It must be used in conjunction with the **DESIGN RESPONSE* option.

Products: Abaqus/Design

Type: History data

Level: Step

References:

- *Direct Design Sensitivity Analysis*
- **DESIGN RESPONSE*

Optional parameters:

MAIN

Set this parameter equal to the name of the main surface for which this output request is being made.

NSET

Set this parameter equal to the name of the node set for which this output request is being made.

SECONDARY

Set this parameter equal to the name of the secondary surface for which this output request is being made.

Data lines to request contact sensitivity output:

First line:

1. Specify the identifying keys for the responses whose sensitivities are to be written to the output database. The valid keys are listed in *Direct Design Sensitivity Analysis*.

Repeat this data line as often as necessary to define the contact responses whose sensitivities are to be written to the output database.

*CONTACT STABILIZATION: Define contact stabilization controls for general contact.

Multiple instances of the option can be used to define contact stabilization controls for general contact in Abaqus/Standard.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *Stabilization for General Contact in Abaqus/Standard*
- **CONTACT*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines a time-dependent scale factor for contact stabilization over the step. If this parameter is omitted, the scale factor ramps linearly from unity to zero over the step.

RANGE

Set this parameter equal to the clearance at which the stabilization becomes zero; no contact stabilization is applied where the separation between surfaces exceeds this value. By default, this clearance is calculated by Abaqus/Standard based on the facet sizes on contact surfaces.

REDUCTION PER INCREMENT

Set this parameter equal to a fraction by which contact stabilization is successively scaled each increment after the first increment. The default value is 0.1 for interactions specified on the data lines of this option, which leads to scaling of contact stabilization by 0.1 in the second increment, scaling of contact stabilization by 0.01 in the third increment, and so forth. This scaling of contact stabilization acts in combination with other scale factors.

RESET

Include this parameter to cancel carryover effects from contact stabilization specifications involving nondefault amplitudes that appeared in previous steps. This parameter cannot be used in conjunction with any other parameters. There are no data lines if this parameter is included.

SCALE FACTOR

Set this parameter equal to a factor by which Abaqus/Standard will scale the contact stabilization coefficient. The default value is unity for the interactions specified on the data lines of this option.

Set SCALE FACTOR=USER ADAPTIVE to scale the contact stabilization coefficient by a factor that decreases over iterations within each increment, according to the pattern specified on the data line.

TANGENT FRACTION

Set this parameter equal to a factor that scales the contact stabilization coefficient in the tangential direction only. The default value is zero, such that no contact stabilization is applied in the tangential direction.

Data lines if both the RESET parameter and SCALE FACTOR=USER ADAPTIVE are omitted:

First line:

1. The name of the first surface. If the first surface name is omitted, a default surface that encompasses the entire general contact domain (including all nodes and facets) is assumed.
2. The name of the second surface. If the second surface name is omitted or is the same as the first surface name, the specified stabilization settings are assigned to contact interactions between the first surface and itself.

Repeat this data line as often as necessary. Specifying named XFEM-based crack surfaces on the data lines is not currently supported.

Data line if SCALE FACTOR=USER ADAPTIVE:

First line:

1. Up to eight scale values in descending order.

*CONTOUR INTEGRAL: Provide contour integral estimates.



Warning: Contour integrals are not calculated accurately for the bending stress in shells. If contour integral values are needed where the bending stress is significant, use second-order solid elements (C3D20 or C3D27) in the crack-tip region where the integral is evaluated instead of shell elements. Contour integrals should not be requested in a linear perturbation step.

The **CONTOUR INTEGRAL* option offers the evaluation of the *J*-integral, the C_t-integral, the stress intensity factors, and the *T*-stress for fracture mechanics studies based on either the conventional finite element method or the extended finite element method (XFEM). The option also computes the crack propagation direction at initiation when the stress intensity factors are evaluated. Contour integrals along several different crack fronts can be evaluated by repeating this option as often as needed in the step definition.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *Contour Integral Evaluation*
- *Modeling Discontinuities as an Enriched Feature Using the Extended Finite Element Method*

Required parameters:

CONTOURS

Set this parameter equal to the number of contours to be used. Each contour provides an evaluation of the contour integral.

Optional parameters:

CRACK NAME

Set this parameter equal to a label to use to refer to the crack. When the extended finite element method is used, set this parameter equal to the name assigned to the enriched feature on the **ENRICHMENT* option.

CRACK TIP NODES

Include this parameter to indicate that the crack tip nodes are specified to form the crack front line. If this parameter is omitted, the crack front line is formed along the first nodes of the crack front node sets. (The first node is the node with the smallest node number for each crack front node set, unless the node set is generated as unsorted.)

This parameter is not relevant when the XFEM parameter is specified.

DIRECTION

This parameter can be used only in combination with the TYPE=K FACTORS parameter.

Set DIRECTION=MTS (default) to choose the maximum tangential stress criterion.

Set DIRECTION=MERR to choose the maximum energy release rate criterion.

Set DIRECTION=KII0 to choose the $K_{II} = 0$ criterion.

ELSET

Set this parameter equal to the name of the element set containing all elements inside the contour integral domain.

By default, Abaqus/Standard searches through all elements in the model to find the ones used for the domain. Therefore, this search is time consuming for extremely large models.

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output is always printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

LENGTH SCALING FACTOR

This parameter is relevant only when XFEM=LINE is used.

Include this parameter to indicate that no adjustments will be made with both the locations and the radii of all nodes along the crack front for the line integral evaluations.

NON-PROPORTIONAL STRESSING

Include this parameter to account for non-proportional stressing.

If this parameter is omitted, the proportional stressing condition is assumed to be satisfied when the contour integrals are calculated.

NORMAL

Include this parameter to indicate that the direction normal to the plane of the crack $\mathbf{n}$ is specified.

Omit this parameter to indicate that the virtual crack extension direction $\mathbf{q}$ is specified.

This parameter is not relevant when the XFEM parameter is specified.

RING RADIUS

This parameter is relevant only when tetrahedral elements are used for fracture mechanics studies based on the conventional finite element method.

Set this parameter equal to a small radius from the crack tip within which rings of elements are used for fracture mechanics studies. The default value of the ring radius is twice the typical element characteristic length of those elements along the crack front. The default value works well for most problems.

OUTPUT

If this parameter is omitted, the contour integral values will be printed in the data (`.dat`) file but not stored in the results (`.fil`) file.

Set OUTPUT=FILE to store the contour integral values in the results file.

Set OUTPUT=BOTH to print the contour integral values in the data file and to store them in the results file.

RESIDUAL STRESS STEP

Use this parameter to account for the effect of residual stress gradients on the contour integral evaluation. Set this parameter equal to the step number from which the stress data in the last available increment of the specified step will be considered as residual stresses. The default is 0, in which case the residual stresses are defined by the specified initial conditions.

This parameter can be set equal to zero only when the XFEM parameter is specified.

SURFACE NORMAL

Set SURFACE NORMAL=NONE (default) if no surface normal is calculated.

Set SURFACE NORMAL=FREE to calculate the surface normals on the free surfaces.

Set SURFACE NORMAL=CRACK to calculate the surface normals on the crack surfaces.

Set SURFACE NORMAL=BOTH to calculate the surface normals on both the free and the crack surfaces.

SYMM

Include this parameter to indicate that the crack front is defined on a symmetry plane, with only half the structure modeled. The change in potential energy calculated from the virtual crack front advance is then doubled to compute the correct contour integral values.

This parameter is not relevant when the XFEM parameter is specified.

TYPE

Set TYPE=J (default) to specify *J*-integral calculations.

Set TYPE=C to specify C_t-integral calculations.

Set TYPE=K FACTORS to specify the calculations of the stress intensity factors.

Set TYPE=T-STRESS to specify the *T*-stress calculations.

XFEM

Include this parameter to indicate the type of integration method to use. This setting is applicable only to cracks modeled as an enriched feature (XFEM).

Set XFEM=DOMAIN to indicate that the fracture parameters are evaluated by using the domain integral method.

Set XFEM=LINE (default) to indicate that the fracture parameters are evaluated by using the line integral method.

Data lines if the NORMAL parameter is included but the CRACK TIP NODES and XFEM parameters are both omitted:

First line:

1. n_x-direction cosine of the normal to the plane of the crack (n_r for axisymmetric cases).
2. n_y-direction cosine of the normal to the plane of the crack (n_z for axisymmetric cases).
3. For three-dimensional cases give the n_z-direction cosine of the normal to the plane of the crack. This field can be left blank for two-dimensional and axisymmetric cases.

Second line:

1. A list of node set names that define the crack front (in two-dimensional cases this will be one node set only). Each node set must contain all the nodes at one position on the crack front.

Repeat the second data line as often as necessary to define the crack front node sets. Up to 16 entries are allowed per line.

Data lines if the NORMAL, CRACK TIP NODES, and XFEM parameters are all omitted:

First line:

1. Node set name. The node set must contain all the nodes at one position on the crack front.
2. q_x-direction cosine of the virtual crack extension direction (q_r for axisymmetric cases).
3. q_y-direction cosine of the virtual crack extension direction (q_z for axisymmetric cases).
4. For three-dimensional cases give the q_z-direction cosine of the virtual crack extension direction. This field can be left blank for two-dimensional and axisymmetric cases.

In two-dimensional cases only one data line is necessary. In three-dimensional cases repeat this data line as often as necessary to define the crack front node sets and virtual crack extension vectors along the crack front.

Data lines if the NORMAL and CRACK TIP NODES parameters are both included but the XFEM parameter is omitted:

First line:

1. n_x-direction cosine of the normal to the plane of the crack (n_r for axisymmetric cases).
2. n_y-direction cosine of the normal to the plane of the crack (n_z for axisymmetric cases).
3. For three-dimensional cases give the n_z-direction cosine of the normal to the plane of the crack. This field can be left blank for two-dimensional and axisymmetric cases.

Second line:

1. First crack front node set.
2. Node number of the first crack tip node or the node set that contains a crack tip node.
3. Second crack front node set.
4. Node number of the second crack tip node or node set that contains a crack tip node.
5. Etc., up to 8 pairs per line.

Repeat the second data line as often as necessary to define the crack front.

Data lines if the NORMAL and XFEM parameters are both omitted but the CRACK TIP NODES parameter is included:

First line:

1. Node set name. The node set must contain all the nodes at one position on the crack front.
2. Node number of the crack tip node or a node set that contains a crack tip node.
3. q_x-direction cosine of the virtual crack extension direction (q_r for axisymmetric cases).
4. q_y-direction cosine of the virtual crack extension direction (q_z for axisymmetric cases).
5. For three-dimensional cases give the q_z-direction cosine of the virtual crack extension direction. This field can be left blank for two-dimensional and axisymmetric cases.

In two-dimensional cases only one data line is necessary. In three-dimensional cases repeat this data line as often as necessary to define the crack front node sets and virtual crack extension vectors along the crack front.

No data lines are needed if the XFEM parameter is included

*CONTROLS: Reset solution controls.



Warning: This option is not needed in most nonlinear analyses, except for use with the parameter ANALYSIS=DISCONTINUOUS. However, if extreme nonlinearities occur, this option may be needed to obtain a solution. *Commonly Used Control Parameters* contains a discussion of the types of problems that may occur and the use of the **CONTROLS* option to overcome these problems.

This option can also be used in some cases to obtain a solution in a more efficient manner. Use of the option for this latter purpose is intended for experienced users only.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *About Convergence and Time Integration Criteria*
- *Commonly Used Control Parameters*
- *Convergence Criteria for Nonlinear Problems*
- *Time Integration Accuracy in Transient Problems*

Required, mutually exclusive parameters:

ANALYSIS

Set ANALYSIS=DISCONTINUOUS to set parameters that will usually improve efficiency for severely discontinuous behavior, such as frictional sliding or concrete cracking, by allowing relatively many iterations prior to beginning any checks on the convergence rate. This parameter overrides any values that may be set for the variables I_0 and I_R on the data lines associated with PARAMETERS=TIME INCREMENTATION. A less efficient solution may result if this parameter is set in problems that do not exhibit severely discontinuous behavior.

PARAMETERS

Set PARAMETERS=FIELD to set parameters for satisfying a field equation. In this case the FIELD parameter can be used to define the field for which the parameters are being given. If the FIELD parameter is omitted, the parameters are being set for all fields that are active in the problem.

Set PARAMETERS=CONSTRAINTS to set tolerances on constraint equations.

Set PARAMETERS=LINE SEARCH to set line search control parameters.

Set PARAMETERS=TIME INCREMENTATION to set time incrementation control parameters.

RESET

Include this parameter to reset all values to their defaults. The option should have no data lines when this parameter is used.

TYPE

Set TYPE=DIRECT CYCLIC to set parameters that will be used to control the stabilized state and plastic ratcheting detections and to specify when to impose the periodicity condition for direct cyclic analysis.

Set TYPE=NO CUTBACK SCALING to set the α and β parameters that will be used with the unstable crack growth criterion in an XFEM analysis, a VCCT analysis, or a PRX-VCCT analysis.

Set TYPE=VCCT LINEAR SCALING to set the β parameter that will be used with linear scaling for a VCCT debonding analysis.

Optional parameters:

CONVERGENCE CHECK

Include this parameter to specify which convergence conditions Abaqus/Standard must satisfy before it accepts the incremental solution as converged. For more information, see *Modifying the Default Convergence Checks*.

Set CONVERGENCE CHECK=STRICT (default) to determine incremental solution convergence based on the following:

- Ratio of largest residual flux to the time averaged flux is less than a tolerance, *and*
- Ratio of largest correction to the largest increment is less than a tolerance.

Set CONVERGENCE CHECK=MODERATE to determine incremental solution convergence based on the following:

- Ratio of largest residual flux to the largest flux is less than a tolerance, *and*
- Ratio of largest correction to the largest increment is less than a tolerance.

Set CONVERGENCE CHECK=RELAXED to determine incremental solution convergence based on the following:

- The ratio of largest correction to largest increment is less than a tolerance.

Set CONVERGENCE CHECK=CONCEPT DESIGN to determine incremental solution convergence based on the following:

- Ratio of largest residual flux to the time averaged flux is less than a tolerance, *or*
- Ratio of largest correction to the largest increment is less than a tolerance.

FIELD

This parameter can be used only with PARAMETERS=FIELD.

Set FIELD=CONCENTRATION to set parameters for the mass concentration field equilibrium equations.

Set FIELD=DISPLACEMENT to set parameters for the displacement field and warping degree of freedom equilibrium equations.

Set FIELD=ELECTRICAL POTENTIAL to set parameters for the electrical potential field equilibrium equations.

Set FIELD=FLUID ELECTRIC POTENTIAL to set parameters for the fluid electric potential field equilibrium equations in any of the coupled electrochemical analyses.

Set FIELD=GLOBAL (default) to define one set of parameters to be used for all active fields.

Set FIELD=HYDROSTATIC FLUID PRESSURE to set parameters for the hydrostatic fluid element volume constraint.

Set FIELD=ION CONCENTRATION to set parameters for the ion concentration field equilibrium equations in any of the coupled electrochemical analyses.

Set FIELD=MATERIAL FLOW to set parameters for the material flow degree of freedom for connector elements.

Set FIELD=PORE FLUID PRESSURE to set parameters for the pore liquid volumetric continuity equations.

Set FIELD=PRESSURE LAGRANGE MULTIPLIER to set parameters for the pressure Lagrange multiplier field equations.

Set FIELD=ROTATION to set parameters for the rotation field equilibrium equations.

Set FIELD=SPECIES CONCENTRATION to set parameters for the species concentration field equilibrium equations in any of the coupled electrochemical analyses.

Set FIELD=SLURRYVF to set parameters for the slurry concentration field continuity equations.

Set FIELD=TEMPERATURE to set parameters for the temperature field equilibrium equations.

Set FIELD=VOLUMETRIC LAGRANGE MULTIPLIER to set parameters for the volumetric Lagrange multiplier field equations.

Data lines for PARAMETERS=FIELD:

First line:

1. R_n^α, convergence criterion for the ratio of the largest (scaled) residual to the corresponding average flux norm for convergence. Default $R_n^\alpha = 5 \times 10^{-3}$.
2. C_n^α, convergence criterion for the ratio of the largest solution correction to the largest corresponding incremental solution value. Default $C_n^\alpha = 10^{-2}$.
3. $\tilde{q}_0^\alpha$, initial value of the time average flux for this step. The default is the time average flux from previous steps or 10^{-2} if this is Step 1.
4. $\tilde{q}_u^\alpha$, user-defined average flux. When this value is defined, $\tilde{q}^\alpha(t) = \tilde{q}_u^\alpha$ for all *t*.

The remaining items rarely need to be reset from their default values.

11. R_P^α, alternative residual convergence criterion to be used after I_P^α iterations. Default $R_P^\alpha = 2 \times 10^{-2}$.
13. ϵ^α, criterion for zero flux compared to $\tilde{q}^\alpha$. Default $\epsilon^\alpha = 10^{-5}$.
15. C_ϵ^α, convergence criterion for the ratio of the largest solution correction to the largest corresponding incremental solution value when there is zero flux in the model. Default $C_\epsilon^\alpha = 10^{-3}$.
17. R_l^α, convergence criterion for the ratio of the largest (scaled) residual to the corresponding average flux norm for convergence to be accepted in one iteration (that is, for a linear case). Default $R_l^\alpha = 10^{-8}$.

Second line:

These items rarely need to be reset from their default values.

1. C_f, field conversion ratio used in scaling the relationship between two active fields when one is of negligible magnitude. Default $C_f = 1.0$.
2. ϵ_l^α, criterion for zero flux compared to the time averaged value of the largest flux $\tilde{q}_{\max}^\alpha$ in the model during the current step. Default $\epsilon_l^\alpha = 10^{-5}$.

3. ϵ_d^α, criterion for zero displacement increment (and/or zero penetration if CONVERT SDI=YES) compared to the characteristic element length in the model. This item is used only when FIELD=DISPLACEMENT. Default $\epsilon_d^\alpha = 10^{-8}$.

Data line for PARAMETERS=CONSTRAINTS:

First (and only) line:

These items rarely need to be reset from their default values. The relevance of certain parameters depends on the value of the CONVERT SDI parameter on the **STEP* option.

1. T^{vol}, volumetric strain compatibility tolerance for hybrid solid elements. Default $T^{vol} = 10^{-5}$.
2. T^{axial}, axial strain compatibility tolerance for hybrid beam elements. Default $T^{axial} = 10^{-5}$.
3. T^{tshear}, transverse shear strain compatibility tolerance for hybrid beam elements. Default $T^{tshear} = 10^{-5}$.
4. T^{cont}, contact and slip compatibility tolerance. For CONVERT SDI=YES, the ratio of the maximum error in the contact or slip constraints to the maximum displacement increment must be less than this tolerance.

   For CONVERT SDI=NO, this is used only with softened contact specified with the **SURFACE BEHAVIOR*, PRESSURE-OVERCLOSURE option. The ratio of the error in the soft contact constraint clearance to the user-specified clearance at which the contact pressure is zero must lie below this tolerance for $p > p^0$, where p^0 is the pressure value at zero clearance. Default $T^{cont} = 5 \times 10^{-3}$.
5. T^{soft}, soft contact compatibility tolerance for low pressure. This tolerance, which is used only if CONVERT SDI=NO, is similar to T^{cont} for softened contact, except that it represents the tolerance when $p = 0.0$. The actual tolerance is interpolated linearly between T^{cont} and T^{soft} for $0 \le p \le p^0$. Default $T^{soft} = 0.1$.
6. T^{disp}, displacement compatibility tolerance for distributing coupling elements. The ratio of the error in the distributing coupling displacement compatibility to a measure of the characteristic length of the coupling arrangement must lie below this tolerance. This characteristic length is twice the average of the coupling node arrangement principal radii of gyration. Default $T^{disp} = 10^{-5}$.
7. T^{rot}, rotation compatibility tolerance for distributing coupling elements. Default $T^{rot} = 10^{-5}$.
8. T^{cfe}, contact force error tolerance for CONVERT SDI=YES. The ratio of the maximum error in the contact force to the time average force must be less than this tolerance. Default $T^{cfe} = 1.0$. This parameter is not used if CONVERT SDI=NO.

Data line for PARAMETERS=LINE SEARCH:

First (and only) line:

1. N^{ls}, maximum number of line search iterations. Default $N^{ls} = 0$ for steps that use the Newton method and $N^{ls} = 5$ for steps that use the quasi-Newton method. A suggested value for activation of the line search algorithm is $N^{ls} = 5$. Specify $N^{ls} = 0$ to forcibly deactivate the method.
2. s^{ls}_{max}, maximum correction scale factor. Default $s^{ls}_{max} = 1.0$.
3. s^{ls}_{min}, minimum correction scale factor. Default $s^{ls}_{min} = 0.0001$.

4. f_s^{ls}, residual reduction factor at which line searching terminates. Default $f_s^{ls} = 0.25$.
5. η^{ls}, ratio of new to old correction scale factors below which line searching terminates. Default $\eta^{ls} = 0.10$.

Data lines for PARAMETERS=TIME INCREMENTATION:

First line:

The relevance of certain parameters depends on the value of the CONVERT SDI parameter on the **STEP* option.

1. I_0, number of equilibrium iterations (without severe discontinuities) after which the check is made whether the residuals are increasing in two consecutive iterations. Minimum value is $I_0 = 3$. Default $I_0 = 4$. If ANALYSIS=DISCONTINUOUS, $I_0 = 8$.
2. I_R, number of consecutive equilibrium iterations (without severe discontinuities) at which logarithmic rate of convergence check begins. Default $I_R = 8$. If ANALYSIS=DISCONTINUOUS, $I_R = 10$. The logarithmic rate of convergence is not checked if fixed time incrementation is used.

The remaining items rarely need to be reset from their default values.

7. I_P, number of consecutive equilibrium iterations (without severe discontinuities) after which the residual tolerance R_p is used instead of R_n. Default $I_P = 9$.
9. I_C, upper limit on the number of consecutive equilibrium iterations (without severe discontinuities), based on prediction of the logarithmic rate of convergence. Default $I_C = 16$.
11. I_L, number of consecutive equilibrium iterations (without severe discontinuities) above which the size of the next increment will be reduced. Default $I_L = 10$.
13. I_G, maximum number of consecutive equilibrium iterations (without severe discontinuities) allowed in consecutive increments for the time increment to be increased. Default $I_G = 4$.
15. I_S, maximum number of severe discontinuity iterations allowed in an increment if CONVERT SDI=NO. Default $I_S = 12$. This parameter is not used if CONVERT SDI=YES.
17. I_A, maximum number of attempts allowed for an increment. Default $I_A = 5$.
19. I_J, maximum number of severe discontinuity iterations allowed in two consecutive increments for the time increment to be increased if CONVERT SDI=NO. Default $I_J = 6$. This parameter is not used if CONVERT SDI=YES.
21. I_T, minimum number of consecutive increments in which the time integration accuracy measure must be satisfied without any cutbacks to allow a time increment increase. Default $I_T = 3$. Maximum allowed $I_T = 10$.
23. I_S^c, maximum number of equilibrium and severe discontinuity iterations allowed in an increment if CONVERT SDI=YES. Default $I_S^c = 50$. This parameter serves only as a protection against failure of the default convergence criteria and should rarely need to be changed. This parameter is not used if CONVERT SDI=NO.
25. I_J^c, maximum number of equilibrium and severe discontinuity iterations allowed in two consecutive increments for the time increment to be increased if CONVERT SDI=YES. Default $I_J^c = 50$. This parameter is not used if CONVERT SDI=NO.

27. I_A^c, maximum number of allowed contact augmentations if the augmented Lagrange contact constraint enforcement method is specified. Default $I_A^c = 50$.

Second line:
These items rarely need to be reset from their default values.

1. D_f, cutback factor used when the solution appears to be diverging. Default $D_f = 0.25$.
2. D_C, cutback factor used when the logarithmic rate of convergence predicts that too many equilibrium iterations will be needed. Default $D_C = 0.5$.
3. D_B, cutback factor for the next increment when too many equilibrium iterations (I_L) are used in the current increment. Default $D_B = 0.75$.
4. D_A, cutback factor used when the time integration accuracy tolerance is exceeded. Default $D_A = 0.85$.
5. D_S, cutback factor used when too many iterations (I_S) arise because of severe discontinuities. Default $D_S = 0.25$.
6. D_H, cutback factor used when element calculations have problems such as excessive distortion in large-displacement problems. Default $D_H = 0.25$.
7. D_D, increase factor when two consecutive increments converge in a small number of equilibrium iterations (I_G). Default $D_D = 1.5$.
8. W_G, ratio of average time integration accuracy measure over I_T increments to the corresponding tolerance for the next allowable time increment to be increased. Default $W_G = 0.75$.

Third line:
These items rarely need to be reset from their default values.

1. D_G, increase factor for the next time increment, as a ratio of the average integration accuracy measure over I_T increments to the corresponding tolerance, when the time integration accuracy measure is less than W_G of the tolerance during I_T consecutive increments. Default $D_G = 0.8$.
2. D_M, maximum time increment increase factor for all cases except dynamic stress analysis and diffusion-dominated processes. Default $D_M = 1.5$.
3. D_M, maximum time increment increase factor for dynamic stress analysis. Default $D_M = 1.25$.
4. D_M, maximum time increment increase factor for diffusion-dominated processes (creep, transient heat transfer, soils consolidation, transient mass diffusion). Default $D_M = 2.0$.
5. D_L, minimum ratio of proposed next time increment to D_M times the current time increment for the proposed time increment to be used in a linear transient problem. This parameter is intended to avoid excessive decomposition of the system matrix and should be less than 1.0. Default $D_L = 0.95$.
6. D_E, minimum ratio of proposed next time increment to the last successful time increment for extrapolation of the solution vector to take place. Default $D_E = 0.1$.
7. D_R, maximum allowable ratio of time increment to stability limit for conditionally stable time integration procedures. Default is 1.0.
8. D_F, fraction of stability limit used as current time increment when the time increment exceeds the above factor times the stability limit. This value cannot exceed 1.0. Default 0.95.

Fourth line:
These items rarely need to be reset from their default values.

1. D_T, increase factor for the time increment directly before a time point or end time of a step is reached. This parameter is used to avoid the small time increment that is sometimes necessary to hit a time point or to complete a step and must be greater than or equal to 1.0. If output or restart data are requested at exact times in a step, the default $D_T = 1.25$; otherwise, the default $D_T = 1.0$.

Data line for TYPE=DIRECT CYCLIC:

First (and only) line:

1. I_{PI}, iteration number at which the periodicity condition is first imposed. Default $I_{PI} = 1$.
2. CR_n^α, stabilized state detection criterion for the ratio of the largest (scaled) residual coefficient on any terms in the Fourier series to the corresponding average flux norm. Default $CR_n^\alpha = 5 \times 10^{-3}$.
3. CU_n^α, stabilized state detection criterion for the ratio of the largest correction to the displacement coefficient on any terms in the Fourier series to the largest displacement coefficient. Default $CU_n^\alpha = 5 \times 10^{-3}$.
4. CR_0^α, plastic ratcheting detection criterion for the ratio of the largest (scaled) residual coefficient on the constant term in the Fourier series to the corresponding average flux norm. Default $CR_0^\alpha = 5 \times 10^{-3}$.
5. CU_0^α, plastic ratcheting detection criterion for the ratio of the largest correction to the displacement coefficient on the constant term in the Fourier series to the largest displacement coefficient. Default $CU_0^\alpha = 5 \times 10^{-3}$.

Data line for TYPE=NO CUTBACK SCALING:

First (and only) line:

1. α parameter. Default $\alpha = 0.5$.
2. β parameter. Default $\beta = 2.0$.
3. n parameter. Default $n = 0$.

Data line for TYPE=VCCT LINEAR SCALING:

First (and only) line:

1. β parameter. Default $\beta = 0.9$.

*CONWEP CHARGE PROPERTY: Define a CONWEP charge for incident waves.

This option defines parameters that create the time history of pressure loading used to simulate an explosion in air. This option must be used in conjunction with the **INCIDENT WAVE INTERACTION PROPERTY* option. The pressure loading is calculated using the CONWEP model empirical data in which mass, length, time, and pressure are given in specific units. Multiplication factors are defined for conversion between the CONWEP data units and the analysis units.

Products: Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Acoustic and Shock Loads*
- **INCIDENT WAVE INTERACTION PROPERTY*

There are no parameters associated with this option.

Data lines to define the CONWEP charge properties:

First line:

1. Equivalent mass of TNT in any preferred mass unit.
2. Multiplication factor to convert from the preferred mass unit to kilograms. The default is 1.0.

Second line (enter a blank line if the analysis uses SI units):

1. Multiplication factor to convert from the analysis length unit to meter.
2. Multiplication factor to convert from the analysis time unit to second.
3. Multiplication factor to convert from the analysis pressure unit to pascal (N/m^2).

*CORRELATION: Define cross-correlation properties for random response loading.

This option is used to define the cross-correlation as part of the definition of random loading for use in the *RANDOM RESPONSE analysis procedure. The *PSD-DEFINITION option is also needed to give the frequency function to be used with the correlation definition.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Random Response Analysis*
- *PSD-DEFINITION*
- *UCORR*

Required parameter for TYPE=CORRELATED and TYPE=UNCORRELATED:

PSD

Set this parameter equal to the name of the frequency function defined on the *PSD-DEFINITION option to be associated with this correlation option.

Optional parameters:

COMPLEX

Set COMPLEX=YES to include both real and imaginary terms in the cross-correlation definition. The alternative is to include real terms only using COMPLEX=NO (default).

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

TYPE

Set TYPE=CORRELATED (default) if all terms in the correlation matrix should be included.

Set TYPE=UNCORRELATED if only diagonal terms should be used.

Set TYPE=MOVING NOISE for moving noise loading. In this case only one *CORRELATION option can be used in the step. The COMPLEX parameter cannot be used with TYPE=MOVING NOISE.

USER

Include this parameter to indicate that user subroutine *UCORR* will be called to obtain the scaling factors for the correlation matrix. If this parameter is included, the TYPE parameter can be set only to CORRELATED or UNCORRELATED.

Data lines for TYPE=CORRELATED or TYPE=UNCORRELATED:

First line:

1. Load case number defined on the loading data lines.

2. Real part of scaling factor.
3. Imaginary part of scaling factor. (Only needed if COMPLEX=YES.)

Repeat this data line as often as necessary to define the load cases and their associated scaling factors.

Data lines if the USER parameter is included:

First line:

1. Load case number defined on the loading data lines.

Repeat this data line as often as necessary to define the load cases to be correlated.

Data lines for TYPE=MOVING NOISE:

First line:

1. Load case number defined on the loading data lines.
2. x-component of noise velocity.
3. y-component of noise velocity.
4. z-component of noise velocity.
5. Name of the power spectral density function, defined on the **PSD-DEFINITION* option, for this noise source.

Repeat this data line as often as necessary to define the random loading.

*CO-SIMULATION: Identify that the current step is a co-simulation step in Abaqus.

This option is used to identify a co-simulation step in Abaqus and the associated co-simulation controls.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *About Co-Simulation*
- *Preparing an Abaqus Analysis for Co-Simulation*
- *Structural-to-Structural Co-Simulation*
- **CO-SIMULATION REGION*

Required parameters:

PROGRAM

Set PROGRAM=MULTIPHYSICS for exchange of data between Abaqus and the SIMULIA Co-Simulation Engine, which in turn can exchange data with third-party analysis programs that support the SIMULIA Co-Simulation Engine.

Optional parameters:

NAME

Set this parameter equal to a label that will be used to refer to the co-simulation event. The co-simulation name adheres to the naming convention for labels (see *Input Syntax Rules*), except that it cannot begin with a number. The default is NAME=COSIM_1.

There are no data lines associated with this option.

*CO-SIMULATION REGION: Identify the interface regions in the Abaqus model and specify the fields to be exchanged during co-simulation.

This option is used to identify the regions across which data will be exchanged and to specify the fields to be passed across those regions. It must be used in conjunction with the *CO-SIMULATION option to identify the analysis program for co-simulation with Abaqus.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *About Co-Simulation*
- *Preparing an Abaqus Analysis for Co-Simulation*
- *CO-SIMULATION

Defining a co-simulation region for *CO-SIMULATION, PROGRAM=MULTIPHYSICS

Required, mutually exclusive parameters:

EXPORT

Include this parameter to specify fields and the accompanying region for export to the coupled analysis.

IMPORT

Include this parameter to specify fields and the accompanying region for import from the coupled analysis.

Optional parameters:

TYPE

Set TYPE=NODE to define a co-simulation region consisting of nodes only.

Set TYPE=SURFACE (default) to define a co-simulation surface region.

Set TYPE=VOLUME to define a co-simulation volume region.

Data lines for TYPE=NODE:

First line:

1. The name of the node set.
2. Field identifier for the field to be passed across this surface. The field identifiers are defined in *Preparing an Abaqus Analysis for Co-Simulation*.
3. Etc., up to seven field identifiers.

Repeat this data line as often as necessary to define additional node set names or more than seven field identifiers for a given node set. Repeat the option to define import and/or export fields.

Data lines for TYPE=SURFACE:

First line:

1. The name of the element-based or node-based surface.
2. Field identifier for the field to be passed across this surface. The field identifiers are defined in *Preparing an Abaqus Analysis for Co-Simulation*.
3. Etc., up to seven field identifiers.

Repeat this data line as often as necessary to define additional surfaces or define more than seven field identifiers for a given surface. Repeat the option to define import and/or export fields.

Data lines for TYPE=VOLUME:

First line:

1. The name of the element set.
2. Field identifier for the field to be passed across this surface. The field identifiers are defined in *Preparing an Abaqus Analysis for Co-Simulation*.
3. Etc., up to seven field identifiers.

Repeat this data line as often as necessary to define additional element set names or more than seven field identifiers for a given element set. Repeat the option to define import and/or export fields.

Defining a co-simulation region for *CO-SIMULATION, PROGRAM=ABAQUS

Optional parameters:

TYPE

Set TYPE=SURFACE (default) to define a surface-based co-simulation region.

Set TYPE=NODE to define a co-simulation region using a node set.

Data line for TYPE=SURFACE:

First (and only) line:

1. The name of the element-based or node-based surface.

Data line for TYPE=NODE:

First (and only) line:

1. The name of the node set.

*COUPLED CONSTITUTIVE RESPONSE: This option is used to introduce a user-defined constitutive behavior involving multiple physics fields.

This option is used in a material data block to introduce a user-defined constitutive response that can couple multiple physics fields (user subroutine *UCOUPLEDCONSTRESP*).

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Coupled Thermal-Electrochemical Analysis*
- *UCOUPLEDCONSTRESP*

Required parameters:

USER

This parameter is required to activate the usage of user subroutine *UCOUPLEDCONSTRESP*.

Optional parameters:

PROPERTIES

Set this parameter equal to the number of real (floating point) property values needed as data in user subroutine *UCOUPLEDCONSTRESP*. The default is PROPERTIES=0.

Data line to define properties (if the value of PROPERTIES is not set to 0):

First line:

1. Give the properties, eight per line.

Repeat this data line as often as necessary.

*COUPLED TEMPERATURE-DISPLACEMENT: Fully coupled, simultaneous heat transfer and stress analysis.

This option is used to analyze problems where the simultaneous solution of the temperature and stress/displacement fields is necessary.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Fully Coupled Thermal-Stress Analysis*
- *Fully Coupled Thermal-Electrical-Structural Analysis*
- *Fully Coupled Thermal-Electrochemical-Structural Analysis*
- *Rate-Dependent Plasticity: Creep and Swelling*
- *Fully Coupled Thermal-Electrochemical-Structural–Pore Pressure Analysis*

Optional parameters:

ADVECTION

This parameter applies only to a coupled thermal-electrochemical-structural–pore pressure analysis. It is valid only when pore pressure is included with electrochemistry and adds an advection contribution (resulting from the velocity field of the electrolyte) to the ion flux equation.

Set ADVECTION=YES (default) to consider the fluid velocity effects.

Set ADVECTION=NO to ignore the fluid velocity effects.

ALLSDTOL

Include this parameter to indicate that an adaptive automatic damping algorithm will be activated in this step. Set this parameter equal to the maximum allowable ratio of the stabilization energy to the total strain energy. The initial damping factor is specified using the STABILIZE parameter or the FACTOR parameter. This damping factor will then be adjusted through the step based on the convergence history and the value of ALLSDTOL. If this parameter is set equal to zero, the adaptive automatic damping algorithm is not activated; a constant damping factor will be used throughout the step. If this parameter is included without a specified value, the default value is 0.05. If this parameter is omitted but the STABILIZE parameter is included with the default value of dissipated energy fraction, the adaptive automatic damping algorithm will be activated automatically with ALLSDTOL=0.05.

This parameter must be used in conjunction with the STABILIZE parameter (see *Solving Nonlinear Problems*).

AMBIENT ION CONCENTRATION

This parameter applies only to a coupled thermal-electrochemical-structural–pore pressure analysis. It is valid only when pore pressure is included with electrochemistry and the fluid velocity effects are modeled using ADVECTION=YES.

This parameter is used to model the presence of an external reservoir of electrolyte at a prescribed ion concentration.

Set AMBIENT ION CONCENTRATION=C_e^{amb} to prescribe the ion concentration of the external electrolyte reservoir.

CONTINUE

Set CONTINUE=NO (default) to specify that this step will not carry over the damping factors from the results of the preceding general step. In this case the initial damping factors will be recalculated based on the declared damping intensity and on the solution of the first increment of the step or can be specified directly.

Set CONTINUE=YES to specify that this step will carry over the damping factors from the end of the immediately preceding general step.

This parameter must be used in conjunction with the ALLSDTOL and the STABILIZE parameters.

CREEP

Set CREEP=EXPLICIT to use explicit integration for creep and swelling effects throughout the step, which might sometimes be computationally less expensive. When CREEP=EXPLICIT, the time increment will be limited by the accuracy tolerances (CETOL and DELTMX) and also by the stability limit of the forward difference operator. See *Rate-Dependent Plasticity: Creep and Swelling* for details on the integration scheme.

Set CREEP=NONE to specify that there is no creep or viscoelastic response occurring during this step even if creep or viscoelastic material properties have been defined.

ELECTRICAL

Include this parameter to model electrical conduction in regions of the model that use thermal-electrical-structural elements.

In this case, Abaqus/Standard solves the heat transfer equations in conjunction with the mechanical equilibrium and the electrical conduction equations.

Omit this parameter to specify that electrical conduction is not modeled in these regions.

ELECTROCHEMICAL

Include this parameter to model electrochemistry in regions of the model that use thermal-electrochemical-structural elements.

In this case, Abaqus/Standard solves the heat transfer equations in conjunction with the mechanical equilibrium and the electrical and electrochemical equations.

Omit this parameter to specify that electrochemistry is not modeled in these regions.

FACTOR

Set this parameter equal to the damping factor to be used in the automatic damping algorithm (see *Solving Nonlinear Problems*) if the problem is expected to be unstable due to local instabilities and the damping factor calculated by Abaqus/Standard is not suitable. This parameter must be used in conjunction with the STABILIZE parameter and overrides the automatic calculation of the damping factor based on a value of the dissipated energy fraction.

PORE PRESSURE

Include this parameter to model pore pressure–mediated fluid (electrolyte) flow in a porous medium along with the thermal, electrochemical, and structural physics in regions of the model that use thermal-electrochemical-structural–pore pressure elements.

In this case, Abaqus/Standard solves the pore fluid continuity equation fully coupled with the thermal, electrochemical, and structural governing equations.

Omit this parameter if pore fluid continuity is not modeled in these regions.

RATE DEPENDENCE

Set RATE DEPENDENCE=ON (default) to consider the strain rate–dependence when evaluating the yield stress of a material and to consider the slip rate–dependence when evaluating the friction coefficient.

Set RATE DEPENDENCE=OFF to ignore the strain rate–dependence when evaluating the yield stress of a material and to ignore the slip rate–dependence when evaluating the friction coefficient.

STABILIZE

Include this parameter to use automatic stabilization if the problem is expected to be unstable due to local instabilities. Set this parameter equal to the dissipated energy fraction of the automatic damping algorithm (see *Solving Nonlinear Problems*). If this parameter is omitted, the stabilization algorithm is not activated. If this parameter is included without a specified value, the default value of the dissipated energy fraction is 2×10^{-4} and the adaptive automatic damping algorithm will be activated by default with ALLSDTOL =0.05 in this step; set ALLSDTOL=0 to deactivate the adaptive automatic damping algorithm. If the FACTOR parameter is used, any value of the dissipated energy fraction is overridden by the damping factor.

STEADY STATE

Include this parameter to choose steady-state analysis. If this parameter is omitted, the step is assumed to involve transient response. If this parameter is included, automatic time incrementation will be used.

Optional parameters to control time incrementation in transient analysis:

CETOL

This parameter is meaningful only when the material response is time dependent (creep and swelling).

Set this parameter equal to the maximum difference in the creep strain increment calculated from the creep strain rates at the beginning and at the end of the increment, thus controlling the accuracy of the creep integration. The tolerance is sometimes calculated by choosing an acceptable stress error tolerance and dividing by a typical elastic modulus.

If both this parameter and the DELTMX parameter are omitted in a transient analysis, fixed time increments are used, with a constant time increment equal to the initial time increment.

DELTMX

Set this parameter equal to the maximum temperature change allowed within an increment. Abaqus/Standard restricts the time step to ensure that this value is not exceeded at any node during any increment of the step.

If the maximum allowable temperature change is not specified, there is no restriction on the maximum temperature change in an increment.

POREPTOL

This parameter is valid only in a coupled thermal-electrochemical-structural–pore pressure analysis.

Set this parameter equal to the maximum pore pressure change allowed in an increment. Abaqus/Standard restricts the time step to ensure that this value is not exceeded at any node during any increment of the step.

If the maximum allowable pore pressure change is not specified, there is no restriction on the maximum pore pressure change in an increment.

Data line to control incrementation in a fully coupled thermal-stress analysis:

First (and only) line:

1. Suggested initial time increment. If automatic incrementation is used, this should be a reasonable suggestion for the initial increment size and will be adjusted as necessary. If direct incrementation is used, this will be the fixed time increment size.
2. Total time period for the step.
3. Minimum time increment allowed. If Abaqus/Standard finds it needs a smaller time increment than this value, the analysis is terminated. If this entry is zero, a default value of the smaller of the suggested initial time increment or 10^{-5} times the total time period is assumed. This value is used only for automatic time incrementation.
4. Maximum time increment allowed. If this value is not specified, the default upper limit is the total time period for the step. This value is used only for automatic time incrementation.

*COUPLED THERMAL-ELECTRICAL: Fully coupled, simultaneous heat transfer and electrical analysis.

This option is used to analyze problems where the electrical potential and temperature fields must be solved simultaneously.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

• *Coupled Thermal-Electrical Analysis*

Optional parameters:

DELTMX

Include this parameter to activate automatic time incrementation in transient analysis. If the DELTMX parameter is omitted in a transient analysis, fixed time increments will be used. Set this parameter equal to the maximum temperature change to be allowed in an increment in a transient analysis. Abaqus/Standard will restrict the time step to ensure that this value will not be exceeded at any node (except nodes with boundary conditions) during any increment of the analysis.

END

Set END=PERIOD (default) to analyze a specific time period in a transient analysis. Set END=SS to end the analysis when steady state is reached.

MXDEM

For problems including cavity radiation heat transfer, set this parameter equal to the maximum allowable emissivity change with temperature and field variables during an increment. If the value of MXDEM is exceeded, Abaqus/Standard will cut back the increment until the maximum change in emissivity is less than the value input. If this parameter is omitted, a default value of 0.1 is used.

This parameter controls the accuracy of changes in emissivity due to temperature since Abaqus/Standard evaluates the emissivity based on the temperature at the start of each increment and uses that emissivity value throughout the increment.

STEADY STATE

Include this parameter to choose steady-state thermal analysis. Transient thermal analysis is assumed if this parameter is omitted. If this parameter is included, automatic time incrementation will be used.

Data line to define incrementation and steady state:

First (and only) line:

1. Initial time increment. If automatic incrementation is used, this value should be a reasonable suggestion for the initial step and will be adjusted as necessary. If direct incrementation is used, this value will be the fixed time increment size.
2. Total time period. If END=SS is chosen, the step ends when steady state is reached or after this time period, whichever occurs first.

3. Minimum time increment allowed. If Abaqus/Standard finds it needs a smaller time increment than this value, the analysis is terminated. If a value is given, Abaqus/Standard will use the minimum of the given value and 0.8 times the initial time increment. If no value is given, Abaqus/Standard sets the minimum increment equal to the minimum of 0.8 times the initial time increment (first data item on this data line) and 10^{-5} times the total time period (second data item on this data line). This value is used only for automatic time incrementation.
4. Maximum time increment allowed. If this value is not specified, the upper limit is the total step time. This value is used only for automatic time incrementation.
5. Temperature change rate (temperature per time) used to define steady-state thermal conditions; only needed if END=SS is chosen. When all nodal temperatures are changing at less than this rate, the solution terminates.

*COUPLED THERMAL-ELECTROCHEMICAL: Define a coupled thermal-electrochemical analysis.

This option is used to analyze problems where temperature, electric potentials in the solid and fluid, and ion concentration fields must be solved simultaneously.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

• *Coupled Thermal-Electrochemical Analysis*

Optional parameters:

DELTMX

Include this parameter to activate automatic time incrementation in a transient analysis.

Set this parameter equal to the maximum temperature change allowed in an increment in a transient analysis. Abaqus/Standard restricts the time step to ensure that this value is not exceeded at any node (except nodes with boundary conditions) during any increment of the analysis.

If the maximum allowable temperature change is not specified, there is no restriction on the maximum temperature change in an increment.

STEADY STATE

Include this parameter to choose a steady-state thermal analysis. Automatic time incrementation is used in this case.

If this parameter is omitted, transient thermal analysis is assumed.

Data line to define incrementation:

First (and only) line:

1. Initial time increment. If automatic incrementation is used, this value should be a reasonable suggestion for the initial step and is adjusted as required. If direct incrementation is used, this value is the fixed time increment size.
2. Total time period.
3. Minimum time increment allowed. If Abaqus/Standard finds it needs a smaller time increment than this value, the analysis ends. If a value is given, Abaqus/Standard uses the minimum of the given value and 0.8 times the initial time increment. If no value is given, Abaqus/Standard sets the minimum increment equal to the minimum of 0.8 times the initial time increment (first data item on this data line) and 10^{-5} times the total time period (second data item on this data line). This value is used only for automatic time incrementation.
4. Maximum time increment allowed. If this value is not specified, the upper limit is the total step time. This value is used only for automatic time incrementation.

*COUPLING: Define a surface-based coupling constraint.

This option is used to impose a kinematic, distributing, or uniform coupling constraint between a reference node and a group of nodes located on a surface. It must be used in conjunction with the *DISTRIBUTING*, the *KINEMATIC*, or the *UNIFORM* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Interaction module

References:

- *Coupling Constraints*
- *Element-Based Surface Definition*
- *Node-Based Surface Definition*

Required parameters:

CONSTRAINT NAME

Set this parameter equal to a label that will be used to refer to this constraint.

REF NODE

Set this parameter equal to either the node number of the reference node or the name of a node set containing the reference nodes.

When the reference node set contains multiple nodes, you should specify an influence radius to avoid overconstraint issues.

SURFACE

Set this parameter equal to the surface name on which the coupling nodes are located.

Optional parameters:

INFLUENCE RADIUS

Set this parameter equal to the radius of influence centered about the reference node. If this parameter is omitted, the entire surface is used to define the coupling constraint.

ORIENTATION

This parameter applies only to distributing and kinematic coupling constraints.

Set this parameter equal to the name given to the *ORIENTATION* definition (*Orientations*) that specifies the initial orientation of the local system in which the constrained degrees of freedom are defined.

There are no data lines associated with this option.

*CRADIATE: Specify radiation conditions and associated sink temperatures at one or more nodes or vertices.

This option is used to apply radiation boundary conditions between a node and a nonreflecting environment in fully coupled thermal-stress analysis. In Abaqus/Standard it is also used for all procedures that solve for nodal temperature degrees of freedom.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

• *Thermal Loads*

Optional parameters:

AMBIENT AMPLITUDE

Set this parameter equal to the name of the **AMPLITUDE* option that gives the variation of the ambient temperature with time.

If this parameter is omitted in an Abaqus/Standard analysis, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (see *Defining an Analysis*). If this parameter is omitted in an Abaqus/Explicit analysis, the reference magnitude is applied immediately at the beginning of the step.

OP

Set OP=MOD (default) for existing **CRADIATE* definitions to remain, with this option modifying existing radiation conditions or defining additional radiation conditions.

Set OP=NEW if all existing **CRADIATE* definitions applied to the model should be removed.

PHANTOM

This parameter applies only to Abaqus/Standard analyses.

Set PHANTOM=NODE to specify radiation conditions and associated sink temperatures at a phantom node that is originally located coincident with the specified real node in an enriched element.

Set PHANTOM=EDGE to specify radiation conditions and associated sink temperatures at a phantom node located on an element edge between the two specified real corner nodes in an enriched element. This setting applies only to nodes with both pore pressure and temperature degrees of freedom.

RADIATION AMPLITUDE

Set this parameter equal to the name of the **AMPLITUDE* option that gives the variation of the total radiation flux with time.

If this parameter is omitted in an Abaqus/Standard analysis, the total radiation flux, accounting for any AMBIENT AMPLITUDE, is applied immediately, independent of the value assigned to the AMPLITUDE parameter on the **STEP* option. If this parameter is omitted in an Abaqus/Explicit analysis, the total radiation flux, accounting for any AMBIENT AMPLITUDE, is applied immediately at the beginning of the step.

REGION TYPE

This parameter applies only to Abaqus/Explicit analyses.

This parameter is relevant only for concentrated radiation conditions applied on the boundary of an adaptive mesh domain. If concentrated radiation conditions are applied to nodes in the interior of an adaptive mesh domain, these nodes will always follow the material.

Set REGION TYPE=LAGRANGIAN (default) to apply a concentrated radiation condition to a node that follows the material (nonadaptive).

Set REGION TYPE=SLIDING to apply a concentrated radiation condition to a node that can slide over the material. Mesh constraints are typically applied to the node to fix it spatially.

Set REGION TYPE=EULERIAN to apply a concentrated radiation condition to a node that can move independently of the material. This option is used only for boundary regions where the material can flow into or out of the adaptive mesh domain. Mesh constraints must be used normal to an Eulerian boundary region to allow material to flow through the region. If no mesh constraints are applied, an Eulerian boundary region will behave in the same way as a sliding boundary region.

Data lines to define radiation conditions:

First line:

1. Node number or node set name.
2. Appropriate area associated with the node where the concentrated radiation condition is applied. The default is 1.0.
3. Reference ambient temperature value, θ^0. (Units of θ.)
4. Emissivity, ϵ.

Repeat this data line as often as necessary to define radiation conditions.

Data lines to define radiation conditions when PHANTOM=NODE:

First line:

1. Node number of a real node that is originally located coincident with the phantom node.
2. Appropriate area associated with the node where the concentrated radiation condition is applied. The default is 1.0.
3. Reference ambient temperature value, θ^0. (Units of θ.)
4. Emissivity, ϵ.

Repeat this data line as often as necessary to define radiation conditions.

Data lines to define radiation conditions when PHANTOM=EDGE:

First line:

1. Node number of one of the two real corner nodes to define an edge in an enriched element.
2. Node number of the other real corner node to define an edge in an enriched element.
3. Appropriate area associated with the node where the concentrated radiation condition is applied. The default is 1.0.
4. Reference ambient temperature value, θ^0. (Units of θ.)
5. Emissivity, ϵ.

Repeat this data line as often as necessary to define radiation conditions.

*CREEP: Define a creep law.

This option is used when metal creep behavior is to be included in a material definition. Metal creep behavior defined is active only during **DIRECT CYCLIC*; **SOILS*, CONSOLIDATION; **COUPLED TEMPERATURE-DISPLACEMENT*; **STEADY STATE TRANSPORT*; and **VISCO* procedures. This option can also be used to define creep behavior in the thickness direction in a gasket; in this case the option is active only during the **VISCO* procedure.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Rate-Dependent Plasticity: Creep and Swelling*
- *Hill Anisotropic Yield/Creep*
- *CREEP*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the creep constants in addition to temperature. If this parameter is omitted, it is assumed that the creep constants have no dependencies or depend only on temperature. See *Material Data Definition* for more information.

LAW

Set LAW=STRAIN (default) to choose a strain hardening law. This parameter setting is superseded by LAW=POWER LAW.

Set LAW=TIME to choose a time hardening law. This parameter setting is superseded by LAW=TIME POWER LAW.

Set LAW=HYPERB to choose a hyperbolic-sine law.

Set LAW=USER to input the creep law using user subroutine *CREEP*.

Set LAW=ANAND to choose an Anand law.

Set LAW=DARVEAUX to choose a Darveaux law.

Set LAW=DOUBLE POWER to choose a double power law.

Set LAW=POWER LAW to choose a power law. It is recommended that you use this parameter setting rather than LAW=STRAIN.

Set LAW=TIME POWER LAW to choose a time power law. It is recommended that you use this parameter setting rather than LAW=TIME.

TIME

This parameter is relevant only when LAW=TIME or LAW=TIME POWER LAW is used.

Set TIME=CREEP to use creep time.

Set TIME=TOTAL (default) to use total time.

Data lines for LAW=TIME or LAW=STRAIN:

First line:

1. A. (Units of $F^{-n}L^{2n}T^{-1-m}$.)
2. n.
3. m.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the creep constants on temperature and other predefined field variables.

Data lines for LAW=HYPERB:

First line:

1. A. (Units of T^{-1}.)
2. B. (Units of $F^{-1}L^2$.)
3. n.
4. ΔH. (Units of JM^{-1}.) (This value can be left blank if temperature dependence is not needed.)
5. R. (Units of $JM^{-1}\theta^{-1}$.)
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the creep constants on predefined field variables.

Data lines for LAW=ANAND:

First line:

1. S_1. (Units of FL^{-2}.)
2. Q/R. (Units of θ.)
3. A. (Units of T^{-1}.)
4. ξ.
5. m.
6. A_0. (Units of FL^{-2}.)
7. $\hat{s}$. (Units of FL^{-2}.)
8. n.

Second line:

1. a.

2. S_2. (Units of $FL^{-2}\theta^{-1}$.)
3. S_3. (Units of $FL^{-2}\theta^{-2}$.)
4. A_1. (Units of $FL^{-2}\theta^{-1}$.)
5. A_2. (Units of $FL^{-2}\theta^{-2}$.)
6. A_3. (Units of $FL^{-2}T$.)
7. A_4. (Units of $FL^{-2}T^2$.)

Data lines for LAW=DARVEAUX:

First line:

1. C_{ss}. (Units of T^{-1}.)
2. Q/R. (Units of θ.)
3. α. (Units of $F^{-1}L^2$.)
4. n.
5. ϵ_T.
6. B.

Data lines for LAW=DOUBLE POWER:

First line:

1. A_1. (Units of T^{-1}.)
2. B_1. (Units of θ.)
3. C_1.
4. A_2. (Units of T^{-1}.)
5. B_2. (Units of θ.)
6. C_2.
7. σ_0. (Units of FL^{-2}.)

Data lines for LAW=POWER LAW:

First line:

1. q_0. (Units of FL^{-2}.)
2. n.
3. m.
4. Blank space.
5. $\dot{\varepsilon}_0$. The default is 1.0. (Units of T^{-1}.)
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the creep constants on temperature and other predefined field variables.

Data lines for LAW=TIME POWER LAW:

First line:

1. q_0. (Units of FL^{-2}.)
2. n.
3. m.
4. $\dot{\varepsilon}_0$. The default is 1.0. (Units of T^{-1}.)
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the creep constants on temperature and other predefined field variables.

*CREEP STRAIN RATE CONTROL: Control loadings based on the maximum equivalent creep strain rate.

This option is used to control loading based on a maximum equivalent creep strain rate calculated in a specified element set.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

• *Rate-Dependent Plasticity: Creep and Swelling*

Required parameters:

AMPLITUDE

Set this parameter equal to the **AMPLITUDE* name (of type DEFINITION=SOLUTION DEPENDENT) that is referenced by the loads being controlled (*Amplitude Curves*).

ELSET

Set this parameter equal to the name of the element set in which the search for the maximum equivalent creep strain rate is made. The **CREEP* option must be part of the **MATERIAL* definition (*Material Data Definition*) for some elements in the set.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the target creep strain rate, in addition to temperature and creep strain. If this parameter is omitted, it is assumed that the target creep strain rate depends only on the equivalent creep strain and, possibly, on temperature. The creep strain dependency curve at each temperature must always start at zero equivalent creep strain. See *Material Data Definition* for more information.

OP

Set OP=MOD (default) for existing target **CREEP STRAIN RATE CONTROL* definitions to remain, with this option defining target creep strain rates to be added or modified.

Set OP=NEW if all target creep strain rates defined in the previous step should be removed.

Data lines to define load control parameters:

First line:

1. Target equivalent creep strain rate.
2. Equivalent creep strain.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.

2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of target strain rate on creep strain, temperature, and other predefined field variables.

*CRUSHABLE FOAM: Specify the crushable foam plasticity model.

This option is used to specify the plastic part of the material behavior for elastic-plastic materials that use the crushable foam plasticity model. It must be used in conjunction with the **CRUSHABLE FOAM HARDENING* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Crushable Foam Plasticity Models*
- **CRUSHABLE FOAM HARDENING*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the crushable foam parameters, in addition to temperature. If this parameter is omitted, it is assumed that the crushable foam parameters are constant or depend only on temperature. See *Material Data Definition* for more information.

HARDENING

Set HARDENING=VOLUMETRIC (default) to specify the volumetric hardening model.

Set HARDENING=ISOTROPIC to specify the isotropic hardening model.

Data lines to define the crushable foam plasticity model with volumetric hardening (HARDENING=VOLUMETRIC):

First line:

1. $k = \sigma_c^0/p_c^0$, yield stress ratio for compression loading; $0 < k < 3$. Enter the ratio of initial yield stress in uniaxial compression to initial yield stress in hydrostatic compression.
2. $k_t = p_t/p_c^0$, yield stress ratio for hydrostatic loading; $k_t \geq 0$. Enter the ratio of yield stress in hydrostatic tension to initial yield stress in hydrostatic compression, given as a positive value. The default value is 1.0.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the crushable foam parameters on temperature and other predefined field variables.

Data lines to define the crushable foam plasticity model with isotropic hardening (HARDENING=ISOTROPIC):

First line:

1. $k = \sigma_c^0/p_c^0$, yield stress ratio for compression loading; $0 \le k < 3$. Enter the ratio of initial yield stress in uniaxial compression to initial yield stress in hydrostatic compression.
2. ν_p, plastic Poisson's ratio; $-1 < \nu_p \le 0.5$.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the crushable foam parameters on temperature and other predefined field variables.

*CRUSHABLE FOAM HARDENING: Specify hardening for the crushable foam plasticity model.

This option is used to define the hardening data for elastic-plastic materials that use the crushable foam plasticity model. It must be used in conjunction with the **CRUSHABLE FOAM* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Crushable Foam Plasticity Models*
- **CRUSHABLE FOAM*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the yield surface size, in addition to temperature. If this parameter is omitted, it is assumed that the size of the yield surface depends only on the volumetric plastic strain and, possibly, on the temperature. See *Material Data Definition* for more information.

Data lines to define crushable foam hardening:

First line:

1. σ_c, yield stress in uniaxial compression, given as a positive value.
2. Absolute value of the corresponding plastic strain. (The first tabular value entered must always be zero.)
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the yield stress in uniaxial compression on the corresponding axial plastic strain and, if needed, on temperature and other predefined field variables.

*CRUSH STRESS: Define the crush stress of a material.



Warning: Crush stress is not a true material property. Where the analysis laminate deviates from the original tested laminate, the ply-by-ply data reduction process is not guaranteed to reliably predict the crush performance on the new laminate. Where crush performance is important to the function of the structure and deviation from the tested laminate is significant, the resulting laminate should be tested to confirm that the crush performance follows the ply-by-ply data reduction values.



Note: All crush stresses should be measured on the complete laminated composite. Data entry is reduced to a ply-by-ply bases, so it is possible to assess the effects of modifying layers within the analysis definition as the structure is developed.

This option is used to define the crush stress of a material. It must be used as a suboption of the **MATERIAL* option. The crush stress can be constant (isotropic) or orientation dependent (anisotropic), defined as a function of the incidence angle. A regularization scheme is used for the material data when the crush stress is dependent.

If specified, temperature and field dependency is evaluated only at the beginning of the analysis. Subsequent changes in temperature and/or predefined field variables will have no effect in the crushing behavior of the material.

Products: Abaqus/Explicit

Type: Model data

Level: Model

References:

- **CRUSH STRESS VELOCITY FACTOR*
- *CZone Analysis*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the crush stress, in addition to temperature.

Data lines to define crush stress:

First line:

1. Crush stress.
2. Material angle in degrees.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of crush stress on the incidence angle and, if needed, on temperature and other predefined variables. The complete profile

must be included, excluding the 180° data point that is assumed to be identical to the first data point at 0°. At minimum, the data should account for the perpendicular crushing impact in each ply direction for the complete composite layup; CZone for Abaqus will interpolate between the data points to account for crushing in other directions.

*CRUSH STRESS VELOCITY FACTOR: Define how the approach velocity at a crushing interface influences a material's resistance to crushing.

The option is used, if necessary, to define how the approach velocity at a crushing interface influences a material's resistance to crushing. It must be used as a suboption of the *CRUSH STRESS option. Abaqus/Explicit uses a regularization scheme for the evaluation of the crush stress scaling factor as a function of relative velocity.

The CZone for Abaqus algorithm requires that the scaling factor be the same for all plies in a composite laminate. Therefore, identical *CRUSH STRESS VELOCITY FACTOR definitions must be used in all materials referenced by the same composite shell section definition.

Products: Abaqus/Explicit

Type: Model data

Level: Model

References:

- *CRUSH STRESS
- *CZone Analysis*

There are no parameters associated with this option.

Data lines to define velocity-dependent crush factors:

First line:

1. Scaling factor to be applied to the crush stress.
2. Relative velocity between the two surfaces.

Repeat this data line as often as necessary to define the crush stress scaling factor as a function of relative velocity.

*CURE GLASS TRANSITION TEMPERATURE: This option is used to specify the glass transition temperature as a function of the degree of cure.

This option is used to specify the glass transition temperature as a function of the degree of cure. It must be used in conjunction with the **CURE KINETICS* option.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Curing Processes in Polymers*

Required parameters:

DEFINITION

Set DEFINITION=DIBENEDETTO to define the glass transition temperature using the DiBenedetto equation.

Set DEFINITION=TABULAR to define the glass transition temperature in tabular form.

Optional parameters:

DEPENDENCIES

This parameter is relevant only for DEFINITION=TABULAR.

Set this parameter equal to the number of field variables included in the specification of the glass transition temperature.

If this parameter is omitted, it is assumed that the glass transition temperature depends only on the degree of cure and, optionally, on temperature. For more information, see *Material Data Definition*.

Data line to define the glass transition temperature using the DiBenedetto equation:

First (and only) line:

1. Glass transition temperature in uncured state, $\theta_{g,0}$. (Units of θ.)
2. Glass transition temperature in fully cured state, $\theta_{g,\infty}$. (Units of θ.)
3. DiBenedetto model parameter, λ ($\lambda > 0$).

Data lines to define the glass transition temperature in tabular form:

First line:

1. Glass transition temperature, θ_g. (Units of θ.)
2. Degree of cure, α.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this data line as often as necessary to define the glass transition temperature as a function of the degree of cure, temperature, and user-defined field variables.

*CURE HEAT GENERATION: Define the specific heat of the reaction for cure modeling capabilities.

This option is used to define the specific heat of the reaction for cure modeling capabilities. It must be used in conjunction with the **CURE KINETICS* option.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Curing Processes in Polymers*

There are no parameters associated with this option.

Data line to define the specific heat of the reaction:

First (and only) line:

1. Specific heat of the reaction. (Units of JM^{-1}.)

*CURE KINETICS: Define the reaction kinetics for cure modeling capabilities.

This option is used to define the reaction kinetics for cure modeling capabilities.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

• *Curing Processes in Polymers*

Required parameters:

DEFINITION

Set DEFINITION=GRINDLING to define the degree of cure rate using the Grindling equation.

Set DEFINITION=KAMAL to define the degree of cure rate using the Kamal equation.

Set DEFINITION=TABULAR to define the degree of cure rate in tabular form.

Set DEFINITION=USER to define the degree of cure rate using user subroutine *UCURE*.

Optional parameters:

DEPENDENCIES

This parameter is relevant only for DEFINITION=TABULAR and for DEFINITION=KAMAL used in combination with the parameter NUMBER TERMS.

Set this parameter equal to the number of field variable dependencies included in the definition of the degree of cure rate, in addition to temperature. If this parameter is omitted, the degree of cure rate does not depend on field variables.

NUMBER TERMS

This parameter is relevant only for DEFINITION=KAMAL.

Set this parameter equal to the number of terms in the Kamal model.

PROPERTIES

This parameter is relevant only for DEFINITION=USER.

Set this parameter equal to the number of property values needed as data in user subroutine *UCURE*. The default is PROPERTIES=0.

Data lines to define the Grindling model coefficients:

First line:

1. Grindling equation coefficient, A_1. (Units of T^{-1}.)
2. Ratio of the activation energy to the universal gas constant, E_1/R. (Units of θ.)
3. Grindling equation exponent, n_1.
4. Grindling equation coefficient, A_2. (Units of T^{-1}.)
5. Ratio of the activation energy to the universal gas constant, E_2/R. (Units of θ.)
6. Grindling equation exponent, n_2.

7. Grindling equation exponent, m.
8. Grindling equation parameter, c_1.

Second line:

1. Grindling equation parameter, c_2. (Units of θ.)
2. Diffusion-controlled reaction rate at the glass transition temperature, $K_D^{\theta=\theta_g}$. (Units of T^{-1}.)
3. Grindling equation parameter, $\Delta\theta_g$. (Units of θ.)

In addition, you must specify the value of absolute zero using the **PHYSICAL CONSTANTS* option and the glass transition temperature using the **CURE GLASS TRANSITION TEMPERATURE* option.

Data lines to define the Kamal model coefficients using DEFINITION=KAMAL without the NUMBER TERMS parameter:

First line:

1. Kamal equation coefficient, Z_1. (Units of T^{-1}.)
2. Ratio of the activation energy to the universal gas constant, E_1/R. (Units of θ.)
3. Kamal equation exponent, m_1.
4. Kamal equation exponent, n_1.
5. Kamal equation parameter, b_1.

In addition, you must specify the value of absolute zero using the **PHYSICAL CONSTANTS* option.

Repeat this data line as often as necessary to define the second, third, etc., terms in the Kamal equation. There is no restriction on the number of terms in the model.

Data lines to define Kamal model coefficients using DEFINITION=KAMAL with the NUMBER TERMS parameter:

First line:

1. Kamal equation coefficient, Z_1. (Units of T^{-1}.)
2. Ratio of the activation energy to the universal gas constant, E_1/R. (Units of θ.)
3. Kamal equation exponent, m_1.
4. Kamal equation exponent, n_1.
5. Kamal equation parameter, b_1.
6. Etc., specify Kamal model parameters Z_k, E_k/R, m_k, n_k, b_k for each term k. (Only needed if the NUMBER TERMS parameter has a value greater than 1.)
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the number of entries is greater than eight):

1. Second field variable.
2. Etc., up to eight field variables per line.

In addition, you must specify the value of absolute zero using the **PHYSICAL CONSTANTS* option.

Repeat this set of data lines as often as necessary to define the dependence of Kamal model coefficients on temperature and other predefined field variables.

Data lines to define the degree of cure rate in tabular form:

First line:

1. Degree of cure rate, $\dot{\alpha}$. (Units of T^{-1}.)
2. Degree of cure, α.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this data line as often as necessary to define the degree of cure rate as a function of the degree of cure, temperature, and user-defined field variables.

Data lines to define the degree of cure rate in user subroutine `UCURE`:

No data lines are needed if the PROPERTIES parameter is omitted or set to 0. Otherwise, first line:

1. Enter the values of the cure kinetics properties, eight per line.

Repeat this data line as often as necessary to define the material properties.

*CURE MAX CONVERSION: Define the maximum value of the degree of conversion for cure modeling capabilities.

This option is used to define the maximum value of the degree of cure (conversion) that can be attained during the curing process. It must be used in conjunction with the **CURE KINETICS* option. If the **CURE MAX CONVERSION* option is not specified, the maximum value of the degree of cure is one (full conversion).

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Curing Processes in Polymers*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the maximum value of degree of cure. If this parameter is omitted, it is assumed that the value is constant or depends only on temperature.

Data lines to define the maximum value of degree of cure:

First line:

1. Maximum value of the degree of cure, α_{max} ($0 < \alpha_{max} \leq 1$).
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this data line as often as necessary to define the maximum value of degree of cure as a function of temperature and user-defined field variables.

*CURE SHRINKAGE: Define the cure shrinkage coefficients for cure modeling capabilities.

This option is used to define the cure shrinkage coefficients for cure modeling capabilities. It must be used in conjunction with the **CURE KINETICS* option.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Curing Processes in Polymers*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables, in addition to temperature, on which the coefficients depend. If this parameter is omitted, it is assumed that the cure shrinkage coefficients are constant or depend only on temperature.

TYPE

Set TYPE=ISO (default) to define isotropic cure shrinkage.

Set TYPE=VOL to define volumetric cure shrinkage.

Set TYPE=ORTHO to define orthotropic cure shrinkage.

Set TYPE=ANISO to define anisotropic cure shrinkage.

Data lines to define isotropic cure shrinkage:

First line:

1. Isotropic cure shrinkage coefficient, γ_{iso}.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Second line:

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this data line as often as necessary to define the isotropic shrinkage coefficient as a function of temperature and user-defined field variables.

Data lines to define volumetric cure shrinkage:

First line:

1. Volumetric cure shrinkage coefficient, γ_{vol}.
2. Temperature.
3. First field variable.

4. Second field variable.
5. Etc., up to six field variables.

Second line:

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this data line as often as necessary to define the volumetric shrinkage coefficient as a function of temperature and user-defined field variables.

Data lines to define orthotropic cure shrinkage:

First line:

1. Orthotropic cure shrinkage coefficient, γ_{11}.
2. Orthotropic cure shrinkage coefficient, γ_{22}.
3. Orthotropic cure shrinkage coefficient, γ_{33}. (Not used for plane stress and shell cases.)
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Second line:

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this data line as often as necessary to define the orthotropic shrinkage coefficients as a function of temperature and user-defined field variables.

Data lines to define anisotropic cure shrinkage:

First line:

1. Anisotropic cure shrinkage coefficient, γ_{11}.
2. Anisotropic cure shrinkage coefficient, γ_{22}.
3. Anisotropic cure shrinkage coefficient, γ_{33}. (Not used for plane stress and shell cases.)
4. Anisotropic cure shrinkage coefficient, γ_{12}.
5. Anisotropic cure shrinkage coefficient, γ_{13}. (Not used for plane stress and shell cases.)
6. Anisotropic cure shrinkage coefficient, γ_{23}. (Not used for plane stress and shell cases.)
7. Temperature.
8. First field variable.

Second line:

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this data line as often as necessary to define the anisotropic shrinkage coefficients as a function of temperature and user-defined field variables.

*CYCLED PLASTIC: Specify cycled yield stress data for the ORNL model.

This option is used to specify the tenth-cycle yield stress and hardening values for the ORNL constitutive model. It is relevant only if the *ORNL option is used.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *ORNL
- *ORNL – Oak Ridge National Laboratory Constitutive Model*

There are no parameters associated with this option.

Data lines to specify cycled yield stress data:

First line:

1. Yield stress.
2. Plastic strain.
3. Temperature.

Repeat this data line as often as necessary to define the dependence of yield stress on plastic strain and, if needed, on temperature.

*CYCLIC: Define cyclic symmetry for a cavity radiation heat transfer analysis.

This option is used to define cavity symmetry by cyclic repetition about a point or an axis. The **CYCLIC* option can be used only following the **RADIATION SYMMETRY* option.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *Cavity Radiation in Abaqus/Standard*
- **RADIATION SYMMETRY*

Required parameters:

NC

Set this parameter equal to the number of cyclically similar images that compose the cavity formed as a result of this symmetry. The angle of rotation (about a point or an axis) used to create cyclically similar images is equal to 360°/NC.

TYPE

Set TYPE=POINT to create a two-dimensional cavity by cyclic repetition of the cavity surface defined in the model by rotation about a point, *l*. See *Figure 1*. The cavity surface defined in the model must be bounded by the line ***lk*** and a line passing through *l* at an angle, measured counterclockwise when looking into the plane of the model, of 360°/NC to ***lk***.

Set TYPE=AXIS to create a three-dimensional cavity by cyclic repetition of the cavity surface defined in the model by rotation about an axis, ***lm***. See *Figure 2*. The cavity surface defined in the model must be bounded by the plane ***lmk*** and a plane passing through line ***lm*** at an angle, measured clockwise when looking from *l* to *m*, of 360°/NC to ***lmk***. Line ***lk*** must be normal to line ***lm***.

Data line to define cyclic symmetry for a two-dimensional cavity (TYPE=POINT):

First (and only) line:

1. *x*-coordinate of rotation point *l* (see *Figure 1*).
2. *y*-coordinate of rotation point *l*.
3. *x*-coordinate of point *k*.
4. *y*-coordinate of point *k*.

Data lines to define cyclic symmetry for a three-dimensional cavity (TYPE=AXIS):

First line:

1. *x*-coordinate of point *l* on rotation axis (see *Figure 2*).
2. *y*-coordinate of point *l* on rotation axis.
3. *z*-coordinate of point *l* on rotation axis.
4. *x*-coordinate of point *m* on rotation axis.
5. *y*-coordinate of point *m* on rotation axis.
6. *z*-coordinate of point *m* on rotation axis.

Second line:

1. x-coordinate of point k.
2. y-coordinate of point k.
3. z-coordinate of point k.

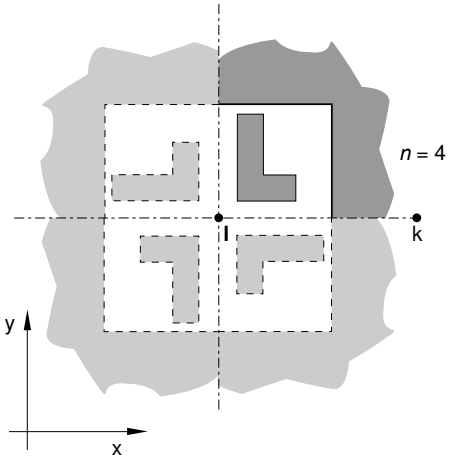


Figure 1: *CYCLIC, TYPE=POINT option.

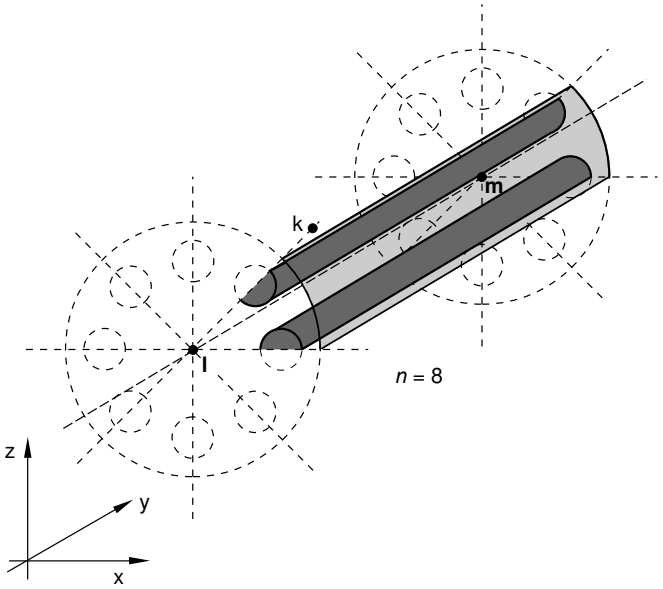


Figure 2: *CYCLIC, TYPE=AXIS option.

*CYCLIC HARDENING: Specify the size of the elastic range for the combined hardening model.

This option is used to define the evolution of the size of the elastic range for the nonlinear combined isotropic/kinematic hardening model. It can be used only in conjunction with the *PLASTIC option. The size of the elastic range remains constant during the analysis if this option is not used.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Models for Metals Subjected to Cyclic Loading*
- *PLASTIC*
- *UHARD*
- *VUHARD*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the cyclic hardening behavior, in addition to temperature. If this parameter is omitted, this behavior does not depend on field variables. See *Material Data Definition* for more information.

PARAMETERS

Include this parameter to provide the material parameters Q_∞ and b directly.

USER

Include this parameter to define the size of the elastic range in user subroutine *UHARD* in Abaqus/Standard analyses and user subroutine *VUHARD* in Abaqus/Explicit analyses. This parameter cannot be included if the kinematic hardening component is specified via half-cycle test data using DATA TYPE=HALF CYCLE on the associated *PLASTIC option.

Optional parameter for use with the USER parameter:

PROPERTIES

Set this parameter equal to the number of property values needed as data in user subroutine *UHARD* in Abaqus/Standard analyses and user subroutine *VUHARD* in Abaqus/Explicit analyses. The default is PROPERTIES=0.

Optional parameters if neither PARAMETERS nor USER is included:

EXTRAPOLATION

This parameter specifies the extrapolation method for the yield stress with respect to the equivalent plastic strain.

If you use the *CYCLIC HARDENING option multiple times to specify test data at different equivalent plastic strain rates, you can use the EXTRAPOLATION parameter only one time, when you specify data at a zero equivalent plastic strain rate (*CYCLIC HARDENING, RATE=0.0). On the subsequent

options, you must either omit the EXTRAPOLATION parameter or specify the same method as that used at a zero equivalent plastic strain rate.

Set EXTRAPOLATION=CONSTANT (default) to use a constant value of the yield stress outside the specified range of the equivalent plastic strains.

Set EXTRAPOLATION=LINEAR to use linear extrapolation (based on the slope) of the yield stress outside the specified range of the equivalent plastic strains.

RATE

Set this parameter equal to the equivalent plastic strain rate, $\dot{\bar{\varepsilon}}^{pl}$, for which this stress-strain curve applies.

Data lines to give tabular material data:

First line:

1. Equivalent stress defining the size of the elastic range.
2. Equivalent plastic strain.
3. Temperature.
4. First field variable.
5. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the values of the isotropic component as a function of plastic strain, temperature, and other predefined variables.

Data lines to define the material parameters directly (PARAMETERS):

First line:

1. Equivalent stress defining the size of the elastic range at zero plastic strain.
2. Isotropic hardening parameter, Q_∞.
3. Isotropic hardening parameter, b.
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the material parameters on temperature and other predefined field variables.

Data lines for USER with PROPERTIES:

First line:

1. Give the hardening properties, eight per line.

Repeat this data line as often as necessary to define all hardening properties.

*CYCLIC SYMMETRY MODEL: Define the number of sectors and the axis of symmetry for a cyclic symmetric structure.

This option is used to define the number of sectors and the axis of symmetry for a cyclic symmetric structure.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Analysis of Models that Exhibit Cyclic Symmetry*
- **SELECT CYCLIC SYMMETRY MODES*
- **TIE*

Required parameters:

N

Set this parameter equal to the number of repetitive datum sectors in the entire 360° structure.

Data line to define the axis of cyclic symmetry:

First (and only) line:

1. *X*-coordinate of the first point defining the cyclic symmetry axis.
2. *Y*-coordinate of the first point defining the cyclic symmetry axis.
3. *Z*-coordinate of the first point defining the cyclic symmetry axis.

The second point is not required for two-dimensional analyses.

9. *X*-coordinate of the second point defining the cyclic symmetry axis.
11. *Y*-coordinate of the second point defining the cyclic symmetry axis.
13. *Z*-coordinate of the second point defining the cyclic symmetry axis.

D

This section describes each Abaqus input option that begins with D, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **D ADDED MASS*
- **DAMAGE EVOLUTION*
- **DAMAGE INITIATION*
- **DAMAGE STABILIZATION*
- **DAMPING*
- **DAMPING CONTROLS*
- **DASHPOT*
- **DEBOND*
- **DECHARGE*
- **DECURRENT*
- **DEFORMATION PLASTICITY*
- **D EM POTENTIAL*
- **DENSITY*
- **DEPVAR*
- **DESIGN GRADIENT*
- **DESIGN PARAMETER*
- **DESIGN RESPONSE*
- **DETONATION POINT*
- **DFLOW*
- **DFLUX*
- **DIAGNOSTICS*
- **DIELECTRIC*
- **DIFFUSIVITY*
- **DIRECT CYCLIC*
- **DISCRETE ELASTICITY*
- **DISCRETE SECTION*
- **DISPLAY BODY*
- **DISTRIBUTING*
- **DISTRIBUTING COUPLING*
- **DISTRIBUTION*
- **DISTRIBUTION TABLE*
- **DLOAD*
- **DOMAIN DECOMPOSITION*
- **DRAG CHAIN*
- **DRUCKER PRAGER*
- **DRUCKER PRAGER CREEP*

- **DRUCKER PRAGER HARDENING*
- **DSA CONTROLS*
- **DSECHARGE*
- **DSECURRENT*
- **DSFLOW*
- **DSFLUX*
- **DSLOAD*
- **DYNAMIC*
- **DYNAMIC TEMPERATURE-DISPLACEMENT*

*D ADDED MASS: Specify distributed added mass in a FREQUENCY step.

This option is used to include the “added mass” contributions due to distributed fluid inertia loads in a *FREQUENCY step.

Products: Abaqus/Aqua

Type: History data

Level: Step

References:

• *Abaqus/Aqua Analysis*

There are no parameters associated with this option.

Data lines to define distributed fluid added mass:

First line:

1. Element number or element set label.
2. Distributed load type label FI.
3. Effective outer diameter of the member.
4. Transverse added-mass coefficient, C_A.

Repeat this data line as often as necessary to define distributed fluid added mass at various elements or element sets.

Data lines to define concentrated fluid added mass:

First line:

1. Element number or element set label.
2. Distributed load type label FI1 or FI2.
3. Added mass coefficient, L_{ts}.
4. Structural acceleration shape factor, F_{2s}.

Repeat this data line as often as necessary to define concentrated fluid added mass at various elements or element sets.

*DAMAGE EVOLUTION: Specify material properties to define the evolution of damage.

This option is used to provide material properties that define the evolution of damage leading to eventual failure. It must be used in conjunction with the **DAMAGE INITIATION* option. It can be utilized for materials defined for cohesive elements, for enriched elements, for elements with plane stress formulations (plane stress, shell, continuum shell, and membrane elements) used with the damage model for fiber-reinforced materials, for ductile bulk materials associated with any element type in a low-cycle fatigue analysis, and, in Abaqus/Explicit, for elastic-plastic materials associated with any element type. It can also be used in conjunction with the **SURFACE INTERACTION* and **DAMAGE INITIATION* options to define a contact property model that allows modeling of progressive failure for cohesive surfaces.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Damage Evolution and Element Removal for Ductile Metals*
- *Damage Evolution and Element Removal for Fiber-Reinforced Composites*
- *Damage Evolution for Fiber-Reinforced Composites Using Multiscale Modeling*
- *Damage Evolution for Ductile Materials in Low-Cycle Fatigue*
- *Defining the Constitutive Response of Cohesive Elements Using a Traction-Separation Description*
- *Contact Cohesive Behavior*
- *Modeling Discontinuities as an Enriched Feature Using the Extended Finite Element Method*

Required parameters:

TYPE

Set TYPE=DISCONTINUOUS to define the residual stiffness scaling factor after the initiation of damage.

Set TYPE=DISPLACEMENT to define the evolution of damage as a function of the total (for elastic materials in cohesive elements) or the plastic (for bulk elastic-plastic materials) displacement after the initiation of damage.

Set TYPE=ENERGY to define the evolution of damage in terms of the energy required for failure (fracture energy) after the initiation of damage.

Set TYPE=HYSTERESIS ENERGY to define the evolution of damage in terms of the inelastic hysteresis energy density dissipated per stabilized cycle after the initiation of damage in a low-cycle fatigue analysis.

Optional parameters:

DEGRADATION

Set DEGRADATION=MAXIMUM (default) to specify that the current damage evolution mechanism will interact with other damage evolution mechanisms in a maximum sense to determine the total damage from multiple mechanisms.

Set DEGRADATION=MULTIPLICATIVE to specify that the current damage evolution mechanism will interact with other damage evolution mechanisms using the same value of the DEGRADATION parameter in a multiplicative manner to determine the total damage from multiple mechanisms.

This parameter is relevant only for cohesive elements.

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of damage evolution. If this parameter is omitted, it is assumed that properties defining the evolution of damage are constant or depend only on temperature. See *Material Data Definition* for more information.

FAILURE INDEX

This parameter can be used only in conjunction with the user-defined or the LARC05 damage initiation criterion.

Set this parameter equal to the corresponding failure mechanism specified in the user-defined or the LARC05 damage initiation criterion for enriched elements.

MIXED MODE BEHAVIOR

This parameter is meaningful only when the **DAMAGE EVOLUTION* option is used to define the evolution of damage for materials associated with cohesive elements or for surface-based cohesive behavior. If this parameter is omitted, Abaqus assumes that the damage evolution behavior is mode independent.

Set MIXED MODE BEHAVIOR=TABULAR to specify the fracture energy or displacement (total or plastic) directly as a function of the shear-normal mode mix for cohesive elements. This method must be used to specify the mixed-mode behavior for cohesive elements when TYPE=DISPLACEMENT. This option can be used with any choice for the MODE MIX RATIO parameter.

Set MIXED MODE BEHAVIOR=POWER LAW to specify the fracture energy as a function of the mode mix by means of a power law mixed mode fracture criterion. This option must be used in conjunction with either the MODE MIX RATIO=ACCUMULATED ENERGY option or the MODE MIX RATIO=ENERGY option.

Set MIXED MODE BEHAVIOR=BK to specify the fracture energy as a function of the mode mix by means of the Benzeggagh-Kenane mixed mode fracture criterion. This option must be used in conjunction with either the MODE MIX RATIO=ACCUMULATED ENERGY option or the MODE MIX RATIO=ENERGY option.

MODE MIX RATIO

This parameter can be used only in conjunction with the MIXED MODE BEHAVIOR parameter. The specification of the damage evolution properties (fracture energy or effective displacement) as a function of the mode mix depends on the value of this parameter. See *Defining Damage Evolution Data as a Tabular Function of Mode Mix* or *Defining Damage Evolution Data as a Tabular Function of Mode Mix* for further details.

Set MODE MIX RATIO=ENERGY (default for all element-based, surface-based, and XFEM-based cohesive traction-separation behaviors) to define the mode mix in terms of a ratio of fracture energy in the different modes. The fracture energy is computed based on the current state of deformation only, not the deformation history. Either this option or the MODE MIX RATIO=ACCUMULATED ENERGY option must be used when MIXED MODE BEHAVIOR=POWER LAW or MIXED MODE BEHAVIOR=BK.

Set MODE MIX RATIO=ACCUMULATED ENERGY (default for surface-based cohesive behavior in Abaqus/Explicit) to define the mode mix in terms of a ratio of fracture energy in the different modes. The fracture energy is computed based on the energy accumulated over the entire deformation history at an integration point. Either this option or the MODE MIX RATIO=ENERGY option must

be used when MIXED MODE BEHAVIOR=POWER LAW or MIXED MODE BEHAVIOR=BK. This option is not available for surface-based cohesive behavior in Abaqus/Explicit.

Set MODE MIX RATIO=TRACTION to define the mode mix in terms of a ratio of traction components.

POWER

This parameter can be used only in conjunction with MIXED MODE BEHAVIOR=POWER LAW or MIXED MODE BEHAVIOR=BK.

Set this parameter equal to the exponent in the power law or the Benzeggagh-Kenane criterion that defines the variation of fracture energy with mode mix for cohesive elements.

RATE DEPENDENT

This parameter can be used only in conjunction with the **DAMAGE INITIATION* option used with CRITERION=MAXE, CRITERION=MAXS, CRITERION=QUADE, or CRITERION=QUADS for either cohesive elements with traction separation behavior or cohesive contact in Abaqus/Explicit.

Include this parameter to define the damage evolution criterion as a function of the effective rate of separation.

REF ENERGY

This parameter can be used only in conjunction with TYPE=HYSTERESIS ENERGY.

Set this parameter equal to a reference value of the accumulated inelastic hysteresis energy density per cycle. The default is 1.0.

SOFTENING

Set SOFTENING=LINEAR (default) to specify a linear softening stress-strain response (after the initiation of damage) for linear elastic materials or a linear evolution of the damage variable with deformation (after the initiation of damage) for elastic-plastic materials.

Set SOFTENING=EXPONENTIAL to specify an exponential softening stress-strain response (after the initiation of damage) for linear elastic materials or an exponential evolution of the damage variable with deformation (after the initiation of damage) for elastic-plastic materials.

Set SOFTENING=TABULAR to specify the evolution of the damage variable with deformation (after the initiation of damage) in tabular form. SOFTENING=TABULAR can be used only in conjunction with TYPE=DISPLACEMENT.

Data lines to specify damage evolution for TYPE=DISCONTINUOUS:

First line:

1. Scaling factor after tension damage initiation, SF^t.
2. Scaling factor after compression damage initiation, SF^c.
3. Temperature, if temperature dependent.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the post-damage scaling factors as a function of temperature and other predefined field variables.

Data lines to specify damage evolution for TYPE=DISPLACEMENT, SOFTENING=LINEAR without the MIXED MODE BEHAVIOR and the RATE DEPENDENT parameters:

First line:

1. Effective total or plastic displacement at failure, measured from the time of damage initiation. (Units of L.)
2. Temperature, if temperature dependent.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the total or the plastic displacement at failure as a function of temperature and other predefined field variables.

Data lines to specify damage evolution for TYPE=DISPLACEMENT, SOFTENING=LINEAR, RATE DEPENDENT without the MIXED MODE BEHAVIOR parameter:

First line:

1. Effective total or plastic displacement at failure, measured from the time of damage initiation. (Units of L.)
2. Effective rate of separation. (Units of LT^{-1}.)
3. Temperature, if temperature dependent.
4. First field variable.
5. Second field variable.
6. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the total or the plastic displacement at failure as a function of temperature and other predefined field variables.

Data lines to specify damage evolution for TYPE=ENERGY, SOFTENING=LINEAR without the MIXED MODE BEHAVIOR and the RATE DEPENDENT parameters:

First line:

1. Fracture energy. (Units of FL^{-1}.)

   Note: Fracture energy refers to energy per unit area.

2. Temperature, if temperature dependent.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the fracture energy as a function of temperature and other predefined field variables.

Data lines to specify damage evolution for TYPE=ENERGY, SOFTENING=LINEAR, RATE DEPENDENT without the MIXED MODE BEHAVIOR parameter:

First line:

1. Fracture energy. (Units of FL^{-1}.)
2. Effective rate of separation. (Units of LT^{-1}.)
3. Temperature, if temperature dependent.
4. First field variable.
5. Second field variable.
6. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the fracture energy as a function of temperature and other predefined field variables.

Data lines to specify damage evolution for TYPE=DISPLACEMENT, SOFTENING=LINEAR, MIXED MODE BEHAVIOR=TABULAR and without the RATE DEPENDENT parameter:

First line:

1. Total displacement at failure, measured from the time of damage initiation. (Units of L.)
2. Appropriate mode mix ratio.
3. Appropriate mode mix ratio (if relevant, for three-dimensional problems with anisotropic shear behavior).
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the total displacement at failure as a function of mode mix, temperature, and other predefined field variables.

Data lines to specify damage evolution for TYPE=DISPLACEMENT, SOFTENING=LINEAR, MIXED MODE BEHAVIOR=TABULAR, RATE DEPENDENT:

First line:

1. Total displacement at failure, measured from the time of damage initiation. (Units of L.)
2. Appropriate mode mix ratio.
3. Appropriate mode mix ratio (if relevant, for three-dimensional problems with anisotropic shear behavior).

4. Effective rate of separation. (Units of LT^{-1}.)
5. Temperature, if temperature dependent.
6. First field variable.
7. Second field variable.
8. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the total displacement at failure as a function of mode mix, temperature, and other predefined field variables.

Data lines to specify damage evolution for TYPE=ENERGY, SOFTENING=LINEAR, MIXED MODE BEHAVIOR=TABULAR and without the RATE DEPENDENT parameter:

First line:

1. Fracture energy. (Units of FL^{-1}.)
2. Appropriate mode mix ratio.
3. Appropriate mode mix ratio (if relevant, for three-dimensional problems with anisotropic shear behavior).
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the fracture energy as a function of mode mix, temperature, and other predefined field variables.

Data lines to specify damage evolution for TYPE=ENERGY, SOFTENING=LINEAR, MIXED MODE BEHAVIOR=TABULAR, RATE DEPENDENT:

First line:

1. Fracture energy. (Units of FL^{-1}.)
2. Appropriate mode mix ratio.
3. Appropriate mode mix ratio (if relevant, for three-dimensional problems with anisotropic shear behavior).
4. Effective rate of separation. (Units of LT^{-1}.)
5. Temperature, if temperature dependent.
6. First field variable.
7. Second field variable.
8. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the fracture energy as a function of mode mix, temperature, and other predefined field variables.

Data lines to specify damage evolution for TYPE=DISPLACEMENT, SOFTENING=EXPONENTIAL without the MIXED MODE BEHAVIOR and the RATE DEPENDENT parameters:

First line:

1. Effective total or plastic displacement at failure, measured from the time of damage initiation. (Units of L.)
2. Exponential law parameter.
3. Temperature, if temperature dependent.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the total or the plastic displacement at failure and the exponential law parameter as a function of temperature and other predefined field variables.

Data lines to specify damage evolution for TYPE=DISPLACEMENT, SOFTENING=EXPONENTIAL, RATE DEPENDENT without the MIXED MODE BEHAVIOR parameter:

First line:

1. Effective total or plastic displacement at failure, measured from the time of damage initiation. (Units of L.)
2. Exponential law parameter.
3. Effective rate of separation. (Units of LT^{-1}.)
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the total or the plastic displacement at failure and the exponential law parameter as a function of temperature and other predefined field variables.

Data lines to specify damage evolution for TYPE=ENERGY, SOFTENING=EXPONENTIAL without the MIXED MODE BEHAVIOR and the RATE DEPENDENT parameters:

First line:

1. Fracture energy. (Units of FL^{-1}.)
2. Temperature, if temperature dependent.
3. First field variable.
4. Second field variable.

5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the fracture energy as a function of temperature and other predefined field variables.

Data lines to specify damage evolution for TYPE=ENERGY, SOFTENING=EXPONENTIAL, RATE DEPENDENT without the MIXED MODE BEHAVIOR parameter:

First line:

1. Fracture energy. (Units of FL^{-1}.)
2. Effective rate of separation. (Units of LT^{-1}.)
3. Temperature, if temperature dependent.
4. First field variable.
5. Second field variable.
6. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the fracture energy as a function of temperature and other predefined field variables.

Data lines to specify damage evolution for TYPE=DISPLACEMENT, SOFTENING=EXPONENTIAL, MIXED MODE BEHAVIOR=TABULAR and without the RATE DEPENDENT parameter:

First line:

1. Total displacement at failure, measured from the time of damage initiation. (Units of L.)
2. Exponential law parameter.
3. Appropriate mode mix ratio.
4. Appropriate mode mix ratio (if relevant, for three-dimensional problems with anisotropic shear behavior).
5. Temperature, if temperature dependent.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the total displacement at failure and the exponential law parameter as a function of mode mix, temperature, and other predefined field variables.

Data lines to specify damage evolution for TYPE=DISPLACEMENT, SOFTENING=EXPONENTIAL, MIXED MODE BEHAVIOR=TABULAR, RATE DEPENDENT:

First line:

1. Total displacement at failure, measured from the time of damage initiation. (Units of L.)
2. Exponential law parameter.

3. Appropriate mode mix ratio.
4. Appropriate mode mix ratio (if relevant, for three-dimensional problems with anisotropic shear behavior).
5. Effective rate of separation. (Units of LT^{-1}.)
6. Temperature, if temperature dependent.
7. First field variable.
8. Second field variable.
9. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the total displacement at failure and the exponential law parameter as a function of mode mix, temperature, and other predefined field variables.

Data lines to specify damage evolution for TYPE=ENERGY, SOFTENING=EXPONENTIAL, MIXED MODE BEHAVIOR=TABULAR and without the RATE DEPENDENT parameter:

First line:

1. Fracture energy. (Units of FL^{-1}.)
2. Appropriate mode mix ratio.
3. Appropriate mode mix ratio (if relevant, for three-dimensional problems with anisotropic shear behavior).
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the fracture energy as a function of mode mix, temperature, and other predefined field variables.

Data lines to specify damage evolution for TYPE=ENERGY, SOFTENING=EXPONENTIAL, MIXED MODE BEHAVIOR=TABULAR, RATE DEPENDENT:

First line:

1. Fracture energy. (Units of FL^{-1}.)
2. Appropriate mode mix ratio.
3. Appropriate mode mix ratio (if relevant, for three-dimensional problems with anisotropic shear behavior).
4. Effective rate of separation. (Units of LT^{-1}.)
5. Temperature, if temperature dependent.
6. First field variable.
7. Second field variable.
8. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the fracture energy as a function of mode mix, temperature, and other predefined field variables.

Data lines to specify damage evolution for TYPE=DISPLACEMENT, SOFTENING=TABULAR without the MIXED MODE BEHAVIOR and the RATE DEPENDENT parameters:

First line:

1. Damage variable.
2. Effective total or plastic displacement, measured from the time of damage initiation. (Units of L.)
3. Temperature, if temperature dependent.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the damage variable as a function of the total or the plastic displacement, temperature, and other predefined field variables.

Data lines to specify damage evolution for TYPE=DISPLACEMENT, SOFTENING=TABULAR, RATE DEPENDENT without the MIXED MODE BEHAVIOR parameter:

First line:

1. Damage variable.
2. Effective total or plastic displacement, measured from the time of damage initiation. (Units of L.)
3. Effective rate of separation. (Units of LT^{-1}.)
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the damage variable as a function of the total or the plastic displacement, temperature, and other predefined field variables.

Data lines to specify damage evolution for TYPE=DISPLACEMENT, SOFTENING=TABULAR, MIXED MODE BEHAVIOR=TABULAR and without the RATE DEPENDENT parameter:

First line:

1. Damage variable.
2. Effective total displacement, measured from the time of damage initiation. (Units of L.)
3. Appropriate mode mix ratio.
4. Appropriate mode mix ratio (if relevant, for three-dimensional problems with anisotropic shear behavior).
5. Temperature, if temperature dependent.

6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the damage variable as a function of the total displacement, mode mix, temperature, and other predefined field variables.

Data lines to specify damage evolution for TYPE=DISPLACEMENT, SOFTENING=TABULAR, MIXED MODE BEHAVIOR=TABULAR, RATE DEPENDENT:

First line:

1. Damage variable.
2. Effective total displacement, measured from the time of damage initiation. (Units of L.)
3. Appropriate mode mix ratio.
4. Appropriate mode mix ratio (if relevant, for three-dimensional problems with anisotropic shear behavior).
5. Effective rate of separation. (Units of LT^{-1}.)
6. Temperature, if temperature dependent.
7. First field variable.
8. Second field variable.
9. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the damage variable as a function of the total displacement, mode mix, temperature, and other predefined field variables.

Data lines to specify damage evolution for TYPE=ENERGY, SOFTENING=LINEAR or EXPONENTIAL, MIXED MODE BEHAVIOR=POWER LAW or BK and without the RATE DEPENDENT parameter:

First line:

1. Normal mode fracture energy. (Units of FL^{-1}.)
2. Shear mode fracture energy for failure in the first shear direction. (Units of FL^{-1}.)
3. Shear mode fracture energy for failure in the second shear direction. (Units of FL^{-1}.)
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the fracture energy as a function of temperature and other predefined field variables.

Data lines to specify damage evolution for TYPE=ENERGY, SOFTENING=LINEAR or EXPONENTIAL, MIXED MODE BEHAVIOR=POWER LAW or BK, RATE DEPENDENT:

First line:

1. Normal mode fracture energy. (Units of FL^{-1}.)
2. Shear mode fracture energy for failure in the first shear direction. (Units of FL^{-1}.)
3. Shear mode fracture energy for failure in the second shear direction. (Units of FL^{-1}.)
4. Effective rate of separation. (Units of LT^{-1}.)
5. Temperature, if temperature dependent.
6. First field variable.
7. Second field variable.
8. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the fracture energy as a function of temperature and other predefined field variables.

Data lines to specify damage evolution for TYPE=ENERGY, SOFTENING=LINEAR for the damage model for unidirectional fiber-reinforced materials:

First line:

1. Fracture energy of the lamina in the longitudinal tensile direction. (Units of FL^{-1}.)
2. Fracture energy of the lamina in the longitudinal compressive direction. (Units of FL^{-1}.)
3. Fracture energy of the lamina in the transverse tensile direction. (Units of FL^{-1}.)
4. Fracture energy of the lamina in the transverse compressive direction. (Units of FL^{-1}.)
5. Temperature, if temperature dependent.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of fracture energies on temperature and other predefined field variables.

Data lines to specify damage evolution for TYPE=ENERGY, SOFTENING=EXPONENTIAL for the damage model for bidirectional fabric-reinforced materials:

First line:

1. Tensile fracture energy in local 1-direction, G_{f1+}. (Units of FL^{-1}.)
2. Compressive fracture energy in local 1-direction, G_{f1-}. (Units of FL^{-1}.)
3. Tensile fracture energy in local 2-direction, G_{f2+}. (Units of FL^{-1}.)

4. Compressive fracture energy in local 2-direction, G_{f2-}. (Units of FL^{-1}.)
5. Parameter for shear damage, α_{12}.
6. Maximum shear damage, $d_{12}^{\max}$.
7. Temperature, if temperature dependent.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to five field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of material constants on temperature and other predefined field variables.

Data lines to specify damage evolution for TYPE=ENERGY, SOFTENING=LINEAR for the damage model for constituents in unidirectional fiber-reinforced materials:

First line:

1. Fracture energy of the constituent in the tensile direction. (Units of FL^{-1}.)
2. Fracture energy of the constituent in the compressive direction. (Units of FL^{-1}.)
3. Temperature, if temperature dependent.
4. First field variable.
5. Second field variable.
6. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of fracture energies on temperature and other predefined field variables.

Data lines to specify damage evolution for TYPE=HYSTERESIS ENERGY in a low-cycle fatigue analysis:

First line:

1. Material constant, c_3. (Units of $L/CYCLE$.)
2. Material constant, c_4.
3. Temperature, if temperature dependent.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables per line.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of material constants on temperature and other predefined field variables.

*DAMAGE INITIATION: Specify material and contact properties to define the initiation of damage.

This option is used to provide material properties that define the initiation of damage. The damage initiation material models can be broadly classified into the following categories based upon their applications:

- continuum damage and fracture in ductile metals,
- continuum damage and instability in sheet metals,
- continuum damage in fiber-reinforced composites,
- continuum damage in fiber-reinforced composites using the multiscale material model,
- continuum damage in ductile materials during low-cycle fatigue,
- traction-separation based damage in cohesive elements,
- damage in enriched elements (in the context of the extended finite element method), and
- general stress/strain-based continuum damage.

This option can also be used in conjunction with the **SURFACE INTERACTION* option to define a contact property model that allows definition of damage initiation for cohesive surfaces.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Damage Initiation for Ductile Metals*
- *Damage Initiation for Fiber-Reinforced Composites*
- *Damage and Failure for Fiber-Reinforced Composites Using Multiscale Modeling*
- *Generalized Damage Initiation Criteria for Anisotropic Materials*
- *Damage Initiation for Ductile Materials in Low-Cycle Fatigue*
- *Defining the Constitutive Response of Cohesive Elements Using a Traction-Separation Description*
- *Modeling Discontinuities as an Enriched Feature Using the Extended Finite Element Method*
- *Contact Cohesive Behavior*
- *UDMGINI*

Defining damage initiation as a material property for continuum damage modeling in ductile materials

Required parameters:

CRITERION

Set CRITERION=DUCTILE to specify a damage initiation criterion based on the ductile failure strain.

Set CRITERION=HC to specify the Hosford-Coulomb damage initiation criterion.

Set CRITERION=JOHNSON COOK to specify a damage initiation criterion based on the Johnson-Cook failure strain. This parameter is available only in Abaqus/Explicit.

Set CRITERION=SHEAR to specify a damage initiation criterion based on the shear failure strain.

Optional parameters:

ACCUMULATION POWER

This parameter can be used only with CRITERION=DUCTILE, CRITERION=HC, CRITERION=SHEAR, and, in Abaqus/Explicit, with CRITERION=JOHNSON COOK.

Set this parameter equal to the value of the power coefficient, n.

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the damage initiation properties. If this parameter is omitted, it is assumed that the damage initiation properties are constant or depend only on temperature.

This parameter cannot be used with CRITERION=JOHNSON COOK.

KS

This parameter can be used only in conjunction with CRITERION=SHEAR.

Set this parameter equal to the value of k_s. The default value is $k_s = 0$.

LODE DEPENDENT

Include this parameter to define a ductile damage initiation criterion that depends on the Lode angle.

Data lines to specify damage initiation for CRITERION=DUCTILE without the LODE DEPENDENT parameter:

First line:

1. Equivalent plastic strain at damage initiation.
2. Stress triaxiality, $(-p/q)$.
3. Strain rate.
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the equivalent plastic strain at damage initiation as a function of triaxiality, strain rate, temperature, and other predefined field variables.

Data lines to specify damage initiation for CRITERION=DUCTILE, LODE DEPENDENT:

First line:

1. Equivalent plastic strain at damage initiation.
2. Stress triaxiality, $\eta = -p/q$.
3. Lode angle term, $\xi = \cos(3\Theta)$.
4. Strain rate.

5. Temperature, if temperature dependent.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the equivalent plastic strain at damage initiation as a function of triaxiality, Lode angle, strain rate, temperature, and other predefined field variables.

Data lines to specify damage initiation for CRITERION=HC:

First line:

1. a.
2. b.
3. c.
4. n.
5. d.
6. $\dot{\varepsilon}_0$.
7. Temperature, if temperature dependent.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the damage initiation as a function of temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=JOHNSON COOK:

First (and only) line:

1. Johnson-Cook failure parameter, d_1.
2. Johnson-Cook failure parameter, d_2.
3. Johnson-Cook failure parameter, d_3.
4. Johnson-Cook failure parameter, d_4.
5. Johnson-Cook failure parameter, d_5.
6. Melting temperature, θ_{melt}.
7. Transition temperature, $\theta_{\mathrm{transition}}$.
8. Reference strain rate, $\dot{\varepsilon}_0$.

Data lines to specify damage initiation for CRITERION=SHEAR:

First line:

1. Equivalent plastic strain at damage initiation.
2. Shear stress ratio, $\theta_s = (q + k_s p)/\tau_{\max}$.

3. Strain rate.
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the equivalent plastic strain at damage initiation as a function of the shear stress ratio, strain rate, temperature, and other predefined field variables.

Defining damage initiation as a material property for modeling instability in sheet metals

Required parameters:

CRITERION

Set CRITERION=FLD to specify a damage initiation criterion based on a forming limit diagram.

Set CRITERION=FLSD to specify a damage initiation criterion based on a forming limit stress diagram.

Set CRITERION=MK to specify a damage initiation criterion based on a Marciniak-Kuczynski analysis. This option is available only in Abaqus/Explicit.

Set CRITERION=MSFLD to specify a damage initiation criterion based on the Müschenborn and Sonne forming limit diagram.

Optional parameters:

DEFINITION

This parameter can be used only in conjunction with CRITERION=MSFLD.

Set DEFINITION=MSFLD (default) to specify the MSFLD damage initiation criterion by providing the limit equivalent plastic strain as a tabular function of α.

Set DEFINITION=FLD to specify the MSFLD damage initiation criterion by providing the limit major strain as a tabular function of minor strain.

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the damage initiation properties. If this parameter is omitted, it is assumed that the damage initiation properties are constant or depend only on temperature.

FEQ

This parameter can be used only in conjunction with CRITERION=MK.

Set this parameter equal to the critical value of the deformation severity index for equivalent plastic strains, f_{eq}^{crit}. The default value is $f_{eq}^{\text{crit}} = 10$.

Set this parameter equal to zero if the deformation severity factor for equivalent plastic strains should not be considered for the evaluation of the Marciniak-Kuczynski criterion.

FNN

This parameter can be used only in conjunction with CRITERION=MK.

Set this parameter equal to the critical value of the deformation severity index for strains normal to the groove direction, f_{nn}^{crit}. The default value is $f_{nn}^{\text{crit}} = 10$.

Set this parameter equal to zero if the deformation severity factor for strains normal to the groove should not be considered for the evaluation of the Marciniak-Kuczynski criterion.

FNT

This parameter can be used only in conjunction with CRITERION=MK.

Set this parameter equal to the critical value of the deformation severity index for shear strains, f_{nt}^{crit}.

The default value is $f_{nt}^{\text{crit}} = 10$.

Set this parameter equal to zero if the deformation severity factor for shear strains should not be considered for the evaluation of the Marciniak-Kuczynski criterion.

FREQUENCY

This parameter can be used only in conjunction with CRITERION=MK.

Set this parameter equal to the frequency, in increments, at which the Marciniak-Kuczynski analysis is going to be performed. By default, the M-K analysis is performed every increment; that is, FREQUENCY=1.

NUMBER IMPERFECTIONS

This parameter can be used only in conjunction with CRITERION=MK.

Set this parameter equal to the number of imperfections to be considered for the evaluation of the Marciniak-Kuczynski analysis. These imperfections are assumed to be equally spaced in the angular direction. By default, four imperfections are used.

OMEGA

This parameter can be used only in conjunction with CRITERION=MSFLD in Abaqus/Explicit.

Set this parameter equal to the factor ω used for filtering the ratio of principal strain rates used for the evaluation of the MSFLD damage initiation criterion. The default value is $\omega = 1.0$.

PEINC

This parameter can be used only in conjunction with CRITERION=MSFLD in Abaqus/Explicit.

Set this parameter equal to the accumulated increment in equivalent plastic strain used to trigger the evaluation of the MSFLD damage initiation criterion. The default value is 0.002 (0.2%).

Data lines to specify damage initiation for CRITERION=FLD:

First line:

1. Major principal strain at damage initiation.
2. Minor principal strain.
3. Temperature, if temperature dependent.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the major principal strain at damage initiation as a function of minor principal strain, temperature, and other predefined field variables.

Data lines to specify damage initiation for CRITERION=FLSD:

First line:

1. Major principal stress at damage initiation.
2. Minor principal stress.
3. Temperature, if temperature dependent.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the major principal stress at damage initiation as a function of minor principal stress, temperature, and other predefined field variables.

Data lines to specify damage initiation for CRITERION=MK:

First line:

1. Groove size relative to nominal thickness of the section, f_0.
2. Angle (in degrees) with respect to the 1-direction of the local material orientation.
3. Temperature, if temperature dependent.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the groove size as a function of angular distance, temperature, and other predefined field variables.

Data lines to specify damage initiation for CRITERION=MSFLD, DEFINITION=MSFLD:

First line:

1. Equivalent plastic strain at initiation of localized necking.
2. Ratio of minor to major principal strains, α.
3. Equivalent plastic strain rate.
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the equivalent plastic strain at damage initiation as a function of α, equivalent plastic strain rate, temperature, and other predefined field variables.

Data lines to specify damage initiation for CRITERION=MSFLD, DEFINITION=FLD:

First line:

1. Major principal strain at initiation of localized necking.
2. Minor principal strain.
3. Equivalent plastic strain rate.
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the major principal strain at damage initiation as a function of minor principal strain, equivalent plastic strain rate, temperature, and other predefined field variables.

Defining damage initiation as a material property for continuum damage modeling in fiber-reinforced composites

Required parameters:

CRITERION

Set CRITERION=HASHIN to specify damage initiation criteria based on the Hashin analysis.

Set CRITERION=LARC05 to specify damage initiation based on the LaRC05 criterion. This option is available only in Abaqus/Standard.

Set CRITERION=PLY FABRIC to specify a damage initiation criterion for a bidirectional fabric-reinforced material. This option is available only in Abaqus/Explicit.

Optional parameters:

ALPHA

This parameter can be used only in conjunction with CRITERION=HASHIN.

Set this parameter equal to the value of the coefficient that will multiply the shear contribution to the Hashin's fiber initiation criterion. The default value is $\alpha = 0.0$.

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the damage initiation properties. If this parameter is omitted, it is assumed that the damage initiation properties are constant or depend only on temperature.

Data lines to specify damage initiation for CRITERION=HASHIN:

First line:

1. Longitudinal tensile strength of the lamina.
2. Longitudinal compressive strength of the lamina.
3. Transverse tensile strength of the lamina.
4. Transverse compressive strength of the lamina.
5. Longitudinal shear strength of the lamina.
6. Transverse shear strength of the lamina.
7. Temperature, if temperature dependent.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the strengths on temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=LARC05:

First line:

1. Longitudinal tensile strength, X_T.
2. Longitudinal compressive strength, X_C.
3. Transverse tensile strength, Y_T.
4. Transverse compressive strength, Y_C.
5. In-plane shear strength, S_L.
6. Fracture plane angle (in degrees) for pure compression, α_0 (default=53°).
7. Misalignment angle (in degrees) at failure for pure compression, φ_0.
8. Transverse shear strength, S_T.

Second line:

1. Longitudinal shear friction coefficient, η_L.
2. Transverse shear friction coefficient, η_T.
3. Temperature, if temperature dependent.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the strengths on temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=PLY FABRIC:

First line:

1. Tensile strength along fiber direction 1, X_{1+}.
2. Compressive strength along fiber direction 1, X_{1-}.
3. Tensile strength along fiber direction 2, X_{2+}.
4. Compressive strength along fiber direction 2, X_{2-}.
5. Shear strength, S.
6. Temperature, if temperature dependent.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the strengths on temperature and other predefined field variables.

Defining damage initiation as a material property for continuum damage modeling in fiber-reinforced composites using the multiscale material model in Abaqus/Standard

Required parameters:

CRITERION

Set CRITERION=HSNFIBER to specify damage initiation criteria in the fiber based on the failure criterion adapted from Hashin theory.

Set CRITERION=HSNMATRIX to specify damage initiation in the matrix based on the failure criterion adapted from Hashin theory.

Set CRITERION=TSINVMATRIX to specify a damage initiation criterion in the matrix material based on the transverse normal stress and the shear stress invariant.

Optional parameters:

ALPHA

This parameter can be used only in conjunction with CRITERION=HSNFIBER.

Set this parameter equal to the value of the coefficient that will multiply the shear contribution to the HSNFIBER initiation criterion for the fiber. The default value is $\alpha = 0.0$.

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the damage initiation properties. If this parameter is omitted, it is assumed that the damage initiation properties are constant or depend only on temperature.

Data lines to specify damage initiation for CRITERION=HSNFIBER:

First line:

1. $K_1^{f,t}$.
2. $K_1^{f,c}$.
3. K_4^f.
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Third field variable.
8. Fourth field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the failure coefficients on temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=HSNMATRIX:

First line:

1. $K_2^{m,t}$.
2. $K_2^{m,c}$.
3. K_3^m.
4. K_4^m.
5. Temperature, if temperature dependent.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the failure coefficients on temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=TSINVMATRIX:

First line:

1. $A_2^{m,t}$.
2. $A_2^{m,c}$.

3. K_4^m.
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables per line.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the failure coefficients on temperature and other predefined field variables.

Defining general stress/strain-based damage initiation as a material property for continuum damage modeling in anisotropic materials

Required parameters:

CRITERION

Set CRITERION=MAX STRAIN to specify a strain-based damage initiation criterion based on the maximum strain theory.

Set CRITERION=MAX STRESS to specify a stress-based damage initiation criterion based on the maximum stress theory.

Set CRITERION=TSAIWU to specify a stress-based damage initiation criterion based on the Tsai-Wu theory.

Set CRITERION=TSAIWUE to specify a strain-based damage initiation criterion based on the Tsai-Wu theory.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the damage initiation properties. If this parameter is omitted, it is assumed that the damage initiation properties are constant or depend only on temperature.

RATE DEPENDENT

This parameter applies only to Abaqus/Explicit analyses.

Include this parameter to define the damage initiation properties to depend either on the equivalent strain rate or on the strain rate components. To define dependency on the equivalent strain rate, the strain rate components on the data line should be left blank. To define dependency on the strain rate components, the equivalent strain rate on the data line should be left blank.

Data lines to specify damage initiation for CRITERION=MAX STRAIN without the RATE DEPENDENT parameter:

First line:

1. Maximum tensile strain in the local X-direction, X_{ϵ_t}.

2. Maximum compressive strain (absolute value) in the local X-direction, X_{ϵ_c}.
3. Maximum tensile strain in the local Y-direction, Y_{ϵ_t}.
4. Maximum compressive strain (absolute value) in the local Y-direction, Y_{ϵ_c}.
5. Maximum tensile strain in the local Z-direction, Z_{ϵ_t}.
6. Maximum compressive strain (absolute value) in the local Z-direction, Z_{ϵ_c}.
7. Maximum shear strain in the local X-Y plane, $S_{\epsilon_{xy}}$.
8. Maximum shear strain in the local X-Z plane, $S_{\epsilon_{xz}}$.

Second line:

1. Maximum shear strain in the local Y-Z plane, $S_{\epsilon_{yz}}$.
2. Temperature, if temperature dependent.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables per line.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the maximum strain at damage initiation as a function of temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=MAX STRAIN with the RATE DEPENDENT parameter:

First line:

1. Maximum tensile strain in the local X-direction, X_{ϵ_t}.
2. Maximum compressive strain (absolute value) in the local X-direction, X_{ϵ_c}.
3. Maximum tensile strain in the local Y-direction, Y_{ϵ_t}.
4. Maximum compressive strain (absolute value) in the local Y-direction, Y_{ϵ_c}.
5. Maximum tensile strain in the local Z-direction, Z_{ϵ_t}.
6. Maximum compressive strain (absolute value) in the local Z-direction, Z_{ϵ_c}.
7. Maximum shear strain in the local X-Y plane, $S_{\epsilon_{xy}}$.
8. Maximum shear strain in the local X-Z plane, $S_{\epsilon_{xz}}$.

Second line:

1. Maximum shear strain in the local Y-Z plane, $S_{\epsilon_{yz}}$.
2. Equivalent strain rate $\dot{\bar{\epsilon}}$.
3. First strain rate component $\dot{\epsilon}_{11}$.
4. Second train rate component $\dot{\epsilon}_{22}$.
5. Third strain rate component $\dot{\epsilon}_{33}$.
6. Fourth strain rate component $\dot{\epsilon}_{12}$.
7. Fifth strain rate component $\dot{\epsilon}_{13}$.

8. Sixth strain rate component $\dot{\epsilon}_{23}$.

Third line (leave blank if there is no dependency on temperature or predefined field variables):

1. Temperature, if temperature dependent.
2. First field variable.
3. Second field variable.
4. Etc., up to seven field variables per line.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than seven):

1. Eighth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the maximum strain at damage initiation as a function of strain rate, temperature, and other predefined field variables.

Data lines to specify damage initiation for CRITERION=MAX STRESS without the RATE DEPENDENT parameter:

First line:

1. Maximum tensile stress in the local X-direction, X_t.
2. Maximum compressive stress (absolute value) in the local X-direction, X_c.
3. Maximum tensile stress in the local Y-direction, Y_t.
4. Maximum compressive stress (absolute value) in the local Y-direction, Y_c.
5. Maximum tensile stress in the local Z-direction, Z_t.
6. Maximum compressive stress (absolute value) in the local Z-direction, Z_c.
7. Maximum shear stress in the local X-Y plane, S_{xy}.
8. Maximum shear stress in the local X-Z plane, S_{xz}.

Second line:

1. Maximum shear stress in the local Y-Z plane, S_{yz}.
2. Temperature, if temperature dependent.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables per line.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the maximum stress at damage initiation as a function of temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=MAX STRESS with the RATE DEPENDENT parameter:

First line:

1. Maximum tensile stress in the local X-direction, X_t.
2. Maximum compressive stress (absolute value) in the local X-direction, X_c.
3. Maximum tensile stress in the local Y-direction, Y_t.

4. Maximum compressive stress (absolute value) in the local Y-direction, Y_c.
5. Maximum tensile stress in the local Z-direction, Z_t.
6. Maximum compressive stress (absolute value) in the local Z-direction, Z_c.
7. Maximum shear stress in the local X-Y plane, S_{xy}.
8. Maximum shear stress in the local X-Z plane, S_{xz}.

Second line:

1. Maximum shear stress in the local Y-Z plane, S_{yz}.
2. Equivalent strain rate $\dot{\bar{\epsilon}}$.
3. First strain rate component $\dot{\epsilon}_{11}$.
4. Second strain rate component $\dot{\epsilon}_{22}$.
5. Third strain rate component $\dot{\epsilon}_{33}$.
6. Fourth strain rate component $\dot{\epsilon}_{12}$.
7. Fifth strain rate component $\dot{\epsilon}_{13}$.
8. Sixth strain rate component $\dot{\epsilon}_{23}$.

Third line (leave blank if there is no dependency on temperature or predefined field variables):

1. Temperature, if temperature dependent.
2. First field variable.
3. Second field variable.
4. Etc., up to seven field variables per line.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than seven):

1. Eighth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the maximum stress at damage initiation as a function of strain rate, temperature, and other predefined field variables.

Data lines to specify damage initiation for CRITERION=TSAIWU without the RATE DEPENDENT parameter:

First line:

1. F_1.
2. F_2.
3. F_3.
4. F_{11}.
5. F_{22}.
6. F_{33}.
7. F_{12}.
8. F_{13}.

Second line:

1. F_{23}.
2. F_{44}.

3. F_{55}.
4. F_{66}.
5. Temperature, if temperature dependent.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the material properties at damage initiation as a function of temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=TSAIWU with the RATE DEPENDENT parameter:

First line:

1. F_1.
2. F_2.
3. F_3.
4. F_{11}.
5. F_{22}.
6. F_{33}.
7. F_{12}.
8. F_{13}.

Second line:

1. F_{23}.
2. F_{44}.
3. F_{55}.
4. F_{66}.
5. Equivalent strain rate $\dot{\bar{\epsilon}}$.
6. First strain rate component $\dot{\epsilon}_{11}$.
7. Second strain rate component $\dot{\epsilon}_{22}$.
8. Third strain rate component $\dot{\epsilon}_{33}$.

Third line:

1. Fourth strain rate component $\dot{\epsilon}_{12}$.
2. Fifth strain rate component $\dot{\epsilon}_{13}$.
3. Sixth strain rate component $\dot{\epsilon}_{23}$.
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Third field variable.
8. Fourth field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the material properties at damage initiation as a function of temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=TSAIWUE without the RATE DEPENDENT parameter:

First line:

1. G_1.
2. G_2.
3. G_3.
4. G_{11}.
5. G_{22}.
6. G_{33}.
7. G_{12}.
8. G_{13}.

Second line:

1. G_{23}.
2. G_{44}.
3. G_{55}.
4. G_{66}.
5. Temperature, if temperature dependent.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the material properties at damage initiation as a function of temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=TSAIWUE with the RATE DEPENDENT parameter:

First line:

1. G_1.
2. G_2.
3. G_3.
4. G_{11}.
5. G_{22}.

6. G_{33}.
7. G_{12}.
8. G_{13}.

Second line:

1. G_{23}.
2. G_{44}.
3. G_{55}.
4. G_{66}.
5. Equivalent strain rate $\dot{\bar{\epsilon}}$.
6. First strain rate component $\dot{\epsilon}_{11}$.
7. Second strain rate component $\dot{\epsilon}_{22}$.
8. Third strain rate component $\dot{\epsilon}_{33}$.

Third line:

1. Fourth strain rate component $\dot{\epsilon}_{12}$.
2. Fifth strain rate component $\dot{\epsilon}_{13}$.
3. Sixth strain rate component $\dot{\epsilon}_{23}$.
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Third field variable.
8. Fourth field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the material properties at damage initiation as a function of temperature and other predefined field variables.

Defining damage initiation as a material property for ductile materials in low-cycle fatigue

Required parameters:

CRITERION

Set CRITERION=HYSTERESIS ENERGY to specify damage initiation criteria based on the inelastic hysteresis energy density dissipated per stabilized cycle in a low-cycle fatigue analysis.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the damage initiation properties. If this parameter is omitted, it is assumed that the damage initiation properties are constant or depend only on temperature.

REF ENERGY

Set this parameter equal to a reference value of the accumulated inelastic hysteresis energy density per cycle. The default is 1.0.

Data lines to specify damage initiation for CRITERION=HYSTERESIS ENERGY:

First line:

1. Material constant, c_1. (Units of cycle.)
2. Material constant, c_2.
3. Temperature, if temperature dependent.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables per line.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of material constants on temperature and other predefined field variables.

Defining damage initiation as a traction-separation–based material property for cohesive elements

Required parameters:

CRITERION

Set CRITERION=MAXE to specify damage initiation based on the maximum nominal strain criterion.

Set CRITERION=MAXS to specify damage initiation based on the maximum nominal stress criterion.

Set CRITERION=QUADE to specify damage initiation based on the quadratic separation-interaction criterion.

Set CRITERION=QUADS to specify damage initiation based on the quadratic traction-interaction criterion.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the damage initiation properties. If this parameter is omitted, it is assumed that the damage initiation properties are constant or depend only on temperature.

RATE DEPENDENT

This parameter can be used to define rate-dependent traction separation behavior only in Abaqus/Explicit.

Data lines to specify damage initiation for CRITERION=QUADE or CRITERION=MAXE without the RATE DEPENDENT parameter:

First line:

1. Nominal strain at damage initiation in a normal-only mode.
2. Nominal strain at damage initiation in a shear-only mode that involves separation only along the first shear direction.
3. Nominal strain at damage initiation in a shear-only mode that involves separation only along the second shear direction.
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the maximum normal and shear strains at damage initiation as a function of temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=QUADE, RATE DEPENDENT or CRITERION=MAXE, RATE DEPENDENT:

First line:

1. Nominal strain at damage initiation in a normal-only mode.
2. Nominal strain at damage initiation in a shear-only mode that involves separation only along the first shear direction.
3. Nominal strain at damage initiation in a shear-only mode that involves separation only along the second shear direction.
4. Effective rate of separation.
5. Temperature, if temperature dependent.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the maximum normal and shear strains at damage initiation as a function of effective rate of separation, temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=QUADS or CRITERION=MAXS without the RATE DEPENDENT parameter:

First line:

1. Maximum nominal stress in the normal-only mode.
2. Maximum nominal stress in the first shear direction (for a mode that involves separation only in this direction).
3. Maximum nominal stress in the second shear direction (for a mode that involves separation only in this direction).

4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the maximum normal and shear tractions at damage initiation as a function of temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=QUADS, RATE DEPENDENT or CRITERION=MAXS, RATE DEPENDENT:

First line:

1. Maximum nominal stress in the normal-only mode.
2. Maximum nominal stress in the first shear direction (for a mode that involves separation only in this direction).
3. Maximum nominal stress in the second shear direction (for a mode that involves separation only in this direction).
4. Effective rate of separation.
5. Temperature, if temperature dependent.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the maximum normal and shear tractions at damage initiation as a function of the effective rate of separation, temperature, and other predefined field variables.

Defining damage initiation as a material property for enriched elements in Abaqus/Standard

Required parameters:

CRITERION

Set CRITERION=LARC05 to specify a damage initiation criterion based on the LaRC05 criterion.

Set CRITERION=MAXE to specify a damage initiation criterion based on the maximum nominal strain.

Set CRITERION=MAXS to specify a damage initiation criterion based on the maximum nominal stress.

Set CRITERION=MAXPE to specify a damage initiation criterion based on the maximum principal strain.

Set CRITERION=MAXPS to specify a damage initiation criterion based on the maximum principal stress.

Set CRITERION=QUADE to specify damage initiation based on the quadratic separation-interaction criterion.

Set CRITERION=QUADS to specify damage initiation based on the quadratic traction-interaction criterion.

Set CRITERION=USER to specify a user-defined damage initiation criterion.

Optional parameters:

ANGLEMAX

This parameter is relevant only for CRITERION=LARC05, CRITERION=MAXPE, CRITERION=MAXPS, or CRITERION=USER.

Set this parameter equal to the maximum allowed change in the crack propagation angle (in degrees) between the new crack propagation direction and the previous crack propagation direction. The default is 85°.

ANGSMOOTH

This parameter can be used only in conjunction with POSITION=NONLOCAL.

Set this parameter equal to the maximum allowed difference (in degrees) below which the normals of the crack facets are included in the moving least-squares approximation to smooth out the crack normals to obtain the crack propagation direction along the crack front. The default is 70°.

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the damage initiation properties. If this parameter is omitted, it is assumed that the damage initiation properties are constant or depend only on temperature.

FAILURE MECHANISMS

This parameter can be used only in conjunction with CRITERION=USER.

Set this parameter equal to the total number of failure mechanisms to be specified in a user-defined damage initiation criterion. This parameter value must be a nonzero.

GROWTH TOLERANCE

Include this parameter to define a separate tolerance used for crack growth.

Set this parameter equal to the tolerance within which the damage initiation criterion in enriched elements with an existing crack must be satisfied. If this parameter is included but no value is specified, the value is set by the TOLERANCE parameter.

INISMOOTH

This parameter can be used only in conjunction with POSITION=NONLOCAL.

Set INISMOOTH=YES (default) to indicate the normals of pre-existing crack facets in elements are included in the moving least-squares approximation to smooth out the crack normals to obtain the crack propagation direction along the crack front.

Set INISMOOTH=NO to indicate the normals of pre-existing crack facets in elements are excluded from the moving least-squares approximation to smooth out the crack normals to obtain the crack propagation direction along the crack front.

NORMAL DIRECTION

This parameter can be used only in conjunction with CRITERION=MAXE, CRITERION=MAXS, CRITERION=QUADE, or CRITERION=QUADS.

Set NORMAL DIRECTION=1 (default) to specify that a new crack orthogonal to the element local 1-direction will be introduced when the damage initiation criterion is satisfied.

Set NORMAL DIRECTION=2 to specify that a new crack orthogonal to the element local 2-direction will be introduced when the damage initiation criterion is satisfied.

Set NORMAL DIRECTION=3 to specify that a new crack orthogonal to the element local 3-direction will be introduced when the damage initiation criterion is satisfied.

NPOLY

This parameter can be used only in conjunction with POSITION=NONLOCAL.

This parameter is used to specify the number of terms in the polynomial used for the moving least-squares approximation to smooth out the crack normals.

Set NPOLY=4 to use a linear polynomial approximation.

Set NPOLY=7 (default) to use a quadratic polynomial approximation.

Set NPOLY=10 to use a cubic polynomial approximation.

Set NPOLY=0 to suppress the moving least-squares approximation.

POSITION

This parameter can be used only in conjunction with CRITERION=MAXPE, CRITERION=MAXPS, CRITERION=MAXE, CRITERION=MAXS, CRITERION=QUADE, CRITERION=QUADS,or CRITERION=USER.

Set POSITION=CENTROID (default) to use the stress/strain at the element centroid ahead of the crack tip to determine if the damage initiation criterion is satisfied and to determine the crack propagation direction (if needed).

Set POSITION=COMBINED to use the stress/strain extrapolated to the crack tip to determine if the damage initiation criterion is satisfied and to use the stress/strain at the element centroid to determine the crack propagation direction (if needed).

Set POSITION=CRACKTIP to use the stress/strain extrapolated to the crack tip to determine if the damage initiation criterion is satisfied and to determine the crack propagation direction (if needed).

Set POSITION=NONLOCAL to use the stress/strain extrapolated to the crack tip to determine if the damage initiation criterion is satisfied and to use the stress/strain averaged over a group of elements around the crack tip in the enriched region to determine the crack propagation direction (if needed). By default, a moving least-squares approximation by polynomials is used to further smooth out the normals of the individual crack facets in elements that satisfy the damage initiation criterion to obtain the crack propagation direction along the crack front. Set NPOLY=0 to suppress this behavior. This option can be used only in conjunction with CRITERION=MAXPE or CRITERION=MAXPS.

PROPERTIES

This parameter can be used only in conjunction with CRITERION=USER.

Set this parameter equal to the number of material constants being specified for a user-defined damage initiation criterion. The parameter value must be a nonzero value.

R CRACK DIRECTION

This parameter can be used only in conjunction with POSITION=NONLOCAL.

Set this parameter equal to the radius around the crack tip within which the elements are included for calculating the averaged stress/strain and for smoothing out the normals of the individual crack facets along the crack front used to obtain the crack propagation direction. The default value is three times the typical element characteristic length in the enriched region.

SMOOTHING

This parameter can be used only in conjunction with POSITION=NONLOCAL.

Set SMOOTHING=NONE (default) to use the stress/strain at the integration points directly for averaging.

Set SMOOTHING=NODAL to assemble the stress/strain to element nodes and then interpolate to the integration points for averaging.

TOLERANCE

Set this parameter equal to the tolerance within which the damage initiation criterion must be satisfied. The default is 0.05.

UNSTABLE GROWTH TOLERANCE

Include this parameter to define a separate tolerance used for unstable crack growth.

Set this parameter equal to the tolerance within which the damage initiation criterion in enriched elements with an existing crack must be satisfied for multiple elements to be allowed to crack without excessively cutting back the increment size in an unstable crack growth analysis.

If this parameter is included but no value is specified, the default value is infinity.

WEIGHTING METHOD

This parameter can be used only in conjunction with POSITION=NONLOCAL.

Set WEIGHTING METHOD=UNIFORM (default) to average the stress/strain with a uniform weight function.

Set WEIGHTING METHOD=GAUSS to average the stress/strain with a Gaussian weight function.

Set WEIGHTING METHOD=CUBIC SPLINE to average the stress/strain with a cubic spline weight function.

Set WEIGHTING METHOD=USER to average the stress/strain with a user-defined weight function.

Data lines to specify damage initiation for CRITERION=LARC05:

First line:

1. Longitudinal tensile strength, X_T.
2. Longitudinal compressive strength, X_C.
3. Transverse tensile strength, Y_T.
4. Transverse compressive strength, Y_C.
5. In-plane shear strength, S_L.
6. Fracture plane angle for pure compression, α_0 (default=53°).
7. Misalignment angle at failure for pure compression, φ_0.
8. Transverse shear strength, S_T.

Second line:

1. Longitudinal shear friction coefficient, η_L.
2. Transverse shear friction coefficient, η_T.
3. Temperature, if temperature dependent.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the strengths on temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=QUADE or CRITERION=MAXE:

First line:

1. Nominal strain at damage initiation in a normal-only mode.
2. Nominal strain at damage initiation in a shear-only mode that involves separation only along the first shear direction.
3. Nominal strain at damage initiation in a shear-only mode that involves separation only along the second shear direction.
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the maximum normal and shear tractions at damage initiation as a function of temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=QUADS or CRITERION=MAXS:

First line:

1. Maximum nominal stress in the normal-only mode.
2. Maximum nominal stress in the first shear direction (for a mode that involves separation only in this direction).
3. Maximum nominal stress in the second shear direction (for a mode that involves separation only in this direction).
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the maximum normal and shear tractions at damage initiation as a function of temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=MAXPE:

First line:

1. Maximum principal strain at damage initiation.
2. Temperature, if temperature dependent.
3. First field variable.

4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the maximum principal strain at damage initiation as a function of temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=MAXPS:

First line:

1. Maximum principal stress at damage initiation.
2. Temperature, if temperature dependent.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the maximum principal stress at damage initiation as a function of temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=USER:

First line:

1. Give the material constants, eight per line.

Repeat this data line as often as necessary to define all material constants.

Defining damage initiation as part of a contact property model

Required parameters:

CRITERION

Set CRITERION=MAXS to specify a damage initiation criterion based on the maximum nominal stress criterion for cohesive surfaces.

Set CRITERION=MAXU to specify a damage initiation criterion based on the maximum separation criterion for cohesive surfaces.

Set CRITERION=QUADS to specify a damage initiation based on the quadratic traction-interaction criterion for cohesive surfaces.

Set CRITERION=QUADU to specify a damage initiation based on the quadratic separation-interaction criterion for cohesive surfaces.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the damage initiation properties. If this parameter is omitted, it is assumed that the damage initiation properties are constant or depend only on temperature.

RATE DEPENDENT

This parameter can be used only in conjunction with CRITERION=MAXU, CRITERION=MAXS, CRITERION=QUADU, or CRITERION=QUADS for cohesive contact in Abaqus/Explicit.

Include this parameter to define the damage initiation criterion as a function of the effective rate of separation.

Data lines to specify damage initiation for CRITERION=QUADU or CRITERION=MAXU without the RATE DEPENDENT parameter:

First line:

1. Separation at damage initiation in a normal-only mode.
2. Separation at damage initiation in a shear-only mode that involves separation only along the first shear direction.
3. Separation at damage initiation in a shear-only mode that involves separation only along the second shear direction.
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the maximum normal and shear separations at damage initiation as a function of temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=QUADU or CRITERION=MAXU, RATE DEPENDENT:

First line:

1. Separation at damage initiation in a normal-only mode.
2. Separation at damage initiation in a shear-only mode that involves separation only along the first shear direction.
3. Separation at damage initiation in a shear-only mode that involves separation only along the second shear direction.
4. Effective rate of separation.
5. Temperature, if temperature dependent.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the maximum normal and shear separations at damage initiation as a function of effective rate of separation, temperature, and other predefined field variables.

Data lines to specify damage initiation for CRITERION=QUADS or CRITERION=MAXS without the RATE DEPENDENT parameter:

First line:

1. Maximum nominal stress in the normal-only mode.
2. Maximum nominal stress in the first shear direction (for a mode that involves separation only in this direction).
3. Maximum nominal stress in the second shear direction (for a mode that involves separation only in this direction).
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the maximum normal and shear tractions at damage initiation as a function of temperature and other predefined field variables.

Data lines to specify damage initiation for CRITERION=QUADS or CRITERION=MAXS, RATE DEPENDENT:

First line:

1. Maximum nominal stress in the normal-only mode.
2. Maximum nominal stress in the first shear direction (for a mode that involves separation only in this direction).
3. Maximum nominal stress in the second shear direction (for a mode that involves separation only in this direction).
4. Effective rate of separation.
5. Temperature, if temperature dependent.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the maximum normal and shear tractions at damage initiation as a function of effective rate of separation, temperature, and other predefined field variables.

*DAMAGE STABILIZATION: Specify viscosity coefficients for the damage model for fiber-reinforced materials, surface-based cohesive behavior or cohesive behavior in enriched elements.

This option is used to specify viscosity coefficients used in the viscous regularization scheme for the damage model for fiber-reinforced materials, surface-based traction-separation behavior in contact or cohesive behavior in enriched elements.

- For fiber-reinforced materials, you can use this option in conjunction with the **DAMAGE INITIATION*, CRITERION=HASHIN and **DAMAGE EVOLUTION* options.
- For constituents of fiber-reinforced materials, you can use this option in conjunction with the **DAMAGE INITIATION*, CRITERION=HSNFIBER, HSNMATRIX, TSINVMATRIX, and **DAMAGE EVOLUTION* options.
- For surface-based traction-separation behavior, you can use this option in conjunction with the **DAMAGE INITIATION*, CRITERION=MAXS, MAXU, QUADS, or QUADU and **DAMAGE EVOLUTION* options.
- For XFEM, you can use this option in conjunction with the **DAMAGE INITIATION*, CRITERION=LARC05, MAXE, MAXS, MAXPE, MAXPS, QUADE, or QUADS and **DAMAGE EVOLUTION* options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Viscous Regularization*
- *Contact Cohesive Behavior*
- *Modeling Discontinuities as an Enriched Feature Using the Extended Finite Element Method*

There are no parameters associated with this option.

Data lines to define viscosity coefficients for fiber-reinforced materials:

First line:

1. Viscosity coefficient in the longitudinal tensile direction.
2. Viscosity coefficient in the longitudinal compressive direction.
3. Viscosity coefficient in the transverse tensile direction.
4. Viscosity coefficient in the transverse compressive direction.

Data lines to define viscosity coefficients for constituents of fiber-reinforced materials:

First line:

1. Viscosity coefficient in the tensile direction.
2. Viscosity coefficient in the compression direction.

Data line to define viscosity coefficients for surfaced-based traction-separation behavior or cohesive behavior in enriched elements:

First (and only) line:

1. Viscosity coefficient.

*DAMPING: Specify material damping.

Warning: The use of stiffness proportional material damping or band-limited damping in Abaqus/Explicit may reduce the stable time increment dramatically and can lead to longer analysis times. See *Material Damping*.

This option is used to provide material damping for mode-based analyses and direct-integration dynamic analyses in Abaqus/Standard and for explicit dynamic analysis in Abaqus/Explicit.

Damping is defined in a material data block except in the case of elements defined with the **BEAM GENERAL SECTION* option, the **SHELL GENERAL SECTION* option, the **ROTARY INERTIA* option, the **MASS* option, or the **SUBSTRUCTURE PROPERTY* option. For the **BEAM GENERAL SECTION*, the **SHELL GENERAL SECTION*, and the **SUBSTRUCTURE PROPERTY* options the **DAMPING* option must be used in conjunction with the property references. For the **MASS* and the **ROTARY INERTIA* options damping must be specified using either the ALPHA or the COMPOSITE parameter associated with these options. Damping can also come from damper elements like connectors and dashpots, and it may also be defined as step data using the **GLOBAL DAMPING* and **MODAL DAMPING* options.

You must use multiple **DAMPING* options if you want to use a tabular definition (to define temperature and/or field dependence) for more than one material damping coefficient.

This option is ignored in mode-based procedures that use Lanczos or subspace iteration eigenvalue extraction that does not use the SIM architecture (**FREQUENCY*, EIGENSOLVER=SUBSPACE, SIM=NO).

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- *Material Damping*
- *About Dynamic Analysis Procedures*
- *Explicit Dynamic Analysis*

Optional parameters:

ALPHA

Set this parameter equal to the α_R factor to create Rayleigh mass proportional damping. The default is ALPHA=0. (Units of T^{-1}.) Rayleigh mass proportional damping can be used in the procedures identified in *Procedures*.

Set ALPHA=TABULAR to specify that the mass proportional damping is dependent on temperature and/or field variables.

BAND LIMITED

Set this parameter equal to the damping ratio, η_d, used in the band-limited damping. The default is BAND LIMITED=0. Band-limited damping can be used in the procedures identified in *Procedures*.

Set BAND LIMITED=TABULAR to specify that the damping ratio in band-limited damping is dependent on temperature and/or field variables.

BETA

Set this parameter equal to the β_R factor to create Rayleigh stiffness proportional damping. The default is BETA=0. (Units of T.) Rayleigh stiffness proportional damping can be used in the procedures identified in *Procedures*.

Set BETA=TABULAR to specify that the stiffness proportional damping is dependent on temperature and/or field dependent variables.

COMPOSITE

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the fraction of critical damping to be used with this material in calculating composite damping factors for the modes. Composite damping is used in mode-based procedures that follow subspace iteration eigenvalue extraction or eigenvalue extraction using the Lanczos eigensolver with SIM=NO, except for **STEADY STATE DYNAMICS*, SUBSPACE PROJECTION. Use the **MODAL DAMPING*, VISCOUS=COMPOSITE option to activate composite modal damping.

The default is COMPOSITE=0.

DEPENDENCIES

This parameter applies when ALPHA=TABULAR, BAND LIMITED=TABULAR, BETA=TABULAR, or STRUCTURAL=TABULAR.

Set this parameter equal to the number of field variables included in the definition of the α_R, η_d, β_R, or s factors, in addition to temperature. If this parameter is omitted, it is assumed that damping does not depend on field variables. See *Specifying Field Variable Dependence* for more information.

FREQUENCY DEPENDENT

This parameter is valid only when STRUCTURAL=TABULAR is used.

Use this parameter to specify that the structural damping is dependent on frequency.

HIGH FREQUENCY CUTOFF

This parameter is valid only when the BAND LIMITED parameter is used.

Set this parameter equal to the frequency value, f_h, to define the upper limit of the frequency range. (Units of cycles/time.)

LOW FREQUENCY CUTOFF

This parameter is valid only when the BAND LIMITED parameter is used.

Set this parameter equal to the frequency value, f_l, to define the lower limit of the frequency range. (Units of cycles/time.)

STRUCTURAL

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the s factor to create structural damping. The default is STRUCTURAL=0. Structural damping can be used in the procedures identified in *Procedures*.

Set STRUCTURAL=TABULAR to specify that the structural damping is dependent on temperature and/or field variables and/or frequency.

Data lines to define temperature and/or field variable-dependent mass proportional damping (ALPHA=TABULAR):

First line:

1. α_R. (Units of T^{-1}.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the alpha damping as a function of temperature and other predefined field variables.

Data lines to define a temperature and/or field variable-dependent damping ratio in band-limited damping (BAND LIMITED=TABULAR):

First line:

1. η_d.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the damping ratio as a function of temperature and other predefined field variables.

Data lines to define temperature and/or field variable-dependent stiffness proportional damping (BETA=TABULAR):

First line:

1. β_R. (Units of T.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the beta damping as a function of temperature and other predefined field variables.

Data lines to define temperature and/or field variable-dependent structural damping (STRUCTURAL=TABULAR) in Abaqus/Standard:

First line:

1. s.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the structural damping as a function of temperature and other predefined field variables.

Data lines to define temperature and/or field variable-dependent frequency-dependent structural damping (STRUCTURAL=TABULAR, FREQUENCY DEPENDENT) in Abaqus/Standard:

First line:

1. s.
2. Frequency f, in cycles per time.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the frequency-dependent structural damping as a function of temperature and other predefined field variables.

*DAMPING CONTROLS: Specify damping controls.

This option is used to control the type (viscous, structural) and source of damping (material, global) within the step definition for the following types of analyses in Abaqus/Standard:

- **STEADY STATE DYNAMICS*, DIRECT
- **STEADY STATE DYNAMICS*, SUBSPACE PROJECTION
- **STEADY STATE DYNAMICS* that supports nondiagonal damping
- **MODAL DYNAMIC* that supports nondiagonal damping
- **MATRIX GENERATE*
- **SUBSTRUCTURE GENERATE*

Damping can be defined at the material level using visco-elastic behavior or using the **DAMPING* option; at the element level using **SPRING*, COMPLEX STIFFNESS or **CONNECTOR DAMPING* options; for acoustic elements using **ACOUSTIC MEDIUM*, VOLUMETRIC DRAG; or using the acoustic impedance definitions (**IMPEDANCE* and **SIMPEDANCE*). Damping is defined at the global level using **GLOBAL DAMPING*. The **DAMPING CONTROLS* option controls which of the supplied damping options will participate in the current step or within a substructure.

Use the **SUBSTRUCTURE DAMPING CONTROLS* option to control damping for substructure elements (see *Defining Substructure Damping* for details).

Products: Abaqus/Standard Abaqus/CAE

Type: Model or history data

Level: Part, Part instance

Abaqus/CAE: Supported in the *Step module* only for substructure generation.

References:

- *Material Damping*
- *Damping in Dynamic Analysis*
- **FREQUENCY*

Optional parameters:

LOW FREQUENCY CUTOFF

This parameter is relevant for transient modal analysis only.

Set this parameter equal to the frequency value below which the eigenmodes will not be damped even if damping is requested at the material, element, and/or modal level. This parameter is useful for free structures or for models with secondary base motions.

If this parameter is omitted, the default value is used. The default value is six orders of magnitude smaller than the frequency of the first deformable eigenmode.

If this parameter is included but no value is given, the low frequency cutoff value is given as the square root of the platform precision value.

A negative value can be assigned to allow damping at the low frequency modes.

The parameter units are cycles/time.

STRUCTURAL

Set this parameter equal to ELEMENT to request the structural damping matrix that includes material and/or element damping properties only.

Set this parameter equal to FACTOR to request the structural damping matrix that includes the global damping factor only.

Set this parameter equal to COMBINED to request the structural damping matrix that includes the combination of both ELEMENT and FACTOR.

Set this parameter equal to NONE to exclude the structural damping matrix at this step.

If this parameter is omitted or the option is not used within the step definition, the default uses all structural damping specified at the model and step levels. If both material and global structural damping are specified, the COMBINED damping is used.

If this parameter is omitted or the option is not used as a suboption of **SUBSTRUCTURE PROPERTY*, the substructure property uses COMBINED as the default with the structural factor specified under the **DAMPING*, STRUCTURAL option.

VISCOUS

Set this parameter equal to ELEMENT to request a viscous damping matrix that includes material and/or element damping properties only.

Set this parameter equal to FACTOR to request a viscous damping matrix that includes the global damping factor only.

Set this parameter equal to COMBINED to request a viscous damping matrix that includes a combination of ELEMENT and FACTOR.

Set this parameter equal to NONE to exclude the viscous damping matrix in this step.

If this parameter is omitted or the option is not used within the step definition, the default uses all viscous damping specified at the model and step levels. If both material and global damping are specified, the COMBINED damping is used.

If this parameter is omitted or the option is not used as a suboption of **SUBSTRUCTURE PROPERTY*, the substructure property uses COMBINED as the default with the mass and stiffness proportional Rayleigh damping factors specified under the **DAMPING*, ALPHA or BETA option.

There are no data lines associated with this option.

*DASHPOT: Define dashpot behavior.

This option is used to define the dashpot behavior for dashpot elements.

In Abaqus/Standard analyses it is also used to define the dashpot behavior for ITS and JOINTC elements. If the **DASHPOT* option is being used to define part of the behavior of ITS or JOINTC elements, it must be used in conjunction with the **ITS* or **JOINT* options and the ELSET and ORIENTATION parameters should not be used.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: *Property module* and *Interaction module*; supported only for linear behavior independent of field variables. For nonlinear behavior or to include field variables, model connectors in the *Interaction module.*

References:

- *Dashpots*
- *Flexible Joint Element*
- *Tube Support Elements*

Required parameter if the behavior of dashpot elements is being defined:

ELSET

Set this parameter equal to the name of the element set containing the dashpot elements for which this behavior is being defined.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the dashpot coefficient, in addition to temperature. If this parameter is omitted, it is assumed that the dashpot coefficient is independent of field variables. See *Material Data Definition* for more information.

NONLINEAR

Include this parameter to define nonlinear dashpot behavior. Omit this parameter to define linear dashpot behavior.

ORIENTATION

This parameter applies only to Abaqus/Standard analyses.

If the option is being used to define the behavior of DASHPOT1 or DASHPOT2 elements, this parameter can be used to refer to an orientation definition so that the dashpot is acting in a local system. Set this parameter equal to the name of the **ORIENTATION* definition (*Orientations*).

RTOL

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the tolerance to be used for regularizing the material data. The default is RTOL=0.03. See *Material Data Definition* for a discussion of data regularization.

Data lines to define linear dashpot behavior for DASHPOTA or ITS elements:

First line:

1. Enter a blank line.

Second line:

1. Dashpot coefficient (force per relative velocity).
2. In an Abaqus/Standard analysis this field corresponds to frequency (in cycles per time, for **STEADY STATE DYNAMICS*, DIRECT and **STEADY STATE DYNAMICS*, SUBSPACE PROJECTION analyses only). Leave this field blank in an Abaqus/Explicit analysis.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dashpot coefficient as a function of frequency, temperature, and other predefined field variables.

Data lines to define nonlinear dashpot behavior for DASHPOTA or ITS elements:

First line:

1. Enter a blank line.

Second line:

1. Force.
2. Relative velocity.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dashpot coefficient as a function of temperature and other predefined field variables.

Data lines to define linear dashpot behavior for DASHPOT1, DASHPOT2, or JOINTC elements:

First line:

1. Give the degree of freedom with which the dashpots are associated at their first nodes or, for JOINTC elements, the degree of freedom in the local corotational system for which the dashpot behavior is being defined.
2. For DASHPOT2 elements give the degree of freedom with which the dashpots are associated at their second nodes.

If the ORIENTATION parameter is included on the **DASHPOT* option when defining dashpot elements or on the **JOINT* option when defining joint elements, the degrees of freedom specified here are in the local system defined by the **ORIENTATION* option referenced.

Second line:

1. Dashpot coefficient (force per relative velocity).

2. Frequency (in cycles per time, for **STEADY STATE DYNAMICS*, DIRECT and **STEADY STATE DYNAMICS*, SUBSPACE PROJECTION analyses only).
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight variables per line.

Repeat this set of data lines as often as necessary to define the dashpot coefficient as a function of frequency, temperature, and other predefined field variables.

Data lines to define nonlinear dashpot behavior for DASHPOT1, DASHPOT2, or JOINTC elements:

First line:

1. Give the degree of freedom with which the dashpots are associated at their first nodes or, for JOINTC elements, the degree of freedom in the local corotational system for which the dashpot behavior is being defined.
2. For DASHPOT2 elements give the degree of freedom with which the dashpots are associated at their second nodes.

If the ORIENTATION parameter is included on the **DASHPOT* option when defining dashpot elements or on the **JOINT* option when defining joint elements, the degrees of freedom specified here are in the local system defined by the **ORIENTATION* option referenced.

Second line:

1. Force.
2. Relative velocity.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dashpot coefficient as a function of temperature and other predefined field variables.

*DEBOND: Activate crack propagation capability and specify debonding amplitude curve.

This option is used to specify that crack propagation may occur between two surfaces that are initially partially bonded. The **FRACTURE CRITERION* option must appear immediately following this option.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

References:

- *Crack Propagation Analysis*
- **FRACTURE CRITERION*

Required parameters:

MAIN

Set this parameter equal to the name of the main surface of the contact pair used in the crack propagation analysis.

SECONDARY

Set this parameter equal to the name of the secondary surface of the contact pair used in the crack propagation analysis.

Optional parameters:

DEBONDING FORCE

This parameter is relevant only when TYPE=VCCT, TYPE=ENHANCED VCCT, TYPE=PRX-VCCT, or TYPE=FATIGUE is used on the **FRACTURE CRITERION* option.

Set DEBONDING FORCE=STEP (default) if the traction between the two surfaces at the crack tip is to be released immediately during the following increment after debonding.

Set DEBONDING FORCE=RAMP if the traction between the two surfaces at the crack tip is to be released gradually during succeeding increments after debonding to avoid a sudden loss of stability.

FREQUENCY

Set this parameter equal to the output frequency, in increments. The crack-tip location and associated quantities will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress this output.

OUTPUT

If this parameter is omitted, crack propagation information will be printed in the data (`.dat`) file but not stored in the results (`.fil`) file.

Set OUTPUT=FILE to store the crack propagation information in the results file.

Set OUTPUT=BOTH to print the crack propagation information in the data file and to store it in the results file.

TIME INCREMENT

Set this parameter equal to the suggested time increment for automatic time incrementation to use for the first increment just after debonding starts. The default is the last relative time given on the data lines below.

For fixed time incrementation the value of this parameter will be used as the time increment after debonding starts if Abaqus/Standard finds it needs a smaller time increment than its current value. The time increment size will be modified as required until debonding is complete.

Data lines to define the debonding amplitude curve:

First line:

1. Time relative to the time at the start of debonding.
2. Relative amplitude of the stresses at the contact interface due to bonding remaining at this time.
3. Etc., up to four time/amplitude pairs per line.

Repeat this data line as often as necessary to define the debonding amplitude curve.

*DECHARGE: Input distributed electric charges for piezoelectric analysis.

This option is used to input distributed electric charges on piezoelectric elements.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

• *Piezoelectric Analysis*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the magnitude of the distributed electric charge during the step. If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (*Defining an Analysis*).

OP

Set OP=MOD (default) for existing **DECHARGE*s to remain, with this option defining electric charges to be added or modified. Set OP=NEW if all existing **DECHARGE*s applied to the model should be removed.

Optional, mutually exclusive parameters for matrix generation and direct-solution, steady-state dynamics analysis:

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the loading.

REAL

Include this parameter (default) to define the real (in-phase) part of the loading.

Data lines to define distributed electric charges:

First line:

1. Element number or element set label.
2. Distributed electric charge type label (see *About the Element Library*).
3. Reference electric charge magnitude. (Units of CL^{-2} for surface charges and CL^{-3} for body charges.)

Repeat this data line as often as necessary to define distributed electric charges for various elements or element sets.

*DECURRENT: Specify distributed current densities in an electromagnetic analysis.

This option is used to input distributed current densities in coupled thermal-electrical, coupled thermal-electrical-structural, coupled thermal-electrochemical, coupled thermal-electrochemical-structural, and coupled thermal-electrochemical-structural-pore pressure analyses or to input volume current densities in an eddy current and magnetostatic analyses.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- **DSECURRENT*
- *Coupled Thermal-Electrical Analysis*
- *Fully Coupled Thermal-Electrical-Structural Analysis*
- *Coupled Thermal-Electrochemical Analysis*
- *Fully Coupled Thermal-Electrochemical-Structural Analysis*
- *Fully Coupled Thermal-Electrochemical-Structural–Pore Pressure Analysis*
- *Eddy Current Analysis*
- *Magnetostatic Analysis*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the magnitude of the electric current density during the step (*Amplitude Curves*). If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (*Defining an Analysis*).

OP

Set OP=MOD (default) for existing **DECURRENT*s to remain, with this option defining distributed current densities to be added or modified.

Set OP=NEW if all existing **DECURRENT*s applied to the model should be removed.

Optional, mutually exclusive parameters for time-harmonic eddy current analyses:

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the volume current density.

REAL

Include this parameter (default) to define the real (in-phase) part of the volume current density.

Data lines to define distributed electrical current densities in coupled thermal-electrical, coupled thermal-electrical-structural, coupled thermal-electrochemical, coupled

thermal-electrochemical-structural, or coupled thermal-electrochemical-structural-pore pressure analyses:

First line:

1. Element number or element set label.
2. Distributed current density type label (see *About the Element Library*).
3. Reference current density magnitude. (Units of $CL^{-2}T^{-1}$ for surface current densities and $CL^{-3}T^{-1}$ for body current sources.) For nonuniform loads, the magnitude must be defined in user subroutine *DECURRENT*. If given, the magnitude will be passed into the user subroutine.
4. Table collection name (optional). Only applicable for nonuniform distributed current densities defined in user subroutine *DECURRENT*.

Repeat this data line as often as necessary to define current densities for various elements or element sets.

Data lines to define volume current densities in eddy current or magnetostatic analyses:

First line:

1. Element number or element set label.
2. Volume current density type label CJ (uniform) or CJNU (nonuniform).
3. Reference volume current density magnitude. (Units of $CL^{-2}T^{-1}$.)
4. 1-component of the volume current density vector direction.
5. 2-component of the volume current density vector direction.
6. 3-component of the volume current density vector direction.

For nonuniform loads the magnitude and direction must be defined in user subroutine *UDECURRENT*. If given, the magnitude and direction will be passed into the user subroutine.

15. Name of the orientation option (*Orientations*) that defines the local coordinate system in which the volume current density vector is specified.

Repeat this data line as often as necessary to define volume current densities for various elements or element sets.

*DEFORMATION PLASTICITY: Specify the deformation plasticity model.

This option is used to define the mechanical behavior of a material as a deformation theory Ramberg-Osgood model.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Deformation Plasticity*

There are no parameters associated with this option.

Data lines to define deformation plasticity:

First line:

1. Young's modulus, E.
2. Poisson's ratio, ν.
3. Yield stress, σ^0.
4. Exponent, n.
5. Yield offset, α.
6. Temperature.

Repeat this data line as often as necessary to define the dependence of the deformation plasticity parameters on temperature.

*D EM POTENTIAL: Specify distributed surface magnetic vector potential.

This option is used to prescribe distributed surface magnetic vector potential in an eddy current or in a magnetostatic analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- *Eddy Current Analysis*
- *Magnetostatic Analysis*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the variation of the magnetic vector potential magnitude during the step with respect to frequency for time-harmonic eddy current analysis or with respect to time for transient eddy current and magnetostatic analyses.

Amplitude references are ignored for nonuniform potentials given by user subroutine *UDEMPOTENTIAL*.

Only the magnitude of the magnetic vector potential is changed with frequency or time. The direction (or both the real and imaginary parts of the direction for a time-harmonic eddy current analysis) of the magnetic vector potential is not changed.

OP

Set OP=MOD (default) for existing **D EM POTENTIAL*s to remain, with this option modifying existing distributed magnetic vector potentials or defining additional distributed magnetic vector potentials.

Set OP=NEW if all existing **D EM POTENTIAL*s applied to the model should be removed. New distributed magnetic vector potentials can be defined.

Data lines to define distributed surface magnetic vector potentials in a time-harmonic eddy current analysis:

First line:

1. Surface name.
2. Region type label S.
3. Distributed potential type label MVP (uniform) or MVPNU (nonuniform).
4. Name of the orientation (*Orientations*) that defines the local coordinate system in which the magnetic vector potential is defined (if left blank, the default is assumed to be the global coordinate system)
5. Magnitude of the real (in-phase) part of the reference magnetic vector potential, which can be modified by using the **AMPLITUDE* option. For nonuniform potentials the magnitude must be defined in user subroutine *UDEMPOTENTIAL*. If given, this value, along with the three components of the direction for the real part of the magnetic vector potential that are provided as the next three data entries, will be passed into the user subroutine.

6. 1-component of the direction of the real part of the magnetic vector potential.
7. 2-component of the direction of the real part of the magnetic vector potential.
8. 3-component of the direction of the real part of the magnetic vector potential.
9. Magnitude of the imaginary (out-of-phase) part of the reference magnetic vector potential, which can be modified by using the **AMPLITUDE* option. For nonuniform potentials the magnitude must be defined in user subroutine *UDEMPOTENTIAL*. If given, this value, along with the three components of the direction for the imaginary part of the magnetic vector potential that are provided as the next three data entries, will be passed into the user subroutine.
10. 1-component of the direction of the imaginary part of the magnetic vector potential.
11. 2-component of the direction of the imaginary part of the magnetic vector potential.
12. 3-component of the direction of the imaginary part of the magnetic vector potential.

Repeat this data line as often as necessary to define distributed magnetic vector potentials on different surfaces.

Data lines to define distributed surface magnetic vector potentials in a transient eddy current analysis or in a magnetostatic analysis:

First line:

1. Surface name.
2. Region type label S.
3. Distributed potential type label MVP (uniform) or MVPNU (nonuniform).
4. Name of the orientation (*Orientations*) that defines the local coordinate system in which the magnetic vector potential is defined (if left blank, the default is assumed to be the global coordinate system).
5. Magnitude of the reference magnetic vector potential, which can be modified by using the **AMPLITUDE* option. For nonuniform potentials the magnitude must be defined in user subroutine *UDEMPOTENTIAL*. If given, this value, along with the three components of the direction of the magnetic vector potential that are provided as the next three data entries, will be passed into the user subroutine.
6. 1-component of the direction of the magnetic vector potential.
7. 2-component of the direction of the magnetic vector potential.
8. 3-component of the direction of the magnetic vector potential.

Repeat this data line as often as necessary to define distributed magnetic vector potentials on different surfaces.

*DENSITY: Specify material mass density.

This option is used to define a material's mass density. In an Abaqus/Standard analysis spatially varying mass density can be defined for solid continuum elements using a distribution (*Distribution Definition*).

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Density*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the density, in addition to temperature. If this parameter is omitted, it is assumed that the density is constant or depends only on temperature. See *Material Data Definition* for more information.

This parameter is not relevant in an Abaqus/Standard analysis if spatially varying density is defined using a distribution. See *Distribution Definition*.

PORE FLUID

Include this parameter if the density of the pore fluid in a porous medium is being defined.

SLURRY

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to define the mass density of the carrier fluid and the solid particles, respectively, as well as the particle diameter. This parameter must always be used together with the PORE FLUID parameter and cannot be used independently.

Data lines to define mass density:

First line:

1. Mass density. (Units of ML^{-3}.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the density as a function of temperature and other predefined field variables.

Data lines to define mass density and the proppant particle diameter with the SLURRY parameter:

First (and only) line:

1. Mass density of the carrier fluid. (Units of ML^{-3}.)
2. Mass density of the proppant particles. (Units of ML^{-3}.)
3. Diameter of the proppant particles.

Data line to define spatially varying mass density for solid continuum elements in an Abaqus/Standard analysis using a distribution:

First (and only) line:

1. Distribution name. The data defined in the distribution must be in units of ML^{-3}.

*DEPVAR: Specify solution-dependent state variables.

This option is used to allocate space at each integration point for solution-dependent state variables. If the **DEPVAR* option is used, it must appear within the **MATERIAL* definition for which solution-dependent state variables are needed. In addition, an output name and a description can be given for some or all of the solution-dependent state variables allocated by this option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *About User Subroutines and Utilities*
- *User-Defined Mechanical Material Behavior*
- *Finite Element Conversion to SPH Particles*

Optional, mutually exclusive parameters:

CONVERT

This parameter applies only to Abaqus/Explicit analyses when continuum finite elements are allowed to convert to SPH particles.

Set this parameter equal to the state variable number controlling the element conversion flag (see *Finite Element Conversion to SPH Particles*). If this parameter is omitted, no element conversion occurs.

DELETE

Set this parameter equal to the state variable number controlling the element deletion flag (see *User-Defined Mechanical Material Behavior*). If this parameter is omitted, no element deletion is controlled by solution-dependent state variables.

TVS DAMAGE

Set this parameter equal to the state variable number controlling the shell element transverse shear damage variable (see *User-Defined Mechanical Material Behavior*). If this parameter is omitted, shell element transverse shear damage is not controlled by a solution-dependent state variable.

Data lines:

First line:

1. Number of solution-dependent state variables required at each integration point.

Second line (required only to specify output descriptions for select solution-dependent state variables):

1. Index of the solution-dependent state variable for which an output name and a description are being given. This value is 1 for the first solution-dependent state variable.
2. The output variable name. The name is treated as a label; therefore, it must adhere to the conventions for labels (see *Input Syntax Rules*), with the exception that case will be preserved.
3. The output variable description. The description is treated as a label; therefore, it must adhere to the conventions for labels (see *Input Syntax Rules*), with the exception that case will be preserved.

Repeat this data line for each solution-dependent state variable for which an output name and a description are being defined. If field or history output of solution-dependent state variables is requested using the **ELEMENT OUTPUT* option, the output identifier for solution-dependent state variables for which a name has been specified under this option will consist of the string “SDV_,” followed by the specified name. Similarly, the descriptions specified under this option will be used in the corresponding field descriptions written to the output database. If an output name and a description are not given for a solution-dependent state variable, the default output identifier SDVn and description “Solution-dependent state variables” are used.

*DESIGN GRADIENT: Directly specify design gradients for design sensitivity analysis.

This option is used to specify directly design gradients with respect to design parameters, excluding design parameters related to shape. (By default, Abaqus/Design will automatically determine the design gradients with respect to non-shape design parameters numerically based on the parameterization of the input file. Design gradients with respect to shape design parameters must be specified via the **PARAMETER SHAPE VARIATION* option.)

Products: Abaqus/Design

Type: Model data

Level: Part, Part instance, Assembly, Model, Step

References:

- *Direct Design Sensitivity Analysis*
- **PARAMETER*
- **DESIGN PARAMETER*

Required parameters:

DEPENDENT

Set this parameter equal to the list of parameter names whose gradients with respect to the design parameter are to be specified. The list must be given inside parentheses as parameter names separated by commas; for example, `(depPar1, depPar2, depPar3)`.

INDEPENDENT

Set this parameter equal to the name of the design parameter with respect to which gradients are specified.

Data lines to define the design gradients:

First line:

1. Python expression giving the gradient of the first dependent parameter.

Repeat this data line as often as necessary to define the gradients of the dependent parameters consecutively with respect to the design parameter. Up to 16 entries are allowed per line.

*DESIGN PARAMETER: Specify design parameters with respect to which sensitivities are calculated.

This option is used to specify design parameters for design sensitivity analysis. Sensitivities of responses specified under the **DESIGN RESPONSE* option will be calculated with respect to these design parameters. The design parameters must be chosen from an existing set of parameters defined on the **PARAMETER* option.

Products: Abaqus/Design

Type: Model data

Level: Model

References:

- *Direct Design Sensitivity Analysis*
- **PARAMETER*

There are no parameters associated with this option.

Data lines to specify design parameters:

First line:

1. List of parameter names chosen from those specified on the **PARAMETER* option. The parameter names associated with this option must be chosen such that they are unique when interpreted in a case insensitive manner.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

*DESIGN RESPONSE: Specify responses for design sensitivity analysis.

This option is used to write the sensitivities in a design sensitivity analysis. The use of this option depends on the type of design sensitivity analysis: direct or adjoint.

For a direct design sensitivity analysis, this option is used to write the sensitivities of contact, element, nodal, and/or eigenmode responses to the output database. The **CONTACT RESPONSE*, **ELEMENT RESPONSE*, and/or **NODE RESPONSE* options can be used in conjunction with this option in a direct design sensitivity analysis.

For an adjoint sensitivity analysis, this option is used to write the design response and sensitivities of global, element, or nodal responses to the output database. The **GLOBAL RESPONSE*, **ELEMENT RESPONSE*, or **NODE RESPONSE* option must be used in conjunction with this option in an adjoint design sensitivity analysis.

Products: Abaqus/Standard Abaqus/Design

Type: History data

Level: Step

References:

- *Direct Design Sensitivity Analysis*
- *Adjoint Design Sensitivity Analysis*
- **CONTACT RESPONSE*
- **ELEMENT RESPONSE*
- **NODE RESPONSE*
- **GLOBAL RESPONSE*

Specifying design responses for a direct design sensitivity analysis

Optional parameters:

FREQUENCY

Set this parameter equal to the output frequency of the response sensitivities. The output will always be written to the output database at the last increment. If this parameter is omitted, output will be written at every increment of the analysis. Set FREQUENCY=0 to suppress output of the response sensitivities. This parameter also controls the frequency of the sensitivity calculations for the total DSA formulation.

MODE LIST

Include this parameter to indicate that a list of eigenmodes for which sensitivities are desired will be listed on the data lines. This parameter is valid only in a **FREQUENCY* procedure.

Data lines to list desired eigenmodes if the MODE LIST parameter is included:

First line:

1. Specify a list of desired eigenmodes.

Repeat this data line as often as necessary to list all desired eigenmodes.

Specifying design responses for an adjoint design sensitivity analysis

Required parameters:

NAME

Set this parameter equal to the label that will be used to refer to this design response on the output database. Design response names in the same input file must be unique.

Optional parameters:

TIME MARKS

This parameter is relevant only in a transient dynamic (**DYNAMIC*) analysis.

Set TIME MARKS=YES (default) to write the design response and corresponding sensitivities to the output database at the exact times dictated by the TIME POINTS parameter.

Set TIME MARKS=NO to write the design response and corresponding sensitivities to the output database at the last increment of the step.

TIME POINTS

This parameter is relevant only in a transient dynamic (**DYNAMIC*) analysis.

Set this parameter equal to the name of the **TIME POINTS* option that defines the time point at which the design response and corresponding sensitivities are to be written. The time points specification can contain only a single time point.

There are no data lines associated with this option.

*DETONATION POINT: Define detonation points for a JWL explosive equation of state.

This option is used to define detonation points for a JWL explosive equation of state. It is required when the *EOS, TYPE=JWL option is used. The *DETONATION POINT option should appear immediately after the *EOS option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *EOS
- *Equation of State*

There are no parameters associated with this option.

Data lines to define detonation points:

First line:

1. Coordinate 1 of detonation point.
2. Coordinate 2 of detonation point.
3. Coordinate 3 of detonation point.
4. Detonation delay time (total time, as defined in *Conventions*). The default is 0.

Repeat this data line as often as necessary to define each detonation point.

*DFLOW: Specify distributed seepage flows for consolidation analysis.

This option is used to input seepage flows (pore fluid velocities normal to surfaces of the model) in consolidation problems.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- *Pore Fluid Flow*
- *DFLOW*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the **AMPLITUDE* curve that defines the magnitude of the seepage flow during the step. If this parameter is omitted for uniform seepage types, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (see *Defining an Analysis*). Amplitude references are ignored for flows defined in user subroutine *DFLOW*.

OP

Set OP=MOD (default) for existing **DFLOW*s to remain, with this option modifying existing flows or defining additional flows.

Set OP=NEW if all existing **DFLOW*s applied to the model should be removed. New flows can be defined.

Data lines to define uniform seepage:

First line:

1. Element number or element set label.
2. Distributed seepage type label (see *About the Element Library*).
3. Reference seepage magnitude. (Units of LT^{-1}.) The seepage magnitude is the pore fluid effective velocity crossing the surface at this point in an outward direction.

Repeat this data line as often as necessary to define uniform seepage for various elements or element sets.

Data lines to define nonuniform seepage:

First line:

1. Element number or element set label.
2. Distributed seepage type label (see *About the Element Library*).
3. Seepage magnitude (optional). If given, this value is passed into user subroutine *DFLOW* in the variable used to define the seepage magnitude.

Nonuniform seepage magnitudes are defined via user subroutine *DFLOW*.

Repeat this data line as often as necessary to define nonuniform seepage for various elements or element sets.

*DFLUX: Specify distributed fluxes in computational fluid dynamics and mass diffusion analyses and specify distributed fluxes and nonuniform concentrated fluxes in heat transfer analyses.

This option is used to apply distributed fluxes in fully coupled thermal-stress analysis. In Abaqus/Standard it is also used for heat transfer, coupled thermal-electrical, and mass diffusion analyses.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- *Thermal Loads*
- *DFLUX*
- *UMDFLUX*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the magnitude of the distributed fluxes during the step (*Amplitude Curves*).

If this parameter is omitted for uniform flux types in an Abaqus/Standard analysis, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (*Defining an Analysis*). If this parameter is omitted in an Abaqus/Explicit analysis, the reference magnitude is applied immediately at the beginning of the step.

In Abaqus/Standard for nonuniform distributed fluxes of type BFNU and S*n*NU the flux magnitude is defined in user subroutine *DFLUX* and AMPLITUDE references are ignored. In Abaqus/Explicit for nonuniform distributed fluxes of type BFNU and S*n*NU the flux magnitude is defined in user subroutine *VDFLUX* and AMPLITUDE references are passed in the user subroutine.

In Abaqus/Standard for nonuniform concentrated fluxes of type MBFNU the flux magnitude is defined in user subroutine *UMDFLUX* and AMPLITUDE references are passed in the user subroutine.

OP

Set OP=MOD (default) for existing **DFLUX*s to remain, with this option modifying existing fluxes or defining additional fluxes.

Set OP=NEW if all existing **DFLUX*s applied to the model should be removed.

Data lines to define a distributed flux:

First line:

1. Element number or element set label.
2. Distributed flux type label (see *About the Element Library*).
3. Reference flux magnitude (units of $JT^{-1}L^{-2}$ for surface fluxes and $JT^{-1}L^{-3}$ for body fluxes). It is needed for uniform fluxes only. If this value is given for nonuniform fluxes, it will be passed into user subroutine *DFLUX*, where the actual flux magnitude is defined.

In heat transfer analysis the units are $JT^{-1}L^{-2}$ for surface fluxes and $JT^{-1}L^{-3}$ for body fluxes. In mass diffusion analysis the units are PLT^{-1} for surface fluxes and PT^{-1} for body fluxes.

Repeat this data line as often as necessary to define distributed fluxes for different element surfaces.

Data lines to define a nonuniform concentrated flux inside the user subroutine UMDFLUX:

First line:

1. Element set label.
2. Distributed flux type label (see *About the Element Library*).
3. Leave this entry blank (any value entered here will be ignored).
4. Table collection name or blank if no table collection is used.

Repeat this data line as often as necessary to define distributed fluxes for different element surfaces.

*DIAGNOSTICS: Control diagnostic messages.

This option is used to request detailed diagnostic output or to cancel specific diagnostic checks. By default, short summaries of diagnostic checks are output if problems are detected during an analysis. In Abaqus/Explicit the diagnostics messages are written to the status (`.sta`) file or to the message (`.msg`) file. In Abaqus/Standard the diagnostic messages are written to the data (`.dat`) file.

For a multistep analysis in Abaqus/Explicit, all parameter values remain the same during the analysis until they are redefined explicitly in the beginning of the next step.

This option cannot appear more than once in the model definition in Abaqus/Standard or in each step definition in Abaqus/Explicit.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data in Abaqus/Standard; History data in Abaqus/Explicit

Level: Model in Abaqus/Standard; Step in Abaqus/Explicit

References:

- *Explicit Dynamic Analysis*
- *Output and Diagnostics for ALE Adaptive Meshing in Abaqus/Explicit*
- *Contact Diagnostics in an Abaqus/Explicit Analysis*
- *Linear Elastic Behavior*
- *Hyperelastic Behavior of Rubberlike Materials*
- **CONTACT CONTROLS*
- *Computational Cost for Constraint Solution in Abaqus/Explicit*

Defining diagnostics in an Abaqus/Explicit analysis

Optional parameters:

ADAPTIVE MESH

Adaptive mesh information is written to the message (`.msg`) file for each adaptive mesh domain in the problem.

Set ADAPTIVE MESH=STEP SUMMARY (default) to obtain a summary at the end of the step. The summary gives the average number of advection sweeps per adaptive mesh increment and the average, maximum, and minimum percentages of nodes moved during the step.

Set ADAPTIVE MESH=SUMMARY to obtain a summary for each adaptive mesh increment. The summary gives the number of mesh sweeps, the average percentage of nodes moved during those mesh sweeps, and the number of advection sweeps performed during the adaptive mesh increment. In addition to this information, the STEP SUMMARY information will be written at the end of each step.

Set ADAPTIVE MESH=DETAIL to obtain detailed information about each adaptive mesh increment. The detailed report gives the number of mesh sweeps performed; the minimum, average, and maximum percentage of nodes moved during those mesh sweeps; the number of advection sweeps performed; the mass and momentum before and after advection; and the percentage volume change

during the adaptive mesh increment. In addition to this information, the STEP SUMMARY information will be written at the end of each step.

Set ADAPTIVE MESH=OFF to suppress all diagnostic messages about adaptive meshing.

CONTACT INITIAL OVERCLOSURE

Set CONTACT INITIAL OVERCLOSURE=DETAIL to write all the initial displacements required to resolve initial overclosures to the message (`.msg`) file and a summary of the maximum initial overclosure for each contact pair to the status (`.sta`) file.

Set CONTACT INITIAL OVERCLOSURE=SUMMARY (default) to obtain only a summary of the maximum initial overclosure for each contact pair in the status (`.sta`) file.

CRITICAL ELEMENTS

Set this parameter equal to the number of critical elements (elements having the smallest stable time increment) written to the output database diagnostic information. The default is 10.

CUTOFF RATIO

Set this parameter equal to the cutoff ratio of deformation speed versus wave speed (the default is 1.0). If the maximum ratio calculated is greater than this value, the analysis ends with an error message. The cutoff check is not applied to a model that has an equation of state material or a user-defined material.

DEEP PENETRATION FACTOR

Set this parameter equal to the fraction of the typical element face dimension in the general contact domain used to detect excessively deep penetrations (the default is 0.5). If during node-to-face contact the penetration of a node into its tracked face exceeds the deep penetration factor times the typical element face dimension in the general contact domain, a diagnostic message is issued. The deep penetration check does not apply to contact penetrations detected by the contact pair algorithm.

DEFORMATION SPEED CHECK

Set DEFORMATION SPEED CHECK=SUMMARY (default) to print messages for only the element with the greatest deformation speed to wave speed ratio in the model. This information is output to the status (`.sta`) file.

Set DEFORMATION SPEED CHECK=DETAIL to print messages for all elements with relatively large deformation speed. This information is output to the message (`.msg`) file.

Set DEFORMATION SPEED CHECK=OFF to suppress the deformation speed check.

DETECT CROSSED SURFACES

This parameter applies only to general contact.

Set DETECT CROSSED SURFACES=ON (default) to issue warning messages for instances of adjacent secondary surfaces being on opposite sides of main surfaces in the initial configuration.

Set DETECT CROSSED SURFACES=OFF to suppress this diagnostic.

LARGE IMPLICIT SYSTEM

Set LARGE IMPLICIT SYSTEM=ERROR (default) to issue an error message when a large implicit constraint system is detected that will cause severe performance degradation.

Set LARGE IMPLICIT SYSTEM=WARNING to issue a warning message when a large implicit constraint system is detected that will cause severe performance degradation.

PLASTICITY

Set PLASTICITY=SUMMARY (default) to obtain a summary of the total number of material points at which the plasticity algorithms have not converged. This information will be printed only at the first occurrence in the status (`.sta`) file.

Set PLASTICITY=DETAIL to obtain detailed information about the elements at which the plasticity algorithms have not converged. This information will be printed in the message (`.msg`) file. This request might cause the analysis to run for a longer time. It is currently available only for Mises plasticity.

Set PLASTICITY=OFF to suppress all the diagnostic messages about the plasticity algorithms.

WARNING RATIO

Set this parameter equal to the warning ratio of deformation speed versus wave speed (the default ratio is 0.3). If the ratio calculated in an element is greater than this value, a warning message will be printed to the status (`.sta`) file or the message (`.msg`) file.

WARPED SURFACE

Set WARPED SURFACE=SUMMARY (default) to obtain a warning message in the status (`.sta`) file when a surface is first considered to contain at least one highly warped facet.

Set WARPED SURFACE=DETAIL to have detailed warning messages also output to the message (`.msg`) file.

Set WARPED SURFACE=OFF to suppress all warnings about warped surfaces.

There are no data lines associated with this option.

Defining diagnostics in an Abaqus/Standard analysis

Optional parameters:

NONHYBRID INCOMPRESSIBLE

If a model has nearly incompressible elastic or hyperelastic materials with an effective initial Poisson's ratio greater than 0.495 (that is, the ratio of the initial bulk modulus to the initial shear modulus is greater than 100) used in nonhybrid continuum elements (except plane stress elements), error messages are written to the data (`.dat`) file during preprocessing.

Set NONHYBRID INCOMPRESSIBLE=WARNING to replace the error messages with corresponding warning messages.

OLD CONTACT CONTROLS

If the model has "old" contact controls defined in the input file (such as MAXCHP and UERRMX), error messages are written to the data (`.dat`) file during preprocessing.

Set OLD CONTACT CONTROLS=WARNING to replace the error messages with corresponding warning messages.

For more information, see "Documentation of Abaqus/Standard contact controls de-emphasized" in the Dassault Systèmes Knowledge Base at *https://support.3ds.com/knowledge-base/*.

There are no data lines associated with this option.

*DIELECTRIC: Specify dielectric material properties.

This option is used to define the dielectric property of a fully constrained material for use in coupled piezoelectric analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Piezoelectric Behavior*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the dielectric property. If this parameter is omitted, the dielectric property is assumed not to depend on any field variables but may still depend on temperature. See *Material Data Definition* for more information.

TYPE

Set TYPE=ISO (default) for isotropic behavior. Set TYPE=ORTHO for orthotropic behavior. Set TYPE=ANISO for fully anisotropic behavior.

Data lines to define isotropic behavior (TYPE=ISO):

First line:

1. Dielectric constant. (Units of $C\varphi^{-1}L^{-1}$.)
2. Temperature, θ.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dielectric property as a function of temperature and other predefined field variables.

Data lines to define orthotropic behavior (TYPE=ORTHO):

First line:

1. $D_{11}^{\varphi(\varepsilon)}$. (Units of $C\varphi^{-1}L^{-1}$.)
2. $D_{22}^{\varphi(\varepsilon)}$.
3. $D_{33}^{\varphi(\varepsilon)}$.
4. Temperature, θ.

5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dielectric property as a function of temperature and other predefined field variables.

Data lines to define anisotropic behavior (TYPE=ANISO):

First line:

1. $D_{11}^{\varphi(\varepsilon)}$. (Units of $C\varphi^{-1}L^{-1}$.)
2. $D_{12}^{\varphi(\varepsilon)}$.
3. $D_{22}^{\varphi(\varepsilon)}$.
4. $D_{13}^{\varphi(\varepsilon)}$.
5. $D_{23}^{\varphi(\varepsilon)}$.
6. $D_{33}^{\varphi(\varepsilon)}$.
7. Temperature, θ.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dielectric property as a function of temperature and other predefined field variables.

*DIFFUSIVITY: Specify mass diffusivity or slurry concentration diffusivity.

This option is used to define the mass diffusivity of a material diffusing through a base material in a mass-diffusion analysis (*Mass Diffusion Analysis*) or to define the diffusivity of particles in a slurry flow during a coupled pore fluid flow and stress analysis (*Coupled Pore Fluid Diffusion and Stress Analysis*).

If the **DIFFUSIVITY* option is used to define mass diffusivity in a mass diffusion analysis, it must be used in conjunction with the **SOLUBILITY* option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Diffusivity*
- **SOLUBILITY*

Defining mass diffusivity of a material diffusing through a base material in a mass-diffusion analysis

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of diffusivity. If this parameter is omitted, the diffusivity is assumed not to depend on any field variables but might still depend on concentration and temperature. See *Material Data Definition* for more information.

LAW

Set LAW=GENERAL (default) to choose general mass diffusion behavior.

Set LAW=FICK to choose Fick's diffusion law.

LAW=FICK and the **KAPPA*, TYPE=TEMP option are mutually exclusive.

TYPE

Set TYPE=ISO (default) to define isotropic diffusivity. Set TYPE=ORTHO to define orthotropic diffusivity. Set TYPE=ANISO to define fully anisotropic diffusivity.

Data lines to define isotropic mass diffusivity (TYPE=ISO):

First line:

1. Diffusivity, D. (Units of L^2T^{-1}.)
2. Concentration, c.
3. Temperature, θ.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the diffusivity as a function of concentration, temperature, and other predefined field variables.

Data lines to define mass orthotropic diffusivity (TYPE=ORTHO):

First line:

1. D_{11}. (Units of L^2T^{-1}.)
2. D_{22}.
3. D_{33}.
4. Concentration, c.
5. Temperature, θ.
6. First field variable.
7. Second field variable.
8. Third field variable

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the diffusivity as a function of concentration, temperature, and other predefined field variables.

Data lines to define anisotropic mass diffusivity (TYPE=ANISO):

First line:

1. D_{11}. (Units of L^2T^{-1}.)
2. D_{12}.
3. D_{22}.
4. D_{13}.
5. D_{23}.
6. D_{33}.
7. Concentration, c.
8. Temperature, θ.

Subsequent lines (only needed if the DEPENDENCIES parameter is used):

1. First field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the diffusivity as a function of concentration, temperature, and other predefined field variables.

Defining the diffusivity of particles in a slurry flow during a coupled pore fluid flow and stress analysis

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of diffusivity. If this parameter is omitted, the diffusivity is assumed not to depend on any field variables but might still depend on concentration and temperature. See *Material Data Definition* for more information.

SLURRY

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to define the diffusivity of the particles in a slurry in a geotechnical analysis that models slurry transport and placement. The diffusivity of the particles must be isotropic.

This parameter cannot be used in conjunction with the LAW and TYPE parameters.

TYPE

Set TYPE=ISO to define isotropic diffusivity.

Data lines to define the diffusivity of the particles in a slurry:

First line:

1. Diffusivity, D. (Units of L^2T^{-1}.)
2. Temperature, θ.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Eighth field variable.

Repeat this set of data lines as often as necessary to define the diffusivity as a function of temperature and other predefined field variables.

*DIRECT CYCLIC: Obtain the stabilized cyclic response of a structure directly.

This option is used to provide a direct cyclic procedure for nonlinear, non-isothermal quasi-static analysis in Abaqus/Standard. It can also be used to predict progressive damage and failure for ductile bulk materials and/or to predict delamination/debonding growth at the interfaces in laminated composites in a low-cycle fatigue analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Direct Cyclic Analysis*
- *Low-Cycle Fatigue Analysis Using the Direct Cyclic Approach*
- **TIME POINTS*

Optional parameters:

CETOL

Set this parameter equal to the maximum difference in the creep strain increment calculated from the creep strain rates based on conditions at the beginning and on conditions at the end of the increment, thus controlling the time integration accuracy of the creep integration.

This parameter can be used in conjunction with the **TIME POINTS* option. In this case Abaqus/Standard will ensure the response will also be evaluated at each time point specified on the **TIME POINTS* option.

If both this parameter and the DELTMX parameter are omitted, fixed time stepping will be used, with a constant time increment equal to the initial time increment or by following precisely the time points specified on the **TIME POINTS* option.

CONTINUE

Set CONTINUE=YES to specify that the current **DIRECT CYCLIC* step is a continuation of the previous direct cyclic step. The displacement solution in the Fourier series obtained in the previous **DIRECT CYCLIC* step is then used as the starting values for the current step.

Set CONTINUE=NO (default) to reset all the displacement Fourier coefficients to zero, thus allowing application of cyclic loading conditions that are very different from those in the previous direct cyclic step.

DELTMX

Set this parameter equal to the maximum temperature change to be allowed in an increment during a direct cyclic analysis. Abaqus/Standard will restrict the time increment to ensure that this value will not be exceeded at any node during any increment of the step.

This parameter can be used in conjunction with the **TIME POINTS* option. In this case Abaqus/Standard will ensure the response will also be evaluated at each time point specified on the **TIME POINTS* option.

If both this parameter and the CETOL parameter are omitted, fixed time stepping will be used, with a constant time increment equal to the initial time increment or by following precisely the time points specified on the *TIME POINTS option.

FATIGUE

Include this parameter to perform a low-cycle fatigue analysis using a direct cyclic approach in conjunction with the damage extrapolation technique. Multiple cycles can be included in a single direct cyclic analysis. The analysis models progressive damage and failure on constitutive points in the bulk materials based on a continuum damage approach. It can also be used to model delamination/debonding growth at the interfaces in laminated composites.

TIME POINTS

Set this parameter equal to the name of the *TIME POINTS option that defines the time points at which the response of the structure will be evaluated.

Data line to control incrementation and Fourier representation in a direct cyclic analysis without the FATIGUE parameter:

First (and only) line:

1. Initial time increment. If this entry is omitted, a default value of 0.1 times the single loading cycle period is assumed. If automatic incrementation is used, this should be a reasonable suggestion for the initial increment size and will be adjusted as necessary. If direct incrementation is used, this entry will be used as the constant time incrementation or will be ignored if the *TIME POINTS option is specified.
2. Time of a single loading cycle.
3. Minimum time increment allowed. This entry is used only if the CETOL or DELTMX parameter is specified. If this entry is omitted, a default value of the smaller of the suggested initial time increment or 10^{-5} times the single loading cycle period is assumed.
4. Maximum time increment allowed. This entry is used only if the CETOL or DELTMX parameter is specified. If this entry is omitted, the upper limit is equal to 0.1 times the single loading cycle period.
5. Initial number of terms in the Fourier series. The value must be greater than 0 and less than 500. It cannot be greater than half of the time of a single loading cycle divided by the initial time increment. If the *TIME POINTS option is used, the number of terms in the Fourier series must be less than half of the number of time points specified. Abaqus/Standard will automatically adjust the number of Fourier terms used in the analysis if such a condition is not satisfied. The default is 11.
6. Maximum number of terms in the Fourier series. It must be greater than 0 and less than 500. The default is 25.
7. Increment in number of terms in the Fourier series. The default is 5.
8. Maximum number of iterations allowed in a step. The default is 200.

Data lines for a low-cycle fatigue analysis using the direct cyclic approach:

First line:

1. Initial time increment. If this entry is omitted, a default value of 0.1 times the single loading cycle period is assumed. If automatic incrementation is used, this should be a reasonable suggestion for the initial increment size and will be adjusted as necessary. If direct incrementation is used, this entry will be used as the constant time incrementation or will be ignored if the *TIME POINTS option is specified.
2. Time of a single loading cycle.

3. Minimum time increment allowed. This entry is used only if the CETOL or DELTMX parameter is specified. If this entry is omitted, a default value of the smaller of the suggested initial time increment or 10^{-5} times the single loading cycle period is assumed.
4. Maximum time increment allowed. This entry is used only if the CETOL or DELTMX parameter is specified. If this entry is omitted, the upper limit is equal to 0.1 times the single loading cycle period.
5. Initial number of terms in the Fourier series. The value must be greater than 0 and less than 500. It cannot be greater than half of the time of a single loading cycle divided by the initial time increment. If the **TIME POINTS* option is used, the number of terms in the Fourier series must be less than half of the number of time points specified. Abaqus/Standard will automatically adjust the number of Fourier terms used in the analysis if such a condition is not satisfied. The default is 11.
6. Maximum number of terms in the Fourier series. It must be greater than 0 and less than 500. The default is 25.
7. Increment in number of terms in the Fourier series. The default is 5.
8. Maximum number of iterations allowed in a step. The default is 200.

Second line:

1. Minimum increment in number of cycles over which the damage is extrapolated forward. It must be greater than 0, and the default is 100. This value is most relevant for progressive damage and failure in bulk materials.
2. Maximum increment in number of cycles over which the damage is extrapolated forward. It must be greater than 0, and the default is 1000. This value is most relevant for progressive damage and failure in bulk materials.
3. Total number of cycles allowed in a step. If this entry is zero or not specified, the default value is equal to one plus half of the maximum increment in number of cycles over which the damage is extrapolated.
4. Damage extrapolation tolerance. The maximum extrapolated damage increment will be limited by this value. The default is 1.0.
5. Cycle increment–based tolerance for the least number of cycles to fracture an element. The default is 0. The damage-based tolerance takes precedence over the cycle increment–based tolerance if both are specified.
6. Damage-based tolerance for the least number of cycles to fracture an element. The damage-based tolerance takes precedence over the cycle increment–based tolerance if both are specified.

*DISCRETE ELASTICITY: Specify effective elastic material properties for discrete particles.

This option is used to define effective elastic moduli for PD3D elements for Hertz normal contact.

Products: Abaqus/Explicit

Type: Model data

Level: Model

Abaqus/CAE: Unsupported

References:

• *Discrete Element Method*

There are no parameters associated with this option.

Data lines to define discrete elasticity:

First (and only) line:

1. Young's modulus, E.
2. Poisson's ratio, ν.

*DISCRETE SECTION: Specify element properties for discrete elements.

This option is used to define properties of discrete elements.

Products: Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

• *Discrete Element Method*

Required parameters:

DENSITY

This parameter applies only to DEM analyses.

Set this parameter equal to a numerical value or to the name of a distribution (see *Distribution Definition*) to be used with these elements.

ELSET

Set this parameter equal to the name of the element set containing the elements for which the section is being defined.

FLUID BEHAVIOR

This parameter applies only to LKM analyses.

Set this parameter equal to the name of the fluid behavior of the gas associated with the lumped gas particles.

SHAPE

This parameter is used to specify the shape of the discrete element.

Set this parameter equal to SPHERE (default).

Optional parameters:

ALPHA

Set this parameter equal to the value of the mass proportional damping factor for discrete elements. The default value is 0.0.

CONTROLS

Set this parameter equal to the name of a section controls definition (see *Section Controls*) to be used to specify the size of the particle tracking box.

Data line for discrete elements:

First (and only) data line:

1. Enter the name of the distribution table or the probability density function for the radius of the discrete elements or the value of the radius. Leave blank when the FLUID BEHAVIOR parameter is included.

*DISPLAY BODY: Define a part instance that will be used for display only.

This option is used to specify that a part instance should be used for display purposes only and should not take part in the analysis. This option must be used in conjunction with the **ASSEMBLY* and **INSTANCE* options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Assembly

Abaqus/CAE: Interaction module

References:

• *Display Body Definition*

Required parameters:

INSTANCE

Set this parameter to the name of the part instance that is to be considered a display body.

Data line to specify the reference nodes (optional; if no data line is given, the display body will remain stationary during the analysis):

First (and only) line:

1. Node number of the first reference node.
2. Node number of the second reference node (optional).
3. Node number of the third reference node (optional; required if a node number for the second reference node is given).

*DISTRIBUTING: Define a distributing coupling constraint.

This option is used to define a distributing coupling constraint. It must be used in conjunction with the *COUPLING option to define the reference node and coupling nodes.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Interaction module

References:

- *Coupling Constraints*
- *COUPLING

Optional parameters:

COUPLING

This parameter controls whether the rotation of the cloud nodes participates in translational coupling constraints for the reference node.

Set COUPLING=CONTINUUM (default) to couple the displacement of the reference node to the displacement (but not rotation) of the cloud nodes within the influence radius.

Set COUPLING=STRUCTURAL to couple the displacement of the reference node to the displacement and rotation of the cloud nodes within the influence radius. This parameter value is available only in three-dimensional analyses.

FREE NODES

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the treatment method for nodes that are not part of any user-defined element but are part of cloud nodes. The default if this parameter is omitted is FREE NODES=ALLOW.

Set FREE NODES=ALLOW to allow the unattached nodes.

Set FREE NODES=ERROR to treat unattached nodes as a user error.

Set FREE NODES=REMOVE to remove the unattached nodes.

ROTATIONAL COUPLING

This parameter controls whether the rotational constraint of the reference node should include the rotations of the cloud nodes in addition to the displacements of the cloud nodes.

Set ROTATIONAL COUPLING=AUTOMATIC (default) to allow Abaqus to couple the rotation of the reference node to the displacement and, if needed, the rotation of the cloud nodes within the influence radius.

Set ROTATIONAL COUPLING=CONTINUUM to couple the rotation of the reference node to the displacement (but not rotation) of the cloud nodes within the influence radius.

Set ROTATIONAL COUPLING=STRUCTURAL to couple the rotation of the reference node to the displacement and rotation of the cloud nodes within the influence radius.

SINGLE FACET

This parameter applies only to Abaqus/Explicit analyses.

This parameter controls whether a distributing coupling that is attached only to a single facet of an element should be converted to a tie constraint.

Set SINGLE FACET=TIE CONVERT (default) to convert a distributing coupling that is attached to only one facet of the element to a tie constraint.

Set SINGLE FACET=DISTRIBUTING to retain the distributing coupling constraint.

WEIGHTING METHOD

Set this parameter equal to the weighting method to modify the default weight distribution at the coupling nodes. The default if this parameter is omitted is WEIGHTING METHOD=UNIFORM.

Set WEIGHTING METHOD=UNIFORM to select a uniform weight distribution equal to 1.0.

Set WEIGHTING METHOD=LINEAR to select a linear decreasing weight distribution with distance from the reference node.

Set WEIGHTING METHOD=QUADRATIC to select a quadratic polynomial decreasing weight distribution with distance from the reference node.

Set WEIGHTING METHOD=CUBIC to select a cubic polynomial monotonic decreasing weight distribution with distance from the reference node.

Data lines to specify the degrees of freedom to be constrained:

First line:

1. First degree of freedom constrained. See *Conventions* for a definition of the numbering of degrees of freedom in Abaqus. If this field is left blank, all degrees of freedom will be constrained.
2. Last degree of freedom constrained. If this field is left blank, the degree of freedom specified in the first field will be the only one constrained.

Only rotational degrees of freedom can be released. All available translational degrees of freedom are constrained. If the user specifies one or more rotation degrees of freedom but not all available translational degrees of freedom, Abaqus will issue a warning message and add all available translational degrees of freedom to the constraint.

Repeat this data line as often as necessary to specify constraints for different degrees of freedom. When the ORIENTATION parameter is specified on the associated **COUPLING* option, the degrees of freedom are in the referenced local system in the initial configuration; otherwise, they are in the global system. In either case these directions will rotate with the reference node in large-displacement analyses (when the NLGEOM parameter is included on the **STEP* option).

*DISTRIBUTING COUPLING: Specify nodes and weighting for distributing coupling elements.

This option is used to define the set of nodes to which forces and mass are distributed according to a specified weighting and to specify the mass of the associated distributing coupling element. The preferred method for defining a distributing constraint is the **COUPLING* option used in conjunction with the **DISTRIBUTING* option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Unsupported; this option has been superseded by coupling constraints used in conjunction with the distributing option.

References:

- *Distributing Coupling Elements*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the distributing coupling elements that interact with the coupling nodes. This element set can contain more than one element, although this would not be a typical case.

Optional parameters:

FREE NODES

Set this parameter equal to the appropriate treatment for nodes that are not part of any user-defined element but are part of cloud nodes. The default if this parameter is omitted is FREE NODES=ALLOW.

Set FREE NODES=ALLOW to allow the unattached nodes.

Set FREE NODES=ERROR to treat unattached nodes as a user error.

Set FREE NODES=REMOVE to remove the unattached nodes.

Data lines to specify coupling nodes and assign weight factors:

First line:

1. Coupling node number or node set label.
2. Weight factor for the coupling node or for the nodes of the coupling node set.

Repeat this data line as often as necessary. A minimum of two coupling nodes must be specified for each distributing coupling definition.

*DISTRIBUTION: Define spatial distributions.

This option is used to define a spatial distribution. Spatially varying field data can be defined either on the data lines or using the *EXTERNAL FIELD option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly, Model

Abaqus/CAE: Property module

References:

- *Distribution Definition*
- *CONTACT CLEARANCE
- *DENSITY
- *DISTRIBUTION TABLE
- *ELASTIC
- *EXPANSION
- *MEMBRANE SECTION
- *ORIENTATION
- *SHELL GENERAL SECTION
- *SHELL SECTION

Required parameters:

LOCATION

Set LOCATION=ELEMENT to define a distribution on elements.

Set LOCATION=NODE to define a distribution on nodes.

Set LOCATION=FACE to define a distribution on element faces.

Set LOCATION=NONE to define a distribution used with a pressure-volume fluid boundary condition.

NAME

Set this parameter equal to a label that will be used to refer to the distribution.

TABLE

Set this parameter equal to the distribution table that defines the format of the data given on the data lines.

Optional parameters:

DESIGN VARIABLE

Include this parameter to specify a spatial distribution of design variables in an adjoint design sensitivity analysis.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

Data lines to define a distribution of the coordinates of points a and b used to define a local coordinate system:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. X-coordinate of point a.
3. Y-coordinate of point a.
4. Z-coordinate of point a.
5. X-coordinate of point b.
6. Y-coordinate of point b.
7. Z-coordinate of point b.

Repeat this data line as often as necessary to define the data for element numbers or element sets. The data can also be read directly from an output database file in SIM format (`.sim`) using the **EXTERNAL FIELD* option. The orientation output variable in the output database file is converted automatically to the coordinates of a and b.

Data lines to define a distribution of additional rotation angles used to define a local coordinate system:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. Angle (in degrees).

Repeat this data line as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of shell, membrane, or composite layer thickness:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. Shell, membrane, or composite layer thickness.

Repeat this data line as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of radii for beams with a solid circular section:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. Beam radius.

Repeat this data line as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of shell offset:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. Shell offset.

Repeat this data line as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of general section stiffnesses:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent uses of this data line.
2. D_{11}.
3. D_{12}.
4. D_{22}.
5. D_{13}.
6. D_{23}.
7. D_{33}.
8. D_{14}.

Second line:

1. D_{24}.
2. D_{34}.
3. D_{44}.
4. D_{15}.
5. D_{25}.
6. D_{35}.
7. D_{45}.
8. D_{55}.

Third line:

1. D_{16}.
2. D_{26}.
3. D_{36}.
4. D_{46}.
5. D_{56}.
6. D_{66}.

Repeat this set of data lines as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of initial contact clearances:

First line:

1. Node number or node set. Default data are not allowed.
2. Initial clearance.

Repeat this data line as often as necessary.

Data lines to define a distribution of isotropic elastic moduli:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent uses of this data line.
2. E.
3. ν.

Repeat this data line as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of orthotropic elastic moduli:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent uses of this data line.
2. D_{1111}.
3. D_{1122}.
4. D_{2222}.
5. D_{1133}.
6. D_{2233}.
7. D_{3333}.
8. D_{1212}.

Second line:

1. D_{1313}.
2. D_{2323}.

Repeat this set of data lines as often as necessary to define data for element numbers or element sets.

Data lines to define a distribution of orthotropic elastic moduli using engineering constants:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent uses of this data line.
2. E_1.
3. E_2.

4. E_3.
5. ν_{12}.
6. ν_{13}.
7. ν_{23}.
8. G_{12}.

Second line:

1. G_{13}.
2. G_{23}.

Repeat this set of data lines as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of orthotropic elastic moduli in plane stress:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. E_1.
3. E_2.
4. ν_{12}.
5. G_{12}.
6. G_{13}. This shear modulus is needed to define transverse shear behavior in shells.
7. G_{23}. This shear modulus is needed to define transverse shear behavior in shells.

Repeat this data line as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of anisotropic elastic moduli:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent uses of this data line.
2. D_{1111}.
3. D_{1122}.
4. D_{2222}.
5. D_{1133}.
6. D_{2233}.
7. D_{3333}.
8. D_{1112}.

Second line:

1. D_{2212}.
2. D_{3312}.
3. D_{1212}.
4. D_{1113}.

5. D_{2213}.
6. D_{3313}.
7. D_{1213}.
8. D_{1313}.

Third line:

1. D_{1123}.
2. D_{2223}.
3. D_{3323}.
4. D_{1223}.
5. D_{1323}.
6. D_{2323}.

Repeat this set of data lines as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of mass density:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. Density.

Repeat this data line as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of isotropic thermal expansion:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. α.

Repeat this data line as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of orthotropic thermal expansion:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. α_{11}.
3. α_{22}.
4. α_{33}. (Not used for plane stress case.)

Repeat this data line as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of anisotropic thermal expansion:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. α_{11}.
3. α_{22}.
4. α_{33}. (Not used for plane stress case.)
5. α_{12}.
6. α_{13}.
7. α_{23}.

Repeat this data line as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of the volume fraction for a multiscale material:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. v_f.

Repeat this data line as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of the aspect ratio for a multiscale material:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. AR.

Repeat this data line as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of the second-order orientation tensor for a multiscale material:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. a_{11}.
3. a_{22}.
4. a_{33}.
5. a_{12}.
6. a_{13}.
7. a_{23}.

Repeat this data line as often as necessary to define the data for element numbers or element sets. Orientation tensor data can also be read directly from an output database file in SIM format (`.sim`) using the **EXTERNAL FIELD* option.

Data lines to define a distribution of scale factors and a quantity related to the scale factor derivatives for one of the following material attributes: mass, stiffness, and design stress; or to define a spatial distribution of scale factors for the material thermal conductivity:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. Scale factor.
3. Derivative of the scale factor divided by the scale factor.

Repeat this data line as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of adjustments (displacements) to nodal coordinates:

First line:

1. Blank space to define default data for the first use of this data line. Node number or node set for subsequent data lines.
2. *X*-displacement.
3. *Y*-displacement.
4. *Z*-displacement.

Repeat this data line as often as necessary to define the data for node numbers or node sets.

Data lines to define a distribution of local material directions (or fiber directions) for an anisotropic material:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. *X*-component of the local material direction.
3. *Y*-component of the local material direction.
4. *Z*-component of the local material direction.

Repeat this data line as often as necessary to define the data for element numbers or element sets.

*DISTRIBUTION TABLE: Define a distribution table.

This option is used to define a distribution table that defines the format of the data given on the data lines for a spatial distribution.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Distribution Definition*
- **DISTRIBUTION*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the distribution table.

Data line to specify the distribution table label for shell or membrane thickness, beam radius for a circular cross-section beam, or initial contact clearance:

First (and only) line:

1. LENGTH.

Data line to specify the distribution table label for shell offset:

First (and only) line:

1. RATIO.

Data lines to specify the distribution table labels for shell stiffness:

First line:

1. SHELLSTIFF1.
2. SHELLSTIFF1.
3. SHELLSTIFF1.
4. SHELLSTIFF1.
5. SHELLSTIFF1.
6. SHELLSTIFF1.
7. SHELLSTIFF2.

Second line:

1. SHELLSTIFF2.
2. SHELLSTIFF2.
3. SHELLSTIFF3.
4. SHELLSTIFF2.
5. SHELLSTIFF2.
6. SHELLSTIFF2.
7. SHELLSTIFF3.
8. SHELLSTIFF3.

Third line:

1. SHELLSTIFF2.
2. SHELLSTIFF2.
3. SHELLSTIFF2.
4. SHELLSTIFF3.
5. SHELLSTIFF3.
6. SHELLSTIFF3.

Data line to specify the distribution table labels for the coordinates of points a and b used to define a local coordinate system:

First (and only) line:

1. COORD3D.
2. COORD3D.

Data line to specify the distribution table label for an additional rotation angle used to define a local coordinate system:

First (and only) line:

1. ANGLE.

Data line to specify the distribution table labels for isotropic elasticity:

First (and only) line:

1. MODULUS.
2. RATIO.

Data lines to specify the distribution table labels for orthotropic elasticity:

First line:

1. MODULUS.
2. MODULUS.
3. MODULUS.
4. MODULUS.
5. MODULUS.
6. MODULUS.
7. MODULUS.

Second line:

1. MODULUS.
2. MODULUS.

Data lines to specify the distribution table labels for orthotropic elasticity with engineering constants:

First line:

1. MODULUS.
2. MODULUS.
3. MODULUS.
4. RATIO.
5. RATIO.
6. RATIO.
7. MODULUS.

Second line:

1. MODULUS.
2. MODULUS.

Data line to specify the distribution table labels for orthotropic elasticity in plane stress:

First (and only) line:

1. MODULUS.
2. MODULUS.
3. RATIO.
4. MODULUS.
5. MODULUS.
6. MODULUS.

Data lines to specify the distribution table labels for anisotropic elasticity:

First line:

1. MODULUS.
2. MODULUS.
3. MODULUS.
4. MODULUS.
5. MODULUS.
6. MODULUS.
7. MODULUS.

Second line:

1. MODULUS.
2. MODULUS.
3. MODULUS.
4. MODULUS.
5. MODULUS.
6. MODULUS.
7. MODULUS.
8. MODULUS.

Third line:

1. MODULUS.
2. MODULUS.
3. MODULUS.
4. MODULUS.
5. MODULUS.
6. MODULUS.

Data line to specify the distribution table label for mass density:

First (and only) line:

1. DENSITY.

Data line to specify the distribution table label for isotropic thermal expansion:

First (and only) line:

1. EXPANSION.

Data line to specify the distribution table labels for orthotropic thermal expansion:

First (and only) line:

1. EXPANSION.
2. EXPANSION.
3. EXPANSION.

Data line to specify the distribution table labels for anisotropic thermal expansion:

First (and only) line:

1. EXPANSION.
2. EXPANSION.
3. EXPANSION.
4. EXPANSION.
5. EXPANSION.
6. EXPANSION.

Data line to specify the distribution table labels for the second-order orientation tensor:

First (and only) line:

1. ORITENS.

Data line to specify the distribution table labels for a scale factor and a quantity related to the scale factor derivatives for one of the following material attributes: mass, stiffness, or design stress:

First (and only) line:

1. RATIO.
2. RATIO.

Data line to specify the distribution table labels for a scale factor for material thermal conductivity:

First (and only) line:

1. RATIO.

Data line to specify the distribution table labels for adjusting (displacing) nodal coordinates:

First (and only) line:

1. DISP3D.

Data line to specify the distribution table labels for local material directions:

First (and only) line:

1. DIR3D.

*DLOAD: Specify distributed loads.

This option is used to prescribe distributed loading. It is also used to apply concentrated or distributed wind, wave, or buoyancy loading in an Abaqus/Aqua analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE Abaqus/Aqua

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- *Distributed Loads*
- *DLOAD*
- *Defining ALE Adaptive Mesh Domains in Abaqus/Explicit*
- *Analysis of Models that Exhibit Cyclic Symmetry*
- *Abaqus/Aqua Analysis*

Applying distributed loads

Required parameter for cyclic symmetry models in steady-state dynamics analyses:

CYCLIC MODE

Set this parameter equal to the cyclic symmetry mode number of loads that are applied in the current steady-state dynamics procedure.

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the variation of the load magnitude during the step.

If this parameter is omitted for uniform load types in an Abaqus/Standard analysis, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (see *Defining an Analysis*). If this parameter is omitted in an Abaqus/Explicit analysis, the reference magnitude is applied immediately at the beginning of the step.

Amplitude references are ignored for nonuniform loads given by user subroutine *DLOAD* in an Abaqus/Standard analysis. Amplitude references are passed into user subroutine *VDLOAD* in an Abaqus/Explicit analysis.

Only the load magnitude is changed with time. Quantities such as the direction of an applied gravity load and the fluid surface level in hydrostatic pressure loading are not changed.

CONSTANT RESULTANT

Set CONSTANT RESULTANT=NO (default) if surface traction vectors, edge traction vectors, or edge moments are to be integrated over the surface in the current configuration.

Set CONSTANT RESULTANT=YES if surface traction vectors, edge traction vectors, or edge moments are to be integrated over the surface in the reference configuration.

The CONSTANT RESULTANT parameter is valid only for uniform and nonuniform surface tractions and edge loads (including edge moments); it is ignored for all other load types.

FOLLOWER

Set FOLLOWER=YES (default) if a prescribed traction or shell-edge load is to rotate with the surface or shell edge in a large-displacement analysis (live load).

Set FOLLOWER=NO if a prescribed traction or edge load is to remain fixed in a large-displacement analysis (dead load).

The FOLLOWER parameter is valid only for traction and edge load labels TRVEC*n*, TRVEC, TRVEC*n*NU, TRVECNU, EDLD*n*, and EDLD*n*NU. It is ignored for all other load labels.

OP

Set OP=MOD (default) for existing **DLOAD*s to remain, with this option modifying existing distributed loads or defining additional distributed loads.

Set OP=NEW if all existing **DLOAD*s applied to the model should be removed. New distributed loads can be defined.

ORIENTATION

Set this parameter equal to the name given for the **ORIENTATION* option (*Orientations*) used to specify the local coordinates in which components of traction or shell-edge loads are specified.

The ORIENTATION parameter is valid only for traction and edge load labels TRSHR*n*, TRSHR, TRSHR*n*NU, TRSHRNU, TRVEC*n*, TRVEC, TRVEC*n*NU, TRVECNU, EDLD*n*, and EDLD*n*NU. It is ignored for all other load labels.

REF NODE

This parameter applies only to Abaqus/Explicit analyses and is relevant only for viscous and stagnation body force and pressure loads when the velocity at the reference node is used.

Set this parameter equal to either the node number of the reference node or the name of a node set containing the reference node. If the name of a node set is chosen, the node set must contain exactly one node. If this parameter is omitted, the reference velocity is assumed to be zero.

REGION TYPE

This parameter applies only to Abaqus/Explicit analyses.

This parameter is relevant only for pressure loads applied to the boundary of an adaptive mesh domain. If a distributed pressure load is applied to a surface in the interior of an adaptive mesh domain, the nodes on the surface will move with the material in all directions (they will be nonadaptive). Abaqus/Explicit will create a boundary region automatically on the surface subjected to the defined pressure load.

Set REGION TYPE=LAGRANGIAN (default) to apply the pressure to a Lagrangian boundary region. The edge of a Lagrangian boundary region will follow the material while allowing adaptive meshing along the edge and within the interior of the region.

Set REGION TYPE=SLIDING to apply the pressure load to a sliding boundary region. The edge of a sliding boundary region will slide over the material. Adaptive meshing will occur along the edge and in the interior of the region. Mesh constraints are typically applied on the edge of a sliding boundary region to fix it spatially.

Set REGION TYPE=EULERIAN to apply the pressure to an Eulerian boundary region. This option is used to create a boundary region across which material can flow. Mesh constraints must be used normal to an Eulerian boundary region to allow material to flow through the region. If no mesh

constraints are applied, an Eulerian boundary region will behave in the same way as a sliding boundary region.

Optional, mutually exclusive parameters for matrix generation and steady-state dynamics analyses (direct, modal, or subspace):

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the loading.

REAL

Include this parameter (default) to define the real (in-phase) part of the loading.

Data lines to define all distributed loads except those special cases described below:

First line:

1. Element number or element set label.
2. Distributed load type label (see *About the Element Library*).
3. Reference load magnitude, which can be modified by the use of the **AMPLITUDE* option. For nonuniform loads the magnitude must be defined in user subroutine *DLOAD* for Abaqus/Standard and *VDLOAD* for Abaqus/Explicit. If given, this value will be passed into the user subroutine in an Abaqus/Standard analysis.

Repeat this data line as often as necessary to define distributed loads for different elements or element sets.

Data lines to define mechanical pore pressure loads (Abaqus/Standard only):

First line:

1. Element number or element set label.
2. Distributed load type label PORMECH*n*.
3. Scaling factor.

Repeat this data line as often as necessary to define mechanical pore pressure loading for different elements or element sets.

Data lines to define a general surface traction vector, a surface shear traction vector, or a general shell-edge traction vector:

First line:

1. Element number or element set label.
2. Distributed load type label TRVEC*n*, TRVEC, TRSHR*n*, TRSHR, EDLD*n*, TRVEC*n*NU, TRVECNU, TRSHR*n*NU, TRSHRNU, or EDLD*n*NU.
3. Reference load magnitude, which can be modified by using the **AMPLITUDE* option.
4. 1-component of the traction vector direction.
5. 2-component of the traction vector direction.
6. 3-component of the traction vector direction.

For a two-dimensional or axisymmetric analysis, only the first two components of the traction vector direction need to be specified. For the shear traction load labels TRSHR*n*, TRSHR, TRSHR*n*NU, or TRSHRNU, the loading direction is computed by projecting the specified traction vector direction down upon the surface in the reference configuration. For nonuniform loads in Abaqus/Standard the magnitude and traction vector direction must be defined in user subroutine *UTRACLOAD*. If given, the magnitude and vector will be passed into the user subroutine in an Abaqus/Standard analysis.

Repeat this data line as often as necessary to define traction vectors for different elements or element sets.

Data lines to define a surface normal traction vector, a shell-edge traction vector (in the normal, transverse, or tangent direction), or a shell-edge moment:

First line:

1. Element number or element set label.
2. Distributed load type EDMOM*n*, EDNOR*n*, EDSHR*n*, EDTRA*n*, EDMOM*n*NU, EDNOR*n*NU, EDSHR*n*NU, or EDTRA*n*NU.
3. Reference load magnitude, which can be modified by using the **AMPLITUDE* option. For nonuniform loads in Abaqus/Standard the magnitude must be defined in user subroutine *UTRACLOAD*. If given, the magnitude will be passed into the user subroutine in an Abaqus/Standard analysis.

Repeat this data line as often as necessary to define traction vectors for different elements or element sets.

Data lines to define centrifugal loads and Coriolis forces (Abaqus/Standard only):

First line:

1. Element number or element set label. If this field is left blank, the load is applied to all elements in the model that support the load type and have mass contributions specified (as required by the load type). An internally created element set called `_Whole_Model_xxx_Elset`, where *xxx* is the load type label, contains all the valid elements.
2. Distributed load type label CENTRIF, CENT, or CORIO.
3. Actual magnitude of the load, which can be modified by the use of the **AMPLITUDE* option.
4. Coordinate 1 of a point on the axis of rotation.
5. Coordinate 2 of a point on the axis of rotation.
6. Coordinate 3 of a point on the axis of rotation.
7. 1-component of the direction cosine of the axis of rotation.
8. 2-component of the direction cosine of the axis of rotation.
9. 3-component of the direction cosine of the axis of rotation.

For axisymmetric elements the axis of rotation must be the global *y*-axis, which must be specified as 0.0, 0.0, 0.0, 0.0, 1.0, 0.0.

Repeat this data line as often as necessary to define centrifugal or Coriolis forces for different elements or element sets.

Data lines to define rotary acceleration loads (Abaqus/Standard only):

First line:

1. Element number or element set label. If this field is left blank, the load is applied to all elements in the model that support the load type and have mass contributions specified. An internally created element set called `_Whole_Model_ROTA_Elset` contains all the valid elements.
2. Distributed load type label ROTA.
3. Actual magnitude of the load, which can be modified by the use of the **AMPLITUDE* option.
4. Coordinate 1 of a point on the axis of rotary acceleration.
5. Coordinate 2 of a point on the axis of rotary acceleration.
6. Coordinate 3 of a point on the axis of rotary acceleration.

7. 1-component of the direction cosine of the axis of rotary acceleration.
8. 2-component of the direction cosine of the axis of rotary acceleration.
9. 3-component of the direction cosine of the axis of rotary acceleration.

For two-dimensional elements the axis of rotation direction must be the global z-axis (out of the plane of the model), which must be specified as 0.0, 0.0, 1.0.

Repeat this data line as often as necessary to define rotary acceleration loading for different elements or element sets.

Data lines to define rotordynamic loads (Abaqus/Standard only):

First line:

1. Element number or element set label. If this field is left blank, the load is applied to all elements in the model that support the load type and have mass contributions specified. An internally created element set called `_Whole_Model_ROTDYNF_Elset` contains all the valid elements.
2. Distributed load type label ROTDYNF.
3. Actual magnitude of the load, which can be modified by the use of the **AMPLITUDE* option.
4. Coordinate 1 of a point on the axis of rotation.
5. Coordinate 2 of a point on the axis of rotation.
6. Coordinate 3 of a point on the axis of rotation.
7. 1-component of the direction cosine of the axis of rotation.
8. 2-component of the direction cosine of the axis of rotation.
9. 3-component of the direction cosine of the axis of rotation.

Rotordynamic loads are supported only for three-dimensional continuum and cylindrical elements, shell elements, membrane elements, beam elements, and rotary inertia elements. The spinning axis defined as part of the load must be the axis of symmetry for the structure. Therefore, beam elements must be aligned with the symmetry axis. In addition, one of the principal directions of each loaded rotary inertia element must be aligned with the symmetry axis, and the inertia components of the rotary inertia elements must be symmetric about this axis.

Repeat this data line as often as necessary to define rotordynamic loads for different elements or element sets.

Data lines to define gravity loading:

First line:

1. The element number or element set label is optional for gravity loads. If this field is left blank, the load is applied to all elements in the model that support the load type and have mass contributions (including point mass elements) specified. An internally created element set called `_Whole_Model_GRAV_Elset` contains all the valid elements.
2. Distributed load type label GRAV.
3. Actual magnitude of the load, which can be modified by the use of the **AMPLITUDE* option.
4. 1-component of the gravity vector.
5. 2-component of the gravity vector.
6. 3-component of the gravity vector.

For axisymmetric elements the gravity load must be in the z-direction; therefore, only component 2 should be nonzero.

Repeat this data line as often as necessary to define gravity loading for different elements or element sets.

Data lines to define external and internal pressure in pipe or elbow elements:

First line:

1. Element number or element set label.
2. Distributed load type label PE, PI, PENU, or PINU.
3. Actual magnitude of the load, which can be modified by the use of the **AMPLITUDE* option. For nonuniform loads the magnitude must be defined in user subroutine *DLOAD*.
4. Effective inner or outer diameter.

Repeat this data line as often as necessary to define internal or external pressure loading for different pipe or elbow elements or element sets.

Data lines to define hydrostatic pressure (Abaqus/Standard only):

First line:

1. Element number or element set label.
2. Distributed load type label HP*n* or HP.
3. Actual magnitude of the load, which can be modified by the use of the **AMPLITUDE* option.
4. *Z*-coordinate of zero pressure level in three-dimensional or axisymmetric cases; *Y*-coordinate of zero pressure level in two-dimensional cases.
5. *Z*-coordinate of the point at which the pressure is defined in three-dimensional or axisymmetric cases; *Y*-coordinate of the point at which the pressure is defined in two-dimensional cases.

Repeat this data line as often as necessary to define hydrostatic pressure loading for different elements or element sets.

Data lines to define external and internal hydrostatic pressure in pipe or elbow elements:

First line:

1. Element number or element set label.
2. Distributed load type label HPE (external) or HPI (internal).
3. Actual magnitude of the load, which can be modified by the use of the **AMPLITUDE* option.
4. *Z*-coordinate of zero pressure level in three-dimensional or axisymmetric cases; *Y*-coordinate of zero pressure level in two-dimensional cases.
5. *Z*-coordinate of the point at which the pressure is defined in three-dimensional or axisymmetric cases; *Y*-coordinate of the point at which the pressure is defined in two-dimensional cases.
6. Effective inner or outer diameter.

Repeat this data line as often as necessary to define internal or external pressure loading for different pipe or elbow elements or element sets.

Data lines to define viscous body force, stagnation pressure, or stagnation body loads (Abaqus/Explicit only):

First line:

1. Element number or element set label.
2. Distributed load type label VBF, SP*n*, SP, or SBF.
3. Reference load magnitude, which can be modified by the use of the **AMPLITUDE* option.

Repeat this data line as often as necessary to define viscous body force, stagnation pressure, or stagnation body loads for different elements or element sets.

Loads used by Abaqus/Aqua

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the variation of the load magnitude during the step. If this parameter is omitted for uniform load types, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (see *Defining an Analysis*). Amplitude references are ignored for nonuniform loads given by user subroutine *DLOAD*.

Only the load magnitude is changed with time. Quantities such as the fluid surface level in hydrostatic pressure loading are not changed.

OP

Set OP=MOD (default) for existing **DLOAD*s to remain, with this option modifying existing loads or defining additional loads.

Set OP=NEW if all existing **DLOAD*s applied to the model should be removed. New distributed loads can be defined.

Data lines to define distributed buoyancy forces:

First line:

1. Element number or element set label.
2. Distributed load type label PB.
3. Magnitude factor, *M* (default value is 1.0). This factor will be scaled by any **AMPLITUDE* specification associated with this **DLOAD* option.
4. Effective outer diameter of the beam, truss, or one-dimensional rigid element (not used for rigid surface elements R3D3 and R3D4).

The following data must be provided only when it is necessary to model the fluid inside an element:

11. Density of fluid inside the element.
13. Effective inner diameter of the element.
15. Free surface elevation of the fluid inside the element.

The following data should be provided only if it is necessary to change the fluid properties provided on the **AQUA* option, as described in *Buoyancy Loads*. Gravity waves do not affect the buoyancy loading when any external fluid property is overridden.

19. Density of the fluid outside the element.
21. Free surface elevation of the fluid outside the element.
23. Constant pressure, added to the hydrostatic pressure outside the element.

Repeat this data line as often as necessary to define buoyancy loading for various elements or element sets.

Data lines to define distributed transverse fluid or wind drag:

First line:

1. Element number or element set label.
2. Distributed load type label FDD (fluid) or WDD (wind).
3. Magnitude factor, *M* (default value is 1.0). This factor will be scaled by any **AMPLITUDE* specification associated with this **DLOAD* option.
4. Effective outer diameter of the member, *D*.
5. Drag coefficient, C_D.
6. Structural velocity factor, α_R. The default value is 1.0 if this entry is left blank or set equal to 0.0.
7. For load type FDD, name of the **AMPLITUDE* curve used for scaling steady current velocities (A_c). For load type WDD, name of the **AMPLITUDE* curve used for scaling the local *x*-direction wind velocity (A_x). If this entry is blank, the velocities are not scaled ($A_c = 1$ or $A_x = 1$).
8. For load type FDD, name of the **AMPLITUDE* curve used for scaling wave velocities (A_w). For load type WDD, name of the **AMPLITUDE* curve used for scaling the local *y*-direction wind velocity (A_y). If this is blank, the velocities are not scaled ($A_w = 1$ or $A_y = 1$).

Repeat this data line as often as necessary to define distributed transverse fluid or wind drag on various elements or element sets.

Data lines to define distributed tangential fluid drag:

First line:

1. Element number or element set label.
2. Distributed load type label FDT.
3. Magnitude factor, *M* (default value is 1.0). This factor will be scaled by any **AMPLITUDE* specification associated with this **DLOAD* option.
4. Effective outer diameter of the member, *D*.
5. Drag coefficient, C_t.
6. Structural velocity factor, α_R. The default value is 1.0 if this entry is left blank or set equal to 0.0.
7. Exponent *h*. The default value is 2.0 if this entry is left blank or set equal to 0.0.
8. Name of the **AMPLITUDE* curve (A_c) used for scaling steady current velocities. If this entry is blank, the current velocities are not scaled ($A_c = 1$).
9. Name of the **AMPLITUDE* curve (A_w) used for scaling wave velocities. If this entry is blank, the wave velocities are not scaled ($A_w = 1$).

Repeat this data line as often as necessary to define distributed tangential fluid drag on various elements or element sets.

Data lines to define distributed fluid inertia loading:

First line:

1. Element number or element set label.
2. Distributed load type label FI.
3. Magnitude factor, *M* (default value is 1.0). This factor will be scaled by any **AMPLITUDE* specification associated with this **DLOAD* option.
4. Effective outer diameter of the member, *D*.
5. Transverse fluid inertia coefficient, C_M.
6. Transverse added-mass coefficient, C_A.
7. Name of the **AMPLITUDE* curve used for scaling fluid particle accelerations (A_w). If this entry is blank, the fluid particle accelerations are not scaled ($A_w = 1$).

Repeat this data line as often as necessary to define fluid inertia loading for various elements or element sets.

Data lines to define concentrated fluid and wind drag loading on the ends of elements:

First line:

1. Element number or element set label.
2. Distributed load type label FD1, FD2, WD1, or WD2.
3. Magnitude factor, *M* (default value is 1.0). This factor will be scaled by any AMPLITUDE specification associated with this **DLOAD* option.
4. Exposed area, ΔA.
5. Drag coefficient, *C*.
6. Structural velocity factor, α_R. The default value is 1.0 if this entry is left blank or set equal to 0.0.
7. For load types FD1 or FD2, name of the **AMPLITUDE* curve used for scaling steady current velocities (A_c). For load types WD1 or WD2, name of the **AMPLITUDE* curve used for scaling the local *x*-direction wind velocity (A_x). If this entry is blank, the velocities are not scaled ($A_c = 1$ or $A_x = 1$).
8. For load types FD1 or FD2, name of the **AMPLITUDE* curve used for scaling wave velocities (A_w). For load types WD1 or WD2, name of the **AMPLITUDE* curve used for scaling the local *y*-direction wind velocity (A_y). If this entry is blank, the velocities are not scaled ($A_w = 1$ or $A_y = 1$).

Repeat this data line as often as necessary to define concentrated fluid or wind drag loading on the ends of elements.

Data lines to define concentrated fluid inertia loading on the ends of elements:

First line:

1. Element number or element set label.
2. Distributed load type label FI1 or FI2.

3. Magnitude factor, *M* (default value is 1.0). This factor will be scaled by any AMPLITUDE specification associated with this **DLOAD* option.
4. Fluid inertia coefficient, K_{ts}.
5. Fluid acceleration shape factor, F_{1s}.
6. Added-mass coefficient, L_{ts}.
7. Structural acceleration shape factor, F_{2s}.
8. Name of the **AMPLITUDE* curve used for scaling fluid particle accelerations. If this entry is blank, the fluid particle accelerations are not scaled.

Repeat this data line as often as necessary to define concentrated fluid inertia loading on the ends of elements.

*DOMAIN DECOMPOSITION: Define a region for domain decomposition and/or define constraints on the domain decomposition.

This option is used to define a domain decomposition region and/or to define constraints on the domain decomposition.

Products: Abaqus/Explicit

Type: Model data

Level: Assembly

References:

• *Parallel Execution in Abaqus/Explicit*

Optional parameters:

DECOMPOSE FACTOR

Set DECOMPOSE FACTOR=*N* to split the decomposition region into $N * n_{domainsUser}$ domains, where $n_{domainsUser}$ is the number of domains specified during job submission (the default of $n_{domainsUser}$ is equal to the number of cpus). The default value of *N* is 1.

DEFINITION

Set DEFINITION=ELSET to specify the domain decomposition region using a user-defined element set.

Set DEFINITION=BOX to specify that Abaqus/Explicit should generate the domain decomposition region consisting of all elements contained within the user-specified box.

ELSET

Set this parameter equal to the name of the element set to define a domain decomposition region if DEFINITION=ELSET. This element set will be split into the user-specified number of parallel domains.

Set this parameter equal to an empty element set if DEFINITION=BOX. The elements will be generated automatically by Abaqus/Explicit.

If this parameter is omitted, this domain decomposition region will include all elements not included in other domain decomposition regions.

METHOD

Set METHOD=RCB (default) to decompose the model via recursive coordinate bisection.

Set METHOD=GRAPH PARTITIONING to decompose the model while minimizing the number of shared nodes.

RCB DIRECTION

Set RCB DIRECTION=XYZ (default) to split the decomposition region into domains using cutting planes normal to the global *x*-, *y*-, and *z*-directions.

Set RCB DIRECTION=X to split the decomposition region into domains using cutting planes normal to the global *x*-direction.

Set RCB DIRECTION=Y to split the decomposition region into domains using cutting planes normal to the global *y*-direction.

Set RCB DIRECTION=Z to split the decomposition region into domains using cutting planes normal to the global *z*-direction.

Set RCB DIRECTION=NOTX to split the decomposition region into domains using cutting planes normal to the global *y*- and *z*-directions.

Set RCB DIRECTION=NOTY to split the decomposition region into domains using cutting planes normal to the global *x*- and *z*-directions.

Set RCB DIRECTION=NOTZ to split the decomposition region into domains using cutting planes normal to the global *x*- and *y*-directions.

Data lines to define constraints on domain decomposition (when the ELSET parameter is omitted or DEFINITION=ELSET is used):

First line:

1. Element set label. This element should be a subset of the domain decomposition region if it is defined by specifying the ELSET parameter.
2. The "words" SAME DOMAIN to specify that all elements in the element set should be constrained to the same domain.

Repeat this data line as often as necessary to define constraints on the domain decomposition.

Data lines for DEFINITION=BOX:

First line:

1. *X*-coordinate of the lower box corner (X_{min}).
2. *Y*-coordinate of the lower box corner (Y_{min}).
3. *Z*-coordinate of the lower box corner (Z_{min}).
4. *X*-coordinate of the opposite box corner (X_{max}).
5. *Y*-coordinate of the opposite box corner (Y_{max}).
6. *Z*-coordinate of the opposite box corner (Z_{max}).

Second line:

1. Element set label. This element should be a subset of the domain decomposition region if it is defined by specifying the ELSET parameter.
2. The "words" SAME DOMAIN to specify that all elements in the element set should be constrained to the same domain.

Repeat this data line as often as necessary to define constraints on the domain decomposition.

*DRAG CHAIN: Specify parameters for drag chain elements.

This option is used to specify the maximum length of a drag chain, the frictional limit between the chain and the seabed, and the weight of the drag chain.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

• *Drag Chains*

Required parameters:

ELSET

Set this parameter equal to the name of the element set with which this behavior is associated.

Data line for DRAG2D elements:

First (and only) line:

1. Horizontal length of chain at slip, ℓ.
2. Friction limit between the chain and the seabed.

Data line for DRAG3D elements:

First (and only) line:

1. Total length of chain.
2. Friction coefficient.
3. Weight of chain (per unit length).

*DRUCKER PRAGER: Specify the extended Drucker-Prager plasticity model.

This option is used to define yield surface and flow potential parameters for elastic-plastic materials that use one of the extended Drucker-Prager plasticity models. It must be used in conjunction with the **DRUCKER PRAGER HARDENING* option and, if creep material behavior is included in an Abaqus/Standard analysis, with the **DRUCKER PRAGER CREEP* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Extended Drucker-Prager Models*
- **DRUCKER PRAGER HARDENING*
- **DRUCKER PRAGER CREEP*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the material parameters other than temperature. If this parameter is omitted, it is assumed that the material parameters depend only on temperature. See *Material Data Definition* for more information.

ECCENTRICITY

This parameter is only for use with SHEAR CRITERION=HYPERBOLIC or SHEAR CRITERION=EXPONENT FORM or if creep material properties are included with SHEAR CRITERION=LINEAR.

It is used to define the flow potential eccentricity, ϵ. The eccentricity is a small positive number that defines the rate at which the hyperbolic flow potential approaches its asymptote. The default is

$\epsilon = 0.1$ for the exponent model; and if $\psi = \beta$, it is set to $\epsilon = (d'|_0 - p_t|_0 \tan\beta) / (\bar{\sigma}|_0 \tan\beta)$ for the hyperbolic model to ensure associated flow (the terms are defined in *Extended Drucker-Prager Models*).

SHEAR CRITERION

Set SHEAR CRITERION=LINEAR (default) to define the linear yield criterion. This is required if creep material behavior is included for an Abaqus/Standard analysis.

Set SHEAR CRITERION=HYPERBOLIC to define the hyperbolic yield criterion.

Set SHEAR CRITERION=EXPONENT FORM to define the exponent form as a yield criterion.

TEST DATA

This parameter is only for use with SHEAR CRITERION=EXPONENT FORM.

Include this parameter if the material constants for the exponent model are to be computed by Abaqus from triaxial test data at different levels of confining pressure. The **TRIAXIAL TEST DATA* option must be used for this purpose.

Data lines to define a linear Drucker-Prager plasticity model (SHEAR CRITERION=LINEAR):

First line:

1. Material angle of friction, β, in the p–t plane. Give the value in degrees.
2. K, the ratio of the flow stress in triaxial tension to the flow stress in triaxial compression.

   $0.778 \leq K \leq 1.0$. If this field is left blank or a value of 0.0 is entered, the default of 1.0 is used. If creep material behavior is included, K should be set to 1.0.
3. Dilation angle, ψ, in the p–t plane. Give the value in degrees.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the material parameters on temperature and other predefined field variables.

Data lines to define a hyperbolic Drucker-Prager plasticity model (SHEAR CRITERION=HYPERBOLIC):

First line:

1. Material angle of friction, β, at high confining pressure in the p–q plane. Give the value in degrees.
2. Initial hydrostatic tension strength, $p_t|_0$. (Units of FL^{-2}.)
3. Not used.
4. Dilation angle, ψ, at high confining pressure in the p–q plane. Give the value in degrees.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the material parameters on temperature and other predefined field variables.

Data lines to define a Drucker-Prager plasticity model with the exponent law (SHEAR CRITERION=EXPONENT FORM) and without test data (TEST DATA):

First line:

1. Material constant a.
2. Exponent b. To ensure a convex yield surface, $b \geq 1$.
3. Not used.
4. Dilation angle, ψ, at high confining pressure in the p–q plane. Give the value in degrees.
5. Temperature.

6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the material parameters on temperature and other predefined field variables.

Data lines to define a Drucker-Prager plasticity model with the exponent law (SHEAR CRITERION=EXPONENT FORM) and with test data (TEST DATA):

First line:

1. Not used.
2. Not used.
3. Not used.
4. Dilation angle, ψ, at high confining pressure in the p–q plane. Give the value in degrees.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the material parameters on temperature and other predefined field variables.

*DRUCKER PRAGER CREEP: Specify a Drucker-Prager creep law and material properties.

This option is used to define a Drucker-Prager creep model and material properties. Creep behavior defined by this option is active only during **SOILS*, CONSOLIDATION; **COUPLED TEMPERATURE-DISPLACEMENT*; and **VISCO* procedures. It must be used in conjunction with the **DRUCKER PRAGER* and **DRUCKER PRAGER HARDENING* options. The data entered must be consistent with the TYPE parameter used on the **DRUCKER PRAGER HARDENING* option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Extended Drucker-Prager Models*
- **DRUCKER PRAGER*
- **DRUCKER PRAGER HARDENING*
- *CREEP*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the creep constants, in addition to temperature. If this parameter is omitted, it is assumed that the creep constants depend only on temperature. See *Material Data Definition* for more information.

LAW

Set LAW=STRAIN (default) to choose a strain hardening law. This parameter setting is superseded by LAW=POWER LAW.

Set LAW=TIME to choose a time hardening law. This parameter setting is superseded by LAW=TIME POWER LAW.

Set LAW=SINGHM to choose a Singh-Mitchell type law.

Set LAW=USER to input the creep law using user subroutine *CREEP*.

Set LAW=POWER LAW to choose a power law. It is recommended that you use this parameter setting rather than LAW=STRAIN.

Set LAW=TIME POWER LAW to choose a time power law. It is recommended that you use this parameter setting rather than LAW=TIME.

TIME

This parameter is relevant only when LAW=TIME, LAW=SINGHM, or LAW=TIME POWER LAW is used.

Set TIME=CREEP to use creep time.

Set TIME=TOTAL (default) to use total time.

Data lines for LAW=TIME or LAW=STRAIN:

First line:

1. A. (Units of $F^{-n}L^{2n}T^{-1-m}$.)
2. n.
3. m.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the creep constants on temperature and other predefined field variables.

Data lines for LAW=SINGHM:

First line:

1. A. (Units of T^{-1}.)
2. α. (Units of $F^{-1}L^2$.)
3. m.
4. t_1. (Units of T.)
5. Temperature.
6. First field variable.
7. Second field variable
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the creep constants on temperature and other predefined field variables.

Data lines for LAW=POWER LAW:

First line:

1. q_0. (Units of FL^{-2}.)
2. n.
3. m.
4. Blank space.
5. $\dot{\varepsilon}_0$. The default is 1.0. (Units of T^{-1}.)
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the creep constants on temperature and other predefined field variables.

Data lines for LAW=TIME POWER LAW:

First line:

1. q_0. (Units of FL^{-2}.)
2. n.
3. m.
4. $\dot{\varepsilon}_0$. The default is 1.0. (Units of T^{-1}.)
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the creep constants on temperature and other predefined field variables.

*DRUCKER PRAGER HARDENING: Specify hardening for Drucker-Prager plasticity models.

This option is used to specify the hardening data for elastic-plastic materials that use any of the generalized Drucker-Prager yield criteria defined in the **DRUCKER PRAGER* option.

This option is also used in Abaqus/Standard analyses to specify the type of creep test with which the creep laws defined in the **DRUCKER PRAGER CREEP* option are measured. It must be used in conjunction with the **DRUCKER PRAGER* option and, if creep material behavior is included in an Abaqus/Standard analysis, with the **DRUCKER PRAGER CREEP* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Extended Drucker-Prager Models*
- **DRUCKER PRAGER*
- **DRUCKER PRAGER CREEP*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the yield stress, in addition to temperature. If this parameter is omitted, the yield stress depends only on the plastic strain and, possibly, on temperature. See "Using the DEPENDENCIES parameter to define field variable dependence" in *Material Data Definition* for more information.

RATE

Set this parameter equal to the equivalent plastic strain rate, $\dot{\bar{\varepsilon}}^{pl}$, for which this hardening curve applies. This parameter should be omitted if the **RATE DEPENDENT* option or the **DRUCKER PRAGER CREEP* option is used. Rate-independent behavior is assumed if the RATE parameter, the **RATE DEPENDENT* option, and the **DRUCKER PRAGER CREEP* option are not used.

TYPE

Set TYPE=COMPRESSION (default) to define the hardening behavior by giving the uniaxial compression yield stress, σ_c, as a function of uniaxial compression plastic strain, $\bar{\varepsilon}^{pl} = \left|\varepsilon_{11}^{pl}\right|$.

Set TYPE=TENSION to define the hardening behavior by giving the uniaxial tension yield stress, σ_t, as a function of uniaxial tension plastic strain, $\bar{\varepsilon}^{pl} = \varepsilon_{11}^{pl}$.

Set TYPE=SHEAR to define the hardening behavior by giving the cohesion, $d = \frac{\sqrt{3}}{2}\tau\left(1 + \frac{1}{K}\right)$, as a function of equivalent shear plastic strain, $\bar{\varepsilon}^{pl} = \gamma^{pl}/\sqrt{3}$, where τ is the yield stress in shear, K is the ratio of flow stress in triaxial tension to the flow stress in triaxial compression, and γ^{pl} is the engineering shear plastic strain.

Data lines to define Drucker-Prager hardening:

First line:

1. Yield stress.
2. Absolute value of the corresponding plastic strain. (The first tabular value entered must always be zero.)
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of yield stress on plastic strain and, if needed, on temperature and other predefined field variables.

*DSA CONTROLS: Set DSA solution controls.

This option can be used to control the accuracy or efficiency of the DSA computations.

Products: Abaqus/Design

Type: Model or history data

Level: Model, Step

References:

• *Direct Design Sensitivity Analysis*

Optional parameters:

FORMULATION

Use this parameter to select the design sensitivity analysis formulation type in a multi-increment analysis. This parameter will be ignored if used as history data.

Set FORMULATION=INCREMENTAL (default) to select incremental design sensitivity analysis.

Set FORMULATION=TOTAL to select total design sensitivity analysis.

RESET

Include this parameter to reset the values to those specified on the model data options or to the original default values if no model data options exist. This action takes effect before applying any additional changes to the values.

SIZING FREQUENCY

Set this parameter equal to the frequency in increments (static steps) or modes (frequency steps) at which the default perturbation sizing algorithm is to be executed. The algorithm will always be executed for the first increment or first eigenmode in each step for which DSA calculations are done, even if SIZING FREQUENCY is set to 0. The default is SIZING FREQUENCY=0.

TOLERANCE

Set this parameter equal to the tolerance to be used with the default perturbation sizing algorithm.

The default is TOLERANCE=1.0×10^{-4}.

Data lines to override the default perturbation sizing algorithm for selected design parameters (The SIZING FREQUENCY and TOLERANCE parameters will be ignored for these design parameters.):

First line:

1. Design parameter.
2. Set this entry to FD to use forward difference. Set this entry to CD to use central difference.
3. Absolute value of perturbation.

Repeat this data line for each design parameter for which the default algorithm is to be overridden.

*DSECHARGE: Input distributed electric surface charges for piezoelectric analysis.

This option is used to input distributed electric surface charges on a surface underlying piezoelectric elements.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

• *Piezoelectric Analysis*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the magnitude of the distributed electric charge during the step. If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (*Defining an Analysis*).

OP

Set OP=MOD (default) for existing **DSECHARGE*s to remain, with this option defining electric charges to be added or modified. Set OP=NEW if all existing **DSECHARGE*s applied to the model should be removed.

Optional, mutually exclusive parameters for matrix generation and direct-solution, steady-state dynamics analysis:

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the loading.

REAL

Include this parameter (default) to define the real (in-phase) part of the loading.

Data lines to define distributed electric charges:

First line:

1. Surface name.
2. Distributed electric charge label ES.
3. Reference electric surface charge magnitude. (Units of CL^{-2}.)

Repeat this data line as often as necessary to define distributed electric charges for various surfaces.

*DSECURRENT: Specify distributed current densities over a surface in an electromagnetic analysis.

This option is used to input distributed current densities over a surface in coupled thermal-electrical, coupled thermal-electrical-structural, coupled thermal-electrochemical, coupled thermal-electrochemical-structural, coupled thermal-electrochemical-structural-pore pressure, eddy current, and magnetostatic analyses.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- *Coupled Thermal-Electrical Analysis*
- *Fully Coupled Thermal-Electrical-Structural Analysis*
- *Coupled Thermal-Electrochemical Analysis*
- *Fully Coupled Thermal-Electrochemical-Structural Analysis*
- *Fully Coupled Thermal-Electrochemical-Structural–Pore Pressure Analysis*
- *Eddy Current Analysis*
- *Magnetostatic Analysis*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the magnitude of the electric current density during the step (*Amplitude Curves*). If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (*Defining an Analysis*).

OP

Set OP=MOD (default) for existing **DSECURRENT*s to remain, with this option defining distributed current densities to be added or modified.

Set OP=NEW if all existing **DSECURRENT*s applied to the model should be removed.

Optional, mutually exclusive parameters for time-harmonic eddy current analyses:

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the surface current density.

REAL

Include this parameter (default) to define the real (in-phase) part of the surface current density.

Data lines to define distributed electrical current densities in coupled thermal-electrical, coupled thermal-electrical-structural, coupled thermal-electrochemical, coupled thermal-electrochemical-structural, or coupled thermal-electrochemical-structural-pore pressure analyses:

First line:

1. Surface name.

2. Distributed current density type label CS or CSNU.
3. Reference surface current density magnitude. (Units of $CL^{-2}T^{-1}$.) For nonuniform loads the magnitude must be defined in user subroutine *DECURRENT*. If given, the magnitude will be passed into the user subroutine.
4. Table collection name (optional). Only applicable for nonuniform distributed current densities defined in user subroutine *DECURRENT*.

   .

Repeat this data line as often as necessary to define current densities for various surfaces.

Data lines to define surface current densities in eddy current or magnetostatic analyses:

First line:

1. Surface name.
2. Surface current density type label CK (uniform) or CKNU (nonuniform).
3. Reference surface current density magnitude. (Units of $CL^{-1}T^{-1}$.)
4. 1-component of the surface current density vector direction.
5. 2-component of the surface current density vector direction.
6. 3-component of the surface current density vector direction.

For nonuniform loads the magnitude and direction must be defined in user subroutine *UDSECURRENT*. If given, the magnitude and direction will be passed into the user subroutine.

15. Name of the orientation option (*Orientations*) that defines the local coordinate system in which the surface current density vector is specified.

Repeat this data line as often as necessary to define surface current densities for various surfaces.

*DSFLOW: Specify distributed seepage flows normal to a surface.

This option is used to input seepage flows (pore fluid velocities) normal to surfaces of the model in consolidation problems.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- *Pore Fluid Flow*
- *DFLOW*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the **AMPLITUDE* curve that defines the magnitude of the seepage flow during the step. If this parameter is omitted for uniform seepage types, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (see *Defining an Analysis*). Amplitude references are ignored for flows defined in user subroutine *DFLOW*.

OP

Set OP=MOD (default) for existing **DSFLOW*s to remain, with this option modifying existing flows or defining additional flows.

Set OP=NEW if all existing **DSFLOW*s applied to the model should be removed. New flows can be defined.

Data lines to define uniform seepage:

First line:

1. Surface name.
2. Distributed uniform seepage type label S.
3. Reference seepage magnitude. (Units of LT^{-1}.) The seepage magnitude is the pore fluid effective velocity crossing the surface at this point in an outward direction.

Repeat this data line as often as necessary to define uniform seepage for various surfaces.

Data lines to define nonuniform seepage:

First line:

1. Surface name.
2. Nonuniform distributed seepage type label SNU.
3. Seepage magnitude (optional). If given, this value is passed into user subroutine *DFLOW* in the variable used to define the seepage magnitude.

Nonuniform seepage magnitudes are defined via user subroutine *DFLOW*.

Repeat this data line as often as necessary to define nonuniform seepage for surfaces.

*DSFLUX: Specify distributed surface fluxes for heat transfer analysis.

This option is used to apply distributed surface fluxes for a fully coupled thermal-stress analysis in Abaqus/Standard and Abaqus/Explicit. In Abaqus/Standard it is also used for heat transfer, coupled thermal-electrical, and coupled thermal-electrical-structural analyses.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- *Thermal Loads*
- *DFLUX*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the magnitude of the distributed fluxes during the step (*Amplitude Curves*).

If this parameter is omitted for uniform flux types in an Abaqus/Standard analysis, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (*Defining an Analysis*). If this parameter is omitted in an Abaqus/Explicit analysis, the reference magnitude is applied immediately at the beginning of the step.

For nonuniform flux type SNU (which is available only in Abaqus/Standard), the flux magnitude is defined in user subroutine *DFLUX*, and AMPLITUDE references are ignored.

OP

Set OP=MOD (default) for existing **DSFLUX*s to remain, with this option modifying existing fluxes or defining additional fluxes.

Set OP=NEW if all existing **DSFLUX*s applied to the model should be removed.

Data lines to define a distributed surface flux:

First line:

1. Surface name.
2. Distributed flux type label S or SNU.
3. Reference flux magnitude (units $JT^{-1}L^{-2}$). This value is needed for uniform fluxes only. If it is given for nonuniform fluxes, it will be passed into user subroutine *DFLUX*, where the actual flux magnitude is defined.

Repeat this data line as often as necessary to define distributed fluxes for different surfaces.

*DSLOAD: Specify distributed surface loads.

This option is used to prescribe distributed surface loading.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- *Distributed Loads*
- *Fluid Pressure Penetration Loads*
- *DLOAD*
- *Analysis of Models that Exhibit Cyclic Symmetry*
- *About Submodeling*
- *Surface-Based Submodeling*
- *Modeling Discontinuities as an Enriched Feature Using the Extended Finite Element Method*

Applying distributed loads

Required parameter for cyclic symmetry models in steady-state dynamics analyses:

CYCLIC MODE

Set this parameter equal to the cyclic symmetry mode number of loads that are applied in the current steady-state dynamics procedure.

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the variation of the load magnitude during the step.

If this parameter is omitted for uniform load types in an Abaqus/Standard analysis, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (see *Defining an Analysis*). If this parameter is omitted in an Abaqus/Explicit analysis, the reference magnitude is applied immediately at the beginning of the step.

Amplitude references are ignored for nonuniform loads given by user subroutine *DLOAD* in an Abaqus/Standard analysis. Amplitude references are passed into user subroutine *VDLOAD* in an Abaqus/Explicit analysis.

Only the load magnitude is changed with time. Quantities such as the fluid surface level in hydrostatic pressure loading are not changed.

CONSTANT RESULTANT

Set CONSTANT RESULTANT=NO (default) if surface traction vectors, edge traction vectors, or edge moments are to be integrated over the surface in the current configuration.

Set CONSTANT RESULTANT=YES if surface traction vectors, edge traction vectors, or edge moments are to be integrated over the surface in the reference configuration.

The CONSTANT RESULTANT parameter is valid only for uniform and nonuniform surface tractions and edge loads (including edge moments); it is ignored for all other load types.

FOLLOWER

Set FOLLOWER=YES (default) if a prescribed traction or shell-edge load is to rotate with the surface or shell edge in a large-displacement analysis (live load).

Set FOLLOWER=NO if a prescribed traction or edge load is to remain fixed in a large-displacement analysis (dead load).

The FOLLOWER parameter is valid only for traction and edge load labels TRVEC, TRVECNU, EDLD, and EDLDNU. It is ignored for all other load labels.

OP

Set OP=MOD (default) for existing **DSLOAD*s to remain, with this option modifying existing distributed loads or defining additional distributed loads.

Set OP=NEW if all existing **DSLOAD*s applied to the model should be removed. New distributed loads can be defined.

The OP parameter is ignored for the nodal pressure distributed load type label (NP). All existing **DSLOAD*s are removed before applying the nodal pressure loads.

ORIENTATION

Set this parameter equal to the name given for the **ORIENTATION* option (*Orientations*) used to specify the local coordinates in which components of traction or shell-edge loads are specified.

The ORIENTATION parameter is valid only for traction and edge load labels TRSHR, TRSHRNU, TRVEC, TRVECNU, EDLD, and EDLDNU. It is ignored for all other load labels.

REF NODE

This parameter applies only to Abaqus/Explicit analyses and is relevant only for viscous and stagnation pressure loads when the velocity at the reference node is used.

Set this parameter equal to either the node number of the reference node or the name of a node set containing the reference node. If the name of a node set is chosen, the node set must contain exactly one node. If this parameter is omitted, the reference velocity is assumed to be zero.

Optional, mutually exclusive parameters for matrix generation and steady-state dynamics analysis (direct, modal, or subspace):

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the loading.

REAL

Include this parameter (default) to define the real (in-phase) part of the loading.

Data lines to define distributed surface pressures:

First line:

1. Surface name.
2. Distributed load type label P, PNU, SP, or VP.

3. Reference load magnitude, which can be modified by using the *AMPLITUDE option. For nonuniform loads, the magnitude must be defined in user subroutine *DLOAD* for an Abaqus/Standard analysis or *VDLOAD* for an Abaqus/Explicit analysis. If given, this value is passed into the user subroutine in an Abaqus/Standard analysis.

Repeat this data line as often as necessary to define distributed loads on different surfaces.

Data lines to define hydrostatic pressure (Abaqus/Standard only):

First line:

1. Surface name.
2. Distributed load type label HP.
3. Actual magnitude of the load, which can be modified by using the *AMPLITUDE option.
4. *Z*-coordinate of zero pressure level.
5. *Z*-coordinate of the point at which the pressure is defined.

Repeat this data line as often as necessary to define hydrostatic pressure loading on different surfaces.

Data lines to define mechanical pore pressure loads (Abaqus/Standard only):

First line:

1. Surface name.
2. Distributed load type label PORMECH.
3. Scaling factor.

Repeat this data line as often as necessary to define mechanical pore pressure loading on different surfaces.

Data lines to define nodal pressure loads (Abaqus/Standard only):

First line:

1. Surface name.
2. Distributed load type label NP.
3. Scaling factor. The default is 1.0.

Repeat this data line as often as necessary to define nodal pressure loading on different surfaces.

Data lines to define a general surface traction vector, a surface shear traction vector, or a general shell-edge traction vector:

First line:

1. Surface name.
2. Distributed load type label TRVEC, TRSHR, EDLD, TRVECNU, TRSHRNU, or EDLDNU.
3. Reference load magnitude, which can be modified by using the *AMPLITUDE option.
4. 1-component of the traction vector direction.
5. 2-component of the traction vector direction.
6. 3-component of the traction vector direction.

For a two-dimensional or axisymmetric analysis, only the first two components of the traction vector direction need to be specified. For the shear traction load labels TRSHR and TRSHRNU, the loading direction is computed by projecting the specified traction vector direction down upon the surface in the reference configuration. For nonuniform loads in Abaqus/Standard the magnitude and traction vector direction must be defined in user

subroutine *UTRACLOAD*. If given, the magnitude and vector will be passed into the user subroutine in an Abaqus/Standard analysis.

Repeat this data line as often as necessary to define traction vectors on different surfaces.

Data lines to define a surface normal traction vector, a shell-edge traction vector (in the normal, transverse, or tangent direction), or a shell-edge moment:

First line:

1. Surface name.
2. Distributed load type label EDMOM, EDNOR, EDSHR, EDTRA, EDMOMNU, EDNORNU, EDSHRNU, or EDTRANU.
3. Reference load magnitude, which can be modified by using the **AMPLITUDE* option. For nonuniform loads in Abaqus/Standard the magnitude must be defined in user subroutine *UTRACLOAD*. If given, the magnitude will be passed into the user subroutine in an Abaqus/Standard analysis.

Repeat this data line as often as necessary to define traction vectors on different surfaces.

Data lines to define stagnation pressure loads (Abaqus/Explicit only):

First line:

1. Surface name.
2. Distributed load type label SP.
3. Reference load magnitude, which can be modified by using the **AMPLITUDE* option.

Repeat this data line as often as necessary to define stagnation pressure loads on different surfaces.

Data lines to define surface tension loads (Abaqus/Explicit only):

First line:

1. Surface name.
2. Distributed load type label ST.
3. Reference surface tension coefficient, which can be modified by using the **AMPLITUDE* option.

If the surface tension property is defined for this surface, the reference surface tension defined here will be ignored, and the amplitude will be applied to the surface tension defined by the surface tension property.

Repeat this data line as often as necessary to define surface tension loads on different surfaces.

Data lines to define fluid pressure penetration surface pressures:

First line:

1. Surface name.
2. Distributed load type label PPEN.
3. Reference magnitude of the fluid pressure, which can be modified by using the AMPLITUDE parameter.
4. Critical contact pressure threshold at which fluid penetration starts to occur. The default value is 0.0, in which case fluid penetration occurs only if contact is lost. You can use an amplitude curve defined in entry 8 of the data line to modify a nonzero value.

5. Specify WETTING ADVANCE or LOCAL to indicate the algorithm for controlling the evolution of the region exposed to the fluid pressure. The WETTING ADVANCE algorithm is currently available only in Abaqus/Explicit.
6. Node or node set on the surface initially exposed to fluid pressure. This node or node set is used only if WETTING ADVANCE is specified.
7. Time period over which the fluid pressure on the newly penetrated contact surface segments is ramped up to the current magnitude. The default penetration time period is chosen as 0.001 of the current step time.

   This scalar value applies only to Abaqus/Standard analyses and is neglected in a linear perturbation analysis, in which case the fluid pressure is applied immediately once the pressure penetration criterion is satisfied.
8. Name of the amplitude curve that defines the variation of the critical contact pressure threshold during the step.

Repeat this data line as often as necessary to define distributed fluid pressure penetration loads on either different surfaces or on the same surface but with different nodes initially exposed to a fluid if WETTING ADVANCE is specified.

Applying submodel boundary conditions (Abaqus/Standard only)

Required parameters:

STEP

Set this parameter equal to the step number in the global analysis for which the values of the driven stresses will be read during this step of the submodel analysis.

SUBMODEL

Include this parameter to specify that the distributed loads are the “driven loads” in a submodel analysis. Surfaces used in this option must be among those listed in the **SUBMODEL* model definition option.

Optional parameters:

INC

This parameter can be used only in a static linear perturbation step (*General and Perturbation Procedures*).

Set this parameter equal to the increment in the selected step of the global analysis at which the solution is used to specify the values of the driven stresses. By default, Abaqus/Standard uses the solution at the last increment of the selected step.

OP

Set OP=MOD (default) for existing **DSLOAD*s to remain, with this option modifying existing distributed loads or defining additional distributed loads.

Set OP=NEW if all existing **DSLOAD*s applied to the model should be removed. New distributed loads can be defined.

Data lines to define submodeling loads:

First line:

1. Surface name

Repeat this data line as often as necessary to specify submodel distributed loads at different surfaces.

*DYNAMIC: Dynamic stress/displacement analysis.

This option is used to provide direct integration of a dynamic stress/displacement response in Abaqus/Standard analyses and is generally used for nonlinear cases. It is used to perform a dynamic stress/displacement analysis using explicit integration in Abaqus/Explicit. The analysis in both Abaqus/Standard and Abaqus/Explicit can also be adiabatic.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Implicit Dynamic Analysis Using Direct Integration*
- *Explicit Dynamic Analysis*
- *Adiabatic Analysis*

Defining a dynamic analysis in Abaqus/Standard

Optional parameter for the subspace projection method:

SUBSPACE

Include this parameter to choose the subspace projection method (explicit integration of the model projected onto the eigenvectors obtained in the last **FREQUENCY* step preceding this step).

If this parameter is omitted, implicit time integration of the dynamic equations for all global level degrees of freedom is used.

Optional parameters for the general implicit integration method:

ADIABATIC

Include this parameter if an adiabatic stress analysis is to be performed. This parameter is relevant only for isotropic metal plasticity materials with a Mises yield surface and when the **INELASTIC HEAT FRACTION* option has been specified.

ALPHA

Set this parameter equal to a nondefault value of the numerical (artificial) damping control parameter, α, in the implicit operator for TIME INTEGRATOR=HHT-TF or HHT-MD. Allowable values are 0 (no damping) to –0.5. The value of –0.333 provides maximum damping. The default for TIME INTEGRATOR=HHT-TF is ALPHA=–0.05, which provides slight numerical damping.

APPLICATION

Use this parameter to choose a time integration method. Other parameter values are determined by the time integration method selected. You can override the defaults by specifying these parameter values directly.

Set APPLICATION=TRANSIENT FIDELITY (default for problems without contact in the model) to choose a method for an accurate solution with slight numerical damping. The TIME

INTEGRATOR=HHT-TF, IMPACT=AVERAGE TIME, and INCREMENTATION=CONSERVATIVE are set.

Set APPLICATION=MODERATE DISSIPATION (default for problems with contact in the model) to choose a method with larger than default numerical damping and a more aggressive time incrementation scheme at the expense of some solution accuracy. The TIME INTEGRATOR=HHT-MD, IMPACT=NO, and INCREMENTATION=AGGRESSIVE are set.

Set APPLICATION=QUASI-STATIC to choose a method with very significant numerical damping that is primarily intended to obtain quasi-static solutions. The TIME INTEGRATOR=BWE, IMPACT=NO, and INCREMENTATION=AGGRESSIVE values are set. In addition, the default step amplitude is set to RAMP instead of STEP.

BETA

Set this parameter equal to a nondefault value, β, in the implicit operator for TIME INTEGRATOR=HHT-TF or HHT-MD. Allowable values are positive.

DIRECT

Include this parameter to choose direct user control of the incrementation through the step. If this parameter is included and no contact impacts or releases occur, constant increments of the size defined on the data line are used. If this parameter is omitted, Abaqus/Standard uses the automatic time incrementation scheme after trying the user's initial time increment for the first attempt at the first increment. The DIRECT parameter and the HAFTOL and HALFINC SCALE FACTOR parameters are mutually exclusive.

The DIRECT parameter may have the value NO STOP. If this value is included, the solution to an increment is accepted after the maximum number of iterations allowed (as defined in the **CONTROLS* option) have been done, even if the equilibrium tolerances are not satisfied. Small increments and a minimum of two iterations are usually necessary if this value is used. *This approach is not generally recommended; it should be used only in special cases when the analyst has a thorough understanding of how to interpret results obtained in this way.*

GAMMA

Set this parameter equal to a nondefault value, γ, in the implicit operator for TIME INTEGRATOR=HHT-TF or HHT-MD. Allowable values are greater or equal to 0.5.

HAFTOL

Set this parameter equal to the half-increment residual tolerance to be used with the automatic time incrementation scheme. For automatic time incrementation this value controls the accuracy of the solution if HALFINC SCALE FACTOR is not specified. It is recommended that the HALFINC SCALE FACTOR parameter be used instead of the HAFTOL parameter. If both are included, the HAFTOL parameter is ignored. The DIRECT and HAFTOL parameters are mutually exclusive.

The HAFTOL parameter has dimensions of force and is usually chosen by comparison with typical actual force values, such as applied forces or expected reaction forces. The following guidelines may be helpful. For problems where considerable plasticity or other dissipation is expected to damp out the high frequency response, choose HAFTOL as 10 to 100 times typical actual force values for moderate accuracy and low cost; choose HAFTOL as 1 to 10 times typical actual force values for higher accuracy. In such cases smaller values of HAFTOL are usually not needed.

For elastic cases with little damping the high frequency modes usually remain important throughout the problem; therefore, HAFTOL values should be smaller than recommended above. Choose HAFTOL as 1 to 10 times typical actual force values for moderate accuracy; choose HAFTOL as 0.1 to 1 times actual force values for higher accuracy.

HALFINC SCALE FACTOR

Set this parameter equal to a scale factor applied to Abaqus/Standard calculated time average force and moment values to be used as the half-increment residual tolerance with the automatic time incrementation solution accuracy checking scheme. The DIRECT and HALFINC SCALE FACTOR parameters are mutually exclusive. The HALFINC SCALE FACTOR is ignored when NOHAF parameter is set.

The HALFINC SCALE FACTOR parameter is unitless. As a guideline, with smaller HALFINC SCALE FACTOR values, more accurate solutions should be obtained at the expense of using finer time increments. By default for APPLICATION=TRANSIENT FIDELITY, it is set to 10000 if contact is present in the model and to 1000 otherwise. These defaults differ from the suggested HAFTOL ratios primarily because the HALFINC SCALE FACTOR is applied to known force averages; hence, they need not be as conservative.

IMPACT

Use this parameter to choose a time incrementation type when contact impacts or releases occur during analysis.

Set IMPACT=AVERAGE TIME to choose a time incrementation scheme that employs average time of impact/release cut backs to enforce energy balance and maintains velocities and accelerations compatible on the active contact interface. The IMPACT=AVERAGE TIME and TIME INTEGRATOR=BWE settings are mutually exclusive.

Set IMPACT=CURRENT TIME to choose a “marching through” scheme without impact/release cut backs. The velocities and accelerations are compatible on the active contact interface.

Set IMPACT=NO to choose a “marching through” scheme without impact/release cut backs and without velocity/acceleration compatibility computations.

INCREMENTATION

Use this parameter to choose a general time incrementation type.

Set INCREMENTATION=CONSERVATIVE to choose a time incrementation scheme that maximizes solution accuracy.

Set INCREMENTATION=AGGRESSIVE to choose a time incrementation scheme based only on convergence history, similar to a scheme typically used in static problems without rate or history dependence. Setting INCREMENTATION=AGGRESSIVE also sets the value of the NOHAF parameter.

INITIAL

By default, Abaqus/Standard will calculate or recalculate accelerations at the beginning of the step if an IMPACT value other than NO is used. Set INITIAL=NO to bypass the calculation of initial accelerations at the beginning of the step.

If INITIAL=NO, Abaqus/Standard assumes that the initial accelerations for the current step are zero if the current step is the first **DYNAMIC* step. If the immediately preceding step was also a **DYNAMIC* step, using INITIAL=NO causes Abaqus/Standard to use the accelerations from the end of the previous step to continue the new step. This is appropriate only if the loading does not change suddenly at the start of the new step.

NOHAF

Include this parameter to suppress calculation of the half-increment residuals and thus skip some accuracy checking for the automatic time incrementation scheme. For fixed time incrementation with the DIRECT parameter included, Abaqus/Standard calculates the half-increment residuals by default; the NOHAF parameter switches off this calculation, saving some of the solution cost.

RATE DEPENDENCE

This parameter can be used only if the quasi-static response is specified (APPLICATION=QUASI-STATIC).

Set RATE DEPENDENCE=ON (default) to consider the strain rate–dependence when evaluating the yield stress of a material and to consider the slip rate–dependence when evaluating the friction coefficient.

Set RATE DEPENDENCE=OFF to ignore the strain rate–dependence when evaluating the yield stress of a material and to ignore the slip rate–dependence when evaluating the friction coefficient.

SINGULAR MASS

Use this parameter to control velocity and acceleration adjustments if a singular global mass matrix is detected during initialization or during contact impact/release computations.

Set SINGULAR MASS=ERROR (default) to issue an error message and stop execution if a singular global mass matrix is detected when calculating the velocity and acceleration adjustments.

Set SINGULAR MASS=WARNING to issue a warning message and avoid these velocity and acceleration adjustments (that is, continue time integration using the current velocities and accelerations) if a singular global mass matrix is detected.

Set SINGULAR MASS=MAKE ADJUSTMENTS to adjust velocities and accelerations even if a singular mass matrix is detected. This setting can result in large, non-physical velocity and/or acceleration adjustments, which can, in turn, cause poor time integration solutions and artificial convergence difficulties. *This approach is not generally recommended; it should be used only in special cases when the analyst has a thorough understanding of how to interpret results obtained in this way.*

TIME INTEGRATOR

Use this parameter to choose the time integration method.

Set TIME INTEGRATOR=BWE to choose the backward Euler time integrator.

Set TIME INTEGRATOR=HHT-TF to choose the Hilber-Hughes-Taylor time integrator with default parameter settings which provide slight numerical damping. This is the default for APPLICATION=TRANSIENT FIDELITY.

Set TIME INTEGRATOR=HHT-MD to choose the Hilber-Hughes-Taylor time integrator with default parameter settings that provide moderate numerical damping. This is the default for APPLICATION=MODERATE DISSIPATION.

Data line for a transient dynamic analysis:

First (and only) line:

1. Suggested initial time increment. For implicit integration, this same time increment will be used throughout the step unless contact impacts or releases occur or the automatic time incrementation scheme is used. If the SUBSPACE parameter is included, the smaller of this time increment or

   80% of $2/\omega_{max}$, where ω_{max} is the circular frequency of the highest mode included in the dynamic response analysis, is used throughout the step.
2. Time period of the step.
3. Minimum time increment allowed. If a smaller time increment than this value is needed, the analysis is terminated. If this entry is zero, a default value of the smaller of the suggested initial time increment or 10^{-5} times the time period of the step is assumed.
4. Maximum time increment allowed. Only useful for automatic time incrementation. If this value is zero, the default depends on the APPLICATION setting. If APPLICATION=TRANSIENT FIDELITY, the maximum time increment allowed is the time period of the step divided by 100.

If APPLICATION=MODERATE DISSIPATION, it is the time period of the step divided by 10. If APPLICATION=QUASI-STATIC, it is the time period of the step.

Defining a dynamic analysis in Abaqus/Explicit

Required parameters:

EXPLICIT

Include this parameter to specify explicit time integration.

Optional, mutually exclusive parameters:

DIRECT USER CONTROL

Include this parameter to specify that this step should use a fixed time increment that is specified by the user.

ELEMENT BY ELEMENT

Include this parameter to indicate that variable, automatic time incrementation using the element-by-element stable time increment estimates should be used. This method will generally require more increments and more computational time than the global time estimator.

FIXED TIME INCREMENTATION

Include this parameter to specify that this step should use a fixed time increment that will be determined by Abaqus/Explicit at the beginning of the step using the element-by-element time estimator.

Optional parameters:

ADIABATIC

Include this parameter to specify that an adiabatic stress analysis is to be performed. This parameter is relevant only for metal plasticity (*Inelastic Behavior*). The **INELASTIC HEAT FRACTION* and **SPECIFIC HEAT* options must be specified in the appropriate material definitions.

IMPROVED DT METHOD

Set IMPROVED DT METHOD=YES (default) to use the “improved” method to estimate the element stable time increment for three-dimensional continuum elements and elements with plane stress formulations (shell, membrane, and two-dimensional plane stress elements).

Set IMPROVED DT METHOD=NO to use the conservative method to estimate the element stable time increment for three-dimensional continuum elements and elements with plane stress formulations.

PORE PRESSURE

Set PORE PRESSURE=UNDRAINED to specify that this step should perform an undrained pore fluid flow and stress analysis. In this case the **POROUS BULK MODULI* and **DENSITY*, PORE FLUID options must be specified in the appropriate material definitions.

If this parameter is omitted, Abaqus/Explicit performs a regular stress analysis (optionally including predefined pore fluid pressure). If this parameter is specified without a value, Abaqus/Explicit issues an error message.

SCALE FACTOR

Set this parameter equal to the factor that is used to scale the time increment computed by Abaqus/Explicit. The default scaling factor is 1.0. This parameter can be used to scale the default global time estimate, and it can be used in conjunction with the ELEMENT BY ELEMENT and FIXED TIME INCREMENTATION parameters. It cannot be used in conjunction with the DIRECT USER CONTROL parameter.

Data line for automatic time incrementation (global or ELEMENT BY ELEMENT estimation):

First (and only) line:

1. Enter a blank field.
2. *T*, time period of the step.
3. Enter a blank field.
4. Δt_{max}, maximum time increment allowed. If this value is not specified, no upper limit is imposed.

Data line for fixed time incrementation using DIRECT USER CONTROL:

First (and only) line:

1. Δt, time increment to be used throughout the step.
2. *T*, time period of the step.

Data line for fixed time incrementation using FIXED TIME INCREMENTATION:

First (and only) line:

1. Enter a blank field.
2. *T*, time period of the step.

*DYNAMIC TEMPERATURE-DISPLACEMENT: Dynamic coupled thermal-stress analysis using explicit integration.

This option is used to indicate that a dynamic coupled thermal-stress analysis is to be performed using explicit integration.

Products: Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

• *Fully Coupled Thermal-Stress Analysis*

Required parameters:

EXPLICIT

Include this parameter to specify explicit time integration.

Optional, mutually exclusive parameters:

DIRECT USER CONTROL

Include this parameter to specify that this step should use a fixed time increment that is specified by the user.

ELEMENT BY ELEMENT

Include this parameter to indicate that variable, automatic time incrementation using the element-by-element stable time increment estimates should be used. This method will generally require more increments and more computational time than the global time estimator.

FIXED TIME INCREMENTATION

Include this parameter to specify that this step should use a fixed time increment that will be determined by Abaqus/Explicit at the beginning of the step using the element-by-element time estimator.

Optional parameters:

IMPROVED DT METHOD

Set IMPROVED DT METHOD=YES (default) to use the “improved” method to estimate the element stable time increment due to the mechanical response for three-dimensional continuum elements and elements with plane stress formulations (shell, membrane, and two-dimensional plane stress elements).

Set IMPROVED DT METHOD=NO to use the conservative method to estimate the element stable time increment due to the mechanical response for three-dimensional continuum elements and elements with plane stress formulations.

SCALE FACTOR

Set this parameter equal to the factor that is used to scale the time increment computed by Abaqus/Explicit. The default scaling factor is 1.0. This parameter can be used to scale the default global time estimate, and it can be used in conjunction with the ELEMENT BY ELEMENT and

FIXED TIME INCREMENTATION parameters. It cannot be used in conjunction with the DIRECT USER CONTROL parameter.

Data line for automatic time incrementation (global or ELEMENT BY ELEMENT estimation):

First (and only) line:

1. Enter a blank field.
2. *T*, time period of the step.
3. Enter a blank field.
4. Δt_{max}, maximum time increment allowed. If this value is not specified, no upper limit is imposed.

Data line for fixed time incrementation using DIRECT USER CONTROL:

First (and only) line:

1. Δt, time increment to be used throughout the step.
2. *T*, time period of the step.

Data line for fixed time incrementation using FIXED TIME INCREMENTATION:

First (and only) line:

1. Enter a blank field.
2. *T*, time period of the step.

E

This section describes each Abaqus input option that begins with E, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **EIGENSTRAIN*
- **ELASTIC*
- **ELCOPY*
- **ELECTRICAL CONDUCTIVITY*
- **ELECTRICAL RESISTIVITY*
- **ELECTRIC MACHINE LOAD*
- **ELECTRIC MACHINE PROPERTY*
- **ELECTROMAGNETIC*
- **ELEMENT*
- **ELEMENT MATRIX OUTPUT*
- **ELEMENT OPERATOR OUTPUT*
- **ELEMENT OUTPUT*
- **ELEMENT PROGRESSIVE ACTIVATION*
- **ELEMENT RECOVERY MATRIX*
- **ELEMENT RESPONSE*
- **ELEMENT SOLUTION-DEPENDENT VARIABLES*
- **ELEMENT USER OUTPUT VARIABLES*
- **EL FILE*
- **ELGEN*
- **EL PRINT*
- **ELSET*
- **EMBEDDED ELEMENT*
- **EMISSIVITY*
- **END ASSEMBLY*
- **END INSTANCE*
- **END LOAD CASE*
- **END PART*
- **END STEP*
- **ENERGY FILE*
- **ENERGY OUTPUT*
- **ENERGY PRINT*
- **ENRICHMENT*
- **ENRICHMENT ACTIVATION*
- **EOS*
- **EOS COMPACTION*
- **EPJOINT*

- *EQUATION
- *EQUIVALENT RADIATED SURFACE PROPERTIES
- *EULERIAN BOUNDARY
- *EULERIAN MESH MOTION
- *EULERIAN SECTION
- *EVENT SERIES
- *EVENT SERIES TYPE
- *EXPANSION
- *EXTERNAL FIELD
- *EXTREME ELEMENT VALUE
- *EXTREME NODE VALUE
- *EXTREME VALUE

*EIGENSTRAIN: Define your eigenstrain based on different physics.

This option is used to define eigenstrains at the material integration point.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Modeling the Cure Process in Thermosetting Polymers*
- *About Electrochemical Analysis Procedures*

Required parameters:

USER

Include this parameter to activate the usage of user subroutine *UEIGENSTRAIN* in procedures that support electrochemical analysis. When this parameter is used with a material name that starts with `"ABQ_"`, the built-in eigenstrain capability will be used.

Optional parameters:

PROPERTIES

Set this parameter equal to the number of real (floating point) property values needed as data in user subroutine *UEIGENSTRAIN*. The default is PROPERTIES=0. This parameter is valid only in procedures that support electrochemical analysis (see *About Electrochemical Analysis Procedures*).

There are no data lines associated with this option.

*ELASTIC: Specify elastic material properties.

This option is used to define linear elastic moduli. In an Abaqus/Standard analysis spatially varying isotropic, orthotropic (including engineering constants and lamina), or anisotropic linear elastic moduli can be defined for solid continuum elements using a distribution (*Distribution Definition*).

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Linear Elastic Behavior*

Optional parameters:

COHESIVE OFFSET

This parameter is relevant only for pore pressure, coupled temperature-pore pressure, and coupled slurry-temperature-pore pressure cohesive elements in Abaqus/Standard. It must be used in conjunction with either TYPE=COUPLED TRACTION or TYPE=TRACTION.

Set COHESIVE OFFSET=USER to define cohesive offset in user subroutine *UCOHESIVEOFFSET*.

COMPRESSION FACTOR

This parameter is meaningful only for uncoupled traction-separation elastic behavior.

Set this parameter equal to the factor by which the elastic modulus, E_{nn}, must be scaled in compression. The use of a factor that is different from 1.0 results in different elastic moduli in tension and compression.

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the moduli. If this parameter is omitted, it is assumed that the moduli are constant or depend only on temperature. See *Material Data Definition* for more information.

This parameter is not relevant in an Abaqus/Standard analysis if spatially varying elastic moduli are defined using a distribution. See *Distribution Definition*.

MODULI

This parameter is applicable only when the **ELASTIC* option is used in conjunction with the **VISCOELASTIC* option.

Set MODULI=INSTANTANEOUS to indicate that the elastic material constants define the instantaneous behavior. This parameter value is not available for frequency domain viscoelasticity in an Abaqus/Standard analysis.

Set MODULI=LONG TERM (default) to indicate that the elastic material constants define the long-term behavior.

TYPE

Set TYPE=ANISOTROPIC to define fully anisotropic behavior.

Set TYPE=BILAMINA to define an orthotropic material with a different Young's modulus and Poisson's ratio in tension and compression in plane stress. This parameter setting is applicable only in Abaqus/Explicit.

Set TYPE=COUPLED TRACTION to define coupled traction behavior for cohesive elements.

Set TYPE=ENGINEERING CONSTANTS to define orthotropic behavior by giving the "engineering constants" (the generalized Young's moduli, the Poisson's ratios, and the shear moduli in the principal directions).

Set TYPE=ISOTROPIC (default) to define isotropic behavior.

Set TYPE=LAMINA to define an orthotropic material in plane stress.

Set TYPE=ORTHOTROPIC to define orthotropic behavior by giving the elastic stiffness matrix directly.

Set TYPE=SHEAR to define the (isotropic) shear elastic modulus. This parameter setting is applicable only in conjunction with the **EOS* option in Abaqus/Explicit.

Set TYPE=SHORT FIBER to define laminate material properties for each layer in each shell element. This parameter setting is applicable only when using Abaqus/Standard in conjunction with the **abaqus moldflow** execution procedure. Any data lines given will be ignored. Material properties will be read from the ASCII neutral file identified as *jobid*`.shf`. See *Translating Moldflow Data to Abaqus Input Files* for more information.

Set TYPE=TRACTION to define orthotropic shear behavior for 1-DOF warping elements or uncoupled traction behavior for cohesive elements.

Set TYPE=TRANSVERSELY ISOTROPIC to define transversely isotropic behavior by giving the five "engineering constants" (the generalized Young's moduli, the Poisson's ratios, and the shear moduli in the principal directions).

When using a distribution to define elastic moduli, the TYPE parameter must be used to indicate the level of anisotropy in the elastic behavior. The level of anisotropy must be consistent with that defined in the distribution. See *Distribution Definition*.

Data lines to define fully anisotropic elasticity directly (TYPE=ANISOTROPIC):

First line:

1. D_{1111}. (Units of FL^{-2}.)
2. D_{1122}.
3. D_{2222}.
4. D_{1133}.
5. D_{2233}.
6. D_{3333}.
7. D_{1112}.
8. D_{2212}.

Second line:

1. D_{3312}.
2. D_{1212}.
3. D_{1113}.
4. D_{2213}.

5. D_{3313}.
6. D_{1213}.
7. D_{1313}.
8. D_{1123}.

Third line:

1. D_{2223}.
2. D_{3323}.
3. D_{1223}.
4. D_{1323}.
5. D_{2323}.
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

Data lines to define bilamina elasticity (TYPE=BILAMINA):

First line:

1. E_{1+}.
2. E_{2+}.
3. v_{12+}.
4. G_{12}.
5. E_{1-}.
6. E_{2-}.
7. v_{12-}.
8. Temperature.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than or equal to one):

1. First field variable.
2. Second field variable.
3. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

Data lines to define coupled traction separation behavior for cohesive elements (TYPE=COUPLED TRACTION):

First line:

1. E_{nn}.

2. E_{ss}.
3. E_{tt}.
4. E_{ns}.
5. E_{nt}.
6. E_{st}.
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

Data lines to define orthotropic elasticity with moduli (TYPE=ENGINEERING CONSTANTS):

First line:

1. E_1.
2. E_2.
3. E_3.
4. ν_{12}.
5. ν_{13}.
6. ν_{23}.
7. G_{12}.
8. G_{13}.

Second line:

1. G_{23}.
2. Temperature, θ.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

Data lines to define isotropic elasticity (TYPE=ISOTROPIC):

First line:

1. Young's modulus, E.
2. Poisson's ratio, ν.
3. Temperature, θ.
4. First field variable.

5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

Data lines to define orthotropic elasticity in plane stress (TYPE=LAMINA):

First line:

1. E_1.
2. E_2.
3. ν_{12}.
4. G_{12}.
5. G_{13}. This shear modulus is needed to define transverse shear behavior in shells.
6. G_{23}. This shear modulus is needed to define transverse shear behavior in shells.
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

Data lines to define orthotropic elasticity directly (TYPE=ORTHOTROPIC):

First line:

1. D_{1111}. (Units of FL^{-2}.)
2. D_{1122}.
3. D_{2222}.
4. D_{1133}.
5. D_{2233}.
6. D_{3333}.
7. D_{1212}.
8. D_{1313}.

Second line:

1. D_{2323}.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

Data lines to define isotropic elastic shear behavior (TYPE=SHEAR):

First line:

1. Shear modulus, G. (Units of FL^{-2}.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic shear modulus as a function of temperature and other predefined field variables.

Data lines to define orthotropic shear behavior for 1-DOF warping elements or uncoupled traction behavior for cohesive elements (TYPE=TRACTION):

First line (only line for defining orthotropic shear behavior for 1-DOF warping elements; in this case the data cannot be defined as functions of temperature and/or field variables):

1. E for 1-DOF warping elements; E_{nn} for cohesive elements.
2. G_1 for 1-DOF warping elements; E_{ss} for cohesive elements.
3. G_2 for 1-DOF warping elements; E_{tt} for cohesive elements.
4. Temperature.
5. First field variable.
6. Etc., up to four field variables per line.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four; relevant only for defining uncoupled traction behavior of cohesive elements):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

Data lines to define transversely isotropic elasticity with moduli (TYPE=TRANSVERSELY ISOTROPIC):

First line:

1. E_1.
2. E_2.
3. ν_{12}.
4. ν_{23}.

5. G_{12}.
6. Temperature, θ.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

Data line to define spatially varying elastic behavior for solid continuum elements in an Abaqus/Standard analysis using a distribution. (Distributions are supported for TYPE=ISOTROPIC, TYPE=ENGINEERING CONSTANTS, TYPE=LAMINA, TYPE=ORTHOTROPIC, and TYPE=ANISOTROPIC):

First line:

1. Distribution name. The data defined in the distribution must be in units that are consistent with the prescribed TYPE.

*ELCOPY: Create elements by copying from an existing element set.

This option is used to copy an element set to create new elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Not applicable; copying portions of sketches and instancing of parts serve similar purposes.

References:

• *Element Definition*

Required parameters:

ELEMENT SHIFT

Set this parameter equal to an integer that will be added to each of the existing element numbers to define the element numbers of the elements being created.

OLD SET

Set this parameter equal to the name of the element set being copied. The elements that are copied are those that belong to this set at the time this option is encountered.

SHIFT NODES

Set this parameter equal to an integer that will be added to each of the node numbers of the existing elements to define the node numbers of the elements being created.

Optional parameters:

NEW SET

Set this parameter equal to the name of the element set to which the elements created by the operation will be assigned. If this parameter is omitted, the newly created elements are not assigned to an element set.

REFLECT

Include this parameter to modify the node numbering sequence on the elements being created, which is necessary in some cases to avoid creating elements that violate the Abaqus convention for counterclockwise element numbering. This parameter can be used only with continuum elements and usually is required only when the nodes have been generated using the **NCOPY* option.

There are no data lines associated with this option.

*ELECTRICAL CONDUCTIVITY: Specify electrical conductivity.

This option is used to define electrical conductivity for elements that support electric potential degree of freedom in a coupled thermal-electrical, coupled thermal-electrical-structural, coupled thermal-electrochemical, coupled thermal-electrochemical-structural, or coupled thermal-electrochemical-structural-pore pressure analysis. This option is also used to define electrical conductivity for electromagnetic elements in eddy current analyses.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Electrical Conductivity*
- *Coupled Thermal-Electrical Analysis*
- *Fully Coupled Thermal-Electrical-Structural Analysis*
- *Eddy Current Analysis*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of electrical conductivity. If this parameter is omitted, the electrical conductivity is assumed not to depend on any field variables but may still depend on temperature and frequency. See *Material Data Definition* for more information.

FREQUENCY

Include this parameter to specify electrical conductivity as a function of frequency in an eddy current analysis.

TYPE

Set TYPE=ISO (default) to define isotropic electrical conductivity. Set TYPE=ORTHO to define orthotropic electrical conductivity. Set TYPE=ANISO to define fully anisotropic electrical conductivity.

Data lines to define isotropic electrical conductivity (TYPE=ISO) if the FREQUENCY parameter is omitted:

First line:

1. Electrical conductivity. (Units of $CT^{-1}L^{-1}\varphi^{-1}$.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define isotropic electrical conductivity as a function of temperature and field variables.

Data lines to define isotropic electrical conductivity (TYPE=ISO) if the FREQUENCY parameter is included:

First line:

1. Electrical conductivity. (Units of $CT^{-1}L^{-1}\varphi^{-1}$.)
2. Frequency, in cycles/time.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define isotropic electrical conductivity as a function of frequency, temperature, and field variables.

Data lines to define orthotropic electrical conductivity (TYPE=ORTHO) if the FREQUENCY parameter is omitted:

First line:

1. σ^E_{11}. (Units of $CT^{-1}L^{-1}\varphi^{-1}$.)
2. σ^E_{22}.
3. σ^E_{33}.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define orthotropic electrical conductivity as a function of temperature and field variables.

Data lines to define orthotropic electrical conductivity (TYPE=ORTHO) if the FREQUENCY parameter is included:

First line:

1. σ^E_{11}. (Units of $CT^{-1}L^{-1}\varphi^{-1}$.)
2. σ^E_{22}.
3. σ^E_{33}.
4. Frequency, in cycles/time.
5. Temperature.

6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define orthotropic electrical conductivity as a function of frequency, temperature, and field variables.

Data lines to define anisotropic electrical conductivity (TYPE=ANISO) if the FREQUENCY parameter is omitted:

First line:

1. σ^E_{11}. (Units of $\mathrm{CT}^{-1}\mathrm{L}^{-1}\varphi^{-1}$.)
2. σ^E_{12}.
3. σ^E_{22}.
4. σ^E_{13}.
5. σ^E_{23}.
6. σ^E_{33}.
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define anisotropic electrical conductivity as a function of temperature and field variables.

Data lines to define anisotropic electrical conductivity (TYPE=ANISO) if the FREQUENCY parameter is included:

First line:

1. σ^E_{11}. (Units of $\mathrm{CT}^{-1}\mathrm{L}^{-1}\varphi^{-1}$.)
2. σ^E_{12}.
3. σ^E_{22}.
4. σ^E_{13}.
5. σ^E_{23}.
6. σ^E_{33}.
7. Frequency in cycles/time.
8. Temperature.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value equal to or greater than one):

1. First field variable.

2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define anisotropic electrical conductivity as a function of frequency, temperature, and field variables.

*ELECTRICAL RESISTIVITY: Specify electrical resistivity.

This option is used to define electrical resistivity for elements that support the electric potential degree of freedom in a coupled thermal-electrical, coupled thermal-electrical-structural, coupled thermal-electrochemical, coupled thermal-electrochemical-structural, or coupled thermal-electrochemical-structural-pore pressure analysis.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Coupled Thermal-Electrical Analysis*
- *Fully Coupled Thermal-Electrical-Structural Analysis*
- *Electrical Resistivity*
- *Piezoresistive Effect*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of electrical resistivity. If this parameter is omitted, the electrical resistivity is assumed not to depend on any field variables but may still depend on temperature. See *Material Data Definition* for more information.

TYPE

Set TYPE=ISOTROPIC (default) to define isotropic electrical resistivity.

Set TYPE=ORTHOTROPIC to define orthotropic electrical resistivity.

Set TYPE=ANISOTROPIC to define fully anisotropic electrical resistivity.

Data lines to define isotropic electrical resistivity (TYPE=ISOTROPIC):

First line:

1. Electrical resistivity. (Units of $C^{-1}TL\varphi$.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define isotropic electrical resistivity as a function of temperature and field variables.

Data lines to define orthotropic electrical resistivity (TYPE=ORTHOTROPIC):

First line:

1. ρ_{11}^{E}. (Units of $C^{-1}TL\varphi$.)
2. ρ_{22}^{E}.
3. ρ_{33}^{E}.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define orthotropic electrical resistivity as a function of temperature and field variables.

Data lines to define anisotropic electrical resistivity (TYPE=ANISOTROPIC):

First line:

1. ρ_{11}^{E}. (Units of $C^{-1}TL\varphi$.)
2. ρ_{12}^{E}.
3. ρ_{22}^{E}.
4. ρ_{13}^{E}.
5. ρ_{23}^{E}.
6. ρ_{33}^{E}.
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define anisotropic electrical resistivity as a function of temperature and field variables.

*ELECTRIC MACHINE LOAD: Define electromagnetic forces loads for an electric machine.

This option is used to define electromagnetic force loads for an electric machine. These forces are usually the result of a previous electromagnetics simulation and are changed to Abaqus loads internally.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

• *Applying Electric Machine Loads*

Required parameters:

PROPERTY

Set this parameter equal to a label to refer to the electric machine property definition in the model data.

Data lines to define an electric machine load:

First line:

1. Rotational speed (revolutions per time) at which the electrical machine operates.
2. Mechanical torque at which the electrical machine operates.
3. Electric machine magnet temperature.

Second line, which defines the eccentricity orientation:

1. *X*-coordinate of point *a*.
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.
4. *X*-coordinate of point *b*.
5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.
7. *X*-coordinate of point *c*.
8. *Y*-coordinate of point *c*.
9. *Z*-coordinate of point *c*.

Third line, which defines the rotor orientation:

1. *X*-coordinate of point *a*.
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.
4. *X*-coordinate of point *b*.
5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.
7. *X*-coordinate of point *c*.
8. *Y*-coordinate of point *c*.
9. *Z*-coordinate of point *c*.

Fourth line to define the electromagnetic forces:

1. Radial force magnitude.
2. Radial force phase shift in radians.

3. Tangential force magnitude.
4. Tangential force phase shift in radians.
5. Torque correction magnitude.
6. Torque correction phase shift in radians.
7. Time order of the forces and the torque.
8. Spatial order of the forces and the torque.
9. Slice number on which the forces and the torque are specified.

Repeat this data line as often as necessary to define different forces and torque for each spatial and temporal order in each slice.

*ELECTRIC MACHINE PROPERTY: Define electric machine properties (or parameters) needed to apply electric machine loads.

This option is used to define electric machine properties (or parameters) that are needed to apply electric machine loads in the step data. These geometry properties include the name of the stator surface, the number of electric machine teeth, the number of two-dimensional electromagnetic slices, and the stator orientation.

Products: Abaqus/Standard

Type: Model data

Level: Assembly

Abaqus/CAE: Property module

References:

• *Applying Electric Machine Loads*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the electric machine property definition.

Electric machine property names in the same input file must be unique.

SURFACE

Set this parameter equal to a label that defines the stator surface name.

Data lines to define an electric machine property:

First line:

1. Number of electric machine teeth.
2. Number of slices.
3. Number of lumping points (the default is one lumping point per tooth).

Second line, which defines the stator orientation (points a and b define the axial direction, and point c defines the middle of the electric machine first tooth line):

1. *X*-coordinate of point *a*.
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.
4. *X*-coordinate of point *b*.
5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.
7. *X*-coordinate of point *c*.
8. *Y*-coordinate of point *c*.
9. *Z*-coordinate of point *c*.

Third line:

1. Width of the first slice.
2. Width of the second slice.
3. Etc., up to the number of slices specified on the first data line.

*ELECTROMAGNETIC: Electromagnetic response.

This option is used to calculate the low-frequency electromagnetic response of a system.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Electromagnetic Analysis Procedures*
- *Eddy Current Analysis*

Required parameters:

LOW FREQUENCY

Include this parameter to specify that the electromagnetic response is calculated based on the standard low-frequency assumption of neglecting the effects of displacement currents in Maxwell's equations.

One of the following mutually exclusive parameters is required:

TIME HARMONIC

Include this parameter to calculate the time harmonic linear electromagnetic response of the system subjected to harmonic excitation.

TRANSIENT

Include this parameter to calculate the transient electromagnetic response of the system.

Optional parameters:

DIRECT

This parameter can be used only with the TRANSIENT parameter to select direct user control of the incrementation through the step. If this parameter is used, constant increments of the size defined by the first item on the data line are used. If this parameter is omitted, Abaqus/Standard will choose the increments (after trying the user's initial time increment for the first attempt at the first increment).

STABILIZATION

Include this parameter to activate the stabilization scheme that may be needed in some situations to obtain an electromagnetic solution. It defines a factor that is used by Abaqus in the stabilization computations. If this parameter is included without a value, the default value is assumed to be 1.0. The parameter may be set to a higher value to increase the stabilization or to a lower value to decrease it.

Data lines to define an electromagnetic analysis if the TIME HARMONIC parameter is included:

First line:

1. Lower limit of frequency range or a single frequency, in cycles/time.
2. Upper limit of frequency range, in cycles/time. If this value is given as zero, it is assumed that results are required at only one frequency and the remaining data items on the line are ignored.

3. Total number of points in the frequency range at which results should be given, including the end points. The minimum value is 2. If the value given is less than 2 (or omitted), the default value of 20 points is assumed.

Repeat this data line as often as necessary to define frequency ranges in which results are required.

Data lines to define an electromagnetic analysis if the TRANSIENT parameter is included:

First (and only) line:

1. Initial time increment. This value will be modified as required if the automatic time stepping scheme is used or will be used as the constant time increment if the DIRECT parameter is used. If this entry is zero or is not specified, a default value that is equal to the total time period of the step is assumed.
2. Time period of the step.
3. Minimum time increment allowed. If a smaller time increment than this value is needed, the analysis is terminated. If this entry is zero, a default value of the smaller of the suggested initial time increment or 10^{-5} times the time period of the step is assumed.
4. Maximum time increment allowed. Only used for automatic time incrementation. If this value is not specified, no upper limit is imposed.

*ELEMENT: Define elements by giving their nodes.

This option is used to define an element directly by specifying its nodes.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Mesh module

References:

• *Element Definition*

Required parameters:

TYPE

Set this parameter equal to the element type, as defined in *About the Element Library*.

For user elements, specify the U*n* type identification (see *User-Defined Elements*). The **USER ELEMENT* option must also appear in the same input file.

For substructures, specify SUBSTR or Z*n* type identification. The preferred setting is TYPE=SUBSTR. For more information, see *Using Substructures*.

Optional parameters:

ELSET

Set this parameter equal to the name of the element set to which these elements will be assigned.

FILE

This parameter applies only to Abaqus/Standard analyses. This parameter is meaningful only for substructures.

For TYPE=SUBSTR, set this parameter equal to the substructure name, which is used to name the substructure database files. If the substructure database files are located outside of the user directory, the FILE parameter value must include the relative path to the file location (see *Using Substructures*).

For TYPE=Z*n*, set this parameter equal to the prefix in the substructure name (see the **SUBSTRUCTURE GENERATE*, LIBRARY option).

See *Input Syntax Rules* for the syntax of such file names.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

OFFSET

When the **ELEMENT* option is used to define the connectivity of axisymmetric elements with asymmetric deformation in Abaqus/Standard, set this parameter equal to a positive offset number for use in specifying the additional nodes required in the connectivity (see *Element Definition* for more information). The default is OFFSET=100000.

When the *ELEMENT option is used to define the connectivity of gasket elements in Abaqus/Standard or cohesive elements, set the OFFSET parameter equal to a positive offset number for use in defining the remaining nodes of the element when only part of the element nodes are defined explicitly. If this parameter is omitted, you must specify the connectivity of the entire gasket or cohesive element on the data lines (see *Defining the Gasket Element's Initial Geometry* and *Defining the Cohesive Element's Initial Geometry*).

When the *ELEMENT option is used to define the connectivity of thermal fluid pipe elements or thermal fluid pipe connector elements in Abaqus/Standard, set the OFFSET parameter equal to a positive offset number for use in defining the remaining nodes of the element when only the first two element nodes are defined explicitly. If this parameter is omitted, you must specify the connectivity of the entire element on the data lines (see *Fluid Pipe Element Library* and *Fluid Pipe Connector Element Library*).

SOLID ELEMENT NUMBERING

This parameter applies only to Abaqus/Standard analyses.

This parameter can be used only when the *ELEMENT option is used to define gasket elements. Use this parameter to specify the connectivity of gasket elements using the node ordering of an equivalent solid element. Set it equal to the face number of the equivalent solid element that corresponds to the first face (SNEG) of the gasket element. If no value is assigned to this parameter, it is assumed that the first face (S1) of the solid element corresponds to the first face of the gasket element.

Data lines to define the elements:

First line:

1. Element number.
2. First node number forming the element.
3. Second node number forming the element.
4. Etc., up to 15 node numbers on this line.

The order of nodes for each element type (the element's connectivity) is given in *About the Element Library*.

Continuation lines (only required if the previous line ends with a comma):

1. Node numbers forming the element.

Repeat this set of data lines as often as necessary, with up to 16 integer values per line.

*ELEMENT MATRIX OUTPUT: Write element stiffness matrices and mass matrices to a file.

This option is used to write element stiffness matrices and, if available, mass matrices to the results file, a user-defined file, or the data file.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

• *About Output*

Required parameters:

ELSET

Set this parameter equal to the name of the element set for which this output request is being made.

Optional parameters:

DLOAD

Set DLOAD=YES to write the load vector from distributed loads on the element. The default is DLOAD=NO.

FILE NAME

This parameter can be used only with the parameter OUTPUT FILE=USER DEFINED. It is used to specify the name of the file (without extension) to which the data will be written. The extension `.mtx` will be added to the file name provided by the user; see *Input Syntax Rules* for the syntax of such file names. If this parameter is not included when OUTPUT FILE=USER DEFINED is specified, the output will be written to the data file.

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be written at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

MASS

Set MASS=YES to write the mass matrix. The default is MASS=NO.

OUTPUT FILE

Set OUTPUT FILE=RESULTS FILE (default) for the data to be written to the regular results file in the format specified in *Results File*.

Set OUTPUT FILE=USER DEFINED for the results to be written to a user-specified file in the format of the **USER ELEMENT*, LINEAR option (*User-Defined Elements*). The name of the file is specified using the FILE NAME parameter.

STIFFNESS

Set STIFFNESS=YES to write the stiffness matrix (or the operator matrix for heat transfer elements). The default is STIFFNESS=NO.

There are no data lines associated with this option.

*ELEMENT OPERATOR OUTPUT: Write element operator output to a SIM document.

This option is used to write thermal matrices to a SIM document in element-by-element or assembled form. It can be used only in uncoupled heat transfer analyses.



Note: This option is deprecated in favor of **OPERATOR OUTPUT* and will be removed in a future release.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

• *Generating Matrices from Nonlinear Procedures*

Optional parameters:

ASSEMBLED

Include this parameter to write assembled matrices. By default, element matrices are written.

DAMPING

Include this parameter to output the heat capacity matrix.

ELSET

Use this parameter to write matrices for a part of the model. Set this parameter equal to the name of an element set that contains all the elements in the selected part of the model. By default, matrices are generated for all supported elements in the whole model, including internal elements.

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be written at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

LOAD

Include this parameter to output the flux selected by the LOADTYPE parameter.

LOADTYPE

Use this parameter to select the type of load for output.

Set LOADTYPE=EXTERNAL (default) to output external heat flux.

Set LOADTYPE=NET to output net heat flux.

STIFFNESS

Include this parameter to output the thermal conductivity matrix.

There are no data lines associated with this option.

*ELEMENT OUTPUT: Define output database requests for element variables.

This option is used to write element variables to the output database. It must be used in conjunction with the *OUTPUT option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Output to the Output Database*
- *OUTPUT

One of the following mutually exclusive parameters is required when the *ELEMENT OUTPUT option is used in conjunction with the *OUTPUT, HISTORY option, unless the request is only for whole model output variables:

ELSET

Set this parameter equal to the name of the element set for which this output request is being made.

TRACER SET

This parameter applies only to Abaqus/Explicit analyses using adaptivity.

Set this parameter equal to the name of the tracer set for which this output request is being made.

Optional parameters when the *ELEMENT OUTPUT option is used in conjunction with the *OUTPUT, FIELD option:

ALLSECTIONPTS

Include this parameter to indicate that output should be written for all section points in shell, beam, or layered solid elements for which output is requested. If this parameter is present, section points specified on the data lines are ignored.

DIRECTIONS

This parameter applies only to Abaqus/Standard and Abaqus/Explicit analyses.

Set DIRECTIONS=YES (default) to write the element material directions to the output database. Set DIRECTIONS=NO to indicate that the element material directions should not be written to the output database.

ELSET

Set this parameter equal to the name of the element set for which this output request is being made.

The ELSET and EXTERIOR parameters are mutually exclusive. If this parameter and the EXTERIOR parameter are omitted, output is written for all the elements in the model.

EXTERIOR

This parameter applies only to Abaqus/Standard and Abaqus/Explicit analyses.

Include this parameter to restrict output to only the exterior three-dimensional elements.

The EXTERIOR and ELSET parameters are mutually exclusive. If this parameter and the ELSET parameter are omitted, output is written for all the elements in the model.

MICROMECHANICS

This parameter applies only to elements with multiscale materials.

Include this parameter to request output at both the macro-level and the micro-level. The constituent names are appended to the output variables for output at the micro-level.

POSITION

Set POSITION=AVERAGED AT NODES if the values being written are the averages of values extrapolated to the nodes of the elements in the set. Averaging occurs only over elements that contribute to a node that have the same element type and properties. This parameter value is valid only in Abaqus/Standard analyses.

Set POSITION=CENTROIDAL if values are being written at the centroid of the element (the centroid of the reference surface of a shell element, the midpoint between the end nodes in a beam element).

Set POSITION=INTEGRATION POINTS (default) if values are being written at the integration points at which the variables are actually calculated.

Set POSITION=NODES if the values being written are extrapolated to the nodes of each element in the set but not averaged at the nodes.

Optional parameters:

REBAR

This parameter applies only to rebar in membrane, shell, and surface elements.

This parameter can be used to obtain output only for the rebar in the element set specified; output for the matrix material is not given. It can be used with or without a value. If it is used without a value, the output is given for all rebar in the element set. Its value can be set to the name assigned to the rebar on the **REBAR LAYER* option to specify output for that particular rebar in the element set.

If this parameter is omitted in a model that includes rebar, the output requests govern the output for the matrix material only (except for section forces, when the forces in the rebar are included in the force calculation).

Rebar output can be obtained only in membrane, shell, or surface elements at the integration points and at the centroid of the element.

VARIABLE

Set VARIABLE=ALL to indicate that all element variables applicable to this procedure and material type should be written to the output database.

Set VARIABLE=PRESELECT to indicate that the default element output variables for the current procedure type should be written to the output database. Additional output variables can be requested on the data lines.

If this parameter is omitted, the element variables requested for output must be specified on the data lines.

Data lines to request element output:

First line (optional, and relevant only if integration point variables are being written for shell, beam, or layered solid elements in an Abaqus/Standard analysis or if integration point variables are being written for shell or beam elements in an Abaqus/Explicit analysis):

1. Specify a list of the section points in the shell, beam, or layered solid at which variables should be written to the output database. If this data line is omitted, the variables are written at the default output points. For section points on a meshed beam cross-section, specify a list of user-defined section point labels. For elbow elements the mid-through-thickness section point must be specified to allow COORD data display in Abaqus/CAE since this point is not among the default output points. A maximum number of 16 section points can be specified. Repeat **ELEMENT OUTPUT* as often as required if output at additional points is required.

Second line:

1. Specify the identifying keys for the output variables to be written to the output database. The keys are defined in *Abaqus/Standard Output Variable Identifiers* and *Abaqus/Explicit Output Variable Identifiers*.

Repeat the second data line as often as necessary to define the list of variables to be output to the output database.

*ELEMENT PROGRESSIVE ACTIVATION: Define the progressive element activation feature and its properties.



Warning: Development of this capability and user interface is expected to evolve in subsequent releases. Therefore, models may not be upward compatible without modification.

This option is used to specify elements that can be activated during an analysis.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance, Assembly

References:

- *Progressive Element Activation*
- *UEPACTIVATIONVOL*
- **ACTIVATE ELEMENTS*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the elements that will be activated during the analysis.

NAME

Set this parameter equal to a label that will be used to refer to this progressive element activation.

Optional parameters:

FOLLOW DEFORMATION

Set FOLLOW DEFORMATION=NO (default) for elements that have not yet been activated to remain in their initial positions.

Set FOLLOW DEFORMATION=YES for elements that have not yet been activated to follow the movement of the active elements to prevent their excessive deformation. This parameter is applicable in a stress displacement analysis; in a heat transfer analysis, where temperature is the only active degree of freedom, setting FOLLOW DEFORMATION=YES is not meaningful and will be changed to FOLLOW DEFORMATION=NO.

FREE SURFACE

Set FREE SURFACE=NONE (default) for convection or radiation boundary conditions to be applied on the full exposed areas of the element facets. If the element is partially filled, the cooling is applied on the full element facet area.

Set FREE SURFACE=FACET for user subroutine *UEPACTIVATIONFACET* to be called at the start of the increment for each element. You can prescribe the facet area and the increment time for which the facet area is exposed for cooling. You can prescribe the exposed areas on the element facets and the internal cut facet for partial element activation. This feature is supported only for lower-order elements.

There are no data lines associated with this option.

*ELEMENT RECOVERY MATRIX: Generate modal recovery matrices for a substructure.

This option is used to generate modal stress and strain recovery matrices for a substructure. These matrices can be used to recover components of the stress and strain tensors within the substructure from a solution given at all substructure degrees of freedom (generalized displacements) including the retained nodal degrees of freedom and the generalized coordinates corresponding to the substructure dynamic modes.

This option can be used only with a substructure generation analysis in Abaqus/Standard (**SUBSTRUCTURE GENERATE* option).

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- **SUBSTRUCTURE GENERATE*
- *Generating Substructures*

Required parameters:

POSITION

Set POSITION=AVERAGED AT NODES if the values being recovered are the averages of values extrapolated to the nodes of the elements in the set. Since variables may be discontinuous between elements with different properties, averaging occurs only over those elements contributing to a node that have the same type with the following priority (in descending order): solid, shell, beam, other element types. Therefore, if solid and shell elements contribute to a node, results averaged over solid elements overwrite the other results at the node.

Set POSITION=CENTROIDAL if values are being recovered at the centroid of the elements (the centroid of the reference surface of a shell element, the midpoint between the end nodes of a beam element).

Optional parameters:

ELSET

Set this parameter equal to the name of the element set for which the element results recovery should be enabled. If this parameter is omitted, the modal recovery matrices are generated for all elements in the model.

Data lines to define section point and tensor variables:

First line (optional):

1. Give a section point in the beam, shell, or layered solid at which tensor components should be recovered. Generation of the recovery matrices is supported for a single section point. If this data line is omitted, the recovery matrices are generated for a default section point number 1.

Second line:

1. Give the identifying keys for the tensor variables. Tensors S and E are supported.

*ELEMENT RESPONSE: Define element responses for design sensitivity analysis.

This option is used in a direct design sensitivity analysis to write element response sensitivities calculated at the integration points to the output database. It is used in an adjoint design sensitivity analysis to write an element response and its sensitivities with respect to the appropriate design variables to the output database. It must be used in conjunction with the **DESIGN RESPONSE* option.

Products: Abaqus/Standard Abaqus/Design

Type: History data

Level: Step

References:

- *Direct Design Sensitivity Analysis*
- *Adjoint Design Sensitivity Analysis*
- **DESIGN RESPONSE*

Optional parameters:

ELSET

Set this parameter equal to the name of the element set for which this sensitivity output is being made.

FATIGUE

This parameter is meaningful only in an adjoint design sensitivity analysis.

Include this parameter to define a fatigue design response that is computed based on a combined Abaqus and fe-safe workflow.

This parameter cannot be used with the NSET, ORIENTATION, and USER parameters.

LOWER BOUND

This parameter is meaningful only in an adjoint design sensitivity analysis.

This parameter is used to specify a lower bound on the design response value below which adjoint sensitivities are not calculated. That is, if the value of the design response value is below the specified lower bound, then only the design response value is written to the output database, but not the corresponding adjoint sensitivities.

NSET

This parameter is meaningful only in an adjoint design sensitivity analysis and only for INTF*n* and INTFMAG design responses.

Set this parameter equal to the name of the node set for which this sensitivity output is being made. The element response is summed over all the nodes in this node set.

OPERATOR

This parameter is meaningful only in an adjoint design sensitivity analysis.

For the stress (MISES), plastic strain magnitude (PEMAG), or user-defined element responses, set this parameter equal to the method used for calculating the values of the responses and the approximated sensitivities based on the values of these quantities at the element integration points of the element set.

For the internal force (INTF*n* and INTFMAG) element responses, set this parameter equal to the method for calculating the values of the responses and the corresponding sensitivities based on the values of these quantities in the specified element set.

Set OPERATOR=MAX (default for STRESS, PEMAG, and user-defined responses) to indicate that the calculation method for the design response represents the maximum value of the element set.

Set OPERATOR=MAXABS to indicate that the calculation method for the design response represents the maximum absolute value of the element set. This parameter is not available for design responses in a steady-state dynamic analysis.

Set OPERATOR=MINABS to indicate that the calculation method for the design response represents the minimum absolute value of the element set. This parameter is not available for design responses in a steady-state dynamic analysis.

Set OPERATOR=MIN to indicate that the calculation method for the design response represents the minimum value for the element set.

Set OPERATOR=SUB to indicate that the calculation method for the design response represents the difference between the maximum and the minimum values for the element set.

Set OPERATOR=AVE to indicate that the calculation method for the design response represents the average value for the element set.

Set OPERATOR=SUM (default for INTF*n* and INTFMAG) to indicate that the calculation method for the design response represents the sum over the element set.

ORIENTATION

This parameter is meaningful only in an adjoint design sensitivity analysis and only for INTF*n* and INTFMAG design responses.

Set this parameter equal to the name of an orientation definition to be used to define a local coordinate system for the element response.

UPPER BOUND

This parameter is meaningful only in an adjoint design sensitivity analysis.

This parameter is used to specify an upper bound on the design response value above which adjoint sensitivities are not calculated. That is, if the value of the design response value is above the specified upper bound, then only the design response value is written to the output database, but not the corresponding adjoint sensitivities.

USER

This parameter is meaningful only in an adjoint design sensitivity analysis.

Include this parameter to define the element design response, based on either the stress or the plastic strain tensor, in user subroutine *UELEMDRESP*.

Data lines to request element sensitivity output if the USER parameter is omitted:

First line (only line in an adjoint design sensitivity analysis):

1. Specify the identifying keys for the responses whose sensitivities are to be written to the output database. Only one response key can be specified in an adjoint design sensitivity analysis. The valid keys are listed in *Direct Design Sensitivity Analysis* and *Adjoint Design Sensitivity Analysis*.

In a direct design sensitivity analysis, repeat this data line as often as necessary to define the element responses whose sensitivities are to be written to the output database.

Data lines to request element sensitivity output if the USER parameter is included:

First (and only) line:

1. Element response identifier label.
2. Table collection name or blank if no table collection is used (see **TABLE COLLECTION*).
3. Output variable identifier (S or PE).

*ELEMENT SOLUTION-DEPENDENT VARIABLES: Allocate space for element solution-dependent variables, and specify optional output names and descriptions for some or all variables allocated.

This option is used to allocate space at each element for element solution-dependent variables. The **ELEMENT SOLUTION-DEPENDENT VARIABLES* option can be used only in conjunction with the **SHELL SECTION* option or the **SOLID SECTION* option. The element solution-dependent variables defined with this option affect only those elements whose section properties are defined by the immediately preceding section option. In addition, an output name and a description can be given for some or all of the element solution-dependent variables allocated by this option.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- **SOLID SECTION*
- **SHELL SECTION*
- *About User Subroutines and Utilities*

Optional parameters:

DELETE

Set this parameter equal to the variable number controlling the element deletion flag. If this parameter is omitted, no element deletion is controlled by element solution-dependent variables.

Data lines:

First line:

1. Number of element solution-dependent variables required at each element.

Second line (required only to specify output descriptions for select element solution-dependent variables):

1. Index of the element solution-dependent variable for which an output name and a description are being given. This value is 1 for the first element solution-dependent variable.
2. The output variable name. The name is treated as a label; therefore, it must adhere to the conventions for labels (see *Input Syntax Rules*), with the exception that case will be preserved.
3. The output variable description. The description is treated as a label; therefore, it must adhere to the conventions for labels (see *Input Syntax Rules*), with the exception that case will be preserved.

Repeat this data line for each element solution-dependent variable for which an output name and a description are being defined. If field or history output of element solution-dependent variables is requested using the **ELEMENT OUTPUT* option, the output identifier for element solution-dependent variables for which a name has been specified under this option will consist of the string “ESDV_,” followed by the specified name. Similarly, the descriptions specified under this option will be used in the corresponding field descriptions written to the output database. If an output name and a description are not given for an element solution-dependent variable, the default output identifier ESDV*n* and description "Element solution-dependent variables" are used.

*ELEMENT USER OUTPUT VARIABLES: Specify the number of user-defined element output variables.

This option is used to allow Abaqus/Standard to allocate space at each element for user output variables, which are defined in user subroutine *UVAREL*.

Products: Abaqus/Standard

Type: Model data

Level: Assembly

References:

• *UVAREL*

One of the following mutually exclusive parameters is required:

If you specify two definitions at the same element (one with the ELSET and another with the GLOBAL parameter), Abaqus/Standard applies the ELSET definition.

ELSET

Set this parameter equal to the name of the element set for which you want to define element user variables.

You cannot use the ELSET parameter to define multiple element user variables at the same element.

GLOBAL

Include this parameter to define element user variables for all continuum and shell elements in the model.

Optional parameter:

PROPERTIES

Set this parameter equal to the number of property values needed as data in user subroutine *UVAREL*.

The default value is 0.

Data line to specify the number of element user variables if the PROPERTIES parameter is omitted or set to 0:

First (and only) line:

1. Number of whole element user-defined output variables required at an element.
2. Number of user-defined output variables required at the integration points of an element.
3. Table collection name associated with the element user-defined variable definition.

Second line (required only to specify output descriptions for select element user-defined variables):

1. Identifier of the element user-defined output variable for which an output key and a description are being given.

   The identifier consists of the element user variable name and index. For instance, use UVARE*n* and UVARPT*n* as identifiers of the n-th user-defined output variables for the whole element and for the integration points of the element, respectively.
2. The element user-defined output variable key.

The key is treated as a label; therefore, it must adhere to the conventions for labels (see *Input Syntax Rules*), with the exception that case is preserved.

3. The element user-defined output variable description.

   The description is treated as a label; therefore, it must adhere to the conventions for labels (see *Input Syntax Rules*), with the exception that case is preserved.

Repeat this data line for each element user-defined output variable for which an output name and a description are being defined. If you use the **ELEMENT OUTPUT* option to request field or history output of element user variables, the output identifier for element user variables for which a name has been specified under this option consists of the string “UVARE_,” or “UVARPT_,” followed by the specified name. Similarly, the descriptions specified under this option are used in the corresponding field descriptions written to the output database. If you do not specify an output name and a description for an element user-defined output variable, Abaqus/Standard uses the output identifier UVAREn and description "User-defined element output variables for whole element" or the output identifier UVARPTn and description "User-defined element output variables at integration points."

Data line to specify the number of element user variables if the PROPERTIES parameter is used:

First line:

1. Number of element user-defined variables required at an element.
2. Number of element user-defined variables required at an integration point.
3. Table collection name associated with the element user-defined varaible definition.

Second line:

1. Specify the properties, eight per line.

Repeat this data line as often as necessary to define the properties.

Subsequent line (required only to specify output descriptions for select element user-defined variables):

1. Identifier of the element user-defined output variable for which an output key and a description are being given.

   The identifier consists of the element user variable name and index. The index value is 1 for the first element user-defined output variable.

2. The element user-defined output variable key.

   The key is treated as a label; therefore, it must adhere to the conventions for labels (see *Input Syntax Rules*), with the exception that case is preserved.

3. The element user-defined output variable description.

   The description is treated as a label; therefore, it must adhere to the conventions for labels (see *Input Syntax Rules*), with the exception that case is preserved.

Repeat this data line for each element user-defined output variable for which an output name and a description are being defined. If you use the **ELEMENT OUTPUT* option to request field or history output of element user variables, the output identifier for element user variables for which a name has been specified under this option consists of the string “UVARE_,” or “UVARPT_,” followed by the specified name. Similarly, the descriptions specified under this option are used in the corresponding field descriptions written to the output database. If you do not specify an output name and a description for an element user-defined output variable, Abaqus/Standard uses the output identifier UVAREn and description "User-defined element output variables for whole element" or the output identifier UVARPTn and description "User-defined element output variables at integration points."

*EL FILE: Define results file requests for element variables.

This option is used to select the element variables that will be written to the results (`.fil`) file in an Abaqus/Standard analysis or to the selected results (`.sel`) file in an Abaqus/Explicit analysis. In an Abaqus/Explicit analysis it must be used in conjunction with the **FILE OUTPUT* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Unsupported; Abaqus/CAE reads output from the output database file only.

References:

- *Output to the Data and Results Files*
- **FILE OUTPUT*

Optional parameters:

DIRECTIONS

This parameter applies only to Abaqus/Standard analyses.

This parameter is used to obtain the directions of local element or material coordinate systems when component output is requested. The directions are written as a separate record for each point at which a local coordinate system is used. See *Results File* for a detailed description.

Set DIRECTIONS=NO (default) if the local coordinate directions should not be written.

Set DIRECTIONS=YES if the local coordinate directions should be written.

ELSET

Set this parameter equal to the name of the element set for which this output request is being made. If this parameter is omitted, the output will be written for all elements in the model. In an Abaqus/Explicit analysis, output will also be written for all of the rebars in the model. The REBAR parameter must be included in an Abaqus/Standard analysis to obtain rebar output.

FREQUENCY

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the output frequency, in increments. The output will always be written to the results file at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

LAST MODE

This parameter applies only to Abaqus/Standard analyses.

This parameter is useful only during eigenvalue extraction for natural frequencies (*Natural Frequency Extraction*) and for eigenvalue buckling estimation (*Eigenvalue Buckling Prediction*). Set this parameter equal to the highest mode number for which output is required.

The default value is LAST MODE=*N*, where *N* is the number of modes extracted. If the MODE parameter is used, the default value is LAST MODE=*M*, where *M* is the value of the MODE parameter.

MODE

This parameter applies only to Abaqus/Standard analyses.

This parameter is useful only during eigenvalue extraction for natural frequencies (*Natural Frequency Extraction*) and for eigenvalue buckling estimation (*Eigenvalue Buckling Prediction*). Set this parameter equal to the first mode number for which output is required. The default is MODE=1. When performing a **FREQUENCY* analysis, the normalization will follow the format set by the NORMALIZATION parameter. Otherwise, the normalization is such that the largest displacement component in the mode has a magnitude of 1.0.

POSITION

This parameter applies only to Abaqus/Standard analyses.

Set POSITION=AVERAGED AT NODES if the values being written are the averages of values extrapolated to the nodes of the elements in the set. Since variables can be discontinuous between elements with different properties, Abaqus/Standard breaks the output into separate tables for different element property definitions within the element set specified. Abaqus/Standard will also output elements of differing types separately. Thus, averaging will occur only over elements that contribute to a node that have the same type.

Set POSITION=CENTROIDAL if values are being written at the centroid of the element (the centroid of the reference surface of a shell element, the midpoint between the end nodes of a beam element).

Set POSITION=INTEGRATION POINTS (default) if values are being written at the integration points at which the variables are actually calculated.

Set POSITION=NODES if the values being written are extrapolated to the nodes of each element in the set but not averaged at the nodes.

REBAR

This parameter applies only to Abaqus/Standard analyses.

This parameter can be used to obtain output only for the rebar in the element set specified; output for the matrix material will not be given. It can be used with or without a value. If it is used without a value, the output will be given for all rebar in the element set. Its value can be set to the name assigned to the rebar on the **REBAR* option to specify output for that particular rebar in the element set.

If this parameter is omitted in a model that includes rebar, the output requests govern the output for the matrix material only (except for section forces, when the forces in the rebar are included in the force calculation). Rebar output can be obtained only at the integration points in continuum and beam elements. In shell and membrane elements rebar output can be obtained at the integration points and at the centroid of the element.

Data lines to request element output in the results file in an Abaqus/Standard analysis:

First line (optional, and relevant only if integration point variables are being printed for shell, beam, or layered solid elements):

1. Give a list of the section points in the beam, shell, or layered solid at which variables should be written to the results file. If this data line is omitted, the variables are written at the default output points defined in *About the Element Library*. A maximum number of 16 section points can be specified. Repeat the **EL FILE* option as often as needed if output at additional points is required. For section points on a meshed beam cross-section, specify a list of user-defined section point labels. If this data line is omitted, all available section points will be written.

Second line:

1. Give the identifying keys for the output variables to be written to the results (`.fil`) file. The keys are defined in *Abaqus/Standard Output Variable Identifiers*.

Repeat the second data line as often as necessary to define the list of variables to be output to the results file.

Data lines to request element output in the selected results file in an Abaqus/Explicit analysis:

First line:

1. Give the identifying keys for the output variables to be written to the selected results (`.sel`) file. The keys are defined in *Abaqus/Explicit Output Variable Identifiers*.

Repeat this data line as often as necessary to define the list of variables to be output to the selected results file.

*ELGEN: Incremental element generation.

This option is used to generate elements incrementally.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Not applicable; elements are generated when you *mesh*the model.

References:

• *Element Definition*

Optional parameters:

ALL NODES

Include this parameter to increment the node numbers of rigid body reference nodes for IRS-type and drag chain elements and nodes used to define the direction of the first cross-section axis for beams in space. By default, these node numbers will not be incremented.

ELSET

Set this parameter equal to the name of the element set to which the elements, including the main element, will be assigned.

Data lines to generate elements incrementally:

First line:

1. Main element number.
2. Number of elements to be defined in the first row generated, including the main element.
3. Increment in node numbers of corresponding nodes from element to element in the row. The default is 1.
4. Increment in element numbers in the row. The default is 1.

If necessary, copy this newly created main row to define a layer of elements.

11. Number of rows to be defined, including the main row. The default is 1.
13. Increment in node numbers of corresponding nodes from row to row.
15. Increment in element numbers of corresponding elements from row to row.

If necessary, copy this newly created main layer to define a block of elements.

19. Number of layers to be defined, including the main layer. The default is 1.
21. Increment in node numbers of corresponding nodes from layer to layer.
23. Increment in element numbers of corresponding elements from layer to layer.

Repeat this data line as often as necessary. Each line will generate $N1 \times N2 \times N3$ elements, where $N1$ is the number of elements in a row, $N2$ is the number of rows in a layer, and $N3$ is the number of layers.

*EL PRINT: Define data file requests for element variables.

This option is used to provide tabular printed output of element variables (stresses, strains, etc.).

Products: Abaqus/Standard

Type: History data

Level: Step

References:

• *Output to the Data and Results Files*

Optional parameters:

ELSET

Set this parameter equal to the name of the element set for which this output request is being made. If this parameter is omitted, the output will be printed for all elements in the model.

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

LAST MODE

This parameter is useful only during eigenvalue extraction for natural frequencies (*Natural Frequency Extraction*), complex eigenvalue extraction (*Complex Eigenvalue Extraction*), and for eigenvalue buckling estimation (*Eigenvalue Buckling Prediction*). Set this parameter equal to the highest mode number for which output is required.

The default value is LAST MODE=*N*, where *N* is the number of modes extracted. If the MODE parameter is used, the default value is LAST MODE=*M*, where *M* is the value of the MODE parameter.

MODE

This parameter is useful only during natural frequency extraction, complex eigenvalue extraction, and eigenvalue buckling estimation. Set this parameter equal to the first mode number for which output is required. The default is MODE=1. When performing a SIM-based **FREQUENCY* analysis (EIGENSOLVER=AMS, EIGENSOLVER=LANCZOS, SIM, or EIGENSOLVER=SUBSPACE, SIM) eigenvectors are always mass normalized. Otherwise, the normalization will follow the format set by the NORMALIZATION parameter, with DISPLACEMENT as the default.

POSITION

Set POSITION=AVERAGED AT NODES if the values being printed are the averages of values extrapolated to the nodes of the elements in the set. Since variables may be discontinuous between elements with different properties, Abaqus/Standard breaks the output into separate tables for different element property definitions within the element set specified. Abaqus/Standard will also output elements of differing types separately. Thus, averaging will occur only over elements that contribute to a node that have the same type.

Set POSITION=CENTROIDAL if values are being printed at the centroid of the element (the centroid of the reference surface of a shell element, the midpoint between the end nodes of a beam element).

Set POSITION=INTEGRATION POINTS (default) if values are being printed at the integration points at which the variables are actually calculated.

Set POSITION=NODES if the values being written are extrapolated to the nodes of each element in the set but not averaged at the nodes.

REBAR

This parameter can be used to obtain output only for the rebar in the element set specified; output for the matrix material will not be given. It can be used with or without a value. If it is used without a value, the output will be given for all rebar in the element set. Its value can be set to the name assigned to the rebar on the **REBAR* option to specify output for that particular rebar in the element set.

If this parameter is omitted in a model that includes rebar, the output requests govern the output for the matrix material only (except for section forces, when the forces in the rebar are included in the force calculation).

Rebar output can be obtained only at the integration points in continuum and beam elements. In shell and membrane elements rebar output can be obtained at the integration points and at the centroid of the element.

SUMMARY

Set SUMMARY=YES (default) to obtain a summary and the locations of the maximum and minimum values in each column of the table.

Set SUMMARY=NO to suppress this summary.

TOTALS

Set TOTALS=YES to print the total of each column in the table. This is useful, for example, to sum the energies of a set of elements. The default is TOTALS=NO.

Data lines to request element output in the data file:

First line (optional, and relevant only if integration point variables are being printed for shell, beam, or layered solid elements):

1. Give a list of the section points in the beam, shell, or layered solid at which variables should be printed. If this line is omitted, the variables are printed at the default output points defined in *About the Element Library*. For section points on a meshed beam cross-section, specify a list of user-defined section point labels. If this data line is omitted, all available section points will be printed. A maximum number of 16 section points can be specified. Repeat the **EL PRINT* option as often as needed if output at additional points is required.

Second line:

1. Give the identifying keys for the variables to be printed in a table for this element set. The keys are defined in *Abaqus/Standard Output Variable Identifiers*. All of the variables in each table must be of the same type (integration point, section point, or whole element variables).

Repeat the second data line as often as necessary: each line defines a table. If this line is omitted, no element output will be printed to the data file.

*ELSET: Assign elements to an element set.

This option is used to assign elements to an element set.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Part, Part instance, Assembly, Model, Step

Abaqus/CAE: Set toolset

References:

• *Element Definition*

Required parameters:

ELSET

Set this parameter equal to the name of the element set to which the elements will be assigned.

Optional parameters:

GENERATE

If this parameter is included, each data line should give a first element, e_1; a last element, e_2; and the increment in element numbers between these elements, i. Then, all elements going from e_1 to e_2 in steps of i will be added to the set. i must be an integer such that $(e_2 - e_1)/i$ is a whole number (not a fraction).

INSTANCE

Set this parameter equal to the name of the part instance that contains the elements listed on the data line. This parameter can be used only at the assembly level and is intended to be used as a shortcut to the naming convention. It can be used only in a model defined in terms of an assembly of part instances.

INTERNAL

Abaqus/CAE uses the INTERNAL parameter to identify sets that are created internally. The INTERNAL parameter is used only in models defined in terms of an assembly of part instances. The default is to omit the INTERNAL parameter.

UNSORTED

If this parameter is included, the elements in this element set will be assigned to the set (or added to the set if it already exists) in the order in which they are given.

If this parameter is omitted, the elements in the set are sorted into ascending order of their element numbers, with duplicates eliminated.

Data lines if the GENERATE parameter is omitted:

First line:

1. List of elements or element set labels to be assigned to this element set. Only previously defined element sets can be assigned to another element set.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

Data lines if the GENERATE parameter is included:

First line:

1. First element in set.
2. Last element in set.
3. Increment in element numbers between elements in the set. The default is 1.

Repeat this data line as often as necessary.

*EMBEDDED ELEMENT: Specify an element or a group of elements that lie embedded in a group of “host” elements in a model.

This option is used to specify an element or a group of elements that lie embedded in a group of “host” elements in a model.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Interaction module

References:

• *Embedded Elements*

Optional parameters:

ABSOLUTE EXTERIOR TOLERANCE

Set this parameter equal to the absolute value (given in the units used in the model) by which a node on the embedded element may lie outside the region of the host elements in the model. If this parameter is omitted or has a value of 0.0, the EXTERIOR TOLERANCE will apply.

EMBED NODES

Include this parameter if the host elements are expected to embed the nodes or node set specified on the data line.

EXTERIOR TOLERANCE

Set this parameter equal to the fraction of the average size of all the non-embedded elements in the model by which a node of the embedded element may lie outside the region of the host elements. The default is 0.05.

If both exterior tolerance parameters are specified by the user, Abaqus will use the smaller of the two tolerances.

HOST ELSET

Set this parameter equal to the name of the host element set in which the specified elements on the data lines are to be embedded. If this parameter is omitted, Abaqus will search all non-embedded elements in the model that lie in the vicinity of specified embedded elements.

PARTIAL EMBED

Set this parameter equal to YES if the host elements are expected to only partially embed the embedded elements. The default is NO.

ROUNDOFF TOLERANCE

Set this parameter equal to a small value below which the weight factors of the nodes on a host element associated with an embedded node will be zeroed out. The small weight factors will be distributed to the other nodes on the host element in proportion to their initial weights. The position of the embedded node will also be adjusted accordingly. The default value is 10^{-6}.

Data lines to define the elements embedded in the host elements:

First line:

1. List of elements or element set labels. Up to 16 entries are allowed per line.

Repeat this data line as often as necessary.

Data lines to define the nodes to be embedded in the host elements (EMBED NODES):

First line:

1. List of nodes or node set labels. Up to 16 entries are allowed per line.

Repeat this data line as often as necessary.

*EMISSIVITY: Specify surface emissivity.

This option is used to define the emissivity of a surface in a cavity radiation problem. It must appear immediately after the **SURFACE PROPERTY* option and must be used in conjunction with the **PHYSICAL CONSTANTS* option, which is used to define the Stefan-Boltzmann constant.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Cavity Radiation in Abaqus/Standard*
- **SURFACE PROPERTY*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of emissivity. If this parameter is omitted, the emissivity is assumed not to depend on any field variables (but may still depend on temperature). See *Material Data Definition* for more information.

Data lines to define the emissivity of a surface:

First line:

1. Emissivity, ϵ.
2. Temperature, if temperature dependent.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the emissivity as a function of temperature and user-defined field variables.

*END ASSEMBLY: End the definition of an assembly.

This option is used to end an assembly definition.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Assembly module

References:

- *Assembly Definition*
- **ASSEMBLY*

There are no parameters or data lines associated with this option.

*END INSTANCE: End the definition of an instance.

This option is used to end an instance definition.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Assembly

Abaqus/CAE: *Assembly module* for part instances not imported from a previous analysis; *Load module* for part instances imported from a previous analysis

References:

- *Assembly Definition*
- **INSTANCE*

There are no parameters or data lines associated with this option.

*END LOAD CASE: End the definition of a load case for multiple load case analysis.

This option is used to end a load case definition.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- *Multiple Load Case Analysis*
- **LOAD CASE*

There are no parameters or data lines associated with this option.

*END PART: End the definition of a part.

This option is used to end a part definition.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Part module

References:

- *Assembly Definition*
- **PART*

There are no parameters or data lines associated with this option.

*END STEP: End the definition of a step.

This option is used to end a step definition.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Model

Abaqus/CAE: Step module

References:

- *Defining an Analysis*
- **STEP*

There are no parameters or data lines associated with this option.

*ENERGY FILE: Write energy output to the results file.

This option is used to write a summary of the total energy content of a model to the results (`.fil`) file in an Abaqus/Standard analysis or to the selected results (`.sel`) file in an Abaqus/Explicit analysis. In an Abaqus/Explicit analysis it must be used in conjunction with the **FILE OUTPUT* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Unsupported; Abaqus/CAE reads output from the output database file only.

References:

- *Output to the Data and Results Files*
- **FILE OUTPUT*

Optional parameters:

ELSET

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the name of the element set for which this output request is being made. If this parameter is omitted, the energy for the whole model will be output.

FREQUENCY

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the output frequency, in increments. The output will always be written to the results file at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

There are no data lines associated with this option.

*ENERGY OUTPUT: Define output database requests for whole model or element set energy data.

This option is used to write whole model or element set energy requests to the output database. It must be used in conjunction with the *OUTPUT, HISTORY option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Output Variables*
- *Output to the Output Database*
- *Mode-Based Steady-State Dynamic Analysis*
- *OUTPUT

Optional parameters:

ELSET

Set this parameter equal to the name of the element set for which this output request is being made.

NSET

This parameter applies only to mode-based and direct-solution steady-state dynamic analyses.

Set this parameter equal to the name of the node set representing the boundary for energy flow, power flow, radiated acoustic energy, and radiated acoustic power computations.

The NSET and SURFACE parameters are mutually exclusive.

SURFACE

This parameter applies only to mode-based and direct-solution steady-state dynamic analyses.

Set this parameter equal to the name of the element-based surface representing the boundary for energy flow, power flow, radiated acoustic energy, and radiated acoustic power computations.

The SURFACE and NSET parameters are mutually exclusive.

PER ELEMENT SET

This parameter applies only to Abaqus/Explicit analyses.

Include this parameter to indicate that the requested energy variables are written to the output database for each user-defined element set (all internal element sets, including the internal element sets defined in Abaqus/CAE and the internal element sets created during the analysis, are excluded).

PER SECTION

This parameter applies only to Abaqus/Explicit analyses.

Include this parameter to indicate that the requested energy variables are written to the output database for every user-defined element set that is associated with a section definition (all internal element sets, including the internal element sets defined in Abaqus/CAE and the internal element sets created during the analysis, are excluded).

VARIABLE

Set VARIABLE=ALL to indicate that all energy variables applicable to this procedure and material type should be written to the output database.

Set VARIABLE=PRESELECT to indicate that the default energy output variables for the current procedure type should be written to the output database. Additional output variables can be requested on the data lines.

If this parameter is omitted and no energy variables are specified on the data lines, all energy variables will be written to the output database.

Data lines to request energy output:

First line:

1. Specify the identifying keys for the variables to be written to the output database. The keys are defined in *Output Variables*.

Repeat this data line as often as necessary to define the energy variables to be written to the output database.

*ENERGY PRINT: Print a summary of the total energies.

This option is used to print a summary of the total energy content of a whole model or part of a model to the data (`.dat`) file.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

• *Output to the Data and Results Files*

Optional parameters:

ELSET

Set this parameter equal to the name of the element set for which this output request is being made. If this parameter is omitted, the energy for the whole model will be output.

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

There are no data lines associated with this option.

*ENRICHMENT: Specify an enriched feature and the properties of the enrichment.

This option is used to define an enriched feature using the extended finite element method (XFEM). Enriched features are effective for modeling discontinuities, such as cracks, without conforming the mesh to the discontinued geometry. Only solid (continuum) elements can be associated with the enriched feature.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Interaction module

References:

• *Modeling Discontinuities as an Enriched Feature Using the Extended Finite Element Method*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the elements in which the degrees of freedom are enriched with special functions. The element set should consist of all the elements that are presently intersected by cracks and those that are likely to be intersected by cracks as the cracks propagate through the model.

NAME

Set this parameter equal to a label that will be used to refer to the name of the enriched feature in the model.

Optional parameters:

ENRICHMENT RADIUS

This parameter is relevant only when TYPE=STATIONARY CRACK.

Set this parameter equal to a small radius from the crack tip within which the elements are used for crack singularity calculations. The elements within the small radius should be included as part of the element set specified with the ELSET parameter. The default enrichment radius is six times the typical element characteristic length of those elements along the crack front in the enriched region.

INTERACTION

Set this parameter equal to the name of the **SURFACE INTERACTION* property definition associated with the contact interaction of cracked element surfaces based on a small-sliding formulation.

TYPE

Set TYPE=PROPAGATION CRACK (default) to model a discrete crack propagation along an arbitrary, solution-dependent path based on the extended finite element method.

Set TYPE=STATIONARY CRACK to model an arbitrary stationary crack based on the extended finite element method.

There are no data lines associated with this option.

*ENRICHMENT ACTIVATION: Activate or deactivate an enriched feature.

This option is used to activate or deactivate an enriched feature within the step definition.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

• *Modeling Discontinuities as an Enriched Feature Using the Extended Finite Element Method*

Required parameters:

NAME

Set this parameter equal to the name assigned to the enriched feature on the **ENRICHMENT* option.

Optional parameters:

ACTIVATE

Set ACTIVATE=ON (default) to activate this enriched feature within the step.

Set ACTIVATE=OFF to deactivate this enriched feature within the step.

Set ACTIVATE=AUTO OFF to deactivate this enriched feature automatically once all the pre-existing cracks (or if there are no pre-existing cracks, all the allowable newly nucleated cracks) have propagated through the boundary of the given enriched feature within the step.

Set ACTIVATE=MULTICRACKS to activate this enriched feature with the possibility of multiple crack nucleations within the step.

CRACK INITIATION RADIUS

This parameter can be used only in conjunction with ACTIVATE=MULTICRACKS.

Set this parameter equal to the relative radius around the crack tip to define a crack initiation suppression zone within which the elements in the enriched feature are excluded from crack nucleation. However, multiple crack nucleations are allowed to occur when the damage initiation criterion is satisfied in the elements lying outside the crack initiation suppression zone in the enriched feature. The default radius value is five times the typical element characteristic length in the enriched region.

TYPE

Set this parameter equal to the type of enriched feature specified on the **ENRICHMENT* option. Only TYPE=PROPAGATION CRACK (default) is supported.

There are no data lines associated with this option.

*EOS: Specify an equation of state model.

This option is used to define a hydrodynamic material model in the form of an equation of state.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Equation of State*

Required parameters:

TYPE

Set TYPE=IDEAL GAS for an ideal gas equation of state.

Set TYPE=IGNITION AND GROWTH for an ignition and growth equation of state; if this equation of state is used, the **REACTION RATE* option and the **GAS SPECIFIC HEAT* option are required.

Set TYPE=JWL for an explosive equation of state; if this equation of state is used, the **DETONATION POINT* option is required.

Set TYPE=TABULAR for a tabulated equation of state that is linear in energy.

Set TYPE=USER for a user-defined equation of state that is defined in user subroutine *VUEOS*.

Set TYPE=USUP for a linear $U_s - U_p$ equation of state.

Optional parameters:

DETONATION ENERGY

This parameter can be used only in combination with TYPE=IGNITION AND GROWTH.

Set this parameter equal to the energy of detonation. The default value is 0.0.

PROPERTIES

This parameter can be used only if the USER parameter is specified.

Set this parameter equal to the number of property values needed as data in user subroutine *VUEOS*. The default value is 0.

Data line for an ideal gas equation of state (TYPE=IDEAL GAS):

First (and only) line:

1. Gas constant, R. (Units of $\mathrm{JM}^{-1}\theta^{-1}$.)
2. The ambient pressure, p_A (Units of FL^{-2}). If this field is left blank, a default of **0.0** is used.

Data lines for an ignition and growth equation of state (TYPE=IGNITION AND GROWTH):

First line:

Material constants used in the equation of state for unreacted explosive.

1. A_s. (Units of FL^{-2}.)
2. B_s. (Units of FL^{-2}.)
3. ω_s. (Dimensionless.)
4. R_{1s}. (Dimensionless.)
5. R_{2s}. (Dimensionless.)

Second line:
Material constants used in the equation of state for reacted products.

1. A_g. (Units of FL^{-2}.)
2. B_g. (Units of FL^{-2}.)
3. ω_g. (Dimensionless.)
4. R_{1g}. (Dimensionless.)
5. R_{2g}. (Dimensionless.)

Data line for an explosive equation of state (TYPE=JWL):

First (and only) line:

1. Detonation wave speed, C_d. (Units of LT^{-1}.)
2. A. (Units of FL^{-2}.)
3. B. (Units of FL^{-2}.)
4. ω. (Dimensionless.)
5. R_1. (Dimensionless.)
6. R_2. (Dimensionless.)
7. Detonation energy density, E_0. (Units of JM^{-1}.)
8. Pre-detonation bulk modulus, K_{pd}. (Units of FL^{-2}.)

Data line for a tabulated equation of state (TYPE=TABULAR), where the volumetric strain values must be arranged in descending order:

First line:

1. f_1. (Units of FL^{-2}.)
2. f_2. (Dimensionless.)
3. Volumetric strain ε_{vol}. (Dimensionless.)

Repeat this data line as often as necessary to define the dependence of f_1 and f_2 on volumetric strain.

Data line for a linear equation of state (TYPE=USUP):

First (and only) line:

1. c_0, reference sound speed. (Units of LT^{-1}.)
2. s, slope of the $U_s - U_p$ curve. (Dimensionless.)
3. Γ_0, Grüneisen ratio. (Dimensionless.)

Data lines to define the material properties for a user-defined equation of state (TYPE=USER):

No data lines are needed if the PROPERTIES parameter is omitted or set to 0. Otherwise, first line:

1. Material properties, eight per line.

Repeat this data line as often as necessary to define the material properties.

*EOS COMPACTION: Specify plastic compaction behavior for an equation of state model.

This option is used to specify plastic compaction behavior for a hydrodynamic material. It must be used in conjunction with the *EOS, TYPE=USUP or *EOS, TYPE=TABULAR options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Equation of State*
- *EOS

There are no parameters associated with this option.

Data line to define the plastic compaction behavior:

First (and only) line:

1. Reference sound speed in the porous material, c_e. (Units of LT^{-1}.)
2. Value of the porosity of the unloaded (virgin) material, n_0 . (Dimensionless.)
3. Pressure required to initialize plastic behavior, p_e. (Units of FL^{-2}.)
4. Compaction pressure at which all pores are crushed, p_s. (Units of FL^{-2}.)

*EPJOINT: Define properties for elastic-plastic joint elements.

This option is used to define the properties for elastic-plastic joint elements. The **JOINT ELASTICITY* and, if plasticity is to be defined, **JOINT PLASTICITY* options must immediately follow this option.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- *Elastic-Plastic Joints*
- **JOINT ELASTICITY*
- **JOINT PLASTICITY*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the elastic-plastic joint elements for which properties are being defined.

ORIENTATION

Set this parameter equal to the name given to the **ORIENTATION* definition (*Orientations*) that gives the orientation of the local system in the joint.

Optional parameters:

SECTION

Set this parameter equal to SPUD CAN if the joint models a spud can. If the joint does not model a spud can, this parameter is not needed.

Data line to define spud can geometry with SECTION=SPUD CAN:

First (and only) line:

1. D_o, diameter of spud can cylindrical portion.
2. θ, conical spud can cone angle in degrees. Enter a blank, zero, or 180 for a cylindrical spud can.

Include the **JOINT ELASTICITY* and **JOINT PLASTICITY* options as needed to define the joint behavior.

*EQUATION: Define linear multi-point constraints.

This option is used to define linear multi-point constraints in the form of an equation.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Interaction module

References:

• *Linear Constraint Equations*

Optional parameters:

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

Data lines to define an equation:

First line:

1. Number of terms, *N*, in the equation.

Second line:

1. Node number or node set label, *P*, of first nodal variable, u_i^P.
2. Degree of freedom, *i*, at above node for variable u_i^P.
3. Value of A_1.
4. Node number or node set label, *Q*, of second nodal variable, u_j^Q.
5. Degree of freedom, *j*, at above node for variable u_j^Q.
6. Value of A_2.
7. Etc., up to four terms per line.

Repeat the second data line as often as necessary to define all of the terms of the equation. No more than four terms can be defined on a line. To define another constraint, repeat the entire set of data lines.

*EQUIVALENT RADIATED SURFACE PROPERTIES: Specify radiated surface properties.

This option is used to specify radiated surface properties. It can be used only as a suboption of the **SURFACE PROPERTY* option.

This option is required to calculate the following output variables:

- ERPWR (equivalent radiated power),
- ERPWRDEN (equivalent radiated power density),
- ERPAC (equivalent radiated acoustic pressure), and
- ALLERPWR (equivalent radiated power of a panel).

Products: Abaqus/Standard

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Radiated Energy and Power*

There are no parameters associated with this option.

Data line to define the acoustic medium properties on a sound-radiating element-based surface:

First (and only) line:

1. Density of the acoustic medium.
2. Phase speed (speed of sound).
3. Radiation power factor (the default is 1).

*EULERIAN BOUNDARY: Define inflow and outflow conditions at Eulerian mesh boundaries.

This option is used to specify inflow and outflow conditions at the boundaries of an Eulerian mesh.

Products: Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

• *Defining Eulerian Boundaries*

Optional parameters:

INFLOW

Set INFLOW=FREE (default) if Eulerian material can flow freely into the Eulerian domain.

Set INFLOW=NONE if neither Eulerian material nor void can flow into the Eulerian domain.

Set INFLOW=VOID if only void can flow into the Eulerian domain.

OP

Set OP=MOD (default) to modify existing inflow/outflow conditions or to define additional inflow/outflow conditions.

Set OP=NEW to remove all existing inflow/outflow conditions.

OUTFLOW

This parameter is used to define boundary conditions in unbounded domain problems.

Set OUTFLOW=FREE (default if INFLOW=VOID) if Eulerian material can flow freely out of the Eulerian domain.

Set OUTFLOW=NONE if Eulerian material cannot flow out of the Eulerian domain. The parameter value must be used with INFLOW=NONE. Use this combination to enforce tangential-only flow directions at the Eulerian boundary.

Set OUTFLOW=NONREFLECTING to specify a nonreflecting radiation boundary condition.

Set OUTFLOW=NONUNIFORM PRESSURE to specify an equilibrium condition at the boundary.

Set OUTFLOW=ZERO PRESSURE (default) to specify a zero pressure at the boundary.

Data lines to define the surface where Eulerian boundary conditions are applied:

First line:

1. Surface name.

Repeat this data line as often as necessary to define inflow/outflow conditions for different surfaces.

*EULERIAN MESH MOTION: Define the motion of an Eulerian mesh.

This option allows an Eulerian mesh to translate with the motion of a specified surface and expand and contract to encompass the surface's extent.

Products: Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- *Eulerian Analysis*
- *Eulerian Mesh Motion*

Required parameters:

ELSET

Set this parameter equal to the element set name given on the **EULERIAN SECTION* definition for which to activate mesh motion.

Required parameter when activating mesh motion for the first time or redefining mesh motion after OP=NEW is used:

SURFACE

Set this parameter equal to the name of a node-based, element-based, or Eulerian material surface used to control the motion of the Eulerian mesh.

Optional parameters:

ASPECT RATIO MAX

Set this parameter equal to the maximum change in the allowed aspect ratio of any of the three bounding box aspects (1–2, 2–3, 3–1). The default is 10.0.

BUFFER

Set this parameter equal to a value to maintain a buffer between the bounding box and surface equal to the value times the maximum Eulerian element size in the mesh. The default is BUFFER=2.0.

Set BUFFER=INITIAL to maintain the initial scaling of the mesh with respect to the surface.

CENTER

Set CENTER=BOUNDING BOX (default) to align the center of the bounding box with the center of the surface's bounding box.

Set CENTER=MASS to align the center of the bounding box with the center of mass of the surface.

CONTRACT

Set CONTRACT=YES (default) to allow the bounding box to contract during the analysis.

Set CONTRACT=NO to disallow contraction of the bounding box.

OP

Set OP=MOD (default) to modify existing mesh motion options or to define additional mesh motion options for the given element set.

Set OP=NEW to remove or overwrite an existing mesh motion definition for the given element set.

ORIENTATION

Set this parameter equal to the name given for the **ORIENTATION* option (*Orientations*) to be used to define the local directions of the bounding box. Only orientations defined with SYSTEM=RECTANGULAR or SYSTEM=Z RECTANGULAR can be specified.

VMAX FACTOR

Set this parameter equal to a fraction of the maximum velocity of the surface nodes to bound the mesh motion velocity. The default is VMAX FACTOR=1.01.

VOLFRAC MIN

Set this parameter equal to the lower bound on the volume fraction used to determine which nodes to include in the bounding box calculation for an Eulerian material surface. The default is VOLFRAC MIN=0.5.

Optional data lines to define bounding box constraints:

First line:

1. Value between 1.0 and (default): maximum scaling of the bounding box in local direction 1.
2. Value between 1.0 and (default): maximum scaling of the bounding box in local direction 2.
3. Value between 1.0 and (default): maximum scaling of the bounding box in local direction 3.
4. Value between 0.0 (default) and 1.0: minimum scaling of the bounding box in local direction 1.
5. Value between 0.0 (default) and 1.0: minimum scaling of the bounding box in local direction 2.
6. Value between 0.0 (default) and 1.0: minimum scaling of the bounding box in local direction 3.

Second line:

1. FREE (default) or FIXED: constraint flag for the negative local direction 1 face of the bounding box.
2. FREE (default) or FIXED: constraint flag for the positive local direction 1 face of the bounding box.
3. FREE (default) or FIXED: constraint flag for the negative local direction 2 face of the bounding box.
4. FREE (default) or FIXED: constraint flag for the positive local direction 2 face of the bounding box.
5. FREE (default) or FIXED: constraint flag for the negative local direction 3 face of the bounding box.
6. FREE (default) or FIXED: constraint flag for the positive local direction 3 face of the bounding box.

Third line:

1. FREE (default) or FIXED: constraint flag for the center of the bounding box in local direction 1.
2. FREE (default) or FIXED: constraint flag for the center of the bounding box in local direction 2.
3. FREE (default) or FIXED: constraint flag for the center of the bounding box in local direction 3.

*EULERIAN SECTION: Specify element properties for Eulerian elements.

This option is used to define properties of Eulerian continuum elements, including the list of materials that may occupy the elements.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- *Eulerian Analysis*
- *Eulerian Elements*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the Eulerian elements.

Optional parameters:

ADVECTION

Set ADVECTION=SECOND ORDER (default) to use a second-order algorithm to remap solution variables after remeshing has been performed.

Set ADVECTION=FIRST ORDER to use a first-order algorithm to remap solution variables after remeshing has been performed.

CONTROLS

Set this parameter equal to the name of a section controls definition (see *Section Controls*) to be used to specify a nondefault hourglass control formulation option or scale factor. The **SECTION CONTROLS* option can be used to select the hourglass control and order of accuracy of the formulation.

FLUX LIMIT RATIO

Set this parameter equal to the ratio between the maximum distance a node is allowed to move during one increment and the characteristic length of the Eulerian element containing the node. The value of this parameter must be positive. The default value is 1.0, and the suggested range for the value is between 0.1 and 1.0.

Data lines to define Eulerian elements:

First line:

1. Material name.
2. Material instance name (optional). The default material instance name is the same as the material name. Material instance names must be unique throughout the entire model. Specify a nondefault material instance name if you refer to the same material definition more than once.
3. Orientation name (optional). If not defined, the global coordinate system is used for the material. All orientation-dependent materials in the Eulerian section must be defined in the same local coordinate system.

Repeat this data line as often as necessary to define the list of all materials that may appear in the Eulerian section.

*EVENT SERIES: Define the event series data.

This option is used to define event series data.

Products: Abaqus/Standard

Type: Model or history data

Level: Model, Step

References:

- *About Event Series*
- *Accessing Abaqus Event Series Data*
- **EVENT SERIES TYPE*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to this event series data.

TYPE

Set this parameter equal to the name of the event series type.

Required parameter for reading event series data from a SIM database (`.sim`) file:

FILE

Set this parameter equal to the name of the SIM database (`.sim`) file from which the data are read. The file extension is optional. The SIM database file must be from the same fix pack (if applicable) of the same general release as the analysis.

Optional parameters:

INPUT

The INPUT and FILE parameters are mutually exclusive.

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

SOURCE NAME

This parameter is relevant only for use with the FILE parameter.

Set this parameter equal to the name of the event series in the SIM database (`.sim`) file. If this parameter is omitted, the name of the event series is used.

TIME

Set TIME=STEP TIME (default) for step time.

Set TIME=TOTAL TIME for total time accumulated over all non-perturbation analysis steps.

See *Conventions* for a discussion of these time measures.

TRANSFORM

Include this parameter to indicate that the event series data are to be translated and/or rotated.

Data lines to define event series data:

First line:

1. Time or frequency of the first event.
2. *X*-coordinate of the location of the first event.
3. *Y*-coordinate of the location of the first event.
4. *Z*-coordinate of the location of the first event.
5. The first field value of the first event.
6. The second field value of the first event.
7. Etc., up to four field values per line.

Subsequent lines (only needed if the number of fields is greater than four):

1. The fifth field value of the first event.
2. Etc., up to eight field values per line.

Repeat this set of data lines as often as necessary to define all of the events.

Data lines to define and translate/rotate event series data:

First line:

1. Value of the time shift. The default value is 0.
2. Value of the translation to be applied in the *X*-direction. The default value is 0.
3. Value of the translation to be applied in the *Y*-direction. The default value is 0.
4. Value of the translation to be applied in the *Z*-direction. The default value is 0.

Leave the first data line empty to apply a pure rotation.

Second line:

1. *X*-coordinate of point *a* on the axis of rotation (see *Figure 1*).
2. *Y*-coordinate of point *a* on the axis of rotation.
3. *Z*-coordinate of point *a* on the axis of rotation.
4. *X*-coordinate of point *b* on the axis of rotation.
5. *Y*-coordinate of point *b* on the axis of rotation.
6. *Z*-coordinate of point *b* on the axis of rotation.
7. Angle of rotation about the axis *a*–*b*, in degrees.

Leave the second data line empty to apply a pure translation. If both translation and rotation are specified, translation is applied before rotation.

Third line:

1. Time or frequency of the first event.
2. *X*-coordinate of the location of the first event.
3. *Y*-coordinate of the location of the first event.
4. *Z*-coordinate of the location of the first event.
5. The first field value of the first event.
6. The second field value of the first event.
7. Etc., up to four field values per line.

Subsequent lines (only needed if the number of fields is greater than four):

1. The fifth field value of the first event.
2. Etc., up to eight field values per line.

Repeat this set of data lines as often as necessary to define all of the events.

*EVENT SERIES TYPE: Define the type of an event series.

This option is used to define the type of an event series.

Products: Abaqus/Standard

Type: Model or history data

Level: Model, Step

References:

- *About Event Series*
- *Accessing Abaqus Event Series Data*
- **EVENT SERIES*

Required parameters:

FIELDS

Set this parameter equal to the number of fields that will be defined.

NAME

Set this parameter equal to a label that will be used to refer to the event series type.

Optional data lines:

First line:

1. Description of the first field. This field can be left blank and is unused in the analysis.

Repeat this data line as often as necessary to define all of the field descriptions as indicated by the value of the FIELDS parameter.

*EXPANSION: Specify thermal or field expansion.

This option is used to define thermal expansion or field expansion in Abaqus/Standard for a material or for the behavior of a gasket. In an Abaqus/Standard analysis spatially varying thermal expansion can be defined for solid continuum elements using a distribution (*Distribution Definition*).

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Thermal Expansion*
- *Field Expansion*
- *UEXPAN*
- *VUEXPAN*
- **TRS*

Optional parameters:

DEFINITION

Set DEFINITION=SECANT (default) to use secant (total) thermal expansion coefficients.

Set DEFINITION=TANGENT to use tangent (differential) thermal expansion coefficients. This parameter value is available only in Abaqus/Standard.

DEFINITION=TANGENT cannot be used if you include the FIELD, LIQUID, or PORE FLUID parameter; and it cannot be used with TYPE=SHORT FIBER.

DEPENDENCIES

This parameter applies only to Abaqus/Standard and Abaqus/Explicit analyses.

Set this parameter equal to the number of field variables, in addition to temperature, on which the coefficients depend. If this parameter is omitted, it is assumed that the thermal expansion is constant or depends only on temperature.

This parameter is not relevant if the USER parameter is included or if in an Abaqus/Standard analysis spatially varying thermal expansion is defined using a distribution (see *Distribution Definition*).

FIELD

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the predefined field variable number for which field expansion is being defined.

LIQUID

This parameter applies only to Abaqus/Standard analyses.

Include this parameter if the thermal expansion of the liquid is being defined for the TNM model.

PORE FLUID

Include this parameter if the thermal or field expansion of the pore fluid in a porous medium is being defined. The secant thermal expansion coefficients must be specified and thermal or field expansion

of a fluid must be isotropic; therefore, you cannot use TYPE=ORTHO and TYPE=ANISO if this parameter is included.

PROPERTIES

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the number of properties being entered. The properties are available for use in user subroutine *VUEXPAN*. The default is PROPERTIES=0.

TYPE

Set TYPE=ISO (default) to define isotropic expansion.

Set TYPE=ORTHO to define orthotropic expansion.

Set TYPE=ANISO to define fully anisotropic expansion.

Set TYPE=SHORT FIBER to define laminate material properties for each layer in each shell element. This parameter setting is applicable only when using Abaqus/Standard in conjunction with the **abaqus moldflow** execution procedure. Any data lines will be ignored. Material properties will be read from the ASCII neutral file identified as *jobid*`.shf`. See *Translating Moldflow Data to Abaqus Input Files* for more information.

Set TYPE=TRANSVERSELY ISOTROPIC to define transversely isotropic expansion.

In an Abaqus/Standard analysis spatially varying isotropic, orthotropic, or anisotropic expansion can be defined using a distribution. When using a distribution, the TYPE parameter must be used to indicate the level of anisotropy of thermal expansion. The level of anisotropy must be consistent with that defined in the distribution. See *Distribution Definition*.

USER

In Abaqus/Standard, include this parameter to indicate that user subroutine *UEXPAN* will be used to define increments of either the thermal strain or the field expansion strain (if the FIELD parameter is also used) or potentially increments of both the thermal and the field expansion strains.

In Abaqus/Explicit, include this parameter to indicate that user subroutine *VUEXPAN* will be used to define increments of thermal strain.

The TYPE parameter should be used to indicate the level of anisotropy of the thermal or the field expansion. TYPE=TRANSVERSELY ISOTROPIC cannot be used if the USER parameter is included.

The DEFINITION, DEPENDENCIES and ZERO parameters are not relevant if this parameter is used.

ZERO

If the thermal or field expansion is temperature- or field-variable dependent, set this parameter equal to the value of θ^0 (or $f_n{}^0$ for field expansion driven by field variable number n).

This parameter is not relevant if the USER parameter or the LIQUID parameter is included, or if the DEFINITION=TANGENT is used.

Data lines to define isotropic thermal expansion coefficients (TYPE=ISO with USER parameter omitted):

First line:

1. α in Abaqus/Standard or Abaqus/Explicit analysis. (Units of θ^{-1}.)
2. Temperature.
3. First field variable.
4. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the thermal expansion coefficient as a function of temperature and other predefined field variables.

Data lines to define orthotropic thermal expansion coefficients (TYPE=ORTHO with USER parameter omitted):

First line:

1. α_{11}. (Units of θ^{-1}.)
2. α_{22}.
3. α_{33}. (Not used for plane stress and shell cases.)
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the thermal expansion coefficients as functions of temperature and other predefined field variables.

Data lines to define anisotropic thermal expansion coefficients (TYPE=ANISO with USER parameter omitted):

First line:

1. α_{11}. (Units of θ^{-1}.)
2. α_{22}.
3. α_{33}. (Not used for plane stress and shell cases.)
4. α_{12}.
5. α_{13}. (Not used for plane stress and shell cases.)
6. α_{23}. (Not used for plane stress and shell cases.)
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the thermal expansion coefficients as functions of temperature and other predefined field variables.

Data lines to define transversely isotropic thermal expansion coefficients (TYPE=TRANSVERSELY ISOTROPIC):

First line:

1. α_{11}. (Units of θ^{-1}.)
2. α_{22}.

3. Temperature.
4. First field variable.
5. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the thermal expansion coefficients as functions of temperature and other predefined field variables.

Data line to define spatially varying thermal expansion in an Abaqus/Standard analysis using a distribution:

First (and only) line:

1. Distribution name. The data defined in the distribution must be in units of θ^{-1} and must be consistent with the level of anisotropy prescribed by the TYPE parameter.

To define thermal expansion by a user subroutine (USER parameter included):

No data lines are used with this option when the USER parameter is specified. Instead, user subroutine *UEXPAN* must be used to define the thermal expansion. In Abaqus/Explicit data lines can be used to specify user properties that are passed in user subroutine *VUEXPAN* to define the thermal expansion.

Data lines to define isotropic field expansion coefficients (TYPE=ISO with USER parameter omitted):

First line:

1. α_f. (Units of FV_n^{-1}.)
2. Temperature.
3. First field variable.
4. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the field expansion coefficient as a function of temperature and other predefined field variables.

Data lines to define orthotropic field expansion coefficients (TYPE=ORTHO with USER parameter omitted):

First line:

1. α_{f11}. (Units of FV_n^{-1}.)
2. α_{f22}.
3. $\alpha_{f_{33}}$.
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.

2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the field expansion coefficients as functions of temperature and other predefined field variables.

Data lines to define anisotropic field expansion coefficients (TYPE=ANISO with USER parameter omitted):

First line:

1. α_{f11}. (Units of FV_n^{-1}.)
2. α_{f22}.
3. α_{f33}. (Not used for plane stress case.)
4. α_{f12}.
5. α_{f13}.
6. α_{f23}.
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the field expansion coefficients as functions of temperature and other predefined field variables.

To define field expansion by a user subroutine (USER parameter included):

No data lines are used with this option when the USER parameter is specified. Instead, user subroutine *UEXPAN* must be used to define the field expansion.

*EXTERNAL FIELD: Import external fields to define distributions, initial conditions, and history-dependent fields or to perform a submodel analysis.

This option is used to import external fields. It can be used in conjunction with the **DISTRIBUTION* option to define a spatial distribution, with the **INITIAL CONDITIONS* option to prescribe initial conditions, or with the **FIELD IMPORT* option to define history-dependent fields (such as loads, boundary conditions, and predefined fields) or to perform a node-based submodel analysis.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model or history data

Level: Part, Part instance, Assembly, Model

References:

- **DISTRIBUTION*
- **INITIAL CONDITIONS*
- **FIELD MAPPER CONTROLS*
- **FIELD OPERATIONS*
- *General Capability for Importing External Fields*
- *Distribution Definition*
- *Initial Conditions*
- *Output Variables*

Using an external field to define distributions and initial conditions

Required parameters:

FILE

Set this parameter equal to the name of the SIM database (`.sim`) file from which results data are to be read.

Optional parameters:

INC

Set this parameter equal to the increment from which to import field values.

The INC parameter and the TIME parameter are mutually exclusive. This parameter is ignored if the MODE parameter is specified.

If this parameter is omitted and neither the MODE parameter or the TIME parameter is specified, the field values are read from the last increment of the specified step or from the last step if the STEP parameter is omitted.

MODE

Set this parameter equal to the mode number from which to import field values.

The MODE parameter and the TIME parameter are mutually exclusive. If the STEP parameter is omitted, the last step from the previous analysis is used.

STEP

Set this parameter equal to the step from which to import field values.

The STEP parameter and the TIME parameter are mutually exclusive.

If this parameter is omitted and the TIME parameter is not specified, the field values are read from the last step.

TIME

Set this parameter equal to the time at which to import field values. Linear interpolation is performed using two saved time points of the previous analysis to obtain the field values at the specified time. The time is specified as the total time.

The TIME parameter and the INC and STEP parameters are mutually exclusive.

If the TIME, STEP, and INC parameters are omitted, the last stored frame is imported.

Data lines to use an external field to define distributions and initial conditions:

First line:

1. Name space of the region in the model used in the current analysis. Use NODES to indicate that the region is a node set; use ELEMENTS to indicate that the region is an element set. If omitted, the default is ELEMENTS.
2. Name of the region (node set or element set) defined in the current model where imported fields are specified. If omitted, imported fields are specified on the entire model in the current analysis.

   The region can be either a node set or an element set regardless of the location where imported fields are specified. When a nodal field is specified on an element set, the field data are specified on all nodes of the elements in the specified element set. When an element field is specified on a node set, the field data are specified on all elements containing the nodes in the node set; elements connected to nodes that are not in the node set are not included.
3. Identifying key for the field imported when **EXTERNAL FIELD* is used with the following initial conditions:

   - **INITIAL CONDITIONS*, TYPE=DAMAGE INITIATION, CRITERION=MSFLD; use MSFLDCRT as the identifying key for the Müschenborn and Sonne forming limit diagram–based damage initiation criterion and use ERPRATIO as the identifying key for the ratio of the principal strain rates.
   - **INITIAL CONDITIONS*, TYPE=ESDV; use ESDV*n* as the identifying key for the n^{th} element solution-dependent variable.
   - **INITIAL CONDITIONS*, TYPE=HARDENING with the NUMBER BACKSTRESSES parameter specified; use PEEQ as the identifying key for the equivalent plastic strain, and use ALPHA*n* as the identifying key for the n^{th} initial backstress.
   - **INITIAL CONDITIONS*, TYPE=SOLUTION; use SDV*n* as the identifying key for the n^{th} solution-dependent state variable.

   Otherwise, not used.
4. Name space of the region in the model used in the previous analysis. Use NODES to indicate that the region is a node set; use ELEMENTS to indicate that the region is an element set. If omitted, the default is ELEMENTS.
5. Name of the region (node set or element set) defined in the previous model from which results data are to be imported. If omitted, results data are imported from the entire model in the previous analysis.

   The region can be either a node set or an element set regardless of the location of the results data to be imported. When a node-based field is imported and an element set is specified, the

node-based field is extracted from nodes of the elements in the specified element set. When an element-based field is imported and a node set is specified, the element field is extracted from elements containing the nodes in the node set; elements connected to nodes that are not in the node set are not included.

6. Output variable Identifying keys for the results field data imported from the referenced output database (`.sim`) file. For information on individual output variable identifiers, see *Output Variables*. In addition, you can specify variables that can be read from the output database (`.sim`) file but for which you cannot request output. For example, you cannot request output for the variable ORIENT, but it is found in the output database (`.sim`) file if an orientation field is associated with any of the other results fields.
7. Specify the name of a mapping controls definition. If omitted, default mapping controls are used.
8. Specify the name of a field operations definition.

Repeat this data line as often as necessary to define imported data for element sets or node sets.

Using an external field to define history-dependent fields

Required parameters:

FILE

Set this parameter equal to the name of the output database file from which to read results data. For non-SIM databases, you must specify the reader executable name.

Optional parameters:

AMPLITUDE

This parameter is meaningful only when importing fields at a specified snap-shot point.

Set AMPLITUDE=RAMP (default) to ramp the incoming fields over the analysis step.

Set AMPLITUDE=STEP to apply the fields at the beginning of the analysis step.

FREQ

This parameter is meaningful only when importing fields at a specified snap-shot point from a frequency-domain analysis. Frequency-domain analysis is supported only for non-SIM databases.

Set this parameter equal to the frequency defining the snap-shot point to import fields. Linear interpolation is performed between two saved frequency points of the upstream analysis to obtain the field values at the specified frequency.

MODE

This parameter is meaningful only when importing fields at a specified snap-shot point from a frequency-domain analysis. Frequency-domain analysis is supported only for non-SIM databases.

Set this parameter equal to the mode number defining the mode to import fields.

READER

This parameter is required when reading results from a supported solver database other than a SIM-database.

Set this parameter equal to the reader executable name. Consult the respective solver user's guide for the reader executable name.

START TIME

This parameter is meaningful only when importing fields over a time range from a time-domain analysis.

Set this parameter equal to the start time to import fields. Field values are interpolated linearly between two saved time points in the upstream analysis results file to obtain the field values at the specified start time.

The time is specified as the step time when the STEP parameter is used; otherwise, total time is assumed.

If the START TIME and the END TIME parameters are omitted, field values are imported over the entire analysis step when the STEP parameter is used. Otherwise, values are imported over the entire analysis.

STEP

Set this parameter equal to the analysis step from which to import field values when reading from the SIM-database file.

If this parameter is specified, the TIME parameter to specify a snap-shot point and the START TIME and END TIME parameters to specify a time range refer to step time. Otherwise, total time is used.

END TIME

This parameter is meaningful only when importing field values over a time range from a time-domain analysis.

Set this parameter equal to the end time to import fields over a specified time range. Field values are linearly interpolated between two saved time points in the upstream analysis results file to obtain the field values at the specified end time.

The time is specified as the step time when the STEP parameter is used; otherwise, total time is assumed.

You must explicitly specify the end time value when the upstream and downstream analyses use different time scales and TIME SCALING=ON is specified.

If the START TIME and the END TIME parameters are omitted, field values are imported over the entire analysis step when the STEP parameter is used. Otherwise, values are imported over the entire analysis.

If the END TIME parameter is omitted, the end time is taken as the last saved fields of the upstream analysis step when the STEP parameter is used. Otherwise, the end time is taken as the last saved fields of the upstream analysis.

TIME

This parameter is meaningful only when importing field values at a snap-shot point from a time-domain analysis.

Set this parameter equal to the time at which to import field values. Field values are interpolated linearly between two saved time points of the upstream analysis to obtain field values at a specified time.

The time is specified as the step time when the STEP parameter is used; otherwise, total time is assumed.

If the TIME, MODE, and FREQ parameters are omitted, the last saved fields of the analysis step are used when the STEP parameter is specified. Otherwise, the last saved values at the end of the analysis are used.

TIME SCALING

This parameter is meaningful only when importing fields over a time range.

Set TIME SCALING=OFF (default) to use the same time scales between the upstream and downstream analyses.

Set TIME SCALING=ON to automatically scale time such that the time range of the upstream results are mapped over the downstream analysis step. You must specify an end time value explicitly in this case.

TYPE

Set TYPE=TIME RANGE (default) to import field data over a time range.

Set TYPE=SNAP SHOT to import fields at a snap-shot point (time, frequency, or mode number).

Data lines to use an external field to define history-dependent fields:

First line:

1. Name space of the region in the model used in the current analysis. Use NODES to indicate that the region is a node set; use ELEMENTS to indicate that the region is an element set; or use SURFACE to indicate that the region is a surface. The default is ELEMENTS.
2. Name of the region (node set, element set, or surface) defined in the current model where imported fields are specified.
3. Identifying key for the field imported. The keys are defined in *Identifying the Target and Source Fields*.
4. Name space of the region in the model used in the previous analysis. Use NODES to indicate that the region is a node set; use ELEMENTS to indicate that the region is an element set; use SURFACE to indicate that the region is a surface; or use MONITOR to read a monitor from a CST Studio analysis. The default is ELEMENTS.
5. Name of the region (node set, element set, or surface) defined in the previous model from which results data are to be imported. If the region is a surface, you must specify a surface name. For node sets and element sets, the default is to import results data from the entire model in the previous analysis.

   The region can be either a node set, an element set, or a surface regardless of the location where imported fields are specified. When you import a node-based field and specify an element set or surface, the node-based field is extracted from the nodes of the elements in the specified element set or facets belonging to the specified surface. When an element-based field is imported and you specify a node set, the element field is extracted from elements containing the nodes in the node set; elements connected to nodes that are not in the node set are not included.
6. Identifying keys for the results field data imported from the referenced output database (`.sim`) file. The keys are defined in *Abaqus/Standard Output Variable Identifiers* and *Abaqus/Explicit Output Variable Identifiers*.
7. Specify the name of a mapping controls definition. If omitted, default mapping controls are used.
8. Specify the name of an optional field operations definition.

Repeat this data line as often as necessary to import external fields.

Using an external field in a node-based submodel analysis

Required parameters:

FILE

Set this parameter equal to the name of the output database (`.sim`) file for the global analysis.

Optional parameters:

AMPLITUDE

This parameter is meaningful only when importing fields at a specified snap-shot point.

Set AMPLITUDE=RAMP (default) to ramp the incoming fields over the analysis step.

Set AMPLITUDE=STEP to apply the fields at the beginning of the analysis step.

START TIME

This parameter is meaningful only when importing fields over a time range.

Set this parameter equal to the start time to import fields. Field values are interpolated linearly between two saved time points in the global analysis results file to obtain the field values at the specified start time.

The time is specified as the step time when the STEP parameter is used; otherwise, total time is assumed.

If the START TIME and the END TIME parameters are omitted, field values are imported over the entire analysis step when the STEP parameter is used. Otherwise, values are imported over the entire analysis.

STEP

Set this parameter equal to the analysis step to import fields from the global model .

If this parameter is specified, the TIME parameter to specify a snap-shot point and the START TIME and END TIME parameters to specify a time range refer to step time. Otherwise, total time is used.

END TIME

This parameter is meaningful only when importing field values over a time range.

Set this parameter equal to the end time to import fields over a specified time range. Field values are interpolated linearly between two saved time points in the global analysis results file to obtain the field values at the specified end time.

The time is specified as the step time when the STEP parameter is used; otherwise, total time is assumed.

You must explicitly specify the end time value when the global and submodel analyses use different time scales and TIME SCALING=ON is specified.

If the START TIME and the END TIME parameters are omitted, field values are imported over the entire analysis step when the STEP parameter is used. Otherwise, values are imported over the entire analysis.

If the END TIME parameter is omitted, the end time is taken as the last saved fields of the global analysis step when the STEP parameter is used. Otherwise, the end time is taken as the last saved fields of the global analysis.

TIME SCALING

This parameter is meaningful only when importing fields over a time range.

Set TIME SCALING=OFF (default) to use the same time scales between the local and global analyses.

Set TIME SCALING=ON to automatically scale the time such that the time range of the global anaysis is mapped over the local analysis step. You must specify an end time value explicitly in this case.

TIME

This parameter is meaningful only when importing field values at a snap-shot point.

Set this parameter equal to the time at which to import field values. Field values are interpolated linearly between two saved time points of the global analysis to obtain field values at specified time.

The time is specified as the step time when the STEP parameter is used; otherwise, total time is assumed.

If the TIMEis omitted, the last saved fields of the analysis step are used when the STEP parameter is specified. Otherwise, the last saved values at the end of the analysis are used.

TYPE

Set TYPE=TIME RANGE (default) to import field data over a time range.

Set TYPE=SNAP SHOT to import fields at a snap-shot point (time, frequency, or mode number).

Data lines to use an external field in a node-based submodel analysis:

First line:

1. Use NODES (default) to indicate that the region is a node set.
2. Name of the region (node set) defining the driving nodes of the submodel.
3. Identifying key for the field imported. The keys are defined in *Identifying the Target and Source Fields*.
4. Use ELEMENTS (default) to indicate that the region is an element set.
5. Name of the region (element set) defined in the previous model from which results data are to be imported. By default, results data are imported from the entire model in the previous analysis.
6. Identifying keys for the results field data imported from the referenced output database (`.sim`) file. The keys are defined in *Abaqus/Standard Output Variable Identifiers* and *Abaqus/Explicit Output Variable Identifiers*.
7. Specify the name of a mapping controls definition. By default, default mapping controls are used.
8. Specify the name of an optional field operations definition.

Repeat this data line as often as necessary to import external fields.

*EXTREME ELEMENT VALUE: Define element variables to be monitored.

This option is used to define element variables that are to be monitored and compared with user-specified values. It must be used in conjunction with the **EXTREME VALUE* option.

Products: Abaqus/Explicit

Type: History data

Level: Step

References:

- *Explicit Dynamic Analysis*
- **EXTREME VALUE*

Required parameters:

ELSET

Set this parameter equal to the name of the element set in which the variables are to be monitored.

Required, mutually exclusive parameters:

ABS

Include this parameter if the user-specified value is to be the upper bound for the absolute value of the variable. At every increment Abaqus/Explicit will check whether the absolute value of the variable has exceeded the specified value.

MAX

Include this parameter if the user-specified value is to be the upper bound for the variable. At every increment Abaqus/Explicit will check whether the variable has exceeded the specified value.

MIN

Include this parameter if the user-specified value is to be the lower bound for the variable. At every increment Abaqus/Explicit will check whether the variable has fallen below the specified value.

Optional parameters:

OUTPUT

Set OUTPUT=YES (default) if the requested field-type output to the output database and an additional restart state are to be written when any variable value exceeds the user-specified bounds for the first time. The output will be written in the increment following the one in which such an occurrence took place.

Set OUTPUT=NO to prevent any output from being written.

Data lines to define element variables and the maxima or minima:

First line (optional, and relevant only if variables are being monitored for shell or beam elements):

1. Specify a list of the section points in the beam or shell at which variables should be monitored. If this data line is omitted, the variables are monitored at the default section points.

Second line:

1. Give the identifying keys for the element integration point and/or element section output variables to be monitored. Any variable available for history-type output from the output database can be specified. The keys are defined in *Abaqus/Explicit Output Variable Identifiers*.
2. Enter the extreme value.

Repeat the second data line as often as necessary to define additional variables to be monitored and their maxima or minima.

*EXTREME NODE VALUE: Define nodal variables to be monitored.

This option is used to define nodal variables that are to be monitored and compared with user-specified values. It must be used in conjunction with the **EXTREME VALUE* option.

Products: Abaqus/Explicit

Type: History data

Level: Step

References:

- *Explicit Dynamic Analysis*
- **EXTREME VALUE*

Required parameters:

NSET

Set this parameter equal to the name of the node set in which the variables are to be monitored.

Required, mutually exclusive parameters:

ABS

Include this parameter if the user-specified value is to be the upper bound for the absolute value of the variable. At every increment Abaqus/Explicit will check whether the absolute value of the variable has exceeded the specified value.

MAX

Include this parameter if the user-specified value is to be the upper bound for the variable. At every increment Abaqus/Explicit will check whether the variable has exceeded the specified value.

MIN

Include this parameter if the user-specified value is to be the lower bound for the variable. At every increment Abaqus/Explicit will check whether the variable has fallen below the specified value.

Optional parameters:

OUTPUT

Set OUTPUT=YES (default) if the requested field-type output to the output database and an additional restart state are to be written when any variable value exceeds the user-specified bounds for the first time. The output will be written in the increment following the one in which such an occurrence took place.

Set OUTPUT=NO to prevent any output from being written.

Data lines to define nodal variables and the maxima or minima:

First line:

1. Give the identifying keys for the nodal variables to be monitored. Any variable available for history-type output to the output database can be specified. The keys are defined in *Abaqus/Explicit Output Variable Identifiers*.
2. Enter the extreme value.

Repeat the data line as often as necessary to define additional variables to be monitored and their maxima or minima.

*EXTREME VALUE: Define element and nodal variables to be monitored.

This option is used in conjunction with the *EXTREME ELEMENT VALUE and/or the *EXTREME NODE VALUE options to indicate that nodal and element variables are to be monitored in the current step and compared with user-specified values. For each variable specified with these options, the maximum, minimum, or absolute maximum value attained during the course of the analysis and the associated element or node number will be written to the status (`.sta`) file at the end of the step. Use the *EXTREME VALUE option without the *EXTREME ELEMENT VALUE or *EXTREME NODE VALUE options and without any parameters to stop monitoring variables in a new step.

Products: Abaqus/Explicit

Type: History data

Level: Step

References:

- *Explicit Dynamic Analysis*
- *EXTREME ELEMENT VALUE
- *EXTREME NODE VALUE

Optional parameters:

HALT

Set HALT=NO (default) if the analysis should continue even if the variables have exceeded the user-specified bounds.

Set HALT=YES to stop the analysis at the first occurrence of a variable exceeding its user-specifed bound. The analysis will be stopped after the increment following the one in which such an occurrence took place.

There are no data lines associated with this option.

F

This section describes each Abaqus input option that begins with F, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **FABRIC*
- **FAIL STRAIN*
- **FAIL STRESS*
- **FAILURE RATIOS*
- **FASTENER*
- **FASTENER PROPERTY*
- **FATIGUE*
- **FIBER DISPERSION*
- **FIELD*
- **FIELD IMPORT*
- **FIELD MAPPER CONTROLS*
- **FIELD OPERATIONS*
- **FILE FORMAT*
- **FILE OUTPUT*
- **FILM*
- **FILM PROPERTY*
- **FILTER*
- **FIXED MASS SCALING*
- **FLEXIBLE BODY*
- **FLOW*
- **FLUID BEHAVIOR*
- **FLUID BULK MODULUS*
- **FLUID CAVITY*
- **FLUID DENSITY*
- **FLUID EXCHANGE*
- **FLUID EXCHANGE ACTIVATION*
- **FLUID EXCHANGE PROPERTY*
- **FLUID EXPANSION*
- **FLUID FLUX*
- **FLUID ID FIELD*
- **FLUID INFLATOR*
- **FLUID INFLATOR ACTIVATION*
- **FLUID INFLATOR MIXTURE*
- **FLUID INFLATOR PROPERTY*
- **FLUID LEAKOFF*
- **FLUID PIPE CONNECTOR LOSS*

- *FLUID PIPE CONNECTOR SECTION
- *FLUID PIPE CONNECTOR THERMAL LOSS
- *FLUID PIPE CONNECTOR WALL
- *FLUID PIPE FLOW LOSS
- *FLUID PIPE SECTION
- *FLUID PIPE THERMAL
- *FLUID PIPE WALL
- *FOUNDATION
- *FRACTURE CRITERION
- *FRAME SECTION
- *FREQUENCY
- *FRICTION

*FABRIC: Specify the in-plane response of a fabric material.

This option is used to define the in-plane behavior of a fabric material under plane stress conditions.

Products: Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Fabric Material Behavior*
- *VFABRIC*
- **DAMPING*
- **DENSITY*
- **DEPVAR*
- **INITIAL CONDITIONS*
- **ORIENTATION*
- **SECTION CONTROLS*
- **UNIAXIAL*

Optional parameters:

PROPERTIES

This parameter can be used only if the USER parameter is specified.

Set this parameter equal to the number of property values needed as data in user subroutine *VFABRIC*. The default value is 0.

You can introduce state variables using the **DEPVAR* option and update these variables within user subroutine *VFABRIC*. You can delete the element, if needed, using one of these state variables.

STRESS FREE INITIAL SLACK

Set STRESS FREE INITIAL SLACK=YES (default) to generate zero stresses in regions under initial compressive strains along the fill and the warp directions (these initial compressive strains may arise from modeling techniques such as the initial metric method—see **INITIAL CONDITIONS*, TYPE=REF COORDINATE). The stress remains zero as the strain is continuously recovered from the initial compressive values toward the strain-free state. Once the initial slack is recovered, any subsequent compressive strains generate stresses as per the material definition.

Set STRESS FREE INITIAL SLACK=NO to generate stresses in the initial configuration as per the material definition even over fabric regions that are under compressive strains.

Abaqus also offers a technique to introduce any initial stresses, both tensile and compressive, in fabric materials gradually over a specified time period (see **SECTION CONTROLS*).

USER

Include this parameter if the fabric stresses in a local system are updated in user subroutine *VFABRIC* given the total and the incremental fabric strains in the local system.

If this parameter is omitted, you must include the **UNIAXIAL* option to define the fabric response using test data in terms of the fabric stresses and the fabric strains in the local system.

The local system for the fabric material defined either through the test data or the user subroutine is initialized to the fill and the warp yarn directions in the reference configuration by using the **ORIENTATION* option. Abaqus updates this local system with deformation to track the fill and the warp directions in the current configuration.

Data lines to define the material properties for the USER fabric model:

No data lines are needed if the PROPERTIES parameter is omitted or set to 0. Otherwise, first line:

1. Give the material properties, eight per line.

Repeat this data line as often as necessary to define the material properties.

*FAIL STRAIN: Define parameters for strain-based failure measures.

This option is used to define the strain limits for strain-based failure measures. It can be used only in conjunction with the *ELASTIC option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Plane Stress Orthotropic Failure Measures*
- *ELASTIC

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the failure criteria, in addition to temperature. If this parameter is omitted, it is assumed that the failure criteria depend only on temperature. See *Material Data Definition* for more information.

Data lines to define strain-based failure criteria:

First line:

1. Tensile strain limit in fiber direction, $X_{\varepsilon t}$.
2. Compressive strain limit in fiber direction, $X_{\varepsilon c}$.
3. Tensile strain limit in transverse direction, $Y_{\varepsilon t}$.
4. Compressive strain limit in transverse direction, $Y_{\varepsilon c}$.
5. Shear strain limit in the *X*–*Y* plane, S_{ε}.
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two.):

1. Third field variable.
2. Fourth field variable.
3. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the failure criteria as a function of temperature and other predefined field variables.

*FAIL STRESS: Define parameters for stress-based failure measures.

This option is used to define the stress limits for stress-based failure measures. It can be used only in conjunction with the *ELASTIC option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Plane Stress Orthotropic Failure Measures*
- *ELASTIC

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the failure criteria, in addition to temperature. If this parameter is omitted, it is assumed that the failure criteria depend only on temperature. See *Material Data Definition* for more information.

Data lines to define stress-based failure criteria:

First line:

1. Tensile stress limit in fiber direction, X_t.
2. Compressive stress limit in fiber direction, X_c.
3. Tensile stress limit in transverse direction, Y_t.
4. Compressive stress limit in transverse direction, Y_c.
5. Shear strength in the *X*–*Y* plane, *S*.
6. Cross product term coefficient, $\overset{*}{f}$ ($-1.0 \le \overset{*}{f} \le 1.0$). This value is used only for the Tsai-Wu theory and is ignored if σ_{biax} is given. The default is zero.
7. Biaxial stress limit, σ_{biax}. This value is used only for the Tsai-Wu theory. If this entry is nonzero, $\overset{*}{f}$ is ignored.
8. Temperature.

Subsequent lines (only needed if the DEPENDENCIES parameter has a nonzero value):

1. First field variable.
2. Second field variable.
3. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the failure criterion as a function of temperature and other predefined field variables.

*FAILURE RATIOS: Define the shape of the failure surface for a CONCRETE model.

This option is used to define the shape of the failure surface for a concrete model. If used, it must appear after the *CONCRETE option. The *FAILURE RATIOS option can also be used with the *TENSION STIFFENING and *SHEAR RETENTION options.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Concrete Smeared Cracking*
- *CONCRETE
- *TENSION STIFFENING
- *SHEAR RETENTION

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the failure ratios, in addition to temperature. If this parameter is omitted, it is assumed that the failure ratios depend only on temperature. See *Material Data Definition* for more information.

Data lines to define the failure surface for a concrete model:

First line:

1. Ratio of the ultimate biaxial compressive stress to the uniaxial compressive ultimate stress. Default is 1.16.
2. Absolute value of the ratio of uniaxial tensile stress at failure to the uniaxial compressive stress at failure. Default is 0.09.
3. Ratio of the magnitude of a principal component of plastic strain at ultimate stress in biaxial compression to the plastic strain at ultimate stress in uniaxial compression. Default is 1.28.
4. Ratio of the tensile principal stress value at cracking in plane stress, when the other nonzero principal stress component is at the ultimate compressive stress value, to the tensile cracking stress under uniaxial tension. Default is 1/3.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the failure ratios on temperature and other predefined field variables.

*FASTENER: Define mesh-independent fasteners.

This option is used to define mesh-independent fasteners.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Interaction module

References:

- *Mesh-Independent Fasteners*
- **FASTENER PROPERTY*

Required parameters:

INTERACTION NAME

Set this parameter equal to a label that will be used to refer to the fastener interaction.

PROPERTY

Set this parameter equal to the name of the property to be used with this fastener definition.

At least one of the following parameters is required:

ELSET

This parameter is applicable only when the fastener is modeled using connector elements.

If the connector elements are defined explicitly, set this parameter equal to the name of the element set containing the connector elements. If the connector elements are to be generated internally by Abaqus, set this parameter equal to an empty element set name.

REFERENCE NODE SET

Use this parameter along with the ELSET parameter if internally generated connector elements are to be used to model the fastener. Use this parameter without the ELSET parameter if internally generated rigid beam MPCs are to be used to model the fastener.

Set this parameter equal to the name of the node set containing the reference nodes for this fastener definition.

Optional parameters:

ADJUST ORIENTATION

Set ADJUST ORIENTATION=YES (default) to have Abaqus adjust the user-defined orientation such that the local z-axis for each fastener is normal to the surface that is closest to the reference node for that fastener.

Set ADJUST ORIENTATION=NO to define the local directions precisely.

ATTACHMENT METHOD

Set this parameter equal to the projection method to be used to find the fastening points for the fastener.

Set ATTACHMENT METHOD=FACETOFACE (default) to select the default projection method of locating fastening points on the specified surface or surfaces. The positioning point is projected onto the nearest surface to create the first fastening point, and normal projection is used to find subsequent fastening points.

Set ATTACHMENT METHOD=FACETOEDGE to find the first fastening point by projecting the normal on the nearest surface and to find subsequent fastening points at the closest points on the specified surface or surfaces.

Set ATTACHMENT METHOD=EDGETOFACE to find the closest point on the nearest surface as the first fastening point and to find subsequent fastening points via normal projections on the remaining surfaces.

Set ATTACHMENT METHOD=EDGETOEDGE to find the closest fastening points on the specified surface or surfaces.

Set ATTACHMENT METHOD=CONNECTORDIRECTION to establish fastening points on the respective surfaces based on the axial direction of the connector element associated with the fastener. This option is valid only if the ELSET parameter is specified without the REFERENCE NODE SET parameter and the specified nodal positions for the connector element are not co-located.

COUPLING

This parameter controls whether the rotations of surface nodes participate in translational coupling constraints for each fastening point.

Set COUPLING=CONTINUUM (default) to couple the displacement and rotation of each fastening point to the displacement (but not rotation) of the surface nodes within the radius of influence.

Set COUPLING=STRUCTURAL to couple the displacement and rotation of each fastening point to the displacement and rotation of the surface nodes within the radius of influence.

NUMBER OF LAYERS

Set this parameter equal to the number of layers for each fastener. If this parameter is omitted and no surface is specified by the user or a single surface is specified by the user, Abaqus will form the maximum possible number of layers for each fastener.

This parameter is ignored if multiple surfaces are specified on the data lines.

ORIENTATION

Set this parameter equal to the name of an orientation definition (see *Orientations*) that defines the orientation of the fastener. If this parameter is omitted, the orientation of each fastener is determined from the default local directions of the surface (see *Conventions*) that is closest to the reference node for that fastener.

Fasteners support only rectangular, cylindrical, and spherical orientation definitions. Additional rotations defined as part of the orientation definition are ignored.

RADIUS OF INFLUENCE

Set this parameter equal to the maximum distance from the projection point on a connected surface within which the nodes on that surface must lie to contribute to the motion of the projection point. If this parameter is omitted, Abaqus will compute a default value of the radius of influence internally, based on the fastener diameter and the surface facet lengths.

ROTATIONAL COUPLING

This parameter controls whether the displacement and the rotation of surface nodes participate in rotational coupling constraints for each fastening point.

Set ROTATIONAL COUPLING=AUTOMATIC (default) to allow Abaqus to couple the rotation of each fastening point to the displacement and, if needed, rotation of the surface nodes within the radius of influence.

Set ROTATIONAL COUPLING=CONTINUUM to couple the rotation of each fastening point to the displacement (but not rotation) of the surface nodes within the radius of influence.

Set ROTATIONAL COUPLING=STRUCTURAL to couple the rotation of each fastening point to the displacement and rotation of the surface nodes within the radius of influence.

SEARCH RADIUS

Set this parameter equal to the distance from the reference nodes within which the connected points must lie. If this parameter is omitted and no surface is specified by the user or a single surface is specified by the user, Abaqus will compute a default search radius based on the facet thickness (for shell element facets) or characteristic facet length (for non-shell element facets) in the vicinity of each positioning point.

UNSORTED

If this parameter is omitted, the connectivity of the fastening points is defined by the relative positions of their associated surfaces along the fastener projection direction.

If this parameter is included, the connectivity of the fastening points is defined by the order in which their associated surfaces appear on the data lines.

This parameter is ignored if no surfaces are specified on the data lines.

WEIGHTING METHOD

Set this parameter equal to the weighting scheme to be used to weight the contribution of the displacements of the surface nodes within the radius of influence to the motion of the fastener projection point.

Set WEIGHTING METHOD=UNIFORM (default) to select a uniform weight distribution.

Set WEIGHTING METHOD=LINEAR to select a linear decreasing weight distribution.

Set WEIGHTING METHOD=QUADRATIC to select a quadratic polynomial decreasing weight distribution.

Set WEIGHTING METHOD=CUBIC to select a cubic polynomial monotonic decreasing weight distribution.

Data lines to define the fastener if the default projection direction is used (ATTACHMENT METHOD=FACETOFACE):

First line (optional):

1. Enter a blank line.

Subsequent lines (optional; if omitted, Abaqus will search for fastening points on all element facets that fall within a search radius of the positioning point):

1. Surface name.
2. Etc., up to eight surface names per line.

Repeat this data line as often as necessary to define all the surfaces to be connected for this fastener interaction.

Data lines to define the fastener if the projection direction for the first fastening point is specified by the user:

First line:

1. First direction cosine of the projection direction.

2. Second direction cosine of the projection direction.
3. Third direction cosine of the projection direction.

Subsequent lines (optional; if omitted, Abaqus will search for fastening points on all element facets that fall within a search radius of the positioning point):

1. Surface name.
2. Etc., up to eight surface names per line.

Repeat this data line as often as necessary to define all the surfaces to be connected for this fastener interaction.

*FASTENER PROPERTY: Prescribe mesh-independent fastener properties.

This option is used to prescribe the properties of a fastener interaction. This option must be used in conjunction with the *FASTENER option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Interaction module

References:

- *Mesh-Independent Fasteners*
- *FASTENER

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the fastener property.

Data lines to specify the fastener properties:

First line:

1. Radius, *r*.

Second line:

1. First degree of freedom constrained. See *Conventions* for a definition of the numbering of degrees of freedom in Abaqus. If this field is left blank, all degrees of freedom will be constrained.
2. Last degree of freedom constrained. If this field is left blank, the degree of freedom specified in the first field will be the only one constrained.

Repeat this data line as often as necessary to specify constraints for different degrees of freedom. When the ORIENTATION parameter is specified on the associated *FASTENER option, the degrees of freedom are in the specified local system in the initial configuration; otherwise, they are in the default local system. In either case these directions will rotate with the reference node in large-displacement analyses (when the NLGEOM parameter on the *STEP option is set equal to YES).

*FATIGUE: Define a fatigue crack growth analysis in bulk brittle materials and in brittle material interfaces.

This option is used to predict delamination/debonding growth at the brittle material interfaces in laminated composites and to predict crack growth in bulk brittle materials under cyclic fatigue loading.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Linear Elastic Fatigue Crack Growth Analysis*
- **TIME POINTS*

Optional parameters:

CETOL

Set this parameter equal to the maximum difference in the creep strain increment calculated from the creep strain rates based on conditions at the beginning and on conditions at the end of the increment, thus controlling the time integration accuracy of the creep integration.

This parameter can be used in conjunction with the **TIME POINTS* option. In this case Abaqus/Standard ensures that the response is also evaluated at each time point specified on the **TIME POINTS* option.

If both this parameter and the DELTMX parameter are omitted, fixed time stepping will be used, with a constant time increment equal to the initial time increment or by following precisely the time points specified on the **TIME POINTS* option.

DELTMX

Set this parameter equal to the maximum temperature change to be allowed in an increment during a fatigue crack growth analysis. Abaqus/Standard restricts the time increment to ensure that this value is not exceeded at any node during any increment of the step.

DIRECT

This parameter selects direct user control of the incrementation through the step. If this parameter is used, constant increments of the size defined by the first item on the data line are used. If this parameter is omitted, Abaqus/Standard will choose the increments (after trying the user's initial time increment for the first attempt at the first increment).

TIME POINTS

Set this parameter equal to the name of the **TIME POINTS* option that defines the time points at which the response of the structure will be evaluated.

TYPE

Set TYPE=CONSTANT AMPLITUDE (default) to use the classical incremental method for each cycle.

Set TYPE=SIMPLIFIED to use a constant load to simplify the fatigue loading. At least two increments are required for each single loading cycle in this case.

Data lines to define fatigue crack growth analysis:

First line:

1. Initial time increment. If this entry is omitted, a default value of 0.1 times the single loading cycle period is assumed.
2. Time of a single loading cycle.
3. Minimum time increment allowed.
4. Maximum time increment allowed.

Second line:

1. Minimum increment in number of cycles over which the damage is extrapolated forward. It must be greater than 0, and the default is 100.
2. Maximum increment in number of cycles over which the damage is extrapolated forward. It must be greater than 0, and the default is 1000.
3. Total number of cycles allowed in a step. If this entry is zero or not specified, the default value is equal to one plus half of the maximum increment in number of cycles over which the damage is extrapolated.
4. Not used.
5. Cycle increment–based tolerance for the least number of cycles to fracture an element. The damage-based tolerance takes precedence over the cycle increment–based tolerance if both are specified. If neither tolerance is specified, a damage-based tolerance with a default value of 0.25 is used.
6. Damage-based tolerance for the least number of cycles to fracture an element. The damage-based tolerance takes precedence over the cycle increment–based tolerance if both are specified. If neither tolerance is specified, a damage-based tolerance with a default value of 0.25 is used.

*FIBER DISPERSION: Define the second-order orientation tensor of fiber in a composite material.

This option is used to define the second-order orientation tensor of fiber in a composite material. The second-order orientation tensor is used to compute the orientation-averaged elastic modulus and thermal expansion coefficient of the composite material. This option is relevant only when used with the **ELASTIC*, **CONDUCTIVITY*, and **EXPANSION* options.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Defining the Elasticity of a Short-Fiber Reinforced Composite*
- *Defining Thermal Expansion for a Short-Fiber Reinforced Composite*
- *Conductivity*
- *Conductivity*

Optional parameters:

CLOSURE

This parameter is used to specify the closure approximation used to compute the fourth-order orientation tensor using the second-order orientation tensor.

Set CLOSURE=FITTED to use the orthotropic fitted (ORF) closure approximation.

Set CLOSURE=HYBRID to use the hybrid closure approximation.

Set CLOSURE=ORE to use the orthotropic equivalent of the natural (ORE) closure approximation.

Set CLOSURE=QUADRATIC (default) to use the quadratic closure approximation.

Set CLOSURE=SMOOTH to use the orthotropic smooth closure approximation.

Data line to define the components of the second-order orientation tensor in a local system aligned with principal directions using a distribution in Abaqus/Standard:

First (and only) line:

1. Distribution name.

Data line to define the components of the second-order orientation tensor in a local system aligned with principal directions:

First (and only) line:

1. Maximum principal value of the orientation tensor, a_1.
2. Mid principal value of the orientation tensor, a_2.
3. Minimum principal value of the orientation tensor, a_3.

*FIELD: Specify predefined field variable values.

This option is used to specify values for predefined field variables used in the analysis. To use this option in a restart analysis of Abaqus/Standard, either **FIELD* or **INITIAL CONDITIONS*, TYPE=FIELD must have been specified in the original analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- *Predefined Fields*
- *UFIELD*
- *VUFIELD*

Optional parameters:

VARIABLE

Set this parameter equal to the field variable number. The user must number the field variables consecutively from 1. The default is VARIABLE=1 unless the NUMBER parameter is used. The VARIABLE and NUMBER parameters are mutually exclusive.

Optional parameters for using the data line format:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that gives the time variation of the field variable throughout the step (see *Amplitude Curves*).

If this parameter is omitted in an Abaqus/Standard analysis, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option. If this parameter is omitted in an Abaqus/Explicit analysis, the reference magnitude is applied linearly over the step.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

OP

Set OP=MOD (default) for existing **FIELD* variable values to remain, with this option modifying existing values or defining additional values.

Set OP=NEW if all existing **FIELD* variable values should be removed. New field variable values can be defined.

For a general analysis step, a field variable that is removed via OP=NEW is reset to the value given on the **INITIAL CONDITIONS* option or to zero if no initial field was defined. For a linear perturbation step, a field variable that is removed via OP=NEW is always reset to zero. If a field variable is being returned to its initial condition values, the AMPLITUDE parameter described above does not apply. Rather, the AMPLITUDE parameter given on the **STEP* option governs the

behavior in an Abaqus/Standard analysis. In a multistep Abaqus/Standard analysis, the default is to reset to zero. In an Abaqus/Explicit analysis the field variable is always linearly ramped back to its initial conditions. If the field variable is being reset to a new value (not to its initial condition) via OP=NEW, the AMPLITUDE parameter described above applies.

SECTION SPECIFICATION

Set SECTION SPECIFICATION=UNIFORM to define uniform field variable values.

Required parameter for reading predefined field variable values from the results or output database file:

FILE

Set this parameter equal to the name of the results (`.fil`) or output database (`.odb`) file from which the data are read. The file extension is optional; however, if both `.fil` and `.odb` files exist, the results file will be used if the INTERPOLATE parameter is omitted. If the INTERPOLATE parameter is used, an output database file must exist. See *Input Syntax Rules* for the syntax of such file names.

This parameter cannot be used in a **STATIC*, RIKS analysis step.

Optional parameters for reading predefined field variable values from the results or output database file:

BSTEP

Set this parameter equal to the step number (of the analysis whose results file is being used as input to this option) that begins the history data to be read. If no value is supplied, Abaqus will begin reading field variable data from the first step available on the file read.

BINC

Set this parameter equal to the increment number (of the analysis whose results or output database file is being used as input to this option) that begins the history data to be read. If no value is supplied, Abaqus will begin reading field variable data from the first increment available (excluding any zero increments if the results file was written in Abaqus/Standard using **FILE FORMAT*, ZERO INCREMENT) for step BSTEP on the results or output database file.

ESTEP

Set this parameter equal to the step number (of the analysis whose results or output database file is being used as input to this option) that ends the history data to be read. If no value is supplied, ESTEP is taken as equal to BSTEP.

EINC

Set this parameter equal to the increment number (of the analysis whose results or output database file is being used as input to this option) that ends the history data to be read. If no value is supplied, EINC is taken as the last available increment of step ESTEP on the results file.

Required parameter for reading predefined field variable values from the output database file:

OUTPUT VARIABLE

Set this variable equal to the scalar nodal output variable that will be read from an output database and used to initialize a specified predefined field. For a list of scalar nodal output variables that can be used to initialize a predefined field, see *Predefined Fields*.

Optional parameter for reading predefined field variable values from the output database file:

INTERPOLATE

Include this parameter to indicate that the scalar nodal output variable (specified by the OUTPUT VARIABLE parameter) being read into a predefined field needs to be interpolated between dissimilar meshes. This feature is used to read nodal values from an output database file generated during a previous Abaqus analysis.

Required parameter for defining data in user subroutine `UFIELD` or `VUFIELD`:

USER

Include this parameter to indicate that user subroutine *UFIELD* or *VUFIELD* will be used to define field variable values. For an Abaqus/Standard analysis *UFIELD* is called for each node given on the data lines. For an Abaqus/Explicit analysis *VUFIELD* is called for each field variable or for a set of field variables when the NUMBER parameter is used.

If values are also given on the data lines, these values will be ignored. If a results file has been specified in addition to the user subroutine, values read from the results file will be passed into the user subroutine for possible modification.

Optional parameters for defining data in user subroutine `UFIELD` or `VUFIELD`:

NUMBER

This parameter permits multiple (possibly all) field variables to be updated simultaneously in user subroutine *UFIELD* or *VUFIELD*; for example, because they are interdependent. Set this parameter equal to the number of field variables to be updated simultaneously at a point. The NUMBER and VARIABLE parameters are mutually exclusive.

BLOCKING

This parameter applies only to Abaqus/Explicit analyses. It is related to the `NBLOCK` variable used in the user subroutine argument list.

Set BLOCKING=YES to enable blocking for a given node set. The blocking size will be set to a predefined value in Abaqus/Explicit.

Set BLOCKING=NO (default) to disable blocking.

Use BLOCKING=*n* to specify the blocking size.

Data lines to define gradients of a predefined field variable in beams and shells:

First line:

1. Node set or node number. If a node set label is given, all nodes in this set must have identical initial field variable values.
2. Reference magnitude of the field variable. If the amplitude parameter is present, this and subsequent values will be modified by the **AMPLITUDE* specification.
3. Gradient in the n_2-direction for beams or gradient through the thickness for shells.
4. Gradient in the n_1-direction for beams.

Repeat this data line as often as necessary to define a field variable at different nodes or node sets.

Data lines to define a predefined field variable at temperature points in beams and shells:

First line:

1. Node set or node number. If a node set label is given, all nodes in this set must have identical initial field variable values.
2. Magnitude at the first temperature point. If the amplitude parameter is present, this and subsequent values will be modified by the **AMPLITUDE* specification.

3. Magnitude of the field variable at the second temperature point.
4. Magnitude of the field variable at the third temperature point.
5. Etc., up to seven values.

Subsequent lines (only needed if there are more than seven temperature points in the element):

1. Magnitude of the field variable at the eighth temperature point.
2. Etc., up to eight values per line.

Repeat this set of data lines as often as necessary to define a field variable at different nodes or node sets.

Data lines to define a predefined field variable for solid elements using the data line format:

First line:

1. Node set or node number. If a node set label is given, all nodes in this set must have identical initial field variable values.
2. Field variable value. If the AMPLITUDE parameter is present, this value will be modified by the AMPLITUDE specification.

Repeat this data line as often as necessary to define a field variable at different nodes or node sets.

Data lines to define uniform field variable values (SECTION SPECIFICATION=UNIFORM):

First line:

1. Node set, node number, or blank.
2. Uniform field variable value.

Repeat this data line as often as necessary to define uniform field variable values at various nodes and node sets. If you omit both node numbers and node sets (that is, enter a blank space), the specified uniform field variable is applied to all nodes in the model.

To read values of a field variable from an Abaqus/Standard results or output database file:

No data lines are used when field variable data are read from a results or output database file.

Data lines to define a field variable using user subroutine `UFIELD` or `VUFIELD`:

First line:

1. Node set or node number. If a node set label is given, all nodes in this set must have identical initial field variable values.

Repeat this data line as often as necessary.

*FIELD IMPORT: Import external fields to apply over an analysis step in a sequentially coupled multiphysics analysis.

This option must be used in conjunction with the *EXTERNAL FIELD option to define history-dependent data.

Products: Abaqus/Standard Abaqus/Explicit

Type: History data

Level: Assembly

References:

- *EXTERNAL FIELD
- *General Capability for Importing External Fields*
- *Node-Based Submodeling Using the Field Import Interface*

Optional parameters:

SUBMODEL

Include this parameter to use the node-based submodeling capability using field import.

There are no data lines associated with this option.

*FIELD MAPPER CONTROLS: Define controls for the mapper.

This option can be used only in conjunction with the **EXTERNAL FIELD* option. It is used to define additional mapping controls for the field mapper when importing a predefined field.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- **EXTERNAL FIELD*
- *Field Mapper*
- *General Capability for Importing External Fields*

Required parameters:

NAME

Set this parameter equal to a label that is used to refer to the field mapper control definition. All field mapper control names in the same input file must be unique.

Optional parameters:

CONFIGURATION

This parameter is relevant only when importing fields to define a distribution or an initial condition.

Set CONFIGURATION=DEFORMED to use the deformed configuration of the source mesh for mapping. This parameter setting is used when the model is remeshed in the second analysis using the deformed shape from the first analysis.

Set CONFIGURATION=REFERENCE (default) to use the original configuration of the source mesh for mapping.

MAPPER ALGORITHM

This parameter is relevant only when importing history-dependent fields when the mapper integrates to convert intensive fields to extensive fields.

Set MAPPER ALGORITHM=SUBELEMENT (default) to select an algorithm based on the subelements method. This algorithm provides conservative mapping with a smooth distribution.

Set MAPPER ALGORITHM=GLOBAL CONSERVATIVE to select an algorithm where the results are conservative over the entire region. This algorithm is numerically less expensive than the subelements method, and the mapped field might show an uneven distribution in some cases.

SEARCH TOLERANCE

Set SEARCH TOLERANCE=RELATIVE (default) to specify the search tolerances as relative values to characteristic region dimensions.

Set SEARCH TOLERANCE=ABSOLUTE to specify absolute search tolerances.

TENSOR AVERAGING

This parameter is relevant only for mapping three-dimensional tensor quantities.

Set TENSOR AVERAGING=COMPONENT to use component-by-component mapping. This value is the default except if output variable S, ORIENT, or ORITENS is specified as the source field on the **EXTERNAL FIELD* option.

Set TENSOR AVERAGING=EIGENVALUE to use a mapping technique that preserves the eigenvalues of the tensor during mapping. This value is the default if output variable ORITENS is specified as the source field on the **EXTERNAL FIELD* option.

Set TENSOR AVERAGING=INVARIANT to use a mapping technique that preserves the Mises invariant of the stress tensor during mapping. This value is the default if output variable S is specified as the source field on the **EXTERNAL FIELD* option.

If output variable ORIENT is specified as the source field on the **EXTERNAL FIELD* option, a special averaging technique to map orientations is used by default. Output variable ORIENT is available in the output database (`.sim`) file if material orientation is defined in the element section definition.

UNMAPPED MEMBERS TREATMENT

This parameter defines how to treat unmapped members (nodes or elements where the search algorithm fails to find intersections).

Set UNMAPPED MEMBERS TREATMENT=ERROR (default) to issue an error message and end the simulation if members cannot be mapped properly when importing history-dependent fields. When importing initial conditions and distributions, no values are applied. Unmapped members are reported as a node set or element set.

Set UNMAPPED MEMBERS TREATMENT=NEAREST NEIGHBOR to assign a field value based on the nearest neighbor.

Set UNMAPPED MEMBERS TREATMENT=CONSTANT VALUE to specify a constant value for the unmapped member.

Data lines to define mapping specifications:

First line to specify searching tolerances:

1. Search position tolerance distance searched on the outward-normal side of a surface facet. For a relative search tolerance, specify a factor relative to the characteristic facet size (default is 0.5). For an absolute search tolerance, specify the search distance (default is 0.0).
2. Search position tolerance distance for the cloud point mapping algorithm in the element face normal directions. For a relative search tolerance, specify a factor relative to the characteristic facet size (default is 0.5). For an absolute search tolerance, specify the search distance (default is 0.0).
3. Search position tolerance distance searched outside the source elements or outside a facet in the plane of the surface. For a relative search tolerance, specify a factor relative to the characteristic facet size (default is 0.5). For an absolute search tolerance, specify the search distance (default is 0.0).
4. Search tolerance radius for the nearest neighbor algorithm. For a relative search tolerance, specify a factor relative to the characteristic facet size (default is 0.5). For an absolute search tolerance, specify the search distance (default is 0.0).

Second line (optional, to be used to specify default values at unmapped member locations where search fails):

1. Value assigned to the first component of the scalar, vector, or tensor field at an unmapped member location.
2. Value assigned to the second component of the vector or tensor field at an unmapped member location.

3. Value assigned to the third component of the vector or tensor field at an unmapped member location.
4. Value assigned to the fourth component of the tensor field at an unmapped member location.
5. Value assigned to the fifth component of the tensor field at an unmapped member location.
6. Value assigned to the sixth component of the tensor field at an unmapped member location.

*FIELD OPERATIONS: Define optional field operations to perform when importing a predefined field.

This option can be used only in conjunction with the **EXTERNAL FIELD* option. It is used to apply field operations prior to importing the predefined field.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- **EXTERNAL FIELD*

Required parameters:

NAME

Set this parameter equal to a label that is used to refer to the field operations definition. All field operation names in the same input file must be unique.

Optional parameters:

COMPONENT

Set this parameter equal to an index, i, referencing the ith component of a vector or tensor field value to import.

Use this parameter only if the target is a scalar field, such as a field variable or a solution-dependent variable. The component of the source field is extracted in the local system and then mapped and imported as a scalar field. For more information, see *Convention Used for Stress and Strain Components*.

If this parameter is omitted, the entire vector or tensor field is imported.

SCALE FACTOR

Set this parameter equal to the factor to scale all field values. The default scaling factor is 1.0.

Optional data lines to define the repositioning of the source region from which results data are imported:

First line:

1. Value of the translation to be applied in the *X*-direction. The default value is zero.
2. Value of the translation to be applied in the *Y*-direction. The default value is zero.
3. Value of the translation to be applied in the *Z*-direction. The default value is zero.

Second line:

1. *X*-coordinate of the first point on the axis of rotation.
2. *Y*-coordinate of the first point on the axis of rotation.
3. *Z*-coordinate of the first point on the axis of rotation.
4. *X*-coordinate of the second point on the axis of rotation.
5. *Y*-coordinate of the second point on the axis of rotation.
6. *Z*-coordinate of the second point on the axis of rotation.
7. Angle of rotation about the axis of rotation, in degrees.

If both translation and rotation are specified, translation is applied before rotation.

*FILE FORMAT: Specify format for results file output and invoke zero-increment results file output.

This option is used to specify the format in which the Abaqus/Standard results file output is written and to invoke zero-increment file output for all valid procedures in the analysis. This option can appear only once in an analysis, and the format cannot be changed upon restart.

Products: Abaqus/Standard Abaqus/CAE

Type: Model or history data

Level: Model, Step

Abaqus/CAE: Unsupported; Abaqus/CAE does not use the results file.

References:

• *About Output*

Optional parameters:

ASCII

Include this parameter to specify that the results file output is to be written in ASCII format. If the **FILE FORMAT* option is omitted or this parameter is not used, the default is to write a binary file.

ZERO INCREMENT

Include this parameter to specify that results file output should be written at the beginning of a step (the zero increment) for all valid procedures in the analysis. If the **FILE FORMAT* option is omitted or this parameter is not used, by default output will not be written at the zero increment.

There are no data lines associated with this option.

*FILE OUTPUT: Define output written to the results file.

 **Warning:** This option can create a very large file.

The **FILE OUTPUT* option provides output of nodal, element, or global data to the selected results file. The **EL FILE*, the **ENERGY FILE*, and/or the **NODE FILE* options must be used in conjunction with the **FILE OUTPUT* option.

Products: Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Unsupported; Abaqus/CAE reads output from the output database file only.

References:

- *Output to the Data and Results Files*
- **EL FILE*
- **ENERGY FILE*
- **NODE FILE*

Required parameters:

NUMBER INTERVAL

Set this parameter equal to the number of intervals during the step at which the file output states are to be written. Abaqus/Explicit will always write the results at the beginning of the step. For example, if NUMBER INTERVAL=10, Abaqus/Explicit will write 11 results states consisting of the values at the beginning of the step and the values at the end of 10 intervals throughout the step. The value of this parameter must be a positive integer.

Optional parameters:

TIME MARKS

Set TIME MARKS=NO (default) to write the results at the increment ending immediately after the time dictated by the NUMBER INTERVAL parameter.

Set TIME MARKS=YES to write results at the exact times dictated by the NUMBER INTERVAL parameter. TIME MARKS=YES cannot be used when either the FIXED TIME INCREMENTATION or the DIRECT USER CONTROL parameter is included on the **DYNAMIC* option.

There are no data lines associated with this option.

*FILM: Define film coefficients and associated sink temperatures.

This option is used to provide film coefficients and sink temperatures over a specified element-based surface for all procedures that solve for nodal temperature degrees of freedom.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *Thermal Loads*
- *FILM*

Optional parameters:

FILM AMPLITUDE

Set this parameter equal to the name of the **AMPLITUDE* option that gives the variation of the film coefficient, h, with time.

If this parameter is omitted in an Abaqus/Standard analysis, the reference film coefficient is applied immediately at the beginning of the step and kept constant over the step, independent of the value assigned to the AMPLITUDE parameter on the **STEP* option. If this parameter is omitted in an Abaqus/Explicit analysis, the reference film coefficient is applied immediately at the beginning of the step.

The FILM AMPLITUDE parameter is ignored if a film coefficient is defined to be a function of temperature and field variables via the **FILM PROPERTY* option.

For nonuniform film coefficients, the film coefficient amplitude is defined in user subroutine *FILM* in Abaqus/Standard or user subroutine *VFILM* in Abaqus/Explicit; and FILM AMPLITUDE references are used only to modify the film coefficient passed into the user subroutine.

OP

Set OP=MOD (default) for existing **FILM*s to remain, with this option modifying existing films or defining additional films.

Set OP=NEW if all existing **FILM*s applied to the model should be removed.

REGION TYPE

This parameter applies only to Abaqus/Explicit analyses.

This parameter is relevant only for film conditions applied to the boundary of an adaptive mesh domain. If a film condition is applied to a surface in the interior of an adaptive mesh domain, the nodes on the surface will move with the material in all directions (they will be nonadaptive). Abaqus/Explicit will create a boundary region automatically on the surface subjected to the defined film load.

Set REGION TYPE=LAGRANGIAN (default) to apply the film condition to a Lagrangian boundary region. The edge of a Lagrangian boundary region will follow the material while allowing adaptive meshing along the edge and within the interior of the region.

Set REGION TYPE=SLIDING to apply the film condition to a sliding boundary region. The edge of a sliding boundary region will slide over the material. Adaptive meshing will occur along the

edge and in the interior of the region. Mesh constraints are typically applied on the edge of a sliding boundary region to fix it spatially.

Set REGION TYPE=EULERIAN to apply the film condition to an Eulerian boundary region. This option is used to create a boundary region across which material can flow. Mesh constraints must be used normal to an Eulerian boundary region to allow material to flow through the region. If no mesh constraints are applied, an Eulerian boundary region will behave in the same way as a sliding boundary region.

SINK AMPLITUDE

Set this parameter equal to the name of the *AMPLITUDE option that gives the variation of the sink temperature, θ^0, with time.

If this parameter is omitted in an Abaqus/Standard analysis, the reference sink temperature is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the *STEP option (*Defining an Analysis*). If this parameter is omitted in an Abaqus/Explicit analysis, the reference sink temperature is applied immediately at the beginning of the step.

For nonuniform film coefficients, the sink temperature amplitude is defined in user subroutine *FILM* in Abaqus/Standard or user subroutine *VFILM* in Abaqus/Explicit; and SINK AMPLITUDE references are used only to modify the sink temperature passed into the user subroutine.

Data lines to define sink temperatures and film coefficients:

First line:

1. Element number or element set label.
2. Film type label (see *About the Element Library*).
3. Reference sink temperature value, θ^0. (Units of θ.) For nonuniform film coefficients the sink temperature must be defined in user subroutine *FILM* in Abaqus/Standard or user subroutine *VFILM* in Abaqus/Explicit. If given, this value will be modified by the SINK AMPLITUDE reference and passed into the user subroutine.
4. Reference film coefficient value, h (units of $JT^{-1}L^{-2}\theta^{-1}$), or name of the film property table defined with the *FILM PROPERTY option. Nonuniform film coefficients must be defined in user subroutine *FILM* in Abaqus/Standard or user subroutine *VFILM* in Abaqus/Explicit. If given, this value will be modified by the FILM AMPLITUDE reference and passed into the user subroutine.

Repeat this data line as often as necessary to define film conditions.

*FILM PROPERTY: Define a film coefficient as a function of temperature and field variables.

This option is used to define a film coefficient as a function of temperature and field variables for fully coupled thermal-stress analyses. In Abaqus/Standard it is also used for heat transfer, coupled thermal-electrical, and coupled thermal-electrical-structural analyses. It can be used only in conjunction with the *FILM, *CFILM, and *SFILM options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Thermal Loads*
- *FILM
- *CFILM

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to this film property. This label is referred to on the data lines of the *FILM or *CFILM options.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the film coefficient. If this parameter is omitted, it is assumed that the film coefficient depends only on temperature. See *Material Data Definition* for more information.

Data lines to define the film coefficient as a function of temperature and field variables:

First line:

1. Film coefficient, h. (Units of $JT^{-1}L^{-2}\theta^{-1}$.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to create a film property table.

*FILTER: Define a filter and/or operator for output filtering and/or operating.

This option defines a digital filter and/or an operator to be used in conjunction with the **OUTPUT* option. It can be used to pre-filter and/or operate on the output as the analysis progresses.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: *Filter toolset*

References:

- *Output to the Output Database*
- **OUTPUT*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to this filter and/or operator.

Optional parameters:

EXTRA OUTPUT FRAME

Include this parameter if you want to request an additional output frame when the value specified with the LIMIT parameter is reached.

Set EXTRA OUTPUT FRAME=YES to write the requested field-type output to the output database when the value specified with the LIMIT parameter is reached. The output is written at the increment following the one in which the limit value is reached.

Set EXTRA OUTPUT FRAME=NO (default) to prevent writing any additional output when the value specified with the LIMIT parameter is reached.

HALT

Include this parameter if you want the analysis or the current step to stop when the value specified with the LIMIT parameter is reached.

Set HALT=ANALYSIS to stop the analysis completely when the value specified with the LIMIT parameter is reached.

Set HALT=STEP to conclude the current step and continue to the next step when the value specified with the LIMIT parameter is reached.

INVARIANT

This parameter can be used only in conjunction with the OPERATOR parameter to indicate that you want to filter and/or operate on the invariant of the element or nodal output field variable.

Set INVARIANT=FIRST to apply the filtering to the first invariant.

Set INVARIANT=SECOND to apply the filtering to the second invariant.

Set INVARIANT=MAXP to apply the filtering to the maximum principal stress or principal strain.

Set INVARIANT=INTERMP to apply the filtering to the intermediate principal stress or principal strain.

Set INVARIANT=MINP to apply the filtering to the minimum principal stress or principal strain.

See *Output to the Output Database* for more information.

LIMIT

Include this parameter if you want to set a limit (cap value) to the output variables.

This parameter must be used in conjunction with the OPERATOR parameter.

OPERATOR

This parameter can be used with or without the TYPE parameter. When it is used with a filter type, it will operate on filtered data; and when it is used without a filter type, it will operate on raw (unfiltered) data.

Set OPERATOR=MAX if you want to obtain the maximum value for the variables for which this filter is used. You can put a cap value on the maximum value by using the LIMIT parameter.

Set OPERATOR=MIN if you want to obtain the minimum value for the variables for which this filter is used. You can put a cap value on the minimum value by using the LIMIT parameter.

Set OPERATOR=ABSMAX if you want to obtain the absolute maximum value for the variables for which this filter is used. You can put a cap value on this value by using the LIMIT parameter.

START CONDITION

This parameter must be used with the TYPE parameter.

Set START CONDITION=DC (default) to pre-charge the filter with constant values equal to the first raw data value.

Set START CONDITION=USER DEFINED to pre-charge the filter with constant values equal to the user-defined value.

TYPE

Set TYPE=BUTTERWORTH (which is the default value when OPERATOR is omitted) to define a Butterworth filter.

Set TYPE=CHEBYS1 to define a Type I Chebyshev filter.

Set TYPE=CHEBYS2 to define a Type II Chebyshev filter.

Data lines to define a Butterworth filter:

First line:

1. Cutoff frequency, fc. (Units of T^{-1}.)
2. Order of the filter, N. Abaqus expects an even number; if an odd number is specified, it will be internally changed to the closest greater even number. The default value is two.

Second line (only needed if START CONDITION=USER DEFINED):

1. Starting value.

Data lines to define a Type I Chebyshev filter:

First line:

1. Cutoff frequency, fc. (Units of T^{-1}.)
2. Ripple factor, ϵ.
3. Order of the filter, N. Abaqus expects an even number; if an odd number is specified, it will be changed internally to the closest greater even number. The default value is two.

Second line (only needed if START CONDITION=USER DEFINED):

1. Starting value.

Data lines to define a Type II Chebyshev filter:

First line:

1. Cutoff frequency, fc. (Units of T^{-1}.)
2. Ripple factor, $1/A$.
3. Order of the filter, *N*. Abaqus expects an even number; if an odd number is specified, it will be changed internally to the closest greater even number. The default value is two.

Second line (only needed if START CONDITION=USER DEFINED):

1. Starting value.

*FIXED MASS SCALING: Specify mass scaling at the beginning of the step.

This option is used to specify mass scaling at the beginning of the step for part or all of the model.

Products: Abaqus/Explicit

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Mass Scaling*
- *About Output*

Optional parameters:

DT

Set this parameter equal to the desired element stable time increment for the element set provided. This parameter must be used in conjunction with the TYPE parameter to define how the mass scaling is to be applied.

If both the FACTOR and DT parameters are omitted, a default mass scaling value of 1.0 is used. If both parameters are included, the mass is first scaled by the value assigned to the FACTOR parameter and then possibly scaled again, depending on the values assigned to the DT and TYPE parameters.

ELSET

Set this parameter equal to the name of the element set for which this mass scaling definition is being applied. If this parameter is omitted, the mass scaling definition will apply to all elements in the model.

The **FIXED MASS SCALING* option can be repeated with different ELSET definitions to define different mass scaling for the specified element sets.

FACTOR

Set this parameter equal to the mass scaling factor. The masses of the elements will be scaled once at the beginning of the step by the value assigned to the FACTOR parameter.

If both the FACTOR and DT parameters are omitted, a default mass scaling value of 1.0 is used. If both parameters are included, the mass is first scaled by the value assigned to the FACTOR parameter and then possibly scaled again, depending on the values assigned to the DT and TYPE parameters.

TYPE

Set TYPE=UNIFORM to scale the masses of the elements equally so that the smallest element stable time increment of the scaled elements equals the value assigned to DT.

Set TYPE=BELOW MIN (default) to scale the masses of only the elements whose element stable time increments are less than the value assigned to DT. The masses of these elements will be scaled so that the element stable time increments equal the value assigned to DT.

Set TYPE=SET EQUAL DT to scale the masses of all elements so that they all have the same element stable time increment equal to the value assigned to DT.

There are no data lines associated with this option.

*FLEXIBLE BODY: Generate a flexible body from an Abaqus/Standard substructure or a natural frequency extraction.

This option is used to generate a flexible body from an Abaqus/Standard substructure or a natural frequency extraction. Abaqus/Standard generates several flexible body types from a substructure for use by external flexible body dynamics solvers. The generated flexible body entities are stored in the substructure `.sim` file and can be postprocessed to generate the input data for external flexible body dynamics solvers.

Alternatively, Abaqus/Standard can generate a flexible body from a natural frequency extraction for use in the Simpack flexible body dynamics solver. In this case, Abaqus creates a separate `.sim` file named *jobname*`_FREQ`*n*`.sim`, where *jobname* is the name of the input file or analysis job and *n* is the number of the Abaqus step that performs the natural frequency extraction. In this file, Abaqus stores a modal system along with the generated flexible body entities.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Generating Substructures*
- *Natural Frequency Extraction*
- **SUBSTRUCTURE GENERATE*
- **FREQUENCY*

Optional parameters:

FORMULATION

Include this parameter with TYPE=EXCITE or TYPE=ADAMS to specify the flexible body formulation. This parameter cannot be used with other supported flexible body types.

If you omit the FORMULATION parameter with TYPE=EXCITE , Abaqus generates the general motion flexible body formulation known as the CON6 flexible body for the AVL EXCITE™ flexible body dynamics solver from AVL LIST GmbH. This formulation requires generation of all inertia invariants of the CON6 flexible body.

Set FORMULATION=REDUCED with TYPE=EXCITE to generate a reduced version of the CON6 flexible body using only inertia invariants of the first order.

Set FORMULATION=SMALL MOTION with TYPE=EXCITE to generate the small-motion flexible body formulation known as the SMOT flexible body for the AVL EXCITE™ flexible body dynamics solver from AVL LIST GmbH.

If you omit the FORMULATION parameter with TYPE=ADAMS, Abaqus generates the most general version of the flexible body for the Adams™ flexible body dynamics solver from MSC.Software Corporation. This formulation includes nine inertia invariants J1, J2, …, J9.

Set FORMULATION=REDUCED with TYPE=ADAMS to generate a reduced version of the flexible body for the Adams™ flexible body dynamics solver from MSC.Software Corporation with only inertia invariants of the first order: J1, J2, J3, J4, J6, J7, and J8.

TYPE

Set TYPE=ADAMS to generate a flexible body entities for the Adams™ flexible body dynamics solver from MSC.Software Corporation.

Set TYPE=EXCITE to generate flexible body entities for the AVL EXCITE™ flexible body dynamics solver from AVL LIST GmbH.

Set TYPE=GENERIC (default) to generate a generic flexible body.

Set TYPE=SID to generate the Standard Input Data representation of the flexible body.

Set TYPE=SIMPACK to generate a Flexible Body Interface (FBI) file from a substructure or modal system for use in the Simpack flexible body dynamics solver. The FBI file is created during the Abaqus/Standard analysis. If the Abaqus model units are not SI units, you must use the **UNIT SYSTEM* option to define the model units so that the FBI file is created properly in SI units. Alternatively, you can use the **abaqus tosimpack** translator to convert the flexible body in the substructure or modal system `.sim` file to an FBI file for use in Simpack.

There are no data lines associated with this option.

*FLOW: Define seepage coefficients and associated sink pore pressures.

This option is used to provide seepage coefficients and sink pore pressures to control pore fluid flow normal to the surface in consolidation analysis.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Pore Fluid Flow*
- *FLOW*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that gives the variation of reference pore pressure with time. If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step or linearly over the step depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (see *Defining an Analysis*). The AMPLITUDE parameter is ignored for nonuniform seepage flow boundary conditions defined in user subroutine *FLOW* and for drainage-only seepage boundary conditions.

OP

Set OP=MOD (default) for existing **FLOW*s to remain, with this option modifying existing flows or defining additional flows.

Set OP=NEW if all existing **FLOW*s applied to the model should be removed. New flows can be defined.

Data lines to define uniform seepage:

First line:

1. Element number or element set label.
2. Seepage flow type label (see *About the Element Library*).
3. Reference pore pressure value, u_w^∞. (Units of FL^{-2}.)
4. Reference seepage coefficient value, k_s. (Units of $F^{-1}L^3T^{-1}$.)

Repeat this data line as often as necessary to define uniform seepage for various elements or element sets.

Data lines to define drainage-only seepage:

First line:

1. Element number or element set label.
2. Seepage flow type label (see *About the Element Library*).
3. Drainage-only seepage coefficient value, k_s. (Units of $F^{-1}L^3T^{-1}$.)

Repeat this data line as often as necessary to define drainage-only seepage for various elements or element sets.

Data lines to define nonuniform seepage:

First line:

1. Element number or element set label.
2. Seepage flow type label (see *About the Element Library*).
3. Optional reference pore pressure value. If given, this value is passed into user subroutine *FLOW* in the variable used to define the sink pore pressure.
4. Optional reference seepage coefficient. If given, this value is passed into user subroutine *FLOW* in the variable used to define the seepage coefficient.

The reference pore pressure value, u_w^∞, and reference seepage coefficient, k_s, are defined in user subroutine *FLOW* for nonuniform flow.

Repeat this data line as often as necessary to define nonuniform seepage for various elements or element sets.

*FLUID BEHAVIOR: Define fluid behavior for a fluid cavity.

This option is used to define the fluid behavior for a surface-based fluid cavity.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

• *Fluid Cavity Definition*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the fluid behavior.

Optional parameters:

USER

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to specify a fluid in which the fluid constitutive model is defined in user subroutine *UFLUID*.

There are no data lines associated with this option.

*FLUID BULK MODULUS: Define compressibility for a hydraulic fluid.

This option is used to define compressibility for the hydraulic fluid model. It can be used only in conjunction with the **FLUID BEHAVIOR* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Fluid Cavity Definition*
- **FLUID BEHAVIOR*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the fluid bulk modulus, in addition to temperature. If this parameter is omitted, it is assumed that the fluid bulk modulus depends only on temperature. See *Material Data Definition* for more information.

Data lines to define compressibility for a hydraulic fluid:

First line:

1. Fluid bulk modulus, K.
2. Temperature.
3. First field variable.
4. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to specify K as a function of temperature and field variables.

*FLUID CAVITY: Define a fluid cavity.

This option is used to define a surface-based fluid cavity.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

• *Fluid Cavity Definition*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the fluid cavity.

REF NODE

Set this parameter equal to either the node number of the fluid cavity reference node or the name of a node set containing the fluid cavity reference node. If the name of a node set is chosen, the node set must contain exactly one node.

Required, mutually exclusive parameters:

BEHAVIOR

Set this parameter equal to the name of the **FLUID BEHAVIOR* option defining the fluid behavior.

MIXTURE

This parameter applies only to Abaqus/Explicit analyses.

Set MIXTURE=MASS FRACTION (default) to use the mass fraction if the fluid in the fluid cavity is a mixture of ideal gases.

Set MIXTURE=MOLAR FRACTION to use the molar fraction if the fluid in the fluid cavity is a mixture of ideal gases.

Optional parameters:

ADDED VOLUME

Set this parameter equal to the magnitude of the additional volume for the fluid. The additional volume will be added to the actual volume of the cavity that is calculated.

ADIABATIC

This parameter applies only to Abaqus/Explicit analyses.

This parameter is relevant only when an ideal gas model is used. Include this parameter if adiabatic behavior is assumed for the ideal gas.

AMBIENT PRESSURE

Set this parameter equal to the magnitude of the ambient pressure. For a pneumatic fluid, the ambient pressure will typically be atmospheric pressure.

AMBIENT TEMPERATURE

This parameter applies only to Abaqus/Explicit analyses and is relevant only when heat energy flow is defined for a pneumatic fluid with adiabatic behavior.

Set this parameter equal to the magnitude of the ambient temperature. The ambient temperature will typically be atmospheric temperature.

CHECK NORMALS

This parameter is relevant only when the surface is defined to form the boundary of the fluid cavity.

Set CHECK NORMALS=YES (default) to check the consistency of the surface normals.

Set CHECK NORMALS=NO to skip the consistency checking for the surface normals.

MINIMUM VOLUME

This parameter applies only to Abaqus/Explicit analyses.

Use this parameter to define the magnitude of the minimum volume for the fluid cavity. If the volume of the cavity (which is equal to the actual volume plus the added volume) drops below the minimum, the minimum value will be used to evaluate the equation of state model.

Set this parameter equal to a positive value to define the minimum volume directly.

Set MINIMUM VOLUME=INITIAL VOLUME to set the minimum volume equal to the initial volume of the cavity. If the initial volume of the fluid cavity is a negative value, the minimum volume will be set equal to zero.

SURFACE

This parameter is always required in Abaqus/Standard analyses. In Abaqus/Explicit analyses this parameter is required if the ADDED VOLUME parameter is omitted.

Set this parameter equal to the name of the surface forming the boundary of the fluid cavity.

SWITCHTIME

This parameter applies only to Abaqus/Explicit analyses to switch from the lumped kinetic molecular procedure to the uniform pressure method.

Set the value of the parameter equal to the time at which Abaqus/Explicit is to switch from the lumped kinetic molecular procedure to the uniform pressure method.

Data line if the BEHAVIOR parameter is included:

First (and only) line:

1. Out-of-plane thickness of the surface for two-dimensional models when the SURFACE parameter is included. If this value is left blank or is entered as zero, the default value of 1.0 will be used. Enter a blank line when the surface is defined by three-dimensional and axisymmetric elements or when the SURFACE parameter is omitted.

Data lines if the MIXTURE parameter is included:

First line:

1. Out-of-plane thickness of the surface for two-dimensional models when the SURFACE parameter is included. If this value is left blank or is entered as zero, the default value of 1.0 will be used. Enter a blank line when the surface is defined by three-dimensional and axisymmetric elements or when the SURFACE parameter is omitted.

Second line:

1. Name of fluid behavior forming the gas mixture.
2. Mass fraction or molar fraction.

Repeat this data line as often as necessary to define the initial gas mixture.

*FLUID DENSITY: Specify hydrostatic fluid density.

This option is used to define the reference fluid density for fluid cavities. It is applicable only for hydraulic fluids and should not be used for pneumatic fluids and user-defined fluids. The **FLUID DENSITY* option can be used only in conjunction with the **FLUID BEHAVIOR* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Fluid Cavity Definition*
- **FLUID BEHAVIOR*

There are no parameters associated with this option.

Data line to define the reference fluid density:

First (and only) line:

1. Reference fluid density, ρ_R.

*FLUID EXCHANGE: Define fluid exchange.

This option is used to define fluid exchange between two fluid cavities or between a fluid cavity and its environment.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Fluid Exchange Definition*
- **FLUID EXCHANGE PROPERTY*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the fluid exchange definition.

PROPERTY

Set this parameter equal to the name of the **FLUID EXCHANGE PROPERTY* option defining the fluid exchange property.

Optional parameters:

CAVITY PRESSURE

This parameter applies only to Abaqus/Explicit analyses.

Set CAVITY PRESSURE=SURFACE (default) to indicate that the fluid pressure from the fluid cavity should be applied on the surface used for fluid exchange.

Set CAVITY PRESSURE=PERIMETER to identify the fluid exchange surface as an open vent and to apply the fluid pressure as an equivalent load on the perimeter of the surface.

CONSTANTS

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the number of fluid exchange constants needed as data to define the effective area for fluid exchange in user subroutine *VUFLUIDEXCHEFFAREA*. The default is CONSTANTS=0.

EFFECTIVE AREA

This parameter applies only to Abaqus/Explicit analyses.

Set EFFECTIVE AREA equal to the total area for the exchange. The default value is 1.0 if the SURFACE parameter is omitted. Otherwise, the default value is equal to the area of the surface. If both the EFFECTIVE AREA and SURFACE parameters are specified, the area of the surface is used only to determine blockage and the effective area is reduced to the extent that the surface is blocked.

Set EFFECTIVE AREA=USER to indicate that user subroutine *VUFLUIDEXCHEFFAREA* in Abaqus/Explicit will be used to define the effective area of the surface taking the local material state

into account. The SURFACE parameter is required if user subroutine *VUFLUIDEXCHEFFAREA* is used.

MAX RUPTURE AREA RATIO

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to a threshold ratio above which the fluid cavity is deactivated. The ratio is defined as the surface area from failed elements over the total surface area associated with this fluid exchange. This parameter is valid only if RUPTURE=ON.

RUPTURE

This parameter applies only to Abaqus/Explicit analyses.

Set RUPTURE=ON to allow fluid and/or heat energy to be exchanged over surfaces from failed elements enclosing a fluid cavity.

Set RUPTURE=OFF (default) to indicate that exchanging fluid and/or heat energy over surfaces from failed elements enclosing the fluid cavity is not allowed.

SURFACE

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the name of the surface on the fluid cavity over which fluid and/or heat energy may be exchanged. If this parameter is omitted, the value specified with the EFFECTIVE AREA parameter is used for the exchange. This parameter is required if EFFECTIVE AREA=USER or RUPTURE=ON.

Data lines to define the fluid exchange:

First line:

1. First reference node number of fluid cavity.
2. Second reference node number of fluid cavity (required in Abaqus/Standard; optional in Abaqus/Explicit). If you specify only one node in an Abaqus/Explicit analysis, fluid exchange occurs between the fluid cavity and its environment.

Second line (needed only if the CONSTANTS parameter is used):

1. Enter the values of the fluid exchange constants to define the effective area for fluid exchange, eight per line.

Repeat this data line as often as necessary to define all properties.

*FLUID EXCHANGE ACTIVATION: Activate fluid exchange definitions.

This option is used to activate fluid exchange definitions between two fluid cavities or between a fluid cavity and its environment.

Products: Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *Fluid Cavity Definition*
- *Fluid Exchange Definition*
- **FLUID EXCHANGE*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve defining a multiplier for the fluid exchange magnitude. See *Amplitude Curves*.

BLOCKAGE

Set BLOCKAGE=YES to consider vent and leakage area obstruction by contacted surfaces. The default value is BLOCKAGE=NO.

DELTA LEAKAGE AREA

Set this parameter equal to the ratio of the actual surface area over the initial surface area at which you want the fluid to leak. This real value should be positive and greater than or equal to one. The effective surface area used for the fluid exchange is the difference between the actual area of the surface and the area of the surface at the beginning of step.

OP

Set OP=MOD (default) for existing **FLUID EXCHANGE ACTIVATION* definitions to remain, with this option defining a fluid exchange activation to be added or modified.

Set OP=NEW if all fluid exchange activations that are currently in effect should be removed. To remove only selected fluid exchange activations, use OP=NEW and respecify all fluid exchange activations that are to be retained.

OUTFLOW ONLY

Include this parameter if the flow is allowed only from the first fluid cavity to the second fluid cavity defined in the **FLUID EXCHANGE* option.

If this parameter is omitted, the flow is allowed from both directions. The reference nodes defined on the data line on the **FLUID EXCHANGE* option should be in the appropriate order to obtain the desired flow direction.

Data lines to define the fluid exchange activation:

First line:

1. List of fluid exchange names.

Repeat this data line as often as necessary. Up to 8 entries are allowed per line.

*FLUID EXCHANGE PROPERTY: Define the fluid exchange property for flow in or out of a fluid cavity.

This option is used to define the fluid exchange property for flow between two fluid cavities or between a fluid cavity and its environment.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Fluid Exchange Definition*
- **FLUID EXCHANGE*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the fluid exchange property.

TYPE

Set TYPE=BULK VISCOSITY to define fluid exchange where the mass flow rate is related to the pressure difference by both viscous and hydrodynamic resistance coefficients.

Set TYPE=ENERGY FLUX to define fluid exchange by specifying the heat energy flow rate leakage explicitly. This parameter value applies only to Abaqus/Explicit analyses.

Set TYPE=ENERGY RATE LEAKAGE to define fluid exchange by specifying the heat energy flow rate as a function of temperature difference and pressure. This parameter value applies only to Abaqus/Explicit analyses.

Set TYPE=FABRIC LEAKAGE to define fluid exchange due to fabric leakage.

Set TYPE=MASS FLUX to define fluid exchange by specifying the mass flow rate leakage explicitly.

Set TYPE=MASS RATE LEAKAGE to define fluid exchange by specifying the mass flow rate as a function of pressure difference and temperature.

Set TYPE=ORIFICE to define fluid exchange through a vent orifice. This parameter value applies only to Abaqus/Explicit analyses.

Set TYPE=VOLUME FLUX to define fluid exchange by specifying the volume rate leakage explicitly.

Set TYPE=VOLUME RATE LEAKAGE to define fluid exchange by specifying the volume rate leakage as a function of pressure difference and temperature.

Set TYPE=USER to indicate that user subroutine *VUFLUIDEXCH* is used in Abaqus/Explicit to define fluid exchange by specifying the mass flow rate and/or heat energy flow rate.

Optional parameters:

CONSTANTS

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the number of constant values needed as data to define the fluid exchange in user subroutine *VUFLUIDEXCH*. The default is CONSTANTS=0.

DEPENDENCIES

Set this parameter equal to the number of field variables included in the specification of the coefficients defined by the TYPE parameter. If this parameter is omitted, the coefficients are assumed not to depend on any field variables.

DEPVAR

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the number of solution-dependent state variables required for user subroutine *VUFLUIDEXCH*. The default is DEPVAR=0.

Data lines for TYPE=BULK VISCOSITY:

First line:

1. Viscous resistance coefficient.
2. Hydrodynamic resistance coefficient.
3. Average absolute pressure, if pressure dependent.
4. Average temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to specify the viscous and hydrodynamic resistance coefficients as functions of average absolute pressure, average temperature, and other predefined field variables.

Data line for TYPE=ENERGY FLUX:

First (and only) line:

1. Heat energy flow rate per unit area.

Data lines for TYPE=ENERGY RATE LEAKAGE:

First line:

1. Absolute value of the heat energy flow rate per unit area. (The first tabular value entered must always be zero.)
2. Temperature difference. (The first tabular value entered must always be zero.)
3. Average absolute pressure, if pressure dependent.
4. Average temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the heat energy flow rate as a function of temperature difference, average absolute pressure, average temperature, and other predefined field variables.

Data lines for TYPE=FABRIC LEAKAGE or TYPE=ORIFICE:

First line:

1. Discharge coefficient that is used to modify the exhaust or leakage surface area. The default value is 1.
2. Absolute pressure, if pressure dependent.
3. Temperature, if temperature dependent.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the discharge coefficient as a function of pressure, temperature, and other predefined field variables.

Data line for TYPE=MASS FLUX:

First (and only) line:

1. Mass flow rate per unit area.

Data lines for TYPE=MASS RATE LEAKAGE:

First line:

1. Absolute value of the mass flow rate per unit area. (The first tabular value entered must always be zero.)
2. Absolute value of the pressure difference. (The first tabular value entered must always be zero.)
3. Average absolute pressure, if pressure dependent.
4. Average temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to specify the mass flow rate as a function of pressure difference, average absolute pressure, average temperature, and other predefined field variables.

Data line for TYPE=VOLUME FLUX:

First (and only) line:

1. Volumetric flow rate per unit area.

Data lines for TYPE=VOLUME RATE LEAKAGE:

First line:

1. Absolute value of the volumetric flow rate per unit area. (The first tabular value entered must always be zero.)
2. Absolute value of the pressure difference. (The first tabular value entered must always be zero.)
3. Average absolute pressure, if pressure dependent.
4. Average temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the volume rate leakage as a function of pressure difference, average absolute pressure, average temperature, and other predefined field variables.

Data lines for TYPE=USER:

First line:

1. Enter the values of the fluid exchange constants, eight per line.

Repeat this data line as often as necessary to define all fluid exchange constants.

*FLUID EXPANSION: Specify the thermal expansion coefficient for a hydraulic fluid.

This option is used to define thermal expansion coefficients for the hydraulic fluid model. It can be used only in conjunction with the **FLUID BEHAVIOR* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Fluid Cavity Definition*
- **FLUID BEHAVIOR*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the thermal expansion coefficient, in addition to temperature. If this parameter is omitted, it is assumed that the thermal expansion coefficient depends only on temperature. See *Material Data Definition* for more information.

ZERO

Set this parameter equal to the value of θ_0. The default is ZERO=0.

Data lines to define the thermal expansion coefficient:

First line:

1. Mean coefficient of thermal expansion, α.
2. Temperature, θ.
3. First field variable.
4. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to specify α as a function of θ and field variables.

*FLUID FLUX: Change the amount of fluid in a fluid-filled cavity.

This option is used to specify a change in the amount of fluid in a fluid-filled cavity modeled with hydrostatic fluid elements.

Products: Abaqus/Standard Abaqus/Explicit

Type: History data

Level: Step

References:

• *Fluid Exchange Definition*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude versus time curve that defines the magnitude of the mass flow rate during the step (*Amplitude Curves*). If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step, regardless of the procedure being used in the step.

OP

Set OP=MOD (default) for existing fluid fluxes to remain, with this option defining fluid fluxes to be added (to cavities with no fluid flux loading) or modified (to cavities with fluid flux loading).

Set OP=NEW if all existing fluid fluxes applied to the model should be removed.

Data line to define the fluid mass flow rate:

First (and only) line:

1. Node number or node set label of the cavity reference node.
2. Reference magnitude of the fluid mass flow rate, q. (Units of MT^{-1}.)

*FLUID ID FIELD: Associate a fluid identifier with a predefined field variable in a multi-fluid definition.

This option is used to facilitate multiple fluids definition in fluid pipe elements, cohesive elements that support fluid and slurry transport (for cases where the constitutive response of the fluid transitions from Darcy to Poiseuille flow), and enriched elements that support fluid and slurry transport. In particular, it is used to associate a fluid identifier with a predefined field variable.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Defining the Constitutive Response of Fluid Transitioning from Darcy Flow to Poiseuille Flow*
- *Defining Slurry Transport and Placement within the Cohesive Element Gap*
- *Fluid Pipe Elements*
- *Thermal Fluid Pipe Elements*
- *Slurry Fluid Pipe Elements*
- *Modeling Discontinuities as an Enriched Feature Using the Extended Finite Element Method*

Data line to define fluid identification field:

First line:

1. Fluid identification number.
2. Predefined field variable number to be associated with the fluid identification number.

Repeat this data line as often as necessary to define the variation.

*FLUID INFLATOR: Define a fluid inflator.

This option is used to define a fluid inflator to model deployment of an airbag.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Interaction module

References:

- *Inflator Definition*
- **FLUID INFLATOR PROPERTY*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the fluid inflator.

PROPERTY

Set this parameter equal to the name of the **FLUID INFLATOR PROPERTY* option defining the fluid inflator property.

Data line to define the fluid inflator:

First (and only) line:

1. Reference node of the fluid cavity.

*FLUID INFLATOR ACTIVATION: Activate fluid inflator definitions.

This option is used to activate fluid inflator definitions.

Products: Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *Inflator Definition*
- **FLUID INFLATOR*

Optional parameters:

INFLATION TIME AMPLITUDE

Set this parameter equal to the name of the amplitude curve defining a mapping between the inflation time and the actual time. If this parameter is omitted, the inflation time will be equal to the actual time elapsed since activation.

MASS FLOW AMPLITUDE

Set this parameter equal to the name of the amplitude curve by which to modify the mass flow rate. This parameter is valid only if the mass flow rate is prescribed directly in the inflator property definition. It will be ignored if the mass flow rate is calculated by using tank test data or the dual pressure method.

OP

Set OP=MOD (default) for existing **FLUID INFLATOR ACTIVATION* definitions to remain, with this option defining a fluid inflator activation to be added or modified.

Set OP=NEW if all fluid inflator activations that are currently in effect should be removed. To remove only selected fluid inflator activations, use OP=NEW and respecify all fluid inflator activations that are to be retained.

Data lines to define the fluid inflator activation:

First line:

1. List of fluid inflator names.

Repeat this data line as often as necessary. Up to 8 entries are allowed per line.

*FLUID INFLATOR MIXTURE: Define gas species used for a fluid inflator.

This option is used to define the gas species used for a fluid inflator. The **FLUID INFLATOR MIXTURE* option can be used only in conjunction with the **FLUID INFLATOR PROPERTY* option.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Interaction module

References:

- *Fluid Cavity Definition*
- *Inflator Definition*
- **FLUID BEHAVIOR*
- **FLUID INFLATOR PROPERTY*

Required parameters:

NUMBER SPECIES

Set this parameter equal to the number of gas species used for this inflator.

Optional parameters:

TYPE

Set TYPE=MASS FRACTION (default) to use the mass fraction for a mixture of ideal gases.

Set TYPE=MOLAR FRACTION to use the molar fraction for a mixture of ideal gases.

Data lines to define gas species for a fluid inflator:

First line:

1. Fluid behavior name.
2. Etc., up to eight fluid behavior names per line.

Repeat this data line as often as necessary to define all gas species for this inflator.

Next line:

1. Inflation time.
2. Mass fraction or molar fraction for the first entry of fluid behavior.
3. Mass fraction or molar fraction for the second entry of fluid behavior.
4. Etc., mass fraction or molar fraction up to the seventh entry of fluid behavior.

Subsequent lines (only needed if the NUMBER SPECIES parameter has a value greater than seven):

1. Mass fraction or molar fraction for the eighth entry of fluid behavior.
2. Etc., mass fraction or molar fraction for up to eight entries of fluid behavior per line.

Repeat this set of data lines as often as necessary to define the mass fraction or molar fraction as a function of inflation time.

*FLUID INFLATOR PROPERTY: Define a fluid inflator property.

This option is used to define a fluid inflator property to model the deployment of an airbag.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Interaction module

References:

- *Inflator Definition*
- **FLUID INFLATOR*

Required parameters:

EFFECTIVE AREA

This parameter is relevant only for TYPE=DUAL PRESSURE and TYPE=PRESSURE AND MASS.

Set this parameter equal to the total inflator orifice area.

NAME

Set this parameter equal to a label that will be used to refer to the fluid inflator property.

TANK VOLUME

This parameter is relevant only for TYPE=DUAL PRESSURE or TYPE=TANK TEST.

Set this parameter equal to the tank volume.

TYPE

Set TYPE=DUAL PRESSURE to use the dual pressure method to obtain the mass flow rate of the gas species.

Set TYPE=PRESSURE AND MASS to use the given mass flow rate and inflator pressure to obtain the gas temperature.

Set TYPE=TANK TEST to use tank test data to obtain the mass flow rate of the gas species.

Set TYPE=TEMPERATURE AND MASS to use the given mass flow rate and inflator gas temperature to obtain the gas pressure.

Optional parameters:

DISCHARGE COEFFICIENT

This parameter is relevant only for TYPE=DUAL PRESSURE and TYPE=PRESSURE AND MASS.

Set this parameter equal to the discharge coefficient of the inflator orifice. The default value is 0.4.

Data lines for TYPE=DUAL PRESSURE:

First line:

1. Inflator time.
2. Inflator pressure.
3. Tank pressure.

Repeat this data line as often as necessary to define the inflator pressure and tank pressure as functions of inflation time.

Data lines for TYPE=PRESSURE AND MASS:

First line:

1. Inflation time.
2. Inflator pressure.
3. Inflator mass flow rate.

Repeat this data line as often as necessary to define the inflator pressure and inflator mass flow rate as functions of inflation time.

Data lines for TYPE=TANK TEST:

First line:

1. Inflation time.
2. Inflator gas temperature.
3. Tank pressure.

Repeat this data line as often as necessary to define the inflator gas temperature and tank pressure as functions of inflation time.

Data lines for TYPE=TEMPERATURE AND MASS:

First line:

1. Inflation time.
2. Inflator gas temperature.
3. Inflator mass flow rate.

Repeat this data line as often as necessary to define the inflator gas temperature and inflator mass flow rate as functions of inflation time.

*FLUID LEAKOFF: Define fluid leak-off coefficients for pore pressure cohesive elements and enriched elements.

This option is used to define leak-off coefficients for pore pressure cohesive elements and enriched elements.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Defining the Constitutive Response of Fluid within the Cohesive Element Gap*
- *Defining the Constitutive Response of Fluid Flow within the Cracked Element Surfaces*
- *UFLUIDLEAKOFF*

Optional, mutually exclusive parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the fluid leak-off coefficients, in addition to temperature. If this parameter is omitted, it is assumed that the leak-off coefficients are constant or depend only on temperature. See *Material Data Definition* for more information.

USER

Include this parameter to indicate that user subroutine *UFLUIDLEAKOFF* will be used to define the fluid leak-off coefficients.

Optional parameters:

SLURRY

Include this parameter to account for the evolution of the filter cake thickness, viscosity, and permeability.

Data lines to define fluid leak-off coefficients if the USER parameter is omitted:

First line:

1. Fluid leak-off coefficient at top element surface (units of $L^3T^{-1}F^{-1}$).
2. Fluid leak-off coefficient at bottom element surface(units of $L^3T^{-1}F^{-1}$).
3. Temperature.
4. First field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to specify K as a function of temperature and field variables.

Data line to define the filter cake coefficients if the SLURRY parameter is included and the USER parameter is omitted:

First (and only) line:

1. Initial filter cake thickness h_0.
2. Correction factor, C_f, to account for variation in particle sizes.
3. Filter cake compressibility coefficient α. Enter a value of 0 if the filter cake is incompressible.
4. Filter cake permeability factor, C_d. This value is calculated automatically if this field is left blank.

There are no data lines when the USER parameter is included

*FLUID PIPE CONNECTOR LOSS: Specify fluid pipe connector element loss.

This option is used to specify the loss that happens during fluid flow through a fluid pipe connector element. It must be used in conjunction with the **FLUID PIPE CONNECTOR SECTION* option.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

• *Fluid Pipe Connector Elements*

Required parameters:

TYPE

Set TYPE=CONNECTION to define the connector loss parameters.

Set TYPE=DARBY3K to compute the connector loss using the Darby 3K method.

Set TYPE=HOOPER2K to compute the connector loss using the Hooper 2K method. The Hooper 2K method is unit-dependent and requires the hydraulic diameter to be specified in inches.

Set TYPE=USER to compute the connector loss parameters using user subroutine *UFLUIDCONNECTORLOSS*.

Optional parameters:

LAMINAR FLOW TRANSITION

This parameter is valid only when TYPE=HOOPER2K or TYPE=DARBY3K.

Set this parameter equal to the Reynold's number for the flow in the connector below which the linear formulation is used. If this parameter is omitted, the default value is 1.0.

VALVE CONTROL

Set this parameter equal to USER to define a valve behavior for the connector in user subroutine *UFLUIDCONNECTORVALVE*. If this parameter is omitted, no valve behavior is defined and the fluid is fully flowing.

Data line to define connector loss (TYPE=CONNECTION):

First (and only) line:

1. Hydraulic area.
2. Hydraulic diameter.
3. Connector loss when the flow is in the direction determined by the element connectivity.
4. Connector loss when the flow is opposite to the direction determined by the element connectivity.

Data line to define connector loss (TYPE=DARBY3K):

First (and only) line:

1. Hydraulic area.
2. Hydraulic diameter.
3. Resistance coefficient K_1 for the 3K method.

4. Resistance coefficient K_∞ for the 3K method.
5. Resistance coefficient K_D for the 3K method.

Data line to define connector loss (TYPE=HOOPER2K):

First (and only) line:

1. Hydraulic area. (Units of square inches.)
2. Hydraulic diameter. (Units of square inches.)
3. Resistance coefficient K_1 for the 2K method.
4. Resistance coefficient K_∞ for the 2K method.

Data line to define connector loss (TYPE=USER):

First (and only) line:

1. Hydraulic area.
2. Hydraulic diameter.

*FLUID PIPE CONNECTOR SECTION: Specify fluid pipe connector element section properties.

This option is used to specify the fluid pipe connector element section properties.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

• *Fluid Pipe Connector Elements*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the fluid pipe connector elements for which the section properties are being defined.

MATERIAL

Set this parameter equal to the name of the material that describes the fluid flowing through the fluid pipe connector element.

There are no data lines associated with this option.

*FLUID PIPE CONNECTOR THERMAL LOSS: Define the method to solve for the convective heat transfer solution in thermal fluid pipe connector elements.

This option is used to define the conduction loss, convective loss, and diffusion loss factors in thermal fluid pipe connector elements. It can be used only in conjunction with the **FLUID PIPE CONNECTOR SECTION* option.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- **FLUID PIPE CONNECTOR SECTION*
- *Thermal Fluid Pipe Connector Elements*

There are no parameters associated with this option.

Data lines to define the thermal losses:

First (and only) line:

1. Conduction thermal loss.
2. Convection thermal loss.
3. Diffusion thermal loss.

*FLUID PIPE CONNECTOR WALL: Specify wall geometry for thermal fluid pipe connector elements.

This option is used to specify the connector wall geometry for thermal fluid pipe connector elements. It can be used only in conjunction with the **FLUID PIPE CONNECTOR SECTION* option.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- **FLUID PIPE CONNECTOR SECTION*
- *Thermal Fluid Pipe Connector Elements*

There are no parameters associated with this option.

Data line to define the pipe wall geometry:

First (and only) line:

1. Inner radius of the connector wall.
2. Outer radius of the connector wall.

*FLUID PIPE FLOW LOSS: Specify fluid pipe element loss.

This option is used to specify the loss that happens during fluid flow through a fluid pipe element. It must be used in conjunction with the **FLUID PIPE SECTION* option.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

• *Fluid Pipe Elements*

Required parameters:

TYPE

Set TYPE=BLASIUS to define the friction parameters using Blasius's equation for the friction factor.

Set TYPE=CHURCHILL to define the friction parameters using Churchill's equation for the friction factor.

Set TYPE=TABULAR to define the Reynold's number-friction values in a tabular form.

Set TYPE=USER to define the connector loss parameters using user subroutine *UFLUIDPIPEFRICTION*.

Optional parameters:

LAMINAR FLOW TRANSITION

Set this parameter equal to the Reynold's number for the flow in the pipe below which the linear formulation is used. If this parameter is omitted, the default value is 1.0.

SLURRY SETTLING

Set SLURRY SETTLING=TURIAN to model the additional pressure loss due to slurry flow.

Data line to define pipe loss (TYPE=BLASIUS):

First (and only) line:

1. Hydraulic area.
2. Hydraulic diameter.
3. Additional pipe length to be added to the element for increasing the friction loss.
4. Scaling factor for the pipe length to increase friction loss in the element.
5. Addition of a connector loss to friction factor when the fluid flows in the direction determined by the element connectivity.
6. Addition of a connector loss to friction factor when the fluid flows opposite to the direction determined by the element connectivity.

Data line to define pipe loss (TYPE=CHURCHILL):

First (and only) line:

1. Hydraulic area.
2. Hydraulic diameter.

3. Additional pipe length to be added to the element for increasing the friction loss.
4. Scaling factor for the pipe length to increase friction loss in the element.
5. Addition of a connector loss to friction factor when the fluid flows in the direction determined by the element connectivity.
6. Addition of a connector loss to friction factor when the fluid flows opposite to the direction determined by the element connectivity.
7. Surface roughness of the pipe used in the friction factor calculation.

Data lines to define pipe loss (TYPE=TABULAR):

First line:

1. Hydraulic area.
2. Hydraulic diameter.
3. Additional pipe length to be added to the element for increasing the friction loss.
4. Scaling factor for the pipe length to increase friction loss in the element.
5. Addition of a connector loss to friction factor when the fluid flows in the direction determined by the element connectivity.
6. Addition of a connector loss to friction factor when the fluid flows opposite to the direction determined by the element connectivity.

Second line:

1. Reynolds number.
2. Friction coefficient.

Repeat this data line as often as necessary.

Data line to define pipe loss (TYPE=USER):

First (and only) line:

1. Hydraulic area.
2. Hydraulic diameter.
3. Surface roughness of the pipe used in the friction factor calculation.

Data lines to redefine the default Turian settling coefficients (SLURRY SETTLING=TURIAN):

These data lines are optional. If you choose to redefine the coefficients, these data lines appear after the data lines required for the various pipe loss definitions. You must redefine all the coefficients for the four different flow regimes as described by Turian. For more information, see *Slurry Fluid Pipe Equations*.

First optional line:

1. Coefficient A_1 for sliding bed (Regime 0).
2. Coefficient A_2 for sliding bed (Regime 0).
3. Coefficient A_3 for sliding bed (Regime 0).
4. Coefficient A_4 for sliding bed (Regime 0).
5. Coefficient A_5 for sliding bed (Regime 0).

Second optional line:

1. Coefficient A_1 for saltation (Regime 1).
2. Coefficient A_2 for saltation (Regime 1).
3. Coefficient A_3 for saltation (Regime 1).
4. Coefficient A_4 for saltation (Regime 1).

5. Coefficient A_5 for saltation (Regime 1).

Third optional line:

1. Coefficient A_1 for heterogeneous suspension (Regime 2).
2. Coefficient A_2 for heterogeneous suspension (Regime 2).
3. Coefficient A_3 for heterogeneous suspension (Regime 2).
4. Coefficient A_4 for heterogeneous suspension (Regime 2).
5. Coefficient A_5 for heterogeneous suspension (Regime 2).

Fourth optional line:

1. Coefficient A_1 for homogeneous suspension (Regime 3).
2. Coefficient A_2 for homogeneous suspension (Regime 3).
3. Coefficient A_3 for homogeneous suspension (Regime 3).
4. Coefficient A_4 for homogeneous suspension (Regime 3).
5. Coefficient A_5 for homogeneous suspension (Regime 3).

*FLUID PIPE SECTION: Specify fluid pipe element section properties.

This option is used to specify the section properties of fluid pipe elements.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

• *Fluid Pipe Elements*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the fluid pipe elements for which the section properties are being defined.

MATERIAL

Set this parameter equal to the name of the material that describes the fluid flowing through the pipe.

Optional parameters:

SYMMETRY

Set this parameter equal to a value greater than 0 and less than or equal to 1 to enforce symmetric modeling of fluid pipe elements. A value of 0.5 corresponds to modeling half symmetry, and a value of 0.25 corresponds to modeling quarter symmetry.

When this parameter is used, you must specify the magnitude of the fluid injected into the pipe elements in the **CFLOW* option as the full model flow scaled by the symmetry value.

There are no data lines associated with this option.

*FLUID PIPE THERMAL: Define the method to solve for the convective heat transfer solution in the thermal fluid pipe elements.

This option is used to define whether a constant flux or constant temperature solution method is chosen for the convective heat transfer solution. It can be used only in conjunction with the **FLUID PIPE SECTION* option.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- **FLUID PIPE SECTION*
- *Thermal Fluid Pipe Elements*

Required parameters:

TYPE

SET TYPE=FLUX to define a constant flux solution for convective heat transfer.

SET TYPE=TEMPERATURE to define a constant temperature solution for convective heat transfer.

There are no data lines associated with this option.

*FLUID PIPE WALL: Specify wall geometry for thermal fluid pipe elements.

This option is used to specify the pipe wall geometry for thermal fluid pipe elements. It can be used only in conjunction with the **FLUID PIPE SECTION* option.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- **FLUID PIPE SECTION*
- *Thermal Fluid Pipe Elements*

There are no parameters associated with this option.

Data line to define the pipe wall geometry:

First (and only) line:

1. Inner radius of pipe wall.
2. Outer radius of pipe wall.

*FOUNDATION: Prescribe element foundations.

This option is used to model foundations on elements.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Interaction module

References:

• *Element Foundations*

There are no parameters associated with this option.

Data lines to define element foundations:

First line:

1. Element number or element set label.
2. Foundation type identification, F*n*.
3. Foundation stiffness per area (or per length for beams).

Repeat this data line as often as necessary to define foundations for various elements or element sets.

*FRACTURE CRITERION: Specify crack propagation criteria.

This option is used to specify the criterion for crack propagation along initially partially bonded surfaces. It must appear immediately following the *DEBOND option in Abaqus/Standard and after the *COHESIVE BEHAVIOR option in Abaqus/Explicit. This option can also be used in Abaqus/Standard to specify a linear elastic fracture mechanics-based criterion for crack propagation in enriched elements. It must appear immediately following the *SURFACE BEHAVIOR option in Abaqus/Standard in this case.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data in Abaqus/Standard; Model data in Abaqus/Explicit

Level: Model or Step in Abaqus/Standard; Model in Abaqus/Explicit

Abaqus/CAE: *Property module* and *Interaction module*

References:

- *Crack Propagation Analysis*
- *Modeling Discontinuities as an Enriched Feature Using the Extended Finite Element Method*
- *Fatigue Crack Growth Laws for Linear Elastic Materials*
- *UMIXMODEFATIGUE*
- *DEBOND
- *COHESIVE BEHAVIOR
- *SURFACE BEHAVIOR

Required parameters:

DISTANCE

This parameter is required only if TYPE=COD or TYPE=CRITICAL STRESS is used.

If TYPE=CRITICAL STRESS, set this parameter equal to the distance along the potential crack surface ahead of the crack tip at which the critical stress criterion is evaluated.

If TYPE=COD, set this parameter equal to the distance behind the crack tip along the secondary surface at which the crack opening displacement is measured.

NSET

This parameter is required only if TYPE=CRACK LENGTH. Set this parameter equal to the name of the node set containing the nodes that are used to define the reference point.

TYPE

Set TYPE=CRITICAL STRESS to use the critical stress criterion at a distance ahead of the crack tip as the crack propagation criterion. This setting is available only in Abaqus/Standard.

Set TYPE=COD to use the critical value of the crack opening displacement at a distance behind the crack tip as the crack propagation criterion. This setting is available only in Abaqus/Standard.

Set TYPE=CRACK LENGTH to specify the crack length as a function of time. This setting is available only in Abaqus/Standard.

Set TYPE=ENHANCED VCCT to use the enhanced VCCT (Virtual Crack Closure Technique) criterion in which the onset and growth of a crack can be controlled by two different critical fracture energy release rates. This setting is available only in Abaqus/Standard.

Set TYPE=FATIGUE to indicate that the onset and fatigue crack growth are characterized by the relative fracture energy release rate or the relative stress intensity factor at the crack tip based on the Paris law. This setting is available only in Abaqus/Standard.

Set TYPE=VCCT to use the VCCT (Virtual Crack Closure Technique) criterion as the crack propagation criterion. The VCCT criterion uses the principles of linear elastic fracture mechanics.

Set TYPE=PRX-VCCT to use the PRX-VCCT (Progressive Release eXplicit - Virtual Crack Closure Technique) criterion as the crack propagation criterion.

Set TYPE=PRX-FATIGUE to use the PRX-FATIGUE criterion as the fatigue crack propagation criterion.

Optional parameters:

ANGLEMAX

This parameter is relevant only for TYPE=ENHANCED VCCT, TYPE=FATIGUE, or TYPE=VCCT in enriched elements.

Set this parameter equal to the maximum allowed change in the crack propagation angle (in degrees) between the new crack propagation direction and the previous crack propagation direction. The default is 85°.

DEPENDENCIES

This parameter is not relevant for TYPE=CRACK LENGTH.

Set this parameter equal to the number of field variable dependencies included in the data lines. If this parameter is omitted, it is assumed that the data are constant or depend only on temperature. See *Material Data Definition* for more information.

K-BASED

This parameter can be used only in conjunction with enriched elements with MIXED MODE BEHAVIOR=TABULAR or MIXED MODE BEHAVIOR=USER.

Include this parameter to specify a fatigue crack growth criterion based on a stress intensity factor rather than the fracture energy release rate.

LAMBDA

This parameter is relevant only for TYPE=PRX-VCCT and TYPE=PRX-FATIGUE.

Set this parameter equal to the maximum increment in damage that is allowed within each time increment. The default is LAMBDA=0.1.

MIXED MODE BEHAVIOR

This parameter is relevant only for TYPE=ENHANCED VCCT, TYPE=FATIGUE, or TYPE=VCCT.

Set MIXED MODE BEHAVIOR=BK to specify the fracture energy as a function of the mode mix by means of the Benzeggagh-Kenane mixed mode fracture criterion.

Set MIXED MODE BEHAVIOR=IRWIN to specify the effective stress intensity factor as a function of the mode mix by means of the Irwin mixed-mode fracture criterion. This option can be used only in conjunction with TYPE=FATIGUE in enriched elements.

Set MIXED MODE BEHAVIOR=POWER to specify the fracture energy as a function of the mode mix by means of a power law mixed mode fracture criterion.

Set MIXED MODE BEHAVIOR=REEDER to specify the fracture energy as a function of the mode mix by means of the REEDER mixed mode fracture criterion.

Set MIXED MODE BEHAVIOR=TABULAR to specify the crack growth criterion directly as a function of the total maximum fracture energy release rate, mode mix ratio, and stress ratio in a

mixed-mode fatigue crack growth analysis. This option can be used only in conjunction with TYPE=FATIGUE.

Set MIXED MODE BEHAVIOR=USER to specify a user-defined crack growth criterion in a mixed-mode fatigue crack growth analysis using user subroutine *UMIXMODEFATIGUE*. This option can be used only in conjunction with TYPE=FATIGUE.

The default is MIXED MODE BEHAVIOR=BK.

NODAL ENERGY RATE

This parameter is relevant only for TYPE=FATIGUE or TYPE=VCCT.

Include this parameter to indicate that the critical energy release rates should not be read from the data lines but should be interpolated from the critical energy release rates specified at the nodes with the **NODAL ENERGY RATE* option. The exponents are still read from the data lines.

NORMAL DIRECTION

This parameter can be used only in conjunction with TYPE=ENHANCED VCCT, TYPE=FATIGUE, or TYPE=VCCT for enriched elements in Abaqus/Standard.

Set NORMAL DIRECTION=MTS (default) to specify that the crack will propagate orthogonal to the direction of the maximum tangential stress when the fracture criterion is satisfied.

Set NORMAL DIRECTION=1 to specify that the crack will propagate orthogonal to the element local 1-direction when the fracture criterion is satisfied.

Set NORMAL DIRECTION=2 to specify that the crack will propagate orthogonal to the element local 2-direction when the fracture criterion is satisfied.

NPOLY

This parameter can be used only in conjunction with POSITION=NONLOCAL in enriched elements or in predefined initially partially bonded surfaces in Abaqus/Standard. It is used to specify the number of terms in the polynomial used for the moving least-squares approximation.

Set NPOLY=4 to use a linear polynomial approximation.

Set NPOLY=7 (default) to use a quadratic polynomial approximation.

Set NPOLY=10 to use a cubic polynomial approximation.

NPOWER

This parameter is relevant only for TYPE=PRX-VCCT and defines the rate of crack growth.

Set this parameter equal to a positive integer. The default is NPOWER=10.

POSITION

This parameter is relevant for TYPE=ENHANCED VCCT, TYPE=FATIGUE, TYPE=PRX-VCCT, or TYPE=VCCT in predefined initially partially bonded surfaces in Abaqus/Standard or when NORMAL DIRECTION=MTS is specified in enriched elements.

In enriched elements, set POSITION=NONLOCAL to use a moving least-squares approximation by polynomials to smooth out the normals of the individual crack facets in elements that satisfy the fracture criterion to obtain the crack propagation direction along the crack front.

In predefined initially partially bonded surfaces, set POSITION=NONLOCAL to use a moving least-squares approximation by polynomials to smooth out the tangential directions of the individual crack segments along the crack front to obtain a smoother pseudocrack tangential direction. This direction is used for calculating energy release rates under mixed-mode loading.

PROPERTIES

This parameter can be used only in conjunction with MIXED MODE BEHAVIOR=USER.

Set this parameter equal to the number of material constants being specified for a user-defined crack growth criterion in a mixed-mode fatigue crack growth analysis. The parameter value must be a nonzero value.

R CRACK DIRECTION

This parameter can be used only in conjunction with POSITION=NONLOCAL in enriched elements or in predefined initially partially bonded surfaces.

In enriched elements, set this parameter equal to the radius around the crack tip within which the elements along the crack front are included for smoothing out the normals of the individual crack facets to obtain the crack propagation direction.

In predefined initially partially bonded surfaces, set this parameter equal to the radius around the crack tip within which the elements along the crack front are included for smoothing out the tangential directions of the individual crack segments along the crack front.

The default value is three times the typical element characteristic length along the crack front in the model.

SYMMETRY

Include this parameter to compare the opening between the secondary surface and the symmetry plane to half the COD value specified. The SYMMETRY parameter is relevant only for TYPE=COD when the user is using symmetry conditions to model the problem. In this case the NORMAL parameter must be specified on the **INITIAL CONDITIONS* option.

TOLERANCE

Set this parameter equal to the tolerance within which the crack propagation criterion must be satisfied. The default is TOLERANCE=0.1 for TYPE=CRITICAL STRESS, TYPE=COD, and TYPE=CRACK LENGTH; for TYPE=ENHANCED VCCT, TYPE=PRX-VCCT, and TYPE=VCCT, the default is TOLERANCE=0.2.

This parameter can also be used in a fatigue crack growth analysis (TYPE=FATIGUE) with gradual release of the debonding forces in debonding nodes when DEBONDING FORCE=RAMP is used on the **DEBOND* option. Set this parameter equal to the tolerance within which the scalar damage parameter reaches the value 1.0, resulting in the complete release of the debonding force. The default is 0.05.

UNSTABLE GROWTH TOLERANCE

Set this parameter equal to the tolerance within which the unstable crack propagation criterion must be satisfied for multiple nodes at and ahead of the crack tip to be allowed to debond without the cut back of increment size in one increment when the TYPE=PRX-VCCT or VCCT criterion is satisfied for an unstable crack problem.

If this parameter is included without a specified value, the default value is infinity.

VISCOSITY

This parameter applies only to Abaqus/Standard analyses and can be used only in combination with TYPE=ENHANCED VCCT or TYPE=VCCT.

Set this parameter equal to the value of the viscosity coefficient used in the viscous regularization. The default value is 0.0.

Data lines to define the critical stress criterion (TYPE=CRITICAL STRESS):

First line:

1. Normal failure stress, σ^f.
2. Shear failure stress, τ_1^f.

3. Shear failure stress, τ_2^f. (Not applicable in two dimensions.)
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the critical stress criterion as a function of temperature and/or field variables.

Data lines to define the crack opening displacement criterion (TYPE=COD):

First line:

1. Critical crack opening displacement, δ_c.
2. Cumulative crack length.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the crack opening displacement criterion as a function of temperature and/or field variables.

Data lines to define the crack length versus time criterion (TYPE=CRACK LENGTH):

First line:

1. Total time (not step time).
2. Crack length, l, from the reference point.
3. Etc., up to four time/length pairs per line. Crack length must be given as an increasing function of time.

Repeat this data line as often as necessary to define the crack length as a function of time.

Data lines to define the onset and growth of a crack for the enhanced VCCT criterion (TYPE=ENHANCED VCCT) for MIXED MODE BEHAVIOR=BK or REEDER:

First line:

1. Mode I critical energy release rate for onset of a crack, G_{IC}.
2. Mode II critical energy release rate for onset of a crack, G_{IIC}.
3. Mode III critical energy release rate for onset of a crack, G_{IIIC}.
4. Mode I critical energy release rate for crack propagation, G_{IC}^P.
5. Mode II critical energy release rate for crack propagation, G_{IIC}^P.

6. Mode III critical energy release rate for crack propagation, G_{IIIC}^{P}.
7. Exponent, η.
8. Temperature.

Subsequent lines (only needed if the DEPENDENCIES parameter has a nonzero value):

1. First field variable.
2. Second field variable.
3. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the critical energy rates and exponent as a function of temperature and field variables.

Data lines to define the onset and growth of a crack for the enhanced VCCT criterion (TYPE=ENHANCED VCCT) for MIXED MODE BEHAVIOR=POWER:

First line:

1. Mode I critical energy release rate for onset of a crack, G_{IC}.
2. Mode II critical energy release rate for onset of a crack, G_{IIC}.
3. Mode III critical energy release rate for onset of a crack, G_{IIIC}.
4. Mode I critical energy release rate for crack propagation, G_{IC}^{P}.
5. Mode II critical energy release rate for crack propagation, G_{IIC}^{P}.
6. Mode III critical energy release rate for crack propagation, G_{IIIC}^{P}.
7. Exponent, a_m.
8. Exponent, a_n.

Second line:

1. Exponent, a_o.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the critical energy rates and exponents as a function of temperature and field variables.

Data lines to define the low-cycle fatigue onset and crack growth criterion (TYPE=FATIGUE or TYPE=PRX-FATIGUE criterion) for MIXED MODE BEHAVIOR=BK or REEDER:

First line:

1. Material constant for fatigue crack initiation, c_1. (Not used for TYPE=PRX-FATIGUE.)
2. Material constant for fatigue crack initiation, c_2. (Not used for TYPE=PRX-FATIGUE.)
3. Material constant for fatigue crack growth, c_3.
4. Material constant for fatigue crack growth, c_4.

5. Ratio of energy release rate threshold used in the Paris law over the equivalent critical energy release rate, $\frac{G_{thresh}}{G_C}$.
6. Ratio of energy release rate upper limit used in the Paris law over the equivalent critical energy release rate, $\frac{G_{pl}}{G_C}$.
7. Mode I critical energy release rate, G_{IC}.
8. Mode II critical energy release rate, G_{IIC}.

Second line:

1. Mode III critical energy release rate, G_{IIIC}.
2. Exponent, η.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc. up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the constants used in the Paris law, critical energy rates, and exponents as a function of temperature and field variables.

Data lines to define the low-cycle fatigue onset and crack growth criterion (TYPE=FATIGUE or TYPE=PRX-FATIGUE criterion) for MIXED MODE BEHAVIOR=POWER:

First line:

1. Material constant for fatigue crack initiation, c_1. (Not used for TYPE=PRX-FATIGUE.)
2. Material constant for fatigue crack initiation, c_2. (Not used for TYPE=PRX-FATIGUE.)
3. Material constant for fatigue crack growth, c_3.
4. Material constant for fatigue crack growth, c_4.
5. Ratio of energy release rate threshold used in the Paris law over the equivalent critical energy release rate, $\frac{G_{thresh}}{G_C}$.
6. Ratio of energy release rate upper limit used in the Paris law over the equivalent critical energy release rate, $\frac{G_{pl}}{G_C}$.
7. Mode I critical energy release rate, G_{IC}.
8. Mode II critical energy release rate, G_{IIC}.

Second line:

1. Mode III critical energy release rate, G_{IIIC}.
2. Exponent, a_m.
3. Exponent, a_n.
4. Exponent, a_o.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the constants used in the Paris law, the critical energy rates, and exponents as a function of temperature and field variables.

Data lines to define fatigue crack growth criterion (TYPE=FATIGUE criterion) for MIXED MODE BEHAVIOR=IRWIN:

First line:

1. Material constant, c_3, for fatigue crack growth based on $\frac{da}{dN} = c_3 \Delta K_{eff}^{c_4}$.
2. Material constant, c_4, for fatigue crack growth based on $\frac{da}{dN} = c_3 \Delta K_{eff}^{c_4}$.
3. Ratio of equivalent stress intensity factor threshold used in the Paris law over the equivalent critical stress intensity factor, $\frac{K_{thresh}}{K_C}$. The default is 0.01.
4. Ratio of equivalent stress intensity factor upper limit used in the Paris law over the equivalent critical stress intensity factor, $\frac{K_{pl}}{K_C}$. The default is 0.85.
5. Equivalent critical stress intensity factor, K_C.
6. Constant coefficient, A, defined in $\Delta K_{eff} = \sqrt{A \times \Delta K_I^2 + B \times \Delta K_{II}^2 + C \times \frac{1}{1-\nu} \Delta K_{III}^2}$.
7. Constant coefficient, B, defined in $\Delta K_{eff} = \sqrt{A \times \Delta K_I^2 + B \times \Delta K_{II}^2 + C \times \frac{1}{1-\nu} \Delta K_{III}^2}$.
8. Constant coefficient, C, defined in $\Delta K_{eff} = \sqrt{A \times \Delta K_I^2 + B \times \Delta K_{II}^2 + C \times \frac{1}{1-\nu} \Delta K_{III}^2}$.

Second line (leave blank if no temperature or field variables are specified):

1. Temperature.
2. First field variable.
3. Second field variable.
4. Etc., up to seven field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than seven):

1. Eighth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the constants used in the Paris law and the critical stress intensity factor as a function of temperature and field variables.

Data lines to define fatigue crack growth criterion (TYPE=FATIGUE or TYPE=PRX-FATIGUE criterion) for MIXED MODE BEHAVIOR=TABULAR:

First line:

1. Crack growth rate, $\frac{da}{dN}$ in log scale.
2. Total maximum energy release rate, G_{TMax} in log scale.
3. Mode mix ratio, $\frac{G_{IIMax}+G_{IIIMax}}{G_{TMax}}$.

4. Local stress ratio, $\frac{G_{TMin}}{G_{TMax}}$.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the crack growth rates as a function of energy release rate, mode mix ratio, local stress ratio, temperature, and field variables.

Data lines to define fatigue crack growth criterion (TYPE=FATIGUE for MIXED MODE BEHAVIOR=TABULAR, K-BASED:

First line:

1. Constant coefficient, A, defined in $\Delta K_{eff} = \sqrt{A \times \Delta K_I^2 + B \times \Delta K_{II}^2 + C \times \frac{1}{1-\nu} \Delta K_{III}^2}$.
2. Constant coefficient, B, defined in $\Delta K_{eff} = \sqrt{A \times \Delta K_I^2 + B \times \Delta K_{II}^2 + C \times \frac{1}{1-\nu} \Delta K_{III}^2}$.
3. Constant coefficient, C, defined in $\Delta K_{eff} = \sqrt{A \times \Delta K_I^2 + B \times \Delta K_{II}^2 + C \times \frac{1}{1-\nu} \Delta K_{III}^2}$.

Second line:

1. Crack growth rate, $\frac{da}{dN}$, in log scale.
2. Effective stress intensity factor range of a load cycle,

   $\Delta K_{eff} = \sqrt{A \times \Delta K_I^2 + B \times \Delta K_{II}^2 + C \times \frac{1}{1-\nu} \Delta K_{III}^2}$, in log scale.

3. Mode mix ratio, $\frac{\sqrt{B \times K_{IIMax}^2 + C \times \frac{1}{1-\nu} K_{IIIMax}^2}}{\sqrt{A \times K_{IMax}^2 + B \times K_{IIMax}^2 + C \times \frac{1}{1-\nu} K_{IIIMax}^2}}$.
4. Local stress ratio, $\frac{\sqrt{A \times K_{IMin}^2 + B \times K_{IIMin}^2 + C \times \frac{1}{1-\nu} K_{IIIMin}^2}}{\sqrt{A \times K_{IMax}^2 + B \times K_{IIMax}^2 + C \times \frac{1}{1-\nu} K_{IIIMax}^2}}$.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines (exclude the first line) as often as necessary to define the crack growth rates as a function of the effective stress intensity factor range, mode mix ratio, local stress ratio, temperature, and field variables.

Data lines to define fatigue crack growth criterion (TYPE=FATIGUE or TYPE=PRX-FATIGUE criterion) for MIXED MODE BEHAVIOR=USER:

First line:

1. Give the material constants, eight per line.

Repeat this data line as often as necessary to define all material constants.

Data lines to define the VCCT criterion (TYPE=VCCT) or PRX-VCCT criterion (TYPE=PRX-VCCT) for MIXED MODE BEHAVIOR=BK or REEDER:

First line:

1. Mode I critical energy release rate, G_{IC}.
2. Mode II critical energy release rate, G_{IIC}.
3. Mode III critical energy release rate, G_{IIIC}.
4. Exponent, η.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the critical energy rates and exponent as a function of temperature and field variables.

Data lines to define the VCCT criterion (TYPE=VCCT) or PRX-VCCT criterion (TYPE=PRX-VCCT) for MIXED MODE BEHAVIOR=POWER:

First line:

1. Mode I critical energy release rate, G_{IC}.
2. Mode II critical energy release rate, G_{IIC}.
3. Mode III critical energy release rate, G_{IIIC}.
4. Exponent, a_m.
5. Exponent, a_n.
6. Exponent, a_o.
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the critical energy rates and exponents as a function of temperature and field variables.

*FRAME SECTION: Specify a frame section.

This option is used to define the cross-section for frame elements. Since frame section geometry and material descriptions are combined, no *MATERIAL reference is associated with this option.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- *Frame Elements*
- *Frame Section Behavior*

Required parameters:

ELSET

Set this parameter equal to the name of the element set for which the section is defined.

Optional parameters:

BUCKLING

Include this parameter to indicate that buckling strut response is permitted for these elements and that the default buckling envelope is to be used. When this parameter is included, the YIELD STRESS parameter is required to determine P_{cr} and P_y on the buckling envelope.

To include buckling strut response with a nondefault buckling envelope, use the *BUCKLING ENVELOPE option in conjunction with the *FRAME SECTION option and the YIELD STRESS parameter. If both the BUCKLING parameter and *BUCKLING ENVELOPE option are present, the user-defined buckling envelope takes precedence.

To define effective length factors and added lengths for the first and second cross-section directions with either the default or nondefault buckling envelope, use the *BUCKLING LENGTH option in conjunction with the *FRAME SECTION option. To define buckling reduction factors for the first and second cross-section directions with either the default or nondefault buckling envelope, use the *BUCKLING REDUCTION FACTORS option in conjunction with the *FRAME SECTION option.

DENSITY

Set this parameter equal to the mass density per unit volume of the frame element material. This parameter is needed only when the mass of the element is required, such as in dynamic analysis or for gravity loading.

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of material properties, in addition to temperature. If this parameter is omitted, it is assumed that the properties are constant or depend only on temperature. See *Material Data Definition* for more information.

PINNED

Include this parameter to indicate that these elements have uniaxial response only; that is, the ends have pinned connections.

If this parameter is used and both the BUCKLING parameter and the **BUCKLING ENVELOPE* option are absent, these elements have linear elastic uniaxial response from the beginning of the analysis. If this parameter is used and the BUCKLING parameter or **BUCKLING ENVELOPE* option are present, these elements have uniaxial response with buckling and postbuckling behavior in compression and isotropic hardening plasticity in tension as described by the buckling envelope option from the beginning of the analysis. The **BUCKLING LENGTH* option can be used with this parameter when the BUCKLING parameter or **BUCKLING ENVELOPE* option is present.

This parameter cannot be used with the PLASTIC DEFAULTS parameter or with any of the **PLASTIC* options.

PLASTIC DEFAULTS

Include this parameter to indicate that elastic-plastic material response is included and that all plastic options are created with default values based on the yield stress defined with the YIELD STRESS parameter. The YIELD STRESS parameter is required when this parameter is used.

To include elastic-plastic material response with user-defined plastic material response, use one or more (as appropriate) of the **PLASTIC AXIAL*, **PLASTIC M1*, **PLASTIC M2*, and **PLASTIC TORQUE* options in conjunction with the **FRAME SECTION* option. If the PLASTIC DEFAULTS and YIELD STRESS parameters are omitted, only those plastic options specified will be included in the elastic-plastic material response.

This parameter cannot be used with the PINNED parameter.

SECTION

Set this parameter equal to the name of a library section to choose a standard library section (see *Beam Cross-Section Library*). The following cross-sections are available for elastic frame elements (when elastic-plastic and buckling strut response are omitted):

- BOX, for a rectangular, hollow box section.
- CIRC, for a solid circular section.
- GENERAL, for a general cross-section (default).
- I, for an I-beam section.
- PIPE, for a hollow, circular section.
- RECT, for a solid rectangular section.

For elastic-plastic material response the only available plastic interaction surface is an ellipsoid, which is recommended for PIPE cross-sections only. Other cross-section types, except the GENERAL section, can be used at the user's discretion.

For buckling strut response only the PIPE cross-section is available.

YIELD STRESS

Set this parameter equal to the yield stress for the material making up the cross-section.

This parameter is required when defining default elastic-plastic material response with the PLASTIC DEFAULTS parameter and when modeling buckling strut response by using the **BUCKLING ENVELOPE* option or the BUCKLING parameter.

ZERO

Set this parameter equal to the reference temperature for thermal expansion (θ^0), if required. The default is ZERO=0.

Data lines for SECTION=GENERAL:

First line:

1. Area, A.
2. Moment of inertia for bending about the 1-axis, I_{11}.
3. Moment of inertia for cross bending, I_{12}.
4. Moment of inertia for bending about the 2-axis, I_{22}.
5. Torsional constant, J.

Second line (optional; enter a blank line if the default values are to be used):

1. First direction cosine of the first element section axis.
2. Second direction cosine of the first element section axis.
3. Third direction cosine of the first element section axis.

The entries on this line must be (0, 0, −1) for FRAME2D elements. The default for FRAME3D elements is (0, 0, −1) if the first element section axis is not defined by an additional node in the element's connectivity. See *Frame Elements* for details.

Third line:

1. Young's modulus, E.
2. Torsional shear modulus, G. (This value is ignored for FRAME2D elements.)
3. Coefficient of thermal expansion.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the properties as a function of temperature and other predefined field variables.

Data lines for BOX, CIRC, I, PIPE, and RECT sections:

First data line:

1. Element section geometric data. Values should be given as specified in *Beam Cross-Section Library* for the chosen section type.
2. Etc.

Second data line (optional; enter a blank line if the default values are to be used):

1. First direction cosine of the first element section axis.
2. Second direction cosine of the first element section axis.
3. Third direction cosine of the first element section axis.

The entries on this line must be (0, 0, −1) for FRAME2D elements. The default for FRAME3D elements is (0, 0, −1) if the first element section axis is not defined by an additional node in the element's connectivity. See *Frame Elements* for details.

Third data line:

1. Young's modulus, E.
2. Torsional shear modulus, G. (This value is ignored for FRAME2D elements.)
3. Coefficient of thermal expansion.
4. Temperature.
5. First field variable.

6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the properties as a function of temperature and other predefined field variables.

*FREQUENCY: Extract natural frequencies and modal vectors.

This option is used to perform eigenvalue extraction to calculate the natural frequencies and corresponding mode shapes of a system.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

• *Natural Frequency Extraction*

Optional parameters:

ACOUSTIC COUPLING

For the AMS eigensolver and Lanczos eigensolver, set ACOUSTIC COUPLING=ON to include the effect of acoustic-structural coupling during the natural frequency extraction procedure in models with acoustic and structural elements coupled using the **TIE* option or in models with ASI-type elements. This is the default option for the Lanczos eigensolver.

For the AMS eigensolver and Lanczos eigensolver based on the SIM architecture, set ACOUSTIC COUPLING=PROJECTION to extract the uncoupled acoustic and structural modes and project the acoustic-structural coupling operator during the natural frequency extraction procedure in models with acoustic and structural elements coupled using the **TIE* option. This is the default option for the AMS eigensolver.

Set ACOUSTIC COUPLING=OFF to omit the projection of the acoustic-structural coupling operator and to ignore the effect of acoustic-structural coupling during natural frequency extraction in models with acoustic and structural elements coupled using the **TIE* option or in models with ASI-type elements.

This parameter is not relevant for the subspace iteration eigensolver.

DAMPING PROJECTION

This parameter is relevant only for the AMS eigensolver or for the Lanczos eigensolver used in conjunction with the SIM parameter.

Set DAMPING PROJECTION=ON (default) to project the viscous and structural damping operators during the natural frequency extraction procedure. If there is no damping defined in the model, Abaqus/Standard does not perform the projection.

Set DAMPING PROJECTION=OFF to omit the projection of damping operators.

EIGENSOLVER

Set EIGENSOLVER=LANCZOS (default) to invoke the Lanczos eigensolver.

Set EIGENSOLVER=AMS to invoke the automatic multilevel substructuring (AMS) eigensolver.

Set EIGENSOLVER=SUBSPACE to invoke the subspace iteration eigensolver.

NORMALIZATION

Set NORMALIZATION=DISPLACEMENT to normalize the eigenvectors so that the largest displacement, rotation, or acoustic pressure (in coupled acoustic-structural extractions) entry in each

vector is unity. Displacement normalization is the default for both the subspace iteration eigensolver and for the Lanczos eigensolver when you do not include the SIM parameter.

Set NORMALIZATION=MASS to normalize the eigenvectors with respect to the structure's mass matrix (Abaqus/Standard scales the eigenvectors so that the generalized mass for each vector is unity). Mass normalization is the default and only option available for the AMS eigensolver. Abaqus/Standard switches on mass normalization for both the Lanczos eigensolver and the subspace iteration eigensolver when you use them in conjunction with the default SIM parameter.

PROPERTY EVALUATION

Set this parameter equal to the frequency at which to evaluate frequency-dependent properties for viscoelasticity, springs, and dashpots during the eigenvalue extraction. If you omit this parameter, Abaqus/Standard evaluates the stiffness associated with frequency-dependent springs and dashpots at zero frequency and does not consider the stiffness contributions from frequency domain viscoelasticity in the **FREQUENCY* step.

RESIDUAL MODES

This parameter is relevant only for the Lanczos and AMS eigensolvers.

Include this parameter to indicate that Abaqus/Standard should compute residual modes.

SIM

This parameter is relevant only for the Lanczos and subspace iteration eigensolvers.

Set the value of this parameter equal to NO if the non-SIM architecture is required for the Lanczos or subspace iteration eigensolvers.

Set the value of this parameter equal to YES (default) if the SIM architecture is required.

The SIM architecture is the only option if the AMS eigensolver is activated.

Optional parameter when EIGENSOLVER=AMS:

NSET

Set this parameter equal to the name of the node set or include the parameter with no value to allow Abaqus/Standard to automatically select the nodes at which to compute eigenvectors. If you omit this parameter, Abaqus/Standard computes eigenvectors at all nodes.

Data line for a natural frequency extraction when EIGENSOLVER=LANCZOS:

First (and only) line:

1. Number of eigenvalues to calculate. You can leave this field blank if you provide the maximum frequency of interest and want to evaluate all the eigenvalues in the given range. You must provide the number of requested eigenmodes in a cyclic symmetry analysis or if the analysis includes more than one natural frequency extraction step.
2. Minimum frequency of interest, in cycles/time. If you leave this field blank, Abaqus/Standard does not set a minimum.
3. Maximum frequency of interest, in cycles/time. If you leave this field blank, Abaqus/Standard does not set a maximum. This value is required if you left the first field blank.
4. Shift point, in squared cycles per time (positive or negative). Abaqus/Standard extracts the eigenvalues closest to this point.
5. Block size. If you omit this entry, Abaqus/Standard creates a default value, which is usually appropriate.
6. Maximum number of block Lanczos steps within each Lanczos run. If you omit this entry, Abaqus/Standard creates a default value, which is usually appropriate.

7. Acoustic range factor. This factor applies only to structural-acoustic problems and is used to set the maximum frequency for the acoustic stage of the uncoupled eigenproblem as a multiple of the nominal maximum frequency of interest. Abaqus/Standard supports this factor only when using the SIM architecture and you provide the maximum frequency of interest. The acoustic range factor must be greater than 0. The default value is 1.0.

Subsequent lines when default residual modes are not sufficient or you request residual modes:

1. Node number or node set label.
2. First degree of freedom for which residual modes are requested.
3. Last degree of freedom for which residual modes are requested. You can leave this field blank if you request residual modes for only one degree of freedom.

Repeat this line as often as necessary to request residual modes.

Data lines for a natural frequency extraction when EIGENSOLVER=AMS:

First line:

1. Number of eigenvalues to calculate. If you leave this field blank, Abaqus evaluates all the eigenvalues from the minimum frequency of interest up to the maximum frequency of interest.
2. Minimum frequency of interest, in cycles/time. If you leave this field blank, Abaqus does not set a minimum.
3. Maximum frequency of interest, in cycles/time.
4. AMS_{cutoff_1} multiplied by the maximum frequency of interest is used to truncate local substructure eigenmodes in the reduction phase.
5. AMS_{cutoff_2} multiplied by the maximum frequency of interest is used to truncate local branch substructure eigenmodes for defining a starting subspace in the reduced eigensolution phase.
6. AMS_{cutoff_3} multiplied by the maximum frequency of interest is used to truncate local leaf substructure eigenmodes for defining a starting subspace in the reduced eigensolution phase.
7. Acoustic range factor. This factor applies only to structural-acoustic problems. Use it to set the maximum frequency for the acoustic stage of the uncoupled eigenproblem as a multiple of the nominal maximum frequency of interest. The acoustic range factor must be greater than 0. The default value is 1.0.

Subsequent lines if default residual modes are not sufficient or you request residual modes:

1. Node number or node set label.
2. First degree of freedom for which residual modes are requested.
3. Last degree of freedom for which residual modes are requested. You can leave this field blank if you request residual modes for only one degree of freedom.

Repeat this line as often as necessary to request residual modes.

Data line for a natural frequency extraction when EIGENSOLVER=SUBSPACE:

First (and only) line:

1. Number of eigenvalues to be calculated.
2. Maximum frequency of interest, in cycles/time. If you leave this field blank, Abaqus/Standard does not set a maximum.

Abaqus/Standard extracts frequencies until either of the above limits is reached.

7. Shift point, in squared cycles per time (positive or negative). Abaqus/Standard extracts the eigenvalues closest to this point.
9. Number of vectors used in the iteration. If you omit this entry, Abaqus/Standard creates a default value, which is usually appropriate. The default number of vectors used is the minimum of (n+

8, 2*n*), where *n* is the number of eigenvalues requested (the first data item on this data line). In general, the convergence is more rapid with more vectors, but the memory requirement is also larger. Therefore, if you know that a particular type of eigenproblem converges slowly, you might reduce the analysis cost by providing more vectors.

11. Maximum number of iterations. The default is 30.

*FRICTION: Specify a friction model.

This option is used to introduce friction properties into a mechanical surface interaction model governing the interaction of contact surfaces, a contact pair, or connector elements. It must be used in conjunction with the *SURFACE INTERACTION option, the *CONNECTOR FRICTION option, or in an Abaqus/Standard analysis with the *CHANGE FRICTION, the *GAP, the *INTERFACE, or the *ITS options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data in Abaqus/Standard; History data in Abaqus/Explicit

Level: Part, Part instance, Assembly, Model in Abaqus/Standard; Step in Abaqus/Explicit

Abaqus/CAE: Interaction module

References:

- *Connector Behavior*
- *About Mechanical Contact Properties*
- *Frictional Behavior*
- *FRIC*
- *FRIC_COEF*
- *VFRIC*
- *VFRIC_COEF*
- *VFRICTION*
- *CHANGE FRICTION
- *CONNECTOR FRICTION
- *GAP
- *INTERFACE
- *ITS
- *SURFACE INTERACTION

Optional, mutually exclusive parameters:

ELASTIC SLIP

This parameter applies only to Abaqus/Standard analyses.

In a steady-state transport analysis set this parameter equal to the absolute magnitude of the allowable elastic slip velocity ($\dot{\gamma}_i$) to be used in the stiffness method for sticking friction. In all other analysis procedures set this parameter equal to the absolute magnitude of the allowable elastic slip (γ_i) to be used in the stiffness method for sticking friction. If this parameter is omitted, the elastic slip (or elastic slip velocity) is defined by the SLIP TOLERANCE value.

LAGRANGE

This parameter applies only to Abaqus/Standard analyses and cannot be used when friction is defined for connector elements.

Include this parameter to choose the Lagrange multiplier formulation for friction.

ROUGH

This parameter cannot be used when friction is defined for connector elements.

Include this parameter to specify completely rough (no slipping) friction.

SLIP TOLERANCE

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the value of F_f (defined as the ratio of allowable maximum elastic slip velocity to angular velocity times the diameter of the spinning body in a steady-state transport analysis or as the ratio of allowable maximum elastic slip to characteristic contact surface face dimension in all other analysis procedures). The default is SLIP TOLERANCE=.005.

When friction is defined for connector elements, F_f is defined (when possible) as the ratio of allowable maximum elastic slip to a characteristic element dimension in the model. In this case the default is SLIP TOLERANCE=.0001.

USER

This parameter cannot be used when friction is defined for connector elements.

In an Abaqus/Standard analysis, set USER=FRIC (default) if the friction model is to be defined in user subroutine *FRIC*. Set USER=COEFFICIENT if the friction coefficient is to be defined in user subroutine *FRIC_COEF*.

In an Abaqus/Explicit analysis, set USER=FRIC (default) if the friction model is to be defined in user subroutine *VFRIC*. Set USER=FRICTION if the friction model is to be defined in user subroutine *VFRICTION*. *VFRIC* is applicable to contact pairs, whereas *VFRICTION* is applicable to general contact. Set USER=COEFFICIENT if the friction coefficient is to be defined in user subroutine *VFRIC_COEF*. *VFRIC_COEF* can be used only with general contact.

Optional parameters:

ANISOTROPIC BEHAVIOR

If this parameter is omitted in an Abaqus/Standard analysis and the USER parameter is included, the user subroutine determines whether directional preference exists in resistance to sliding. Otherwise, the friction model remains isotropic.

If this parameter is omitted in an Abaqus/Explicit analysis, anisotropic frictional behavior occurs if directional preference is specified as a surface property.

Set ANISOTROPIC BEHAVIOR=LEGACY to specify anisotropic friction with directional preference associated with a contact orientation. This parameter value applies only to Abaqus/Standard analyses and cannot apply to friction associated with connector elements.

Set ANISOTROPIC BEHAVIOR=NONE to specify that the friction model remains isotropic, even if directional preference is assigned as a surface property.

Set ANISOTROPIC BEHAVIOR=SURFACE PROPERTY to allow anisotropic frictional behavior to occur if directional preference is specified as a surface property. This parameter value applies only to Abaqus/Explicit analyses.

COEFFICIENT PRECEDENCE

This parameter applies only to Abaqus/Explicit analyses and is used to control the precedence of specifying friction coefficients.

Set COEFFICIENT PRECEDENCE=CONTACT PROPERTY (default) to indicate that friction coefficients assigned as contact properties take precedence over friction coefficients derived from values specified as surface properties.

Set COEFFICIENT PRECEDENCE=SURFACE PROPERTY COMBINATION to indicate that friction coefficients derived from values specified as surface properties take precedence over friction coefficients assigned as contact properties.

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the friction coefficient in addition to slip rate, contact pressure, and temperature. If this parameter is omitted, it is assumed that the friction coefficients have no dependencies or depend only on slip rate, contact pressure, and temperature. See *Material Data Definition* for more information.

DEPVAR

This parameter is valid only if the USER parameter is included.

Set DEPVAR equal to the number of state-dependent variables required for user subroutines *FRIC_COEF* and *FRIC* in an Abaqus/Standard analysis or for user subroutines *VFRIC_COEF*, *VFRICTION*, and *VFRIC* in an Abaqus/Explicit analysis.

The default is DEPVAR=0.

EXPONENTIAL DECAY

Include this parameter to specify separate static and kinetic friction coefficients with a smooth transition zone defined by an exponential curve.

The ANISOTROPIC BEHAVIOR and TAUMAX parameters cannot be used with this parameter.

NOMINAL

This parameter applies only to Abaqus/Explicit analyses and is significant only if directional preferences are introduced as surface properties.

Set NOMINAL=AVERAGE (default) to specify an average friction coefficient on the data lines.

Set NOMINAL=MINIMUM to specify a minimum friction coefficient on the data lines.

Set NOMINAL=MAXIMUM to specify a maximum friction coefficient on the data lines.

PROPERTIES

This parameter is valid only if the USER parameter is included.

Set this parameter equal to the number of property values needed as data to define the friction model in user subroutine *FRIC* and *FRIC_COEF* in an Abaqus/Standard analysis or in user subroutines *VFRIC*, *VFRIC_COEF*, and *VFRICTION* in an Abaqus/Explicit analysis. The default is PROPERTIES=0.

SHEAR TRACTION SLOPE

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the slope of the curve that defines the shear traction as a function of the elastic slip between the two surfaces. If this parameter is omitted or frictional forces are not present, shear softening will not be activated. This parameter cannot be used in conjunction with user subroutines *VFRIC*, *VFRIC_COEF*, and *VFRICTION*.

TAUMAX

Set this parameter equal to the equivalent shear stress limit; that is, the maximum achievable value of the equivalent shear stress. The value given must be greater than zero.

TEST DATA

This parameter is valid only if the EXPONENTIAL DECAY parameter is used.

Include this parameter if the exponential decay coefficient, d_c, is to be computed by Abaqus. If this parameter is omitted, the decay coefficient must be given directly on the data line.

Data lines to define the coefficient of friction if the USER, ROUGH, EXPONENTIAL DECAY, and ANISOTROPIC BEHAVIOR parameters are omitted:

First line:

1. Friction coefficient, μ.
2. Slip rate, $\dot{\gamma}_{eq}$. If this value is omitted, the friction coefficient is assumed to be independent of the slip rate.
3. Contact pressure, p. If this value is omitted, the friction coefficient is assumed to be independent of the contact pressure.
4. Average temperature at the contact point, $\overline{\theta}$, between the two contact surfaces. If this value is omitted, the friction coefficient is assumed to be independent of the surface temperature.
5. Average value of the first field variable, $\overline{f}^1$.
6. Average value of the second field variable, $\overline{f}^2$.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Average value of the fifth field variable, $\overline{f}^5$.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the friction coefficient as a function of contact pressure, slip rate, average surface temperature, and other predefined field variables.

Data lines to define the coefficient of friction if the ANISOTROPIC BEHAVIOR=LEGACY parameter is used and the USER, ROUGH, and EXPONENTIAL DECAY parameters are omitted:

First line:

1. Friction coefficient in the first slip direction, μ_1.
2. Friction coefficient in the second slip direction, μ_2.
3. Slip rate, $\dot{\gamma}_{eq}$. If this value is omitted, the friction coefficient is assumed to be independent of the slip rate.
4. Contact pressure, p. If this value is omitted, the friction coefficient is assumed to be independent of the contact pressure.
5. Average temperature at the contact point, $\overline{\theta}$, between the two contact surfaces. If this value is omitted, the friction coefficient is assumed to be independent of the temperature.
6. Average value of the first field variable, $\overline{f}^1$.
7. Average value of the second field variable, $\overline{f}^2$.
8. Etc., up to three field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Average value of the fourth field variable, $\overline{f}^4$.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the friction coefficient as a function of contact pressure, slip rate, average surface temperature, and other predefined field variables.

Data lines to define the coefficient of friction if the ANISOTROPIC BEHAVIOR=SURFACE PROPERTY parameter is used and the USER, ROUGH, and EXPONENTIAL DECAY parameters are omitted:

First line:

1. Average (μ_{nom}), minimum ($\mu_{\min}$), or maximum ($\mu_{\max}$) friction coefficient, depending on the parameter used at the keyword level.
2. Slip rate, $\dot{\gamma}_{eq}$. If this value is omitted, the average friction coefficient is assumed to be independent of the slip rate.
3. Contact pressure, p. If this value is omitted, the average friction coefficient is assumed to be independent of the contact pressure.
4. Average temperature at the contact point, $\overline{\theta}$, between the two contact surfaces. If this value is omitted, the average friction coefficient is assumed to be independent of the temperature.
5. Average value of the first field variable, $\overline{f}^1$.
6. Average value of the second field variable, $\overline{f}^2$.
7. Etc., up to three field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Average value of the fourth field variable, $\overline{f}^4$.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the average friction coefficient as a function of contact pressure, slip rate, average surface temperature, and other predefined field variables.

Data line to define the static and kinetic friction coefficients if the EXPONENTIAL DECAY parameter is used and the decay coefficient is specified directly:

First (and only) line:

1. Static friction coefficient, μ_s.
2. Kinetic friction coefficient, μ_k.
3. Decay coefficient, d_c. The default value is zero.

Data lines if the EXPONENTIAL DECAY and TEST DATA parameters are used:

First line:

1. Friction coefficient for the first data point, μ_1. This value corresponds to the static friction coefficient.

Second line:

1. Friction coefficient for the second data point, μ_2. This value corresponds to the dynamic friction coefficient measured at the reference slip rate, $\dot{\gamma}_2$.
2. Slip rate of the second data point, $\dot{\gamma}_2$. This value corresponds to the reference slip rate used to measure the dynamic friction coefficient.

Third line (optional):

1. Kinetic friction coefficient, μ_∞. This value corresponds to the asymptotic value of the friction coefficient at infinite slip rate, $\dot{\gamma}_\infty$. If this data line is omitted, Abaqus/Standard automatically calculates μ_∞ such that $(\mu_2 - \mu_\infty) / (\mu_1 - \mu_\infty) = 0.05$.

There are no data lines when the ROUGH parameter is used

Data lines to define the user subroutine properties if the PROPERTIES parameter is used:

First line:

1. Enter the values of the friction properties, eight per line.

Repeat this data line as often as necessary to completely define all of the properties needed by user subroutines *FRIC*, *FRIC_COEF*, *VFRIC*, *VFRIC_COEF*, and *VFRICTION* as indicated by the value of PROPERTIES.

There are no data lines when the USER parameter is used without the PROPERTIES parameter

G

This section describes each Abaqus input option that begins with G, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **GAP*
- **GAP CONDUCTANCE*
- **GAP CONVECTION*
- **GAP DIFFUSIVITY*
- **GAP ELECTRICAL CONDUCTANCE*
- **GAP FLOW*
- **GAP HEAT GENERATION*
- **GAP RADIATION*
- **GASKET BEHAVIOR*
- **GASKET CONTACT AREA*
- **GASKET ELASTICITY*
- **GASKET SECTION*
- **GASKET THICKNESS BEHAVIOR*
- **GAS SPECIFIC HEAT*
- **GEL*
- **GEOSTATIC*
- **GLOBAL DAMPING*
- **GLOBAL RESPONSE*

*GAP: Specify clearance and local geometry for GAP-type elements.

This option is used to define the behavior of GAP-type elements.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Unsupported; similar functionality is available by modeling connectors.

References:

• *Gap Contact Elements*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the GAP-type contact elements for which properties are being defined.

Data line for GAPUNI and GAPUNIT elements:

First (and only) line:

1. Initial clearance, *d*.

If the remaining fields are omitted or specified as zero, the contact direction will be computed from the nodal coordinates.

5. Global *X*-direction cosine of the contact direction.
7. Global *Y*-direction cosine of the contact direction.
9. Global *Z*-direction cosine of the contact direction.
11. Element cross-sectional area.

Data line for GAPCYL elements:

First (and only) line:

1. Minimum/maximum separation distance, *d*.
2. Global *X*-direction cosine of the cylinder axis.
3. Global *Y*-direction cosine of the cylinder axis.
4. Global *Z*-direction cosine of the cylinder axis.
5. Element cross-sectional area.

Data line for GAPSPHER elements:

First (and only) line:

1. Minimum/maximum separation distance, *d*.
2. Enter a blank field.
3. Enter a blank field.
4. Enter a blank field.
5. Element cross-sectional area.

Data line for DGAP elements:

First (and only) line:

1. Clearance, *d*.
2. Enter a blank field.
3. Enter a blank field.
4. Enter a blank field.
5. Element cross-sectional area.

*GAP CONDUCTANCE: Introduce heat conductance between interface surfaces.

This option is used to provide conductive heat transfer between closely adjacent (or contacting) surfaces. It must be used in conjunction with the **SURFACE INTERACTION* option or in an Abaqus/Standard analysis with the **GAP* option.

For coupled temperature-displacement gasket elements, this option must be used in conjunction with:

- the **MATERIAL* option if the gasket behavior is defined by a material model, or
- the **GASKET BEHAVIOR* option if the gasket behavior is defined by a gasket behavior model (using **GASKET SECTION*).

For coupled temperature-displacement cohesive elements, this option must be used in conjunction with the **MATERIAL* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data in Abaqus/Standard; History data in Abaqus/Explicit

Level: Part, Part instance, Assembly, Model in Abaqus/Standard; Step in Abaqus/Explicit

Abaqus/CAE: Interaction module

References:

- *Thermal Contact Properties*
- *Defining a Thermal Interaction for a Gasket Element*
- **GAP*
- **INTERFACE*
- **SURFACE INTERACTION*
- **GASKET BEHAVIOR*
- **MATERIAL*
- **GASKET SECTION*
- *GAPCON*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables on which the gap conductance, k, depends.

PRESSURE

Include this parameter to indicate that k is a function of the contact pressure between the surfaces, p. Omit this parameter to define k as a function of the clearance, d, between the surfaces, or, for coupled temperature-displacement gasket elements, as a function of the closure, c, between the surfaces.

USER

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to define k in user subroutine *GAPCON*. Using this parameter will cause the DEPENDENCIES and PRESSURE parameters and any data lines to be ignored.

Data lines to define the gap conductance (k) directly:

First line:

1. Gap conductance or cohesive separation, k. (Units of $JT^{-1}L^{-2}\theta^{-1}$.)
2. Gap clearance, d; gap pressure, p; or closure, c (for coupled temperature-displacement gasket elements).
3. Average temperature, $\overline{\theta}$.
4. In an Abaqus/Standard analysis this data item corresponds to the average mass flow rate per unit area, $\overline{|\dot{m}|}$. In an Abaqus/Explicit analysis enter a blank field.
5. Average value of the first field variable, $\overline{f}_1$.
6. Average value of the second field variable, $\overline{f}_2$.
7. Etc.

Repeat this data line as often as necessary to define the dependence of gap conductance on gap clearance, gap pressure, closure, average surface temperature, average mass flow rate, and any predefined field variables. At least two data lines must be specified for each combination of average surface temperature, average mass flow rate, and any predefined field variables. When gap conductance is defined as a function of clearance or closure, the value of the conductance drops to zero when the clearance or closure is greater than the value corresponding to the data point.

There are no data lines when the USER parameter is used. Instead, define the gap conductance in user subroutine GAPCON

*GAP CONVECTION: Define Nusselt number for gap convection.

This option is used to provide the Nusselt number ($\mathbf{Nu}$) to calculate the convective coefficient for heat transfer between the gap flow and both the top and bottom surfaces of a coupled temperature-pore pressure cohesive element. It must be used in conjunction with the **MATERIAL* option.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance, Assembly

References:

• *Defining Thermal Convection for a Cohesive Element*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables on which the gap convection depends.

TYPE

Set TYPE=FLUX (default) to define a constant flux solution for convective heat transfer ($\mathbf{Nu = 8.235}$).

Set TYPE=TEMPERATURE to define a constant temperature solution for convective heat transfer ($\mathbf{Nu = 7.54}$).

Set TYPE=TABULAR to specify that the Nusselt number is dependent on temperature and/or field variables.

Data lines to define the temperature and/or field variable-dependent Nusselt number for TYPE=TABULAR:

First line:

1. Nusselt number.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the number of field variables is greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the Nusselt number as a function of temperature and other predefined field variables.

*GAP DIFFUSIVITY: Specify ion or species diffusion contact properties.

This option is used to modify ion or species diffusion in a surface interaction model. It must be used in conjunction with the **SURFACE INTERACTION* option.

If this option is omitted, the intended idealized behavior is no resistance to ionic or species flux flow (infinite diffusivity) across an interface while contact is active and for clearances up to a default tolerance distance. Abaqus/Standard approximates this behavior with a penalty method (large diffusivity) for general contact and direct enforcement for contact pairs.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- **SURFACE INTERACTION*
- *Electrical Contact Properties*

Required parameters:

TYPE

Set TYPE=ION CONCENTRATION to define the contact diffusivity, D_g, for ion concentration.

Set TYPE=SPECIES CONCENTRATION to define the contact diffusivity, D_g^s, for species concentration.

Optional parameters:

CUTOFF FLOW ACROSS

Set this parameter equal to a cutoff clearance distance above which no ion or species diffusion occurs across a contact interface.

CUTOFF GAP FILL

Set this parameter equal to a cutoff clearance distance above which no ion or species diffusion occurs into or out of a contact interface due to changes in the clearance distance.

DEPENDENCIES

Set this parameter equal to the number of field variables on which the contact diffusivity, D_g or D_g^s, depends.

Data lines to define the contact diffusivity, D_g, directly for TYPE=ION CONCENTRATION:

First line:

1. Contact diffusivity, D_g. (Units of $T^{-1}L$.)
2. Contact pressure, $p_{contact}$.
3. Average ion concentration, $\overline{C_e}$.
4. Average temperature, $\overline{\theta}$.

5. Average value of the first field variable, $\overline{f_1}$.
6. Average value of the second field variable, $\overline{f_2}$.
7. Etc.

Repeat this data line as often as necessary to define the dependence of contact diffusivity on the contact pressure, average surface ion concentration, average surface temperature, and the average of any predefined field variables on the surfaces.

Data lines to define the contact diffusivity, D_g^s, directly for TYPE=SPECIES CONCENTRATION:

First line:

1. Contact diffusivity, D_g^s. (Units of $T^{-1}L$.)
2. Contact pressure, $p_{contact}$.
3. Average species concentration, $\overline{C_S}$.
4. Average temperature, $\overline{\theta}$.
5. Average value of the first field variable, $\overline{f_1}$.
6. Average value of the second field variable, $\overline{f_2}$.
7. Etc.

Repeat this data line as often as necessary to define the dependence of contact diffusivity on the contact pressure, average surface ion concentration, average surface temperature, and the average of any predefined field variables on the surfaces.

*GAP ELECTRICAL CONDUCTANCE: Specify electrical conductance between surfaces.

This option is used to introduce gap electrical conductance in a surface interaction model in a coupled thermal-electrical or a coupled thermal-electrical-structural simulation. It must be used in conjunction with the *SURFACE INTERACTION option.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Fully Coupled Thermal-Stress Analysis*
- *Fully Coupled Thermal-Electrical-Structural Analysis*
- *Electrical Contact Properties*
- *SURFACE INTERACTION
- *GAPELECTR*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables on which σ_g depends.

PRESSURE

Include this parameter to indicate that σ_g is a function of the contact pressure between the surfaces, p. Omit this parameter to define σ_g as a function of the clearance, d, between the surfaces.

TYPE

Set TYPE=SOLID (default) to define the electrical conductivity in the solid material.

Set TYPE=LIQUID to define the electrical conductivity in the electrolyte material.

USER

Include this parameter to define σ_g in user subroutine *GAPELECTR*. In this case the DEPENDENCIES parameter and any data lines are ignored.

Data lines to define the gap electrical conductance directly for TYPE=SOLID:

First line:

1. Gap electrical conductivity, σ_g. (Units of $CT^{-1}L^{-2}\varphi^{-1}$.)
2. Surface separation, d, or surface pressure, p.
3. Average temperature, $\overline{\theta}$.
4. Average value of the first field variable, $\overline{f_1}$.
5. Average value of the second field variable, $\overline{f_2}$.
6. Etc.

Repeat this data line as often as necessary to define the dependence of gap electrical conductance on the surface separation, average surface temperature, and the average of any predefined field variables on the surfaces.

Data lines to define the gap electrical conductance directly for TYPE=LIQUID:

First line:

1. Gap electrical conductivity, σ_g. (Units of $CT^{-1}L^{-2}\varphi^{-1}$.)
2. Surface separation, d, or surface pressure, p.
3. Average temperature, $\overline{\theta}$.
4. Average ion concentration, $\overline{C_e}$.
5. Average value of the first field variable, $\overline{f_1}$.
6. Average value of the second field variable, $\overline{f_2}$.
7. Etc.

Repeat this data line as often as necessary to define the dependence of gap electrical conductance on the surface separation, average surface temperature, average ion concentration, and the average of any predefined field variables on the surfaces.

*GAP FLOW: Define constitutive parameters for tangential flow in pore pressure cohesive elements and enriched elements.

This option is used to define tangential flow constitutive parameters for pore pressure cohesive elements and enriched elements. It is usually used in a model with hydraulically driven fracture.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Defining the Constitutive Response of Fluid within the Cohesive Element Gap*
- *Defining the Constitutive Response of Fluid Flow within the Cracked Element Surfaces*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the constitutive parameters, in addition to temperature. If this parameter is omitted, it is assumed that the constitutive parameters are constant or depend only on temperature. See *Material Data Definition* for more information.

FLUID ID

Set this parameter equal to the fluid identifier when multi-fluid tangential flow is specified.

KMAX

Set this parameter equal to the maximum permeability value that should be used. This parameter is meaningful only when TYPE=NEWTONIAN. If this parameter is omitted, Abaqus assumes that the permeability is not bounded.

PTAU

Set this parameter equal to a scalar value P_τ used to compute the still fluid viscosity before yielding as $\mu = P_\tau \times K$. The default value of P_τ is 10^7.

This parameter can be used only with TYPE=BINGHAM PLASTIC and TYPE=HERSCHEL-BULKLEY.

SETTLING

This parameter specifies the formulation used to define the proppant settling behavior during slurry transport. It can be used only in conjunction with the SLURRY parameter.

If this parameter is omitted or used without a value (default), a built-in settling mechanism is applied.

Set SETTLING=NONE to neglect settling effects.

Set SETTLING=USER to specify a user-defined settling velocity field through user subroutine *USETTLING*.

SLURRY

Include this parameter to define tangential flow constitutive properties that depend on slurry concentration.

TYPE

Set TYPE=NEWTONIAN (default) to define the viscosity for a Newtonian fluid.

Set TYPE=POWER LAW to define the consistency and exponent for a power law fluid.

Set TYPE=BINGHAM PLASTIC to define the consistency and the yield stress for a Bingham plastic fluid.

Set TYPE=HERSCHEL-BULKLEY to define the consistency, exponent, and yield stress for a Herschel-Bulkley fluid.

Data lines to define the pore fluid viscosity (μ) (TYPE=NEWTONIAN):

First line:

1. μ.
2. Temperature, θ.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the variation.

Data lines to define the consistency, K, and exponent, α (TYPE=POWER LAW):

First line:

1. K.
2. α.
3. Temperature, θ.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the variation.

Data lines to define the consistency, K, and the yield stress, τ_0 (TYPE=BINGHAM PLASTIC):

First line:

1. K.
2. τ_0.
3. Temperature, θ.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the variation.

Data lines to define the consistency, K; exponent, α; and yield stress, τ_0 (TYPE=HERSCHEL-BULKLEY):

First line:

1. K.
2. α.
3. τ_0.
4. Temperature, θ.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the variation.

Data lines to define the pore fluid viscosity (μ) (TYPE=NEWTONIAN with the SLURRY parameter):

First line:

1. μ.
2. Slurry concentration, C_v.
3. Temperature, θ.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the variation.

Data lines to define the consistency, K, and exponent, α (TYPE=POWER LAW with the SLURRY parameter):

First line:

1. K.
2. α.
3. Slurry concentration, C_v.
4. Temperature, θ.
5. First field variable.
6. Second field variable.

7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the variation.

Data lines to define the consistency, K, and the yield stress, τ_0 (TYPE=BINGHAM PLASTIC with the SLURRY parameter):

First line:

1. K.
2. τ_0.
3. Slurry concentration, C_v.
4. Temperature, θ.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the variation.

Data lines to define the consistency, K; exponent, α; and yield stress, τ_0 (TYPE=HERSCHEL-BULKLEY with the SLURRY parameter):

First line:

1. K.
2. α.
3. τ_0.
4. Slurry concentration, C_v.
5. Temperature, θ.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the variation.

*GAP HEAT GENERATION: Introduce heat generation due to energy dissipation at the interface.

This option is used to modify the default gap heat generation model used to dissipate energy created by nonthermal surface interactions, such as frictional sliding or electric currents. The default is to convert all of the dissipated energy to heat and to distribute it evenly between the two interacting surfaces.

The **GAP HEAT GENERATION* option must be used in conjunction with the **SURFACE INTERACTION* option or in an Abaqus/Standard analysis with the **GAP* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data in Abaqus/Standard; History data in Abaqus/Explicit

Level: Part, Part instance, Assembly, Model in Abaqus/Standard; Step in Abaqus/Explicit

Abaqus/CAE: Interaction module

References:

- *Frictional Behavior*
- *Thermal Contact Properties*
- *Electrical Contact Properties*
- **GAP*
- **INTERFACE*
- **SURFACE INTERACTION*

There are no parameters associated with this option.

Data line to define the gap heat generation in Abaqus/Standard:

First (and only) line:

1. η, fraction of dissipated energy converted into heat generated by mechanical interaction (η_M), electrical conduction in solids (η_{ES}), and electrical conduction in liquids (η_{EL}). If η_M, η_{ES}, or η_{EL} is omitted, it is replaced by η. The default value is 1.0.
2. f, weighting factor for the distribution of heat between the interacting surfaces generated by mechanical interaction (f_M), electrical conduction in solids (f_{ES}), and electrical conduction in liquids (f_{EL}). If f_M, f_{ES}, or f_{EL} is omitted, it is replaced by f. The heat flux into the secondary surface is weighted by f, and the heat flux into the main surface is weighted by $1 - f$. By default, the heat is distributed equally between the two surfaces, $f = 0.5$.
3. η_M, fraction of dissipated energy converted into heat generated by mechanical interaction.
4. f_M, weighting factor for the distribution of heat between the interacting surfaces generated by mechanical interaction.
5. η_{ES}, fraction of dissipated energy converted into heat generated by electrical conduction in the solids.
6. f_{ES}, weighting factor for the distribution of heat between the interacting surfaces generated by electrical conduction in the solids.

7. η_{EL}, fraction of dissipated energy converted into heat generated by electrical conduction in the liquids.
8. f_{EL}, weighting factor for the distribution of heat between the interacting surfaces generated by electrical conduction in the liquids.

Data line to define the gap heat generation in Abaqus/Explicit:

First (and only) line:

1. η, fraction of dissipated energy converted into heat including any unit conversion factor. The default value is 1.0.
2. f, weighting factor for the distribution of heat between the interacting surfaces. The heat flux into the secondary surface is weighted by f, and the heat flux into the main surface is weighted by $1 - f$. By default, the heat is distributed equally between the two surfaces, $f = 0.5$.

*GAP RADIATION: Introduce heat radiation between surfaces.

This option is used to provide radiative heat transfer between closely adjacent surfaces. It must be used in conjunction with the *SURFACE INTERACTION option or in an Abaqus/Standard analysis with the *GAP option.

For coupled temperature-displacement cohesive elements, this option must be used in conjunction with the *MATERIAL option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data in Abaqus/Standard; History data in Abaqus/Explicit

Level: Part, Part instance, Assembly, Model in Abaqus/Standard; Step in Abaqus/Explicit

Abaqus/CAE: Interaction module

References:

- *Thermal Contact Properties*
- *GAP
- *INTERFACE
- *SURFACE INTERACTION

There are no parameters associated with this option.

Data lines to define surface constants for radiative heat transfer:

First line:

1. Emissivity, ϵ_A.
2. Emissivity, ϵ_B.

Second line:

1. Effective view factor, F. $(0 \le F \le 1)$
2. Gap clearance, d.

Repeat this data line as often as necessary to define the dependence of the view factor on gap clearance.

*GASKET BEHAVIOR: Begin the specification of a gasket behavior.

This option is used to indicate the start of a gasket behavior definition.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Defining the Gasket Behavior Directly Using a Gasket Behavior Model*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the behavior when it is referenced in the **GASKET SECTION* option. Gasket behavior names in the same input file must be unique.

There are no data lines associated with this option.

*GASKET CONTACT AREA: Specify a gasket contact area or contact width for average pressure output.

This option is used to define contact area or contact width versus closure curves to output an average pressure through variable CS11. It can be used only with gasket link and three-dimensional line gasket elements that have their thickness-direction behavior defined in terms of force or force per unit length.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Defining the Gasket Behavior Directly Using a Gasket Behavior Model*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the data, in addition to temperature. If this parameter is omitted, it is assumed that the data depend only on temperature. See *Material Data Definition* for more information.

Data lines to define a contact area for average pressure output:

First line:

1. Contact area or width. (This value cannot be negative.)
2. Closure. (This value cannot be negative.)
3. Temperature, θ.
4. First field variable.
5. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the contact area or width versus closure curves on temperature and field variables.

*GASKET ELASTICITY: Specify elastic properties for the membrane and transverse shear behaviors of a gasket.

This option is used to define the elastic parameters for the membrane and transverse shear behaviors of a gasket.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Defining the Gasket Behavior Directly Using a Gasket Behavior Model*

Optional parameters:

COMPONENT

Set COMPONENT=MEMBRANE to define the membrane behavior of a gasket.

Set COMPONENT=TRANSVERSE SHEAR (default) to define the transverse shear behavior of a gasket.

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the elastic parameters, in addition to temperature. If this parameter is omitted, it is assumed that the elastic parameters depend only on temperature. See *Material Data Definition* for more information.

VARIABLE

This parameter is used only with COMPONENT=TRANSVERSE SHEAR to specify the unit system in which the transverse shear behavior will be defined.

Set VARIABLE=FORCE to define the transverse shear stiffness in terms of force per unit displacement or force per unit length per unit displacement, depending on the element type to which this behavior refers.

Set VARIABLE=STRESS (default) to define the transverse shear stiffness in terms of stress per unit displacement.

Data lines for COMPONENT=TRANSVERSE SHEAR:

First line:

1. Shear stiffness. (This value cannot be negative.)
2. Temperature, θ.
3. First field variable.
4. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the shear stiffness on temperature and field variables.

Data lines for COMPONENT=MEMBRANE:

First line:

1. Young's modulus, E.
2. Poisson's ratio, ν.
3. Temperature, θ.
4. First field variable.
5. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of Young's modulus and Poisson's ratio on temperature and field variables.

*GASKET SECTION: Specify element properties for gasket elements.

This option is used to define the properties of gasket elements.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- *About Gasket Elements*
- *Defining the Gasket Behavior Using a Material Model*
- *Defining the Gasket Behavior Directly Using a Gasket Behavior Model*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the elements for which the gasket behavior is being defined.

Required, mutually exclusive parameters:

BEHAVIOR

Set this parameter equal to the name of the gasket behavior to which the specified element set refers.

MATERIAL

Set this parameter equal to the name of the material of which the gasket is made.

Optional parameters:

ORIENTATION

Set this parameter equal to the name given for the **ORIENTATION* option (*Orientations*) to be used to define a local coordinate system for integration point calculations in the gasket elements in the specified element set.

STABILIZATION STIFFNESS

This parameter is usually not needed. It is used to change the default stabilization stiffness used in all but link elements to stabilize gasket elements that are not supported at all nodes, such as those that extend outside neighboring components. The default value is set equal to 10^{-9} times the initial compressive stiffness in the thickness direction. To change the default, set this parameter equal to the desired stabilization stiffness. The units are stress (FL^{-2}). In case the thickness behavior is defined in terms of force versus closure, the initial compressive stiffness needs to be divided by the cross-sectional area to convert the units to stress before it can be used to set the desired stabilization stiffness.

Data line to define the attributes of gasket elements:

First (and only) line:

1. Initial gasket thickness (obtained from nodal coordinates if this field is blank or zero).
2. Initial gap (default of 0).

3. Initial void (default of 0).
4. Cross-sectional area, width, or out-of-plane thickness, depending on the gasket element type. The default is 1.0. The value is ignored for gasket elements that do not require this input.
5. First component of the thickness direction of the elements.
6. Second component of the thickness direction of the elements.
7. Third component of the thickness direction of the elements.

*GASKET THICKNESS BEHAVIOR: Specify a gasket thickness-direction behavior.

This option is used to define the behavior in the thickness direction for a gasket.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Defining the Gasket Behavior Directly Using a Gasket Behavior Model*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the data, in addition to temperature. If this parameter is omitted, it is assumed that the data depend only on temperature. See *Material Data Definition* for more information.

DIRECTION

Set DIRECTION=LOADING (default) to prescribe the loading curve of the model used to define the gasket thickness-direction behavior.

Set DIRECTION=UNLOADING to prescribe the unloading curves of the model used to define the gasket thickness-direction behavior.

INTERPOLATION

This parameter is relevant only when used in conjunction with TYPE=DAMAGE and DIRECTION=UNLOADING.

Set INTERPOLATION=NORMALIZED (default) to choose the interpolation scheme based on the normalized curves.

Set INTERPOLATION=DIRECT to choose the interpolation scheme based on the specified curves.

TENSILE STIFFNESS FACTOR

Set this parameter equal to the fraction of the initial compressive stiffness that defines the stiffness in tension. The default value is 10^{-3}. This parameter can be used only with DIRECTION=LOADING.

TYPE

Set TYPE=DAMAGE to define a damage elasticity model for the gasket thickness-direction behavior.

Set TYPE=ELASTIC-PLASTIC (default) to define an elastic-plastic model for the gasket thickness-direction behavior.

VARIABLE

Set VARIABLE=FORCE to define the behavior in terms of force versus closure or force per unit length versus closure, depending on the element type with which this behavior is being used.

Set VARIABLE=STRESS (default) to define the behavior in terms of pressure versus closure.

The following parameters are optional, mutually exclusive, and can be used only with DIRECTION=LOADING:

SLOPE DROP

Set this parameter equal to the relative drop in slope on the loading curve that defines the onset of plastic deformation. The default value is 0.1.

YIELD ONSET

Set this parameter equal to the closure value at which the onset of yield occurs. The specified value must correspond to a point on the loading curve at which the slope decreases.

Data lines to define the loading in terms of pressure versus closure (DIRECTION=LOADING and VARIABLE=STRESS):

First line:

1. Pressure. (This value cannot be negative.)
2. Closure. (Units of L.) (This value cannot be negative.)
3. Temperature, θ.
4. First field variable.
5. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the loading curve on temperature and field variables.

Data lines to define the loading in terms of force or force per unit length per closure (DIRECTION=LOADING and VARIABLE=FORCE):

First line:

1. Force or force per unit length. (This value cannot be negative.)
2. Closure. (Units of L.) (This value cannot be negative.)
3. Temperature, θ.
4. First field variable.
5. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the loading curve on temperature and field variables.

Data lines to define the unloading in terms of pressure versus closure for an elastic-plastic model (TYPE=ELASTIC-PLASTIC, DIRECTION=UNLOADING, and VARIABLE=STRESS):

First line:

1. Pressure. (This value cannot be negative.)
2. Closure. (Units of L.) (This value must be positive.)
3. Plastic closure. (Units of L.) (This value must be positive.)
4. Temperature, θ.
5. First field variable.

6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the unloading curve of the elastic-plastic model on temperature and field variables.

Data lines to define the unloading in terms of force or force per unit length versus closure for an elastic-plastic model (TYPE=ELASTIC-PLASTIC, DIRECTION=UNLOADING, and VARIABLE=FORCE):

First line:

1. Force or force per unit length. (This value cannot be negative.)
2. Closure. (Units of L.) (This value must be positive.)
3. Plastic closure. (Units of L.) (This value must be positive.)
4. Temperature, θ.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the unloading curve of the elastic-plastic model on temperature and field variables.

Data lines to define the unloading in terms of pressure versus closure for a damage model (TYPE=DAMAGE, DIRECTION=UNLOADING, and VARIABLE=STRESS):

First line:

1. Pressure. (This value cannot be negative.)
2. Closure. (Units of L.) (This value cannot be negative.)
3. Maximum closure reached while loading the gasket. (Units of L.) (This value must be positive.)
4. Temperature, θ.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the unloading curve of the damage model on temperature and field variables.

Data lines to define the unloading in terms of force or force per unit length versus closure for a damage model (TYPE=DAMAGE, DIRECTION=UNLOADING, and VARIABLE=FORCE):

First line:

1. Force or force per unit length. (This value cannot be negative.)
2. Closure. (Units of L.) (This value cannot be negative.)
3. Maximum closure reached while loading. (Units of L.) (This value must be positive.)

4. Temperature, θ.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the unloading curve of the damage model on temperature and field variables.

*GAS SPECIFIC HEAT: Define reacted product's specific heat for an ignition and growth equation of state.

This option is used to specify the specific heat of reacted gas products for an ignition and growth equation of state. It is required when the *EOS, TYPE=IGNITION AND GROWTH option is used. The *GAS SPECIFIC HEAT option should appear immediately after the *EOS or the *REACTION RATE option.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Equation of State*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the reacted product's specific heat. If this parameter is omitted, it is assumed that the reacted product's specific heat is constant or depends only on temperature.

Data lines to specify the reacted product's specific heat:

First line:

1. Specific heat per unit mass. (Units of $JM^{-1}\theta^{-1}$.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the specific heat as a function of temperature and other predefined field variables.

*GEL: Define a swelling gel.

This option is used to define the growth of the gel particles that swell and trap wetting liquid in a partially saturated porous medium in the analysis of coupled wetting liquid flow and porous medium stress.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Swelling Gel*

There are no parameters associated with this option.

Data line to define a swelling gel:

First (and only) line:

1. Radius of gel particles when completely dry, r_a^{dry}. (Units of L.)
2. Fully swollen radius of gel particles, r_a^f. (Units of L.)
3. Number of gel particles per unit volume, k_a. (Units of L^{-3}.)
4. Relaxation time constant for long-term swelling of gel particles, τ_1. (Units of T.)

*GEOSTATIC: Obtain a geostatic stress field.

This option is used to verify that the geostatic stress field is in equilibrium with the applied loads and boundary conditions on the model and to iterate, if needed, to obtain equilibrium.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

• *Geostatic Stress State*

Optional parameters:

HEAT

This parameter is relevant if there are regions in the model that use coupled temperature–pore pressure elements; it specifies whether heat transfer effects are to be modeled in these regions.

Set HEAT=YES (default) to specify that heat transfer effects are to be modeled in these regions. In this case Abaqus/Standard solves the heat transfer equation in conjunction with the mechanical equilibrium and the fluid flow continuity equations.

Set HEAT=NO to specify that heat transfer will not be modeled in these regions.

This parameter is not relevant if only coupled pore pressure–displacement elements are used in a model.

SLURRY FLOW

Set SLURRY FLOW=YES (default) to activate the solution of the slurry continuity equation in a step. In this case, the slurry concentration field will evolve with time within the step.

Set SLURRY FLOW=NO to deactivate the solution of the slurry continuity equation in a step. In this case, the slurry concentration field will not evolve with time within the step.

This parameter is relevant only when the slurry concentration degree of freedom is active.

SLURRY MAX

Set this parameter equal to the maximum allowed value of the slurry concentration at a material point. The default is 0.67.

This parameter is relevant only when the slurry concentration degree of freedom is active.

SLURRY MIN

Set this parameter equal to the minimum allowed value of the slurry concentration at a material point. The default is zero.

This parameter is relevant only when the slurry concentration degree of freedom is active.

UTOL

This parameter will invoke automatic time incrementation.

Set this parameter equal to the tolerance for the maximum change of displacements. Abaqus/Standard will ensure that the maximum absolute value of a displacement at a node is smaller than the tolerance times the characteristic element length in the model. If this parameter is used without any value

specified, the default value of 10^{-5} is used. If this parameter is omitted, no restrictions are imposed on the displacement values.

Data line to define automatic time incrementation:

First (and only) line:

1. Initial time increment. This value will be modified as required. If this entry is zero or is not specified, a default value that is equal to the total time period of the step is assumed.
2. Time period of the step. If this entry is zero or is not specified, a default value of 1.0 is assumed.
3. Minimum time increment allowed. If Abaqus/Standard finds it needs a smaller time increment than this value, the analysis is terminated. If this entry is zero, a default value of the smaller of the suggested initial time increment or 10^{-5} times the total time period is assumed.
4. Maximum time increment allowed. If this value is zero or is not specified, no upper limit is imposed.

*GLOBAL DAMPING: Specify global damping.

This option is used to provide global damping factors for the following procedures in Abaqus/Standard:

- **COMPLEX FREQUENCY*
- **MODAL DYNAMIC*
- **RANDOM RESPONSE*
- **RESPONSE SPECTRUM*
- **STEADY STATE DYNAMICS*
- **STEADY STATE DYNAMICS*, DIRECT
- **STEADY STATE DYNAMICS*, SUBSPACE PROJECTION
- **MATRIX GENERATE*
- **SUBSTRUCTURE GENERATE*

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Supported in the *Step module* only for substructure generation.

References:

- *Material Damping*
- *Damping in Dynamic Analysis*
- *Coupled Acoustic-Structural Analysis*

Optional parameters:

ALPHA

Set this parameter equal to the α_{global} factor to create global Rayleigh mass proportional damping $D_{\alpha} = \alpha_{global} * M$, where M denotes the model mass matrix. The default is ALPHA=0. (Units of T^{-1}.)

BETA

Set this parameter equal to the β_{global} factor to create global Rayleigh stiffness proportional damping $D_{\beta} = \beta_{global} * K$, where K denotes the model stiffness matrix. The default is BETA=0. (Units of T.)

FIELD

Set FIELD=ACOUSTIC to apply the global damping only to the acoustic fields in the model.

Set FIELD=ALL (default) to apply the global damping to all of the valid displacement, rotation, and acoustic fields in the model.

Set FIELD=MECHANICAL to apply the global damping only to the valid displacement and rotation fields in the model.

This parameter is not supported in a mode-based steady-state dynamic analysis that uses coupled acoustic-structural modes.

STRUCTURAL

Set this parameter equal to the s_{global} factor to create frequency-independent stiffness proportional structural damping $D_s = is_{global} * K$, where K denotes the model stiffness matrix. The default is STRUCTURAL=0.

There are no data lines associated with this option.

*GLOBAL RESPONSE: Specify global design response.

This option is used in an adjoint sensitivity analysis to write a global design response and its corresponding sensitivities with respect to the appropriate design variables to the output database. It must be used in conjunction with the **DESIGN RESPONSE* option.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

• *Adjoint Design Sensitivity Analysis*

Optional parameters:

LOWER BOUND

This parameter is used to specify a lower bound on the design response value below which adjoint sensitivities are not calculated. That is, if the value of the design response value is below the specified lower bound, then only the design response value is written to the output database, but not the corresponding adjoint sensitivities.

This parameter is not supported in an eigenfrequency sensitivity analysis.

UPPER BOUND

This parameter is used to specify an upper bound on the design response value above which adjoint sensitivities are not calculated. That is, if the value of the design response value is above the specified upper bound, then only the design response value is written to the output database, but not the corresponding adjoint sensitivities.

This parameter is not supported in an eigenfrequency sensitivity analysis.

Data lines:

First line:

1. Specify an identifying key for the global response whose sensitivities are to be written to the output database. The valid keys are listed in *Adjoint Design Sensitivity Analysis*. Only one identifying key can be specified for each use of this option.

Second line:

1. Mode number for eigenfrequency sensitivity.

H

This section describes each Abaqus input option that begins with H, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **HEADING*
- **HEATCAP*
- **HEAT GENERATION*
- **HEAT TRANSFER*
- **HOURGLASS STIFFNESS*
- **HYPERELASTIC*
- **HYPERFOAM*
- **HYPOELASTIC*
- **HYSTERESIS*

*HEADING: Print a heading on the output.

This option is used to define a title for the analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Job module

References:

• *Abaqus Model Definition*

There are no parameters associated with this option.

Data lines to print a heading:

First line:

1. The heading.

The heading can be several lines long, but only the first 80 characters of the first line will be saved and printed as a heading.

*HEATCAP: Specify a point capacitance.

This option is used to define lumped heat capacitance values associated with HEATCAP elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: *Property module* and *Interaction module*.

References:

• *Point Capacitance*

Required parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the point capacitance. If this parameter is omitted, it is assumed that the point capacitance is constant or depends only on temperature. See *Material Data Definition* for more information.

ELSET

Set this parameter equal to the name of the element set containing the HEATCAP elements for which the value is being given.

Data lines to define the capacitance magnitude:

First line:

1. Capacitance magnitude. Capacitance (ρcV), not specific heat, should be given.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the point capacitance as a function of temperature and other predefined field variables. Abaqus does not use any specific physical units, so the user's choice must be consistent.

*HEAT GENERATION: Include volumetric heat generation in heat transfer analyses.

This option is used in a material data block to include user-defined heat generation in heat transfer, coupled thermal-electrical, coupled temperature-displacement, or coupled thermal-electrical-structural analyses. It must be used in conjunction with user subroutine *HETVAL* (Abaqus/Standard) or *VHETVAL* (Abaqus/Explicit).

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Uncoupled Heat Transfer Analysis*
- *Fully Coupled Thermal-Stress Analysis*
- *Fully Coupled Thermal-Electrical-Structural Analysis*
- *HETVAL*
- *VHETVAL*

Optional parameters:

PROPERTIES

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the number of properties being entered. The properties are available for use in user subroutine *VHETVAL*. The default is PROPERTIES=0.

There are no parameters or data lines associated with this option in Abaqus/Standard analyses:

Properties and parameters can be either defined directly in *HETVAL* or, more generally, defined in an input file and accessed from *HETVAL* (see *Accessing Abaqus Table Collections* for more information).

Data lines to define properties in Abaqus/Explicit analyses:

No data lines are needed if the PROPERTIES parameter is omitted or set to 0. Otherwise, first line:

1. Give the properties, eight per line.

Repeat this data line as often as necessary to define all properties.

*HEAT TRANSFER: Transient or steady-state uncoupled heat transfer analysis.

This option is used to control uncoupled heat transfer for either transient or steady-state response.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Uncoupled Heat Transfer Analysis*
- *Cavity Radiation in Abaqus/Standard*

Optional parameters:

DELTMX

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the maximum temperature change to be allowed in an increment during a transient heat transfer analysis. Abaqus/Standard will restrict the time step to ensure that this value will not be exceeded at any node (except nodes whose temperature degree of freedom is constrained via boundary conditions, MPC's, etc.) during any increment of the step. If the DELTMX parameter is omitted, fixed time increments will be used.

END

This parameter applies only to Abaqus/Standard analyses.

Set END=PERIOD (default) to analyze a specific time period. Set END=SS to end the analysis when steady state is reached.

This parameter is relevant only for transient analysis.

STEADY STATE

Include this parameter to choose steady-state analysis. Transient analysis is assumed if this parameter is omitted.

Optional parameter for cavity radiation analysis:

MXDEM

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the maximum allowable emissivity change with temperature and field variables during an increment. If this value is exceeded, Abaqus/Standard will cut back the increment until the maximum change in emissivity is less than the specified value. If this parameter is omitted, a default value of 0.1 is used.

Data line to control incrementation and steady-state conditions in a pure heat transfer analysis:

First (and only) line:

1. Initial time increment. If automatic incrementation is used, this should be a reasonable suggestion for the initial increment size and will be adjusted as necessary. If direct incrementation is used, this will be the fixed time increment size.
2. Total time period. If END=SS is chosen, the step ends when steady state is reached or after this time period, whichever occurs first.
3. Minimum time increment allowed. If Abaqus/Standard finds it needs a smaller time increment than this value, the analysis is terminated. If a value is given, Abaqus/Standard will use the minimum of the given value and 0.8 times the initial time increment. If no value is given, Abaqus/Standard sets the minimum increment to the minimum of 0.8 times the initial time increment (first data item on this data line) and 1×10^{-5} times the total time period (second data item on this data line). This value is used only for automatic time incrementation.
4. Maximum time increment allowed. If this value is not specified, no upper limit is imposed. This value is used only for automatic time incrementation.
5. Temperature change rate (temperature per time) used to define steady state; only needed if END=SS is chosen. When all nodal temperatures are changing at less than this rate, the solution terminates.

*HOURGLASS STIFFNESS: Specify nondefault hourglass stiffness.

This option is relevant for first-order, reduced-integration elements; second-order, reduced-integration element types M3D9R, S8R5, and S9R5; and modified tetrahedral and triangular elements. It can also be used to define an hourglass scaling factor for the stiffness associated with the drill degree of freedom (rotation about the surface normal) in shell elements and to modify the hourglass stiffness factor for the pressure Lagrange multiplier degrees of freedom for C3D4H elements.

The **HOURGLASS STIFFNESS* option can be used only in conjunction with the **MEMBRANE SECTION* option, the **SOLID SECTION* option, the **SHELL SECTION* option, or the **SHELL GENERAL SECTION* option. The hourglass control defined with this option affects only those elements whose section properties are defined by the immediately preceding section option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Mesh module

References:

- *Section Controls*
- **MEMBRANE SECTION*
- **SHELL GENERAL SECTION*
- **SHELL SECTION*
- **SOLID SECTION*

There are no parameters associated with this option.

Data line to define a nondefault hourglass stiffness:

First (and only) line:

1. Hourglass control stiffness parameter $(r_F G)$ for use with membrane and solid elements and for membrane hourglass mode control in shells. Units are stress (FL^{-2}). If this value is left blank or entered as zero, Abaqus/Standard will use the default value.
2. Hourglass control stiffness parameter $(r_F K)$ for use with element type C3D4H. Units of this parameter depend on the material property assigned to the element. For nearly incompressible elastomers (**HYPERELASTIC*) and elastometric foams (**HYPERFOAM*) the units are stress (FL^{-2}); for all other remaining materials, including fully incompressible elastomers, the units are stress compliance ($F^{-1}L^{2}$). If this value is left blank or entered as zero, Abaqus/Standard will use the default value.
3. Hourglass control stiffness parameter ($r_\theta G$) for bending hourglass mode control in shells. Units are stress (FL^{-2}). If this value is left blank or entered as zero, Abaqus/Standard will use the default value.
4. Factor by which the default stiffness for rotation about the shell surface normal is to be scaled (for shell nodes where six degrees of freedom are active). If this value is not entered or is entered as zero, Abaqus/Standard will use the default value.

*HYPERELASTIC: Specify elastic properties for approximately incompressible elastomers.

This option is used to define material constants for a general hyperelastic material.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Hyperelastic Behavior of Rubberlike Materials*
- *UHYPER*
- *UHYPER_STRETCH*
- **BIAXIAL TEST DATA*
- **PLANAR TEST DATA*
- **UNIAXIAL TEST DATA*
- **VOLUMETRIC TEST DATA*
- *VUHYPER*

Optional, mutually exclusive parameters:

ARRUDA-BOYCE

Include this parameter to use the Arruda-Boyce model, also known as the eight-chain model.

HENCKY

Include this parameter to use the Hencky model.

MARLOW

Include this parameter to use the Marlow model.

MOONEY-RIVLIN

Include this parameter to use the Mooney-Rivlin model. This method is equivalent to using the POLYNOMIAL parameter with N=1.

NEO HOOKE

Include this parameter to use the neo-Hookean model. This method is equivalent to using the REDUCED POLYNOMIAL parameter with N=1.

OGDEN

Include this parameter to use the Ogden strain energy potential.

POLYNOMIAL

Include this parameter to use the polynomial strain energy potential. This method is the default method of defining the strain energy potential.

REDUCED POLYNOMIAL

Include this parameter to use the reduced polynomial strain energy potential. This method is equivalent to using the POLYNOMIAL parameter with $C_{ij} = 0$ for $j \neq 0$.

USER

Include this parameter if the derivatives of the strain energy potential with respect to the strain invariants are defined in user subroutine *UHYPER* (Abaqus/Standard) or *VUHYPER* (Abaqus/Explicit), or if the derivatives of the strain energy potential with respect to the principal stretches are defined in user subroutine *UHYPER_STRETCH* in Abaqus/Standard.

VALANIS-LANDEL

Include this parameter to use the Valanis-Landel model.

VAN DER WAALS

Include this parameter to use the Van der Waals model, also known as the Kilian model.

YEOH

Include this parameter to use the Yeoh model. This method is equivalent to using the REDUCED POLYNOMIAL parameter with N=3.

Required parameters if the USER parameter is included:

FORMULATION

This parameter applies only to Abaqus/Standard analyses.

Set FORMULATION=INVARIANT (default) to indicate that the hyperelastic energy potential is formulated in terms of invariants and is defined in user subroutine *UHYPER*.

Set FORMULATION=STRETCH to indicate that the hyperelastic energy potential is formulated in terms of the principal stretches and is defined in user subroutine *UHYPER_STRETCH*.

TYPE

This parameter applies only to Abaqus/Standard analyses.

Set TYPE=INCOMPRESSIBLE to indicate that the hyperelastic material defined by *UHYPER* or by *UHYPER_STRETCH* is incompressible.

Set TYPE=COMPRESSIBLE to indicate that the hyperelastic material defined by *UHYPER* or by *UHYPER_STRETCH* is compressible.

Optional parameters:

BETA

This parameter can be used only when both the VAN DER WAALS and TEST DATA INPUT parameters are used; it defines the value of β while the other coefficients of the Van der Waals model are fitted from the test data given by the user. If this parameter is omitted, β will be determined from a nonlinear, least-squares fit of the test data. Allowable values of BETA are $0 \leq \beta \leq 1$. It is recommended to set $\beta = 0$ if only one type of test data is available.

MODULI

This parameter is applicable only when the **HYPERELASTIC* option is used in conjunction with the **VISCOELASTIC* or the **HYSTERESIS* option.

Set MODULI=INSTANTANEOUS to indicate that the hyperelastic material constants define the instantaneous behavior. This is the only option available if you define the hyperelastic material in user subroutine *UHYPER*, user subroutine *VUHYPER*, or *UHYPER_STRETCH*.

Set MODULI=LONG TERM to indicate that the hyperelastic material constants define the long-term behavior. This option is not available when you define the hyperelastic material in user subroutine

UHYPER, user subroutine *VUHYPER* or user subroutine *UHYPER_STRETCH*. It is the default for all other hyperelastic models.

N

This parameter can be used only with the OGDEN, POLYNOMIAL, and REDUCED POLYNOMIAL parameters. Include this parameter to define the order of the strain energy potential. The default is N=1.

If the TEST DATA INPUT parameter is used, the parameter N can take only the values 1 or 2 for the POLYNOMIAL form and up to 6 for the OGDEN and REDUCED POLYNOMIAL forms.

If the TEST DATA INPUT parameter is omitted, the maximum value of N is 6 for either form.

POISSON

Set this parameter equal to the Poisson's ratio, ν, to account for compressibility. This parameter cannot be used if the material coefficients are specified directly or if volumetric behavior is defined by entering nonzero values for D_i on the data line or by specifying the **VOLUMETRIC TEST DATA* option. In addition, this parameter cannot be used for the Marlow model or for the Valanis-Landel model if the nominal lateral strains are specified on the **UNIAXIAL TEST DATA*, **BIAXIAL TEST DATA*, or **PLANAR TEST DATA* option.

PROPERTIES

This parameter can be used only if the USER parameter is specified. Set this parameter equal to the number of property values needed as data in user subroutine *UHYPER*, in user subroutine *VUHYPER*, or in user subroutine *UHYPER_STRETCH*. The default value is 0.

TEST DATA INPUT

Include this parameter if the material constants are to be computed by Abaqus from data taken from simple tests on a material specimen.

If this parameter is omitted, the material constants must be given directly on the data lines. This parameter cannot be used with the Hencky model. It is not relevant for the Marlow model or the Valanis-Landel model; you must specify the test data for these models.

To define the material behavior by giving test data:

Alternative options for specifying test data rather than specifying relevant material constants on the data lines of the **HYPERELASTIC* option are applicable to all hyperelastic material models except the user-defined model. No data lines are used with the **HYPERELASTIC* option when the MARLOW, VALANIS-LANDEL, or TEST DATA INPUT parameter is specified. In this case the test data are specified with the **BIAXIAL TEST DATA*, **PLANAR TEST DATA*, **UNIAXIAL TEST DATA*, and **VOLUMETRIC TEST DATA* options.

Data lines to define the material constants for the ARRUDA-BOYCE model:

First line:

1. μ.
2. λ_m.
3. D.
4. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

Data lines to define the material constants for the HENCKY model:
First line:

1. Young's modulus, E. (Units of FL^{-2}.)
2. Poisson's ratio, ν.
3. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

Data lines to define the material constants for the MOONEY-RIVLIN model:
First line:

1. C_{10}.
2. C_{01}.
3. D_1.
4. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

Data lines to define the material constants for the NEO HOOKE model:
First line:

1. C_{10}.
2. D_1.
3. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

Data lines to define the material constants for the OGDEN strain energy potential:
First line if N=1:

1. μ_1.
2. α_1.
3. D_1.
4. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

First line if N=2:

1. μ_1.
2. α_1.
3. μ_2.
4. α_2.
5. D_1.
6. D_2.
7. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

First line if N=3:

1. μ_1.
2. α_1.
3. μ_2.
4. α_2.
5. μ_3.
6. α_3.
7. D_1.
8. D_2.

Second line if N=3:

1. D_3.
2. Temperature.

Repeat this pair of data lines as often as necessary to define the material constants as a function of temperature.

Data lines for higher values of N (up to 6):

1. The data lines for higher values of *N* follow the same pattern. First, give the μ_i and α_i for *i* from *1* to *N*. Then, give the *N* coefficients D_i. Finally, give the temperature. Exactly eight data values should be given on each line.

Data lines to define the material constants for the POLYNOMIAL strain energy potential:

First line if N=1:

1. C_{10}.
2. C_{01}.
3. D_1.
4. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

First line if N=2:

1. C_{10}.
2. C_{01}.
3. C_{20}.
4. C_{11}.
5. C_{02}.
6. D_1.
7. D_2.
8. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

First line if N=3:

1. C_{10}.
2. C_{01}.
3. C_{20}.
4. C_{11}.
5. C_{02}.
6. C_{30}.
7. C_{21}.
8. C_{12}.

Second line if N=3:

1. C_{03}.
2. D_1.
3. D_2.
4. D_3.
5. Temperature.

Repeat this pair of data lines as often as necessary to define the material constants as a function of temperature.

Data lines for higher values of N (up to 6):

1. The data lines for higher values of *N* follow the same pattern. For each value of $(i + j)$ from *1* to *N*, give the C_{ij}, with *i* decreasing from $(i + j)$ to zero and *j* increasing from zero to $(i + j)$. Then, give the *N* coefficients D_i. Finally, give the temperature. Exactly eight data values should be given on each line.

Data lines to define the material constants for the REDUCED POLYNOMIAL strain energy potential:

First line if N=1:

1. C_{10}.
2. D_1.
3. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

First line if N=2:

1. C_{10}.
2. C_{20}.
3. D_1.
4. D_2.
5. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

First line if N=3:

1. C_{10}.
2. C_{20}.
3. C_{30}.
4. D_1.
5. D_2.
6. D_3.
7. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

Data lines for higher values of N (up to 6):

1. The data lines for the higher values of N follow the same pattern. First, give the C_{i0} for i from 1 to N. Then, give the N coefficients D_i. Finally, give the temperature. Exactly eight data values should be given on each line.

Data lines to define the material properties for the USER hyperelasticity model:

No data lines are needed if the PROPERTIES parameter is omitted or set to 0. Otherwise, first line:

1. Give the material properties, eight per line. If this option is used in conjunction with the **VISCOELASTIC* or **HYSTERESIS* option, the material properties must define the instantaneous behavior. If this option is used in conjunction with the **MULLINS EFFECT* option, the material properties must define the primary response.

Repeat this data line as often as necessary to define the material properties.

Data lines to define the material constants for the VAN DER WAALS model:

First line:

1. μ.
2. λ_m.
3. a.
4. β.
5. D.
6. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

Data lines to define the material constants for the YEOH model:

First line:

1. C_{10}.
2. C_{20}.
3. C_{30}.
4. D_1.

5. D_2.
6. D_3.
7. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

*HYPERFOAM: Specify elastic properties for a hyperelastic foam.

This option is used to define material constants for a general elastomeric foam. This material is distinct from the regular hyperelastic materials in that it is highly compressible.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Hyperelastic Behavior in Elastomeric Foams*
- **BIAXIAL TEST DATA*
- **PLANAR TEST DATA*
- **SIMPLE SHEAR TEST DATA*
- **UNIAXIAL TEST DATA*
- **VOLUMETRIC TEST DATA*

Optional parameters:

MODULI

This parameter is applicable only when the **HYPERFOAM* option is used in conjunction with the **VISCOELASTIC* option.

Set MODULI=INSTANTANEOUS to indicate that the hyperfoam material constants define the instantaneous behavior. This parameter value is not available for frequency domain viscoelasticity in an Abaqus/Standard analysis.

Set MODULI=LONG TERM (default) to indicate that the hyperfoam material constants define the long-term behavior.

N

Set this parameter equal to the order of the strain energy potential. The maximum value is N=6. The default is N=1.

POISSON

Set this parameter equal to the effective Poisson's ratio, ν, of the material. This parameter is valid only when the TEST DATA INPUT parameter is specified.

If this parameter is included, it is assumed that $\nu_i = \nu$ for all i and that lateral strain test data and volumetric test data are not required. If this parameter is omitted, the ν_i are calculated from lateral strain data and/or volumetric test data.

TEST DATA INPUT

Include this parameter if the μ_i, the α_i, and the ν_i material constants are to be computed by Abaqus from data taken from simple tests on a material specimen. If this parameter is omitted, μ_i, α_i, and ν_i must be given directly on the data lines.

To define elastomeric foam behavior by giving test data:

No data lines are used with this option when the TEST DATA INPUT parameter is specified. The data are given instead under the **BIAXIAL TEST DATA*, the **PLANAR TEST DATA*, the **SIMPLE SHEAR TEST DATA*, the **UNIAXIAL TEST DATA*, and the **VOLUMETRIC TEST DATA* options.

Data lines to define the elastic properties directly:

First line if N=1:

1. μ_1.
2. α_1.
3. ν_1.
4. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

First line if N=2:

1. μ_1.
2. α_1.
3. μ_2.
4. α_2.
5. ν_1.
6. ν_2.
7. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

First line if N=3:

1. μ_1.
2. α_1.
3. μ_2.
4. α_2.
5. μ_3.
6. α_3.
7. ν_1.
8. ν_2.

Second line if N=3:

1. ν_3.
2. Temperature.

Repeat this pair of data lines as often as necessary to define the material constants as a function of temperature.

Data lines for higher values of N (up to 6):

1. The data lines for higher values of N follow the same pattern. First, give the μ_i and α_i for i from 1 to N. Then, give the $N\nu_i$. Finally, give the temperature. Exactly eight data values should be given on each line.

*HYPOELASTIC: Specify hypoelastic material properties.

This option is used to define a nonlinear, small-strain elastic material.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Hypoelastic Behavior*
- *UHYPEL*

Optional parameters:

USER

Include this parameter if the moduli are defined in user subroutine *UHYPEL*. Omit this parameter if the moduli are defined on the data lines.

Data lines to define hypoelasticity by specifying the material constants directly:

First line:

1. Instantaneous Young's modulus, E.
2. Instantaneous Poisson's ratio, ν.
3. First strain invariant, I_1.
4. Second strain invariant, I_2.
5. Third strain invariant, I_3.

Repeat this data line as often as necessary to define the variation of the moduli with the strain invariants.

To define hypoelasticity by a user subroutine:

No data lines are used with this option when the USER parameter is specified. Instead, user subroutine *UHYPEL* must be used to define the hypoelasticity.

*HYSTERESIS: Specify a rate-dependent elastomer model.

This option is used to specify the creep part of the material model for the hysteretic behavior of elastomers.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Hysteresis in Elastomers*

There are no parameters associated with this option.

Data line to define the creep behavior:

First (and only) line:

1. Stress scaling factor, S.
2. Creep parameter, A.
3. Effective stress exponent, m.
4. Creep strain exponent, C.
5. Constant, E, for regularizing the creep strain rate near the undeformed state. The value of E should be non-negative. If this field is left blank, the default value of 0.01 is used.

Typical values of these parameters for common elastomers are given in *Hysteresis in Elastomers*.

I

This section describes each Abaqus input option that begins with I, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **IMPEDANCE*
- **IMPEDANCE PROPERTY*
- **IMPERFECTION*
- **IMPORT*
- **IMPORT CONTROLS*
- **IMPORT ELSET*
- **IMPORT NSET*
- **IMPORT SURFACE*
- **INCIDENT WAVE*
- **INCIDENT WAVE FLUID PROPERTY*
- **INCIDENT WAVE INTERACTION*
- **INCIDENT WAVE INTERACTION PROPERTY*
- **INCIDENT WAVE PROPERTY*
- **INCIDENT WAVE REFLECTION*
- **INCLUDE*
- **INCREMENTATION OUTPUT*
- **INELASTIC HEAT FRACTION*
- **INERTIA RELIEF*
- **INITIAL CONDITIONS*
- **INSTANCE*
- **INTEGRATED OUTPUT*
- **INTEGRATED OUTPUT SECTION*
- **INTERFACE*
- **INTERFACE REACTION*
- **INTERNAL FLUX GENERATION*
- **ISOTROPIZATION PARAMETERS*
- **ITS*

*IMPEDANCE: Define impedances for acoustic analysis.

This option is used to provide boundary impedances or nonreflecting boundaries for acoustic and coupled acoustic-structural analyses.

Products: Abaqus/Standard Abaqus/Explicit

Type: History data

Level: Step

References:

- *Coupled Acoustic-Structural Analysis*
- *Acoustic and Shock Loads*
- **IMPEDANCE PROPERTY*
- **SIMPEDANCE*

Required, mutually exclusive parameters:

PROPERTY

Set this parameter equal to the name of the **IMPEDANCE PROPERTY* option defining the table of impedance values to be used.

NONREFLECTING

Set NONREFLECTING=PLANAR (default) to specify the impedance corresponding to that of a normal incidence plane wave.

Set NONREFLECTING=IMPROVED to specify the impedance corresponding to that of a plane wave at an arbitrary angle of incidence. This parameter can be used only for transient dynamics.

Set NONREFLECTING=CIRCULAR to specify a radiation condition appropriate for a circular boundary in two dimensions or a right circular cylinder in three dimensions.

Set NONREFLECTING=SPHERICAL to specify a radiation condition appropriate for a spherical boundary.

Set NONREFLECTING=ELLIPTICAL to specify a radiation condition appropriate for an elliptical boundary in two dimensions or a right elliptical cylinder in three dimensions.

Set NONREFLECTING=PROLATE SPHEROIDAL to specify a radiation condition appropriate for a prolate spheroidal boundary.

Optional parameters:

OP

Set OP=MOD (default) to modify existing impedances or to define additional impedances.

Set OP=NEW if all existing impedances applied to the model should be removed. To remove only selected impedances, use OP=NEW and respecify all impedances that are to be retained.

Data line to define an impedance for PROPERTY, NONREFLECTING=PLANAR, or NONREFLECTING=IMPROVED:

First (and only) line:

1. Element number or element set label.

2. Surface impedance type label, I*n*, for impedance on face *n*.

Data line to define an absorbing boundary impedance for NONREFLECTING=CIRCULAR or NONREFLECTING=SPHERICAL:

First (and only) line:

1. Element number or element set label.
2. Surface impedance type label, I*n*, for impedance on face *n*.
3. r_1, the radius of the circle or sphere defining the absorbing boundary surface.

Data line to define an absorbing boundary impedance for NONREFLECTING=ELLIPTICAL or NONREFLECTING=PROLATE SPHEROIDAL:

First (and only) line:

1. Element number or element set label.
2. Surface impedance type label, I*n*, for impedance on face *n*.
3. The semimajor axis, *a*, of the ellipse or prolate spheroid defining the surface. *a* is 1/2 of the maximum distance between two points on the ellipse or spheroid, analogous to the radius of a circle or sphere.
4. The eccentricity, ϵ, of the ellipse or prolate spheroid. The eccentricity is the square root of one minus the square of the ratio of the semiminor axis, *b*, to the semimajor axis, *a*: $\epsilon = \sqrt{1-(b/a)^2}$.
5. Global *X*-coordinate of the center of the ellipse or prolate spheroid defining the radiating surface.
6. Global *Y*-coordinate of the center of the ellipse or prolate spheroid defining the radiating surface.
7. Global *Z*-coordinate of the center of the ellipse or prolate spheroid defining the radiating surface.
8. *X*-component of the direction cosine of the major axis of the ellipse or prolate spheroid defining the radiating surface. The components of this vector need not be normalized to unit magnitude.
9. *Y*-component of the direction cosine of the major axis of the ellipse or prolate spheroid defining the radiating surface.
10. *Z*-component of the direction cosine of the major axis of the ellipse or prolate spheroid defining the radiating surface.

*IMPEDANCE PROPERTY: Define the impedance parameters for an acoustic medium boundary.

This option is used to define the proportionality factors between the pressure and the normal components of surface displacement and velocity in acoustic analysis. The **IMPEDANCE PROPERTY* option must be used in conjunction with the **IMPEDANCE* or **SIMPEDANCE* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Acoustic and Shock Loads*
- **IMPEDANCE*
- **SIMPEDANCE*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the impedance property on the **IMPEDANCE* or **SIMPEDANCE* option.

Optional parameters:

DATA

Set DATA=ADMITTANCE (default) to specify an impedance using a table of admittance values.

Set DATA=IMPEDANCE to specify an impedance using a table of real and imaginary parts of the impedance.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

Data lines to define an impedance using DATA=ADMITTANCE (default):

First line:

1. $1/k_1$, the proportionality factor between pressure and displacement of the surface in the normal direction. This quantity is the imaginary part of the complex admittance, divided by the angular frequency; see *Acoustic and Shock Loads*. (Units of $F^{-1}L^3$.)
2. $1/c_1$, the proportionality factor between pressure and velocity of the surface in the normal direction. This quantity is the real part of the complex admittance. (Units of $F^{-1}L^3T^{-1}$.)
3. Frequency. (Cycles/time.) Frequency dependence is active only during frequency domain analysis in Abaqus/Standard.

Repeat this data line as often as necessary in Abaqus/Standard to describe the variation of the coefficients with frequency. Only the first line entered will be used in direct-integration procedures.

Data lines to define an impedance using DATA=IMPEDANCE:

First line:

1. **Re (Z)**, the real part of the surface impedance. (Units of F L^{-3}T.)
2. **Im (Z)**, the imaginary part of the surface impedance. (Units of F L^{-3}T.)
3. Frequency. (Cycles/time.) Frequency dependence is active only during frequency domain analysis in Abaqus/Standard.

Repeat this data line as often as necessary in Abaqus/Standard to describe the variation of the coefficients with frequency. Only the first line entered will be used in direct-integration procedures.

*IMPERFECTION: Introduce geometric imperfections for postbuckling analysis.

This option is used to introduce a geometric imperfection into a model for a postbuckling analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Introducing a Geometric Imperfection into a Model*
- *Unstable Collapse and Postbuckling Analysis*
- *Eigenvalue Buckling Prediction*

Optional parameters (mutually exclusive—if neither parameter is specified, Abaqus assumes that the imperfection data will be entered directly on the data lines):

FILE

Set this parameter equal to the name of the results (`.fil`) file or an output database (`.odb` or `.sim`) file from a previous Abaqus/Standard analysis containing either the mode shapes from a **BUCKLE* or **FREQUENCY* analysis or the nodal displacements from a **STATIC* analysis. The file extension is optional. However, if both a results file and output database file exist, the results file is used. If there is no results file but both the `.odb` and `.sim` files exist, the `.odb` file is used.

INPUT

Set this parameter equal to the name of the alternate input file containing the imperfection data, in general, as the node number and imperfection values in the global coordinate system. See *Input Syntax Rules* for the syntax of such file names.

Required parameter if the FILE parameter is used:

STEP

Set this parameter equal to the step number (in the analysis whose file is being used as input to this option) from which the modal or displacement data are to be read.

Optional parameters if the FILE parameter is used:

INC

Set this parameter equal to the increment number (in the analysis whose file is being used as input to this option) from which the displacement data are to be read. If this parameter is omitted, Abaqus will read the data from the last increment available for the specified step in the results or output database file.

NSET

Set this parameter equal to the node set to which the geometric imperfection values are to be applied. If this parameter is omitted, the imperfection will be applied to all nodes in the model.

Optional parameter if the FILE parameter is omitted:

SYSTEM

Set SYSTEM=R (default) to specify the imperfection as perturbation values of Cartesian coordinates. Set SYSTEM=C to specify the imperfection as perturbation values of cylindrical coordinates. Set SYSTEM=S to specify the imperfection as perturbation values of spherical coordinates. See *Figure 1*.

The SYSTEM parameter is entirely local to this option and should not be confused with the **SYSTEM* option. As the data lines are read, the imperfection values specified are transformed to the global rectangular Cartesian coordinate system. This transformation requires that the object be centered about the origin of the global coordinate system; i.e., the **SYSTEM* option should be off when specifying imperfections as perturbation values using either cylindrical or spherical coordinates.

Data lines to define the imperfection as a linear superposition of mode shapes from the results file:

First line:

1. Mode number.
2. Scaling factor for this mode.

Repeat this data line as often as necessary to define the imperfection as a linear combination of mode shapes.

Data line to define the imperfection based on the solution of a static analysis from the results file:

First (and only) line:

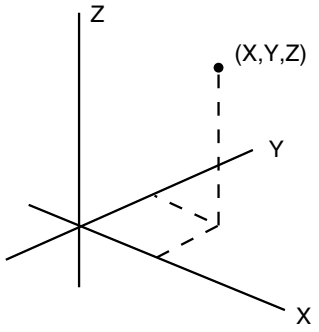
1. Set to 1.
2. Scaling factor.

Data lines to define the imperfection if the FILE and INPUT parameters are omitted:

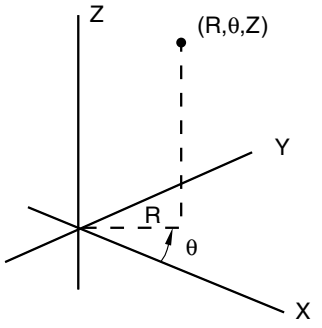
First line:

1. Node number.
2. Component of imperfection in the first coordinate direction.
3. Component of imperfection in the second coordinate direction.
4. Component of imperfection in the third coordinate direction.

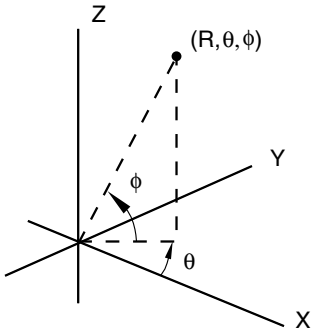
Repeat this data line as often as necessary to define the imperfection.



Rectangular Cartesian
(SYSTEM=R)
(default)



Cylindrical
(SYSTEM=C)



Spherical
(SYSTEM=S)

(θ and ϕ are given in degrees)

Figure 1: Coordinate systems.

*IMPORT: Import information from a previous Abaqus/Explicit or Abaqus/Standard analysis.

This option is used to define the time in a previous Abaqus/Standard or Abaqus/Explicit analysis at which the specified node and element information is imported. The **IMPORT* option must be used in conjunction with the **INSTANCE* option when importing a part instance from a previous analysis.

You can define new positions for the imported elements and import an element or a part instance more than once. In an Abaqus/Explicit import analysis you can also import element sets and part instances from multiple previous analyses.

You can use the full-model input format to specify that all import model data (including elements and nodes, element sets and node sets, sections and materials, etc.) are given in the input. The full-model input format is specified in the **IMPORT CONTROLS* option. This format cannot be used in conjunction with the **INSTANCE* option.

You can input a blank line as data to specify import of all element sets in the previous analysis with a section definition that is supported for import. This format provides a simple way to import all elements in a model without the need to list the element sets to be imported explicitly. You can use the blank line input format with repositioning. However, you cannot use the blank line input format with the RENAME parameter or in conjunction with the **INSTANCE* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part instance

Abaqus/CAE: Supported for use in conjunction with part instances; importing selected part instances stored in an output database is supported using the **File** menu and importing the initial state of part instances is supported in the *Load module*.

References:

- *About Transferring Results between Abaqus Analyses*
- **INSTANCE*

Required parameters:

UPDATE

Set UPDATE=NO to continue the analysis without resetting the reference configuration.

Set UPDATE=YES to continue the analysis by resetting the reference configuration to be the imported configuration. In this case displacement and strain values are calculated from the new reference configuration.

Required parameters if the imported elements are to be renamed (not applicable for import of part instances):

EOFFSET

Set this parameter equal to an integer that specifies the offset to be used to renumber the imported elements.

If you use the full-model input format, you must number the elements for the imported element sets input in the data lines below by their numbers in the previous analysis plus the value assigned to the EOFFSET parameter.

NOFFSET

Set this parameter equal to an integer that specifies the offset to be used to renumber the imported nodes.

If you use the full-model input format, you must number the nodes for the imported node sets input in the data lines below by their numbers in the previous analysis plus the value assigned to the NOFFSET parameter.

RENAME

Include this parameter to specify new labels for the element sets to be imported from the previous analysis.

This parameter is not applicable if you use the full-model input format because the names of the imported element sets are given in the input.

Required parameter if importing from multiple previous analyses (Abaqus/Explicit only; not applicable for import of part instances) or if the full-model input format is used; optional parameter if importing from a single analysis:

LIBRARY

Set this parameter equal to a value that specifies the previous analysis from which to import the element sets. You can specify the name of the previous analysis or the name including a full path. If no path is specified, all input files and results files from the previous analyses must reside in the current (working) directory.

When importing from a single previous analysis, if the LIBRARY parameter is omitted, you must specify the job name of the previous analysis in the command line using the **oldjob** option. If both methods are used, the LIBRARY parameter takes precedence over the command line specification.

When importing from multiple previous analyses or when the full-model input format is specified, set this parameter equal to the job name. In this case, you should not use the **oldjob** option on the command line.

Optional parameters:

INCREMENT

When importing an analysis from Abaqus/Standard, set this parameter equal to the increment of the specified step on the Abaqus/Standard restart file from which the analysis is to be imported. If this parameter is omitted, the analysis is imported from the last available increment of the specified step.

The INCREMENT, INTERVAL, and ITERATION parameters are mutually exclusive.

INTERVAL

When importing an analysis from Abaqus/Explicit, set this parameter equal to the interval of the specified step on the Abaqus/Explicit state file from which the analysis is to be imported. If this parameter is omitted, the analysis is imported from the last available interval of the specified step.

The INCREMENT, INTERVAL, and ITERATION parameters are mutually exclusive.

ITERATION

This parameter is relevant only when the results are imported from a previous direct cyclic Abaqus/Standard analysis.

Set this parameter equal to the iteration number of the specified step on the Abaqus/Standard restart file from which the analysis is to be imported. Since restart information can be written only at the end of an iteration in a direct cyclic analysis, the INCREMENT parameter is irrelevant and is ignored

if the ITERATION parameter is specified. If this parameter is omitted, the analysis is imported from the last available iteration of the specified step.

The INCREMENT, INTERVAL, and ITERATION parameters are mutually exclusive.

STATE

Set STATE=YES (default) to import the current material state of the elements at the specified step and the specified interval, increment, or iteration.

Set STATE=NO if no material state is to be imported. In this case the elements start with no initial state or with the state as defined by the **INITIAL CONDITIONS* option.

STEP

Set this parameter equal to the step number on the Abaqus/Explicit state file or on the Abaqus/Standard restart file from which the analysis is being imported.

If both the STEP and STEP NAME parameters are omitted, the analysis is imported from the last available step on the state file or the restart file at the specified increment, interval, or iteration.

The STEP and STEP NAME parameters are mutually exclusive.

STEP NAME

Set this parameter equal to the step name on the Abaqus/Explicit state file or on the Abaqus/Standard restart file from which the analysis is being imported. This parameter value is case sensitive.

If both the STEP and STEP NAME parameters are omitted, the analysis is imported from the last available step on the state file or the restart file at the specified increment, interval, or iteration.

The STEP and STEP NAME parameters are mutually exclusive.

Data lines to specify the elements to be imported and optionally repositioned:

First line if the RENAME parameter is not specified:

1. List of element sets that are to be imported. Specify only element set names that are used in the previous Abaqus/Explicit or Abaqus/Standard analysis.

   Enter a single blank line in place of a list of element sets to include all element sets with a section definition that are supported for import from the previous Abaqus/Explicit or Abaqus/Standard analysis.

Repeat this data line if it is not blank as often as necessary to define the element sets to be imported. Up to 16 element sets can be listed per data line.

First line if the RENAME parameter is specified:

1. The name of the element set to be imported. Specify only element set names that are used in the previous Abaqus/Explicit or Abaqus/Standard analysis.
2. The new name of the element set in the import analysis.

Repeat this data line as often as necessary to specify the old and new names of the element sets to be imported.

Subsequent line to translate the imported element sets (optional if rotation is not specified):

1. Value of the translation to be applied in the *X*-direction.
2. Value of the translation to be applied in the *Y*-direction.
3. Value of the translation to be applied in the *Z*-direction.

Enter values of zero to apply a pure rotation.

Subsequent line to rotate the imported element sets (optional):

1. *X*-coordinate of point *a* on the axis of rotation (see *Figure 1*).
2. *Y*-coordinate of point *a* on the axis of rotation.
3. *Z*-coordinate of point *a* on the axis of rotation.
4. *X*-coordinate of point *b* on the axis of rotation.
5. *Y*-coordinate of point *b* on the axis of rotation.
6. *Z*-coordinate of point *b* on the axis of rotation.
7. Angle of rotation about the axis *a*–*b*, in degrees.

If both translation and rotation are specified, translation is applied before rotation.

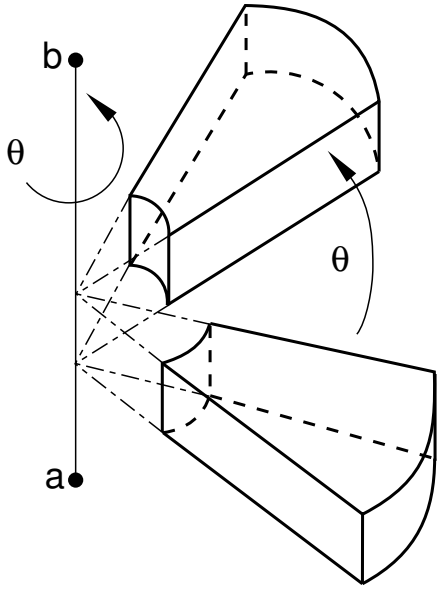


Figure 1: Rotation definition for import.

There are no data lines for importing a part instance

*IMPORT CONTROLS: Specify the full-model input format, the tie constraint formation configuration, and the tolerance for error checking on shell normals.

This option is used to specify the full-model input format, the tie constraint formation configuration, and the tolerance for error checking on shell normals in Abaqus/Standard or Abaqus/Explicit when the *IMPORT, UPDATE=YES option is used.

If the *IMPORT CONTROLS option is used, it must appear after the *IMPORT option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Transferring Results between Abaqus/Explicit and Abaqus/Standard*
- *IMPORT

Optional parameters:

MODEL

Set MODEL=YES to use the full-model input format.

This parameter cannot be used in conjunction with the *INSTANCE option.

NORMAL TOL

Set this parameter equal to the tolerance required for the error checking on shell normals. The default value is 0.1.

TIE CONFIG

This parameter applies only to Abaqus/Explicit import analysis.

Set TIE CONFIG=OLD if the original configuration is to be used in the tie constraint formation for all surfaces of the imported model.

Set TIE CONFIG=NEW if the updated configuration is to be used in the tie constraint formation for all surfaces of the imported model.

There are no data lines associated with this option.

*IMPORT ELSET: Import element set definitions from a previous Abaqus/Explicit or Abaqus/Standard analysis.

This option is used to import element set definitions that were defined in a previous Abaqus/Explicit or Abaqus/Standard analysis. If the *IMPORT ELSET option is used, it must appear after the *IMPORT option.

If the *IMPORT ELSET option is used with the RENAME parameter, it must appear after an *IMPORT option with active RENAME and EOFFSET parameters.

If this option is omitted or is specified without any data lines and all *IMPORT options that reference a previous analysis do not include the RENAME parameter, all the element sets relevant to the analysis are imported. References to elements that are not imported are removed from imported sets.

This option is ignored if you specify the full-model input format in the *IMPORT CONTROLS* option because data for all imported element sets are given in the input.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Transferring Results between Abaqus/Explicit and Abaqus/Standard*
- *IMPORT

Required parameter if the imported element sets are to be renamed:

RENAME

Include this parameter to indicate that the imported element sets will be given new names.

Data lines to specify the element set definitions to be imported but not renamed:

First line:

1. List of element sets that are to be imported. Specify only element set names that are used in the previous Abaqus/Explicit or Abaqus/Standard analysis.

Repeat this data line as often as necessary to define the element sets to be imported. Up to 16 element sets can be listed per line.

Data lines to specify the element set definitions to be imported and renamed:

First line:

1. The name of the element set to be imported. Specify only element set names that are used in the previous Abaqus/Explicit or Abaqus/Standard analysis.
2. The new name of the element set in the import analysis.

Repeat this data line as often as necessary to specify the old and new names of the element sets to be imported.

*IMPORT NSET: Import node set definitions from a previous Abaqus/Explicit or Abaqus/Standard analysis.

This option is used to import node set definitions that were defined in a previous Abaqus/Explicit or Abaqus/Standard analysis. If the **IMPORT NSET* option is used, it must appear after the **IMPORT* option.

If the **IMPORT NSET* option is used with the RENAME parameter, it must appear after an **IMPORT* option with active RENAME and NOFFSET parameters.

If this option is omitted or is specified without any data lines and all **IMPORT* options that reference a previous analysis do not include the RENAME parameter, all the node sets relevant to the analysis are imported. References to nodes that are not imported are removed from imported sets.

This option is ignored if you specify the full-model input format in the *IMPORT CONTROLS* option because data for all imported node sets are given in the input.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Transferring Results between Abaqus/Explicit and Abaqus/Standard*
- **IMPORT*

Required parameter if the imported node sets are to be renamed:

RENAME

Include this parameter to indicate that the imported node sets will be given new names.

Data lines to specify the node set definitions to be imported but not renamed:

First line:

1. List of node sets that are to be imported. Specify only node set names that are used in the previous Abaqus/Explicit or Abaqus/Standard analysis.

Repeat this data line as often as necessary to specify the node sets to be imported. Up to 16 node sets can be listed per line.

Data lines to specify the node set definitions to be imported and renamed:

First line:

1. The name of the node set to be imported. Specify only node set names that are used in the previous Abaqus/Explicit or Abaqus/Standard analysis.
2. The new name of the node set in the import analysis.

Repeat this data line as often as necessary to specify the old and new names of the node sets to be imported.

*IMPORT SURFACE: Import surface definitions from a previous Abaqus/Explicit or Abaqus/Standard analysis.

This option is used to import surface definitions that were defined in a previous Abaqus/Explicit or Abaqus/Standard analysis. If the *IMPORT SURFACE option is used, it must appear after the *IMPORT or the *INSTANCE option.

If this option is specified after an *IMPORT option without the RENAME parameter and with no data lines, all surfaces relevant to the analysis are imported. If this option is omitted in an Abaqus/Standard to Abaqus/Standard import analysis and the *INSTANCE option is not used, all surfaces relevant to the analysis are imported.

If this option is specified with the RENAME parameter, it must appear after an *IMPORT option with the RENAME parameter specified. The values of the EOFFSET and NOFFSET parameters of the *IMPORT option apply to the *IMPORT SURFACE option.

If this option is specified without any data lines, all surfaces relevant to the analysis are imported.

If this option is specified after the *INSTANCE option, the RENAME parameter cannot be used, and this option cannot have any data lines. All surfaces defined within the instance are imported if you include this option, and none of the surfaces are imported if you omit this option.

In an Abaqus/Explicit analysis, you cannot import surfaces defined for a *CONTACT PAIR to or from an Abaqus/Explicit or Abaqus/Standard analysis. If these surfaces are required, you must redefine them in the import analysis.

This option is ignored if you specify the full-model input format in the IMPORT CONTROLS option because data for all imported surfaces are given in the input.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Transferring Results between Abaqus/Explicit and Abaqus/Standard*
- *IMPORT
- *INSTANCE

Required parameter if the imported surfaces are to be renamed:

RENAME

This parameter is not applicable if the *IMPORT SURFACE option is specified after an *INSTANCE option.

Include this parameter to indicate that the imported surfaces will be given new names.

Optional data lines to specify the surface definitions to be imported but not renamed:

First line:

1. List of surfaces that are to be imported. Specify only surface names that are used in the previous Abaqus/Explicit or Abaqus/Standard analysis.

Repeat this data line as often as necessary to define the surfaces to be imported. Up to 16 surfaces can be listed per line.

Data lines to specify the surface definitions to be imported and renamed:

First line:

1. The name of the surface to be imported. Specify only surface names that are used in the previous Abaqus/Explicit or Abaqus/Standard analysis.
2. The new name of the surface in the import analysis.

Repeat this data line as often as necessary to specify the old and new names of the surfaces to be imported.

*INCIDENT WAVE: Define incident wave loading for a blast or scattering load on a boundary.

The preferred interface for applying incident wave loading is the *INCIDENT WAVE INTERACTION option used in conjunction with the *INCIDENT WAVE INTERACTION PROPERTY option. The alternative interface uses the *INCIDENT WAVE option to apply incident wave loading.

The *INCIDENT WAVE PROPERTY option must be used in conjunction with the *INCIDENT WAVE option. If the incident wave field includes a reflection off a plane outside the boundaries of the mesh, this effect can be modeled with the *INCIDENT WAVE REFLECTION option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Unsupported; this option has been superseded by incident wave interactions.

References:

- *Coupled Acoustic-Structural Analysis*
- *Acoustic and Shock Loads*
- *INCIDENT WAVE PROPERTY
- *INCIDENT WAVE REFLECTION

Required parameters:

PROPERTY

Set this parameter equal to the name of the *INCIDENT WAVE PROPERTY option defining the incident wave field.

Required, mutually exclusive parameters:

ACCELERATION AMPLITUDE

Set this parameter equal to the name of the amplitude curve defining the fluid particle acceleration time history at the standoff point (*Amplitude Curves*). This amplitude curve will be used to compute the fluid traction *only*: a solid surface requiring a pressure load cannot be specified on the data line of an *INCIDENT WAVE option if the ACCELERATION AMPLITUDE parameter is used.

This parameter is valid only for planar incident waves using the *INCIDENT WAVE PROPERTY, TYPE=PLANE option. Reflected loads, using the *INCIDENT WAVE REFLECTION option, are not permitted in this case.

PRESSURE AMPLITUDE

Set this parameter equal to the name of the amplitude curve defining the fluid pressure time history at the standoff point (*Amplitude Curves*). The corresponding fluid traction, if required, will be computed from the pressure amplitude reference.

Data lines to define an incident wave:

First line:

1. Surface name.
2. Reference magnitude.

Repeat this data line as often as necessary to describe the loading on the surfaces due to the incident wave. In problems involving fluid-solid boundaries, both the fluid surface and the solid surface comprising the boundary must have an incident wave load specified, using the appropriate load type.

*INCIDENT WAVE FLUID PROPERTY: Define the fluid properties associated with an incident wave.

The preferred interface for defining the fluid properties for an incident wave is the *INCIDENT WAVE INTERACTION PROPERTY option used in conjunction with the *INCIDENT WAVE INTERACTION option. The alternative interface uses the *INCIDENT WAVE FLUID PROPERTY option to define the fluid properties used to define an incident wave.

The *INCIDENT WAVE FLUID PROPERTY must be used in conjunction with the *INCIDENT WAVE PROPERTY option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Unsupported; this option has been superseded by incident wave interaction properties.

References:

- *Acoustic Medium*
- *Acoustic and Shock Loads*
- *INCIDENT WAVE PROPERTY

There are no parameters associated with this option.

Data line to define incident wave fluid properties:

First (and only) line:

1. K_f, the fluid bulk modulus.
2. ρ_f, the fluid mass density.

*INCIDENT WAVE INTERACTION: Define incident wave loading for a blast or scattering load on a surface.

This option is used to apply incident wave loading. The **INCIDENT WAVE INTERACTION PROPERTY* option must be used in conjunction with the **INCIDENT WAVE INTERACTION* option. If the incident wave field includes a reflection off a plane outside the boundaries of the mesh, this effect can be modeled with the **INCIDENT WAVE REFLECTION* option. The incident wave interaction can be used in steady-state dynamic procedures to define individual wave sources or loading from diffuse sources.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *Coupled Acoustic-Structural Analysis*
- *Acoustic and Shock Loads*
- **INCIDENT WAVE INTERACTION PROPERTY*
- **INCIDENT WAVE REFLECTION*

Required parameters:

PROPERTY

Set this parameter equal to the name of the **INCIDENT WAVE INTERACTION PROPERTY* option defining the incident wave field.

Required, mutually exclusive parameters:

ACCELERATION AMPLITUDE

Set this parameter equal to the name of the amplitude curve defining the fluid particle acceleration time history at the standoff point (*Amplitude Curves*). This amplitude curve will be used to compute the fluid traction *only*: a solid surface requiring a pressure load cannot be specified on the data line of an **INCIDENT WAVE INTERACTION* option if the ACCELERATION AMPLITUDE parameter is used.

This parameter is valid only for transient planar incident waves using the **INCIDENT WAVE INTERACTION PROPERTY*, TYPE=PLANE option. Reflected loads, using the **INCIDENT WAVE REFLECTION* option, are not permitted in this case.

CONWEP

This parameter applies only to Abaqus/Explicit analyses.

Include this parameter to define an incident wave using the **CONWEP CHARGE PROPERTY* option. This parameter is valid only for blast waves using the **INCIDENT WAVE INTERACTION PROPERTY*, TYPE=AIR BLAST or TYPE=SURFACE BLAST option.

PRESSURE AMPLITUDE

Set this parameter equal to the name of the amplitude curve defining the fluid pressure time history at the standoff point (*Amplitude Curves*). The corresponding fluid traction, if required, will be computed from the pressure amplitude reference.

UNDEX

Include this parameter to define a spherical incident wave using the **UNDEX CHARGE PROPERTY* option. This parameter is valid only for spherical incident waves using the **INCIDENT WAVE INTERACTION PROPERTY*, TYPE=SPHERE option.

Optional, mutually exclusive parameters for matrix generation and steady-state dynamics analysis (direct or subspace):

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the loading at the standoff point.

REAL

Include this parameter (default) to define the real (in-phase) part of the loading at the standoff point.

Data line to define an incident wave when the CONWEP parameter is omitted:

First (and only) line:

1. Surface name.
2. The node number or set name of the source node.
3. The node number or set name of the standoff node.
4. Reference magnitude.

Data line to define an incident wave when the CONWEP parameter is included:

First (and only) line:

1. Surface name.
2. The node number or set name of the source node.
3. Blank field.
4. Time of detonation given in total time. See *Conventions* for a discussion of the total time measure.
5. Magnitude scale factor. The default is 1.0.

*INCIDENT WAVE INTERACTION PROPERTY: Define the geometric data and fluid properties describing an incident wave.

This option defines the geometric data and fluid properties used to define incident waves. Each **INCIDENT WAVE INTERACTION* option must refer to an **INCIDENT WAVE INTERACTION PROPERTY* definition.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Acoustic and Shock Loads*
- **INCIDENT WAVE INTERACTION*
- **UNDEX CHARGE PROPERTY*
- **CONWEP CHARGE PROPERTY*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the incident wave interaction property in the **INCIDENT WAVE INTERACTION* option.

Optional parameters:

TYPE

Set TYPE=PLANE (default) to specify a planar incident wave.

Set TYPE=SPHERE to specify a spherical incident wave.

Set TYPE=DIFFUSE to specify a field of planar waves incident from multiple angles.

Set TYPE=AIR BLAST to specify a spherical blast wave. This option is applicable only in an Abaqus/Explicit analysis.

Set TYPE=SURFACE BLAST to specify a hemispherical blast wave. This option is applicable only in an Abaqus/Explicit analysis.

Data line to define an incident wave interaction property with TYPE=PLANE, TYPE=SPHERE (acoustic decay), or TYPE=SPHERE used in conjunction with the *UNDEX CHARGE PROPERTY option:

First (and only) line:

1. c_f, the speed of sound in the fluid, defining the rate of propagation of the wave.
2. ρ_f, the fluid mass density.

Data line to define an incident wave interaction property with TYPE=SPHERE and generalized spatial decay:

First (and only) line:

1. c_f, the speed of sound in the fluid, defining the rate of propagation of the wave.

2. ρ_f, the fluid mass density.
3. A, dimensionless decay parameter ($A > -1$). The default is 0.
4. B, dimensionless decay parameter ($B > -1$). The default is 0.
5. C, dimensionless decay parameter ($C \geq 0$). The default is 0.

Data line to define incident wave loading from diffuse sources (TYPE=DIFFUSE):

First (and only) line:

1. c_f, the speed of sound in the fluid, defining the propagation speed of the waves.
2. ρ_f, the fluid mass density.
3. *N*, the seed number for the diffuse source calculation: N^2 sources will be used in the simulation.

No data lines are required for TYPE=AIR BLAST or TYPE=SURFACE BLAST used in conjunction with the *CONWEP CHARGE PROPERTY option

*INCIDENT WAVE PROPERTY: Define the geometric data describing an incident wave.

The preferred interface for defining the geometric data for an incident wave is the **INCIDENT WAVE INTERACTION PROPERTY* option used in conjunction with the **INCIDENT WAVE INTERACTION* option. The alternative interface uses the **INCIDENT WAVE PROPERTY* option to define the geometric data for incident waves.

Each **INCIDENT WAVE* option must refer to an **INCIDENT WAVE PROPERTY* definition. The **INCIDENT WAVE PROPERTY* option must be followed by the **INCIDENT WAVE FLUID PROPERTY* option, which defines the fluid properties used in the incident wave loading.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Unsupported; this option has been superseded by incident wave interaction properties.

References:

- *Acoustic and Shock Loads*
- **INCIDENT WAVE*
- **INCIDENT WAVE FLUID PROPERTY*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the incident wave property in the **INCIDENT WAVE* option.

Optional parameters:

TYPE

Set TYPE=PLANE (default) to specify a planar incident wave.

Set TYPE=SPHERE to specify a spherical incident wave.

Data lines to define an incident wave property:

First line:

1. *X*-coordinate of $\mathbf{x_0}$, the position of the incident wave standoff point.
2. *Y*-coordinate of $\mathbf{x_0}$, the position of the incident wave standoff point.
3. *Z*-coordinate of $\mathbf{x_0}$, the position of the incident wave standoff point.
4. *X*-component of $\mathbf{v_0}$, the velocity of the incident wave standoff point.
5. *Y*-component of $\mathbf{v_0}$, the velocity of the incident wave standoff point.
6. *Z*-component of $\mathbf{v_0}$, the velocity of the incident wave standoff point.

Second line:

1. *X*-coordinate of $\mathbf{x_S}$, the position of the incident wave source point. Alternatively, specify the name of an **AMPLITUDE* definition describing the time history of this coordinate.

2. *Y*-coordinate of $\mathbf{x_S}$, the position of the incident wave source point. Alternatively, specify the name of an **AMPLITUDE* definition describing the time history of this coordinate.
3. *Z*-coordinate of $\mathbf{x_S}$, the position of the incident wave source point. Alternatively, specify the name of an **AMPLITUDE* definition describing the time history of this coordinate.

If TYPE=PLANE, the vector from $\mathbf{x_S}$ to $\mathbf{x_0}$ defines the direction of the incoming wave; the distance between the two points is unimportant. For incident wave loads using bubble amplitudes, the source positions defined by the user with the **INCIDENT WAVE PROPERTY* option are interpreted as the initial positions of the source.

*INCIDENT WAVE REFLECTION: Define the reflection load on a surface caused by incident wave fields.

This option is used to define reflected incident wave fields. It must be used in conjunction with the *INCIDENT WAVE INTERACTION option (preferred interface for applying incident wave loading) or the *INCIDENT WAVE option (alternative interface).

Products: Abaqus/Standard Abaqus/Explicit

Type: History data

Level: Step

References:

- *Acoustic and Shock Loads*
- *INCIDENT WAVE INTERACTION
- *INCIDENT WAVE

There are no parameters associated with this option.

Data lines to define an incident wave reflection:

First line:

1. Distance from the source point to the first reflecting plane.
2. *X*-direction cosine of the normal to the reflecting plane, pointing away from the source point.
3. *Y*-direction cosine of the normal to the reflecting plane, pointing away from the source point.
4. *Z*-direction cosine of the normal to the reflecting plane, pointing away from the source point.
5. Reflection coefficient. The default value is zero, meaning no reflected wave will be applied.

Repeat this data line as often as necessary to describe the planes that reflect the incident wave.

*INCLUDE: Reference an external file containing Abaqus input data.

This option is used to reference an external file containing a portion of the Abaqus input file.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Part, Part instance, Assembly, Model, Step

Abaqus/CAE: Several input data options in Abaqus/CAE provide the capability to reference external files; for example, the material editor can read material properties from an ASCII file.

References:

• *Abaqus Model Definition*

Required parameters:

INPUT

Set this parameter equal to the name of the file containing the input data. See *Input Syntax Rules* for the syntax of such file names.

Optional parameters:

PASSWORD

When the external file is encrypted, set this parameter equal to the file's password. Passwords are case-sensitive.

There are no data lines associated with this option.

*INCREMENTATION OUTPUT: Define output database requests for time incrementation data.

This option is used to write incrementation variables to the output database. It must be used in conjunction with the *OUTPUT, HISTORY option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Output to the Output Database*
- *OUTPUT

Optional parameters:

VARIABLE

Set VARIABLE=ALL to indicate that all incrementation variables applicable to this procedure type should be written to the output database.

Set VARIABLE=PRESELECT to indicate that the default incrementation output variables for the current procedure type should be written to the output database. Additional output variables can be requested on the data lines.

If this parameter is omitted, the incrementation variables requested for output must be specified on the data lines.

Data lines to request incrementation output:

First line:

1. Give the identifying keys for the variables to be written to the output database. The keys are defined in *Abaqus/Explicit Output Variable Identifiers*.

Repeat this data line as often as necessary to define the time incrementation variables to be written to the output database.

*INELASTIC HEAT FRACTION: Define the fraction of the rate of inelastic dissipation that appears as a heat source.

This option is used to provide for inelastic energy dissipation to act as a heat source in adiabatic thermomechanical problems. It is relevant when the ADIABATIC parameter is included on the *DYNAMIC or the *STATIC option. It is also relevant when a coupled thermal-electrical-structural or fully coupled thermal-stress analysis is conducted using the *COUPLED TEMPERATURE-DISPLACEMENT or *DYNAMIC TEMPERATURE-DISPLACEMENT options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Adiabatic Analysis*
- *Fully Coupled Thermal-Stress Analysis*
- *Fully Coupled Thermal-Electrical-Structural Analysis*

There are no parameters associated with this option.

Data line to define the inelastic heat fraction:

First (and only) line:

1. Fraction of inelastic dissipation rate that appears as a heat flux per unit volume. The fraction may include a unit conversion factor if required. The default value is 0.9.

*INERTIA RELIEF: Apply inertia-based load balancing.

This option is used to apply inertia-based loads on a free or partially constrained body.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- *Inertia Relief*
- *Distributed Loads*

Optional parameters:

ORIENTATION

Set this parameter equal to the name given to the **ORIENTATION* definition (*Orientations*) that specifies the orientation of the local system for rigid body degrees of freedom.

Optional, mutually exclusive parameters:

FIXED

Include this parameter (default in a dynamic or buckling analysis) to indicate that the inertia relief load from a previous step should remain fixed at its value from the beginning of the current step.

REMOVE

Include this parameter to indicate that the inertia relief load from a previous step should be removed in the current step.

Optional data lines to specify global (or local, if the ORIENTATION parameter is used) degrees of freedom that define the free directions along which inertia relief loads are applied:

First line:

1. Integer list of degrees of freedom identifying the free directions.

Second line (only needed to define a reference point for the rigid body direction vectors when the user-chosen combination of free directions requires such a point):

1. Global *X*-coordinate of the reference point.
2. Global *Y*-coordinate of the reference point.
3. Global *Z*-coordinate of the reference point.

These data lines are needed only if rigid body motions are constrained in some directions.

There are no data lines when the FIXED or REMOVE parameters are specified

*INITIAL CONDITIONS: Specify initial conditions for the model.

This option is used to prescribe initial conditions for an analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE Abaqus/Aqua

Type: Model data

Level: Model

Abaqus/CAE: Load module

References:

• *Initial Conditions*

Required parameters:

TYPE

Set TYPE=ACOUSTIC STATIC PRESSURE to define initial static pressure values at acoustic nodes for use in evaluating the cavitation status of the acoustic element nodes in Abaqus/Explicit.

Set TYPE=ACTIVATION to define the initial volume fraction for elements used in progressive element activation in an Abaqus/Standard analysis. The value of the volume fraction must be equal to zero or one, which means that an element at the beginning of an analysis must be either inactive or fully active.

Set TYPE=BINDERVF to give the initial volume fraction of the binder phase in the electrodes of a battery cell for any of the coupled electrochemical analyses.

Set TYPE=CONCENTRATION to give initial normalized concentrations for a mass diffusion analysis in Abaqus/Standard.

Set TYPE=CONTACT to specify initial bonded contact conditions on part of the secondary surface identified by a node set in an Abaqus/Standard analysis.

Set TYPE=CURE to specify the initial degree of cure in Abaqus/Standard.

Set TYPE=DAMAGE INITIATION to specify initial values of the damage initiation measure. The CRITERION parameter must also be used to specify the damage initiation criterion for which initial conditions are being specified. The REBAR and SECTION POINTS parameters can be used with this parameter when CRITERION=DUCTILE or CRITERION=SHEAR.

Set TYPE=ENRICHMENT to specify initial location of an enriched feature, such as a crack, in an Abaqus/Standard analysis. Two signed distance functions per node are generally required to describe the crack location, including the location of crack tips, in a cracked geometry. The first describes the crack surface while the second is used to construct an orthogonal surface so that the intersection of the two surfaces gives the crack front. The first signed distance function is assigned only to nodes of elements intersected by the crack while the second signed distance function is assigned only to nodes of elements containing the crack tips. No explicit representation of the crack is needed as the crack is entirely described by the nodal data.

Set TYPE=ESDV to give initial values for element solution-dependent variables in an Abaqus/Standard analysis.

Set TYPE=FIELD to specify initial values of field variables. The VARIABLE parameter can be used with this parameter to define the field variable number. The STEP and INC parameters can be used in conjunction with the FILE parameter to define initial values of field variables from a results

(`.fil`) or output database (`.odb`) file. The STEP and INC parameters can also be used in conjunction with the FILE and OUTPUT VARIABLE parameters to define initial values of field variables based on scalar nodal output variables read from an output database file.

Set TYPE=FLUID ELECTRIC POTENTIAL to give initial electric potential in the electrolyte for a coupled thermal-electrochemical analysis.

Set TYPE=FLUID PRESSURE to give initial pressures for hydrostatic fluid-filled cavities.

Set TYPE=HARDENING to prescribe initial equivalent plastic strain and, if relevant, the initial backstress tensor or to prescribe initial volumetric compacting plastic strain for the crushable foam model. The REBAR and, in Abaqus/Standard, SECTION POINTS and USER parameters can be used with this parameter. If the USER parameter is used, the initial conditions on equivalent plastic strain and, if relevant, the backstress tensor must be specified via user subroutine *HARDINI* for each section point and for each rebar. Consequently, in this case the REBAR and SECTION POINTS parameters do not have any effect and are ignored. If the USER parameter is omitted, Abaqus/Standard assumes that the initial conditions are defined on the data lines.

Set TYPE=INITIAL GAP to identify the elements within which tangential fluid flow exists initially and to set the material initial damage variables at the integration points.

Set TYPE=ION CONCENTRATION to give initial concentrations in the electrolyte for any of the coupled electrochemical analyses.

Set TYPE=MASS FLOW RATE to specify initial values of mass flow rates in Abaqus/Standard heat transfer analyses involving forced convection modeled with the forced convection/diffusion heat transfer elements.

Set TYPE=NODE REF COORDINATE to define the reference mesh (initial metric) for membrane elements and three-dimensional solid elements in Abaqus/Explicit using node numbers and the coordinates of each node. If a reference mesh is specified for an element, no initial stress or strain can be specified for the same element. The initial stress and strain are computed automatically to account for deformation from the reference to the initial configuration.

Set TYPE=PARTICLE SPECIES CONCENTRATION to give initial species concentrations in the particles present in the active electrodes for any of the coupled electrochemical analyses.

Set TYPE=PLASTIC STRAIN to specify initial plastic strains. The SECTION POINTS and REBAR parameters can be used with this parameter. It is assumed that the plastic strain components are defined on each data line in the order given for the element type, as defined in *About the Element Library*.

Set TYPE=PORE PRESSURE to give initial pore fluid pressures for a coupled pore fluid diffusion/stress analysis in Abaqus/Standard. The STEP and INC parameters can also be used with the FILE parameter to define initial values of pore fluid pressures based on scalar nodal output variables read from an output database (`.odb`) file. In Abaqus/Explicit, you can specify initial pore fluid pressures for an undrained pore fluid flow and stress analysis.

Set TYPE=POROSITY to give initial porosity values for materials defined with the **EOS COMPACTION* option in Abaqus/Explicit, materials defined using the poroelastic acoustic models in Abaqus/Standard, or modified Drucker-Prager/Cap plasticity materials. In Abaqus/Explicit, you can also specify initial porosity values in an undrained pore fluid flow and stress analysis. In Abaqus/Standard, you can specify initial porosity values in a coupled thermal-electrochemical-structural–pore pressure analysis.

Set TYPE=PRESSURE STRESS to give initial pressure stresses for a mass diffusion analysis in Abaqus/Standard. The STEP and INC parameters can be used in conjunction with the FILE parameter to define initial values of pressure stress from the results (`.fil`) file of a previous Abaqus/Standard stress/displacement analysis.

Set TYPE=RATIO to give initial void ratio values for a coupled pore fluid diffusion/stress analysis in Abaqus/Standard. The STEP and INC parameters can be used in conjunction with the FILE parameter to define initial values of void ratio from the output database (`.odb`) file of a previous Abaqus/Standard soil analysis. The USER parameter can be used with this parameter to define initial void ratio values in user subroutine *VOIDRI*. In Abaqus/Explicit, you can specify initial void ratio values for an undrained pore fluid flow and stress analysis.

Set TYPE=REF COORDINATE to define the reference mesh (initial metric) for membrane elements in Abaqus/Explicit using the element number and the coordinates of all of the nodes associated with the element. If a reference mesh is specified for an element, no initial stress or strain can be specified for the same element. The initial stress and strain are computed automatically to account for deformation from the reference to the initial configuration.

Set TYPE=RELATIVE DENSITY to give initial relative density values for materials defined with the **POROUS METAL PLASTICITY* option.

Set TYPE=ROTATING VELOCITY to prescribe initial velocities in terms of an angular velocity and a global translational velocity.

Set TYPE=SATURATION to give initial saturation values for the analysis of flow through a porous medium in Abaqus/Standard. If no initial saturation values are given on this option, the default is fully saturated conditions (saturation of 1.0). For partial saturation the initial saturation and the pore fluid pressure must be consistent in the sense that the pore fluid pressure must lie within the range of absorption and exsorption values for the initial saturation value. If this is not the case, Abaqus/Standard will adjust the saturation value as needed to satisfy this requirement.

Set TYPE=SLURRYVF to give initial slurry concentration in a geotechnical analysis that models slurry transport and placement.

Set TYPE=SOLIDVF to give the initial volume fraction of the solid phase for any of the coupled electrochemical analyses.

Set TYPE=SOLID ELECTRIC POTENTIAL to give initial electric potential in the solid phase for any of the coupled electrochemical analyses.

Set TYPE=SPECIES CONCENTRATION to give initial species concentrations in the solid electrodes for any of the coupled electrochemical analyses.

Set TYPE=SOLUTION to give initial values of solution-dependent state variables. The REBAR and USER parameters can be used with this parameter. If TYPE=SOLUTION is used without the USER parameter, element average quantities of the solution-dependent state variables must be defined on each data line.

Set TYPE=SPECIFIC ENERGY to give initial specific energy values for materials defined with the **EOS* option in Abaqus/Explicit.

Set TYPE=SPUD EMBEDMENT to give the initial embedment for a spud can in an Abaqus/Aqua analysis.

Set TYPE=SPUD PRELOAD to give the initial preload value for a spud can in an Abaqus/Aqua analysis.

Set TYPE=STRESS to give initial stresses. (These stresses are effective stresses when the analysis includes pore fluid flow.) The GEOSTATIC; the REBAR; the SECTION POINTS; and, in Abaqus/Standard, the USER parameters can be used with this parameter. If TYPE=STRESS is used without the USER parameter, it is assumed that the stress components are defined on each data line in the order given for the element type, as defined in *About the Element Library*. The STEP and INC parameters can also be used with the FILE parameter to define initial stress values based on stress output variables read from an output database (`.odb` or `.sim`) file.

Set TYPE=TEMPERATURE to give initial temperatures. The STEP and INC parameters can be used in conjunction with the FILE parameter to define initial temperatures from the results (`.fil`) or output database (`.odb`) file of a previous Abaqus/Standard heat transfer analysis.

Set TYPE=UNFOLD COORDINATE to give initial coordinates in an Abaqus/Standard one-step inverse analysis.

Set TYPE=VELOCITY to prescribe initial velocities. Initial velocities should be defined in the global directions, regardless of the use of the **TRANSFORM* option.

Set TYPE=VOLUME FRACTION to define the initial material content of Eulerian elements in an Abaqus/Explicit analysis.

Optional parameters:

ABSOLUTE EXTERIOR TOLERANCE

This parameter is relevant only for use with the INTERPOLATE parameter. Set this parameter equal to the absolute value (given in the units used in the model) by which nodes of the current model might lie outside the region of the model in the output database specified by the FILE parameter. If this parameter is not used or has a value of 0.0, the EXTERIOR TOLERANCE parameter will apply.

CORRESPONDING CONTACT STRESS

This parameter applies only to Abaqus/Standard analyses.

This parameter is used only with TYPE=STRESS.

Set CORRESPONDING CONTACT STRESS=YES (default) to activate initial contact stress calculations.

Set CORRESPONDING CONTACT STRESS=NO to turn off initial contact stress calculations.

This parameter is a global setting and affects all contact definitions in a model. Also, if the **INITIAL CONDITIONS* options is repeated multiple times in an input file, the CORRESPONDING CONTACT STRESS parameter setting on the last occurrence prevails.

CRITERION

Set CRITERION=DUCTILE to provide the damage initiation measure for the ductile damage initiation criterion.

Set CRITERION=MSFLD to provide the damage initiation measure for the Müschenborn and Sonne forming limit diagram based damage initiation criterion.

Set CRITERION=SHEAR to provide the damage initiation measure for the shear damage initiation criterion.

DEFINITION

Set DEFINITION=COORDINATES (default) to define the axis of rotation for TYPE=ROTATING VELOCITY by giving the coordinates of the two points, *a* and *b*.

Set DEFINITION=NODES to define the axis of rotation for TYPE=ROTATING VELOCITY by giving global node numbers for points *a* and *b*.

DRIVING ELSETS

This parameter is relevant only for use with the INTERPOLATE parameter. Include this parameter to indicate that the field (temperature, void ratio, and pore pressure only) is interpolated from a user-specified element set from the previous analysis to a user-specified node set in the current job. This parameter is used to eliminate mapping ambiguity in cases where element regions in the previous

analysis are close or touching. To accomplish part instance to part instance mapping, define your element and node sets to correspond to the respective instances in the previous and current analysis.

EXTERIOR TOLERANCE

This parameter is relevant only for use with the INTERPOLATE parameter. Set this parameter equal to the fraction of the average element size by which nodes of the current model might lie outside the region of the elements of the model in the output database specified by the FILE parameter. The default value is 0.05.

If both tolerance parameters are specified, Abaqus uses the tighter tolerance.

FILE

Set this parameter equal to the name of the results (`.fil`) file or output database (`.odb`) file from which initial field variable, void ratio, pore pressure, or pressure stress data are to be read.

Set this parameter equal to the name of output database (`.odb` or `.sim`) file from which initial stress is to be read.

This parameter must be used in conjunction with the STEP and INC parameters. For more information, see *File Extension Definitions*.

This parameter is ignored if the initial field values are specified with the **EXTERNAL FIELD* option.

FULL TENSOR

Include this parameter if the kinematic shift tensor (backstress) components are specified using the full tensor format, regardless of the element type to which the initial conditions are applied.

This parameter can be used only in conjunction with the parameter TYPE=HARDENING. It cannot be used if any of the parameters REBAR, SECTION POINTS, or USER has been used.

GEOSTATIC

This parameter is used only with TYPE=STRESS to specify that a geostatic stress state, in which stresses vary with elevation only, is being defined.

INC

This parameter is used only with the FILE parameter. If this parameter is omitted, the initial conditions will be read from the last increment of the step specified on the STEP parameter or from the last step if the STEP parameter is omitted.

The parameter specifies the increment in the results (`.fil`) file of a previous Abaqus analysis from which prescribed fields of TYPE=FIELD, TYPE=PRESSURE STRESS, or TYPE=TEMPERATURE are to be read. It can also specify the increment in the output database (`.odb`) file of a previous Abaqus analysis from which prescribed fields of TYPE=FIELD, TYPE=PORE PRESSURE, TYPE=RATIO, or TYPE=TEMPERATURE are to be read. For TYPE=STRESS, this parameter specifies the increment in the output database (`.odb` or `.sim`) file of a previous Abaqus analysis from which initial stress is to be read.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

INTERPOLATE

Include this parameter in conjunction with the FILE, STEP, and INC parameters to indicate that the nodal temperatures being read into the temperature field or the scalar nodal output variable being read into a predefined field needs to be interpolated between dissimilar meshes. This feature is used to read nodal values from an output database (`.odb`) file generated during a previous Abaqus analysis.

For void ratio initialization from a previous output database file, this parameter is automatically activated and the old void ratios from either the element integration points or the element nodes are read and mapped onto the current nodes.

For temperature fields this parameter and the MIDSIDE parameter are mutually exclusive. For temperature fields if the initial analysis uses first-order elements and the current mesh is the same but uses second-order elements, use the MIDSIDE parameter instead. The MIDSIDE parameter is not supported for predefined fields; therefore, the INTERPOLATE parameter is the only option for initializing predefined fields using scalar nodal output values from a dissimilar mesh.

MIDSIDE

This parameter applies only to Abaqus/Standard analyses.

Include this parameter in conjunction with the FILE, STEP, and INC parameters to indicate that midside node temperatures in second-order elements are to be interpolated from corner node temperatures. This feature is used to read temperatures from a results (`.fil`) or output database (`.odb`) file generated during a heat transfer analysis using first-order elements. This parameter and the INTERPOLATE parameter are mutually exclusive.

NORMAL

This parameter applies only to Abaqus/Standard analyses.

This parameter can be used only with TYPE=CONTACT to specify that the nodes in the node set (or the contact pair, if a node set is not defined) are bonded only in the normal (contact) direction and are allowed to move freely in the tangential direction. If the nodes in the node set (or the contact pair) are to be bonded in all directions, this parameter should be omitted.

NUMBER BACKSTRESSES

Set this parameter equal to the number of backstresses. This parameter can be used only in conjunction with TYPE=HARDENING. The default number of backstresses is 0 if the initial conditions are defined with the **EXTERNAL FIELD* option; otherwise, the default number is 1. The maximum allowed is 10.

OUTPUT VARIABLE

This parameter is required when TYPE=FIELD and the FILE parameter references an output database.

Set this parameter equal to the scalar nodal output variable that will be read from an output database and used to initialize a specified predefined field. For a list of scalar nodal output variables that can be used to initialize a predefined field, see *Predefined Fields*.

REBAR

This parameter can be used with TYPE=DAMAGE INITIATION, TYPE=HARDENING, TYPE=PLASTIC STRAIN, TYPE=SOLUTION, or TYPE=STRESS.

When used with TYPE=DAMAGE INITIATION, it specifies the initial value of the damage initiation measure in the rebar.

When used with TYPE=HARDENING, it specifies that rebars are in a work hardened state, with initial equivalent plastic strain and, possibly, initial backstress.

When used with TYPE=PLASTIC STRAIN, it specifies the initial plastic strain in the rebar.

When used with TYPE=SOLUTION, it specifies that rebars are being assigned initial solution-dependent state variable values.

When used with TYPE=STRESS, it specifies that prestress in rebars is being defined. When performing an Abaqus/Standard analysis, some iteration will usually be needed in this case to

establish a self-equilibrating stress state in the rebar and concrete. The **PRESTRESS HOLD* option can be useful for post-tensioning simulations (see *Defining Rebar as an Element Property*).

SECTION POINTS

This parameter is used only with TYPE=DAMAGE INITIATION, TYPE=HARDENING, TYPE=PLASTIC STRAIN, and TYPE=STRESS to specify damage initiation measures, hardening variables, plastic strains, and stresses at individual section points through the thickness of a shell element. This parameter can be used only when shell properties are defined using the **SHELL SECTION* option. It cannot be used when properties are defined using the **SHELL GENERAL SECTION* option.

SECTION SPECIFICATION

This parameter is relevant only for TYPE=TEMPERATURE and TYPE=FIELD.

Set SECTION SPECIFICATION=UNIFORM to specify uniform temperatures and field variables.

The INPUT and VARIABLE parameters can be used with this parameter.

STEP

This parameter is used only with the FILE parameter. If this parameter is omitted, the initial conditions will be read from the last step.

The parameter specifies the step in the results (`.fil`) file of a previous Abaqus analysis from which prescribed fields of TYPE=FIELD, TYPE=PRESSURE STRESS, or TYPE=TEMPERATURE are to be read. It can also specify the step in the output database (`.odb`) file of a previous Abaqus analysis from which prescribed fields of TYPE=FIELD, TYPE=PORE PRESSURE, TYPE=RATIO, or TYPE=TEMPERATURE are to be read. For TYPE=STRESS, this parameter specifies the step in the output database (`.odb` or `.sim`) file of a previous Abaqus analysis from which initial stress is to be read.

UNBALANCED STRESS

This parameter applies only to Abaqus/Standard analyses.

This parameter is used only with TYPE=STRESS.

Set UNBALANCED STRESS=RAMP (default) if the unbalanced stress is to be resolved linearly over the step.

Set UNBALANCED STRESS=STEP if the unbalanced stress is to be resolved in the first increment.

If the **INITIAL CONDITIONS* option is repeated multiple times in an input file, the UNBALANCED STRESS parameter setting on the last occurrence prevails.

USER

In Abaqus/Standard, this parameter can be used with TYPE=HARDENING, TYPE=PORE PRESSURE, TYPE=RATIO, TYPE=SOLUTION, or TYPE=STRESS.

When used with TYPE=HARDENING, it specifies that the initial conditions on equivalent plastic strain and, if relevant, backstress tensor are to be given via user subroutine *HARDINI*.

When used with TYPE=PORE PRESSURE, it specifies that initial pore pressures are to be given via user subroutine *UPOREP*.

When used with TYPE=RATIO, it specifies that initial void ratios are to be given via user subroutine *VOIDRI*.

When used with TYPE=SOLUTION, it specifies that initial solution-dependent state variable fields are to be given via user subroutine *SDVINI*.

When used with TYPE=STRESS, it specifies that stresses are to be given in user subroutine *SIGINI*.

In Abaqus/Explicit, this parameter can be used only with TYPE=SOLUTION. It specifies that initial solution-dependent state variable fields are to be given in user subroutine *VSDVINI*.

VARIABLE

This parameter is used only with TYPE=FIELD when it is used to define the field variable number. The default is VARIABLE=1. Any number of separate field variables can be used: each must be numbered consecutively (1, 2, 3, etc.)

No data lines are required if the initial conditions are read from a file (the FILE parameter is included) unless the DRIVING ELSETS parameter is used; if the initial conditions are defined with the *EXTERNAL FIELD option; or if the initial conditions are defined in a user subroutine (the USER parameter is included)

Data line for TYPE=ACOUSTIC STATIC PRESSURE:

First (and only) line:

1. Node set or node number.
2. Hydrostatic pressure at the first reference point.
3. *X*-coordinate of the first reference point.
4. *Y*-coordinate of the first reference point.
5. *Z*-coordinate of the first reference point.
6. Hydrostatic pressure at the second reference point.
7. *X*-coordinate of the second reference point.
8. *Y*-coordinate of the second reference point.
9. *Z*-coordinate of the second reference point.

Data lines for TYPE=ACTIVATION:

First line:

1. Element set or element number.
2. Initial volume fraction of material in the element.

Repeat this data line as often as necessary to define the initial volume fraction of material in various elements or element sets.

Data lines for TYPE=BINDERVF:

First line:

1. Element set or element number.
2. Initial volume fraction of the binder phase.

Repeat this data line as often as necessary to define the initial volume fraction of the binder phase at various elements or element sets.

Data lines for TYPE=CONCENTRATION:

First line:

1. Node set or node number.
2. Initial normalized concentration value at the node.

Repeat this data line as often as necessary to define the initial normalized concentration at various nodes or node sets.

Data lines for TYPE=CONTACT:

First line:

1. Secondary surface name.
2. Main surface name.
3. Name of the node set associated with the secondary surface.

Repeat this data line as often as necessary to define partially bonded surfaces.

Data lines for TYPE=CURE:

First line:

1. Element number or element set label.
2. Initial value of the degree of cure.

Repeat this data line as often as necessary to define the initial degree of cure in various elements or element sets.

Data lines for TYPE=DAMAGE INITIATION, CRITERION=DUCTILE or CRITERION=SHEAR if the REBAR and SECTION POINTS parameters are omitted:

First line:

1. Element number or element set label.
2. Damage initiation measure for either the ductile or the shear damage initiation criterion, ω_D or ω_S.

Repeat this data line as often as necessary to define initial damage initiation measures in various elements or element sets.

Data lines for TYPE=DAMAGE INITIATION, CRITERION=DUCTILE or CRITERION=SHEAR with the REBAR parameter included:

First line:

1. Element number or element set label.
2. Rebar name. If this field is left blank, the initial conditions will be applied to all rebars in the model.
3. Damage initiation measure for either the ductile or the shear damage initiation criterion, ω_D or ω_S.

Repeat this data line as often as necessary to define initial damage initiation measures for rebars in various elements or element sets.

Data lines for TYPE=DAMAGE INITIATION, CRITERION=DUCTILE or CRITERION=SHEAR with the SECTION POINTS parameter included:

First line:

1. Element number or element set label.
2. Section point number.

3. Damage initiation measure for either the ductile or the shear damage initiation criterion, ω_D or ω_S.

Repeat this data line as often as necessary to define initial damage initiation measures in various elements or element sets. The initial damage initiation measures must be defined at all section points within an element.

Data lines for TYPE=DAMAGE INITIATION, CRITERION=MSFLD:

First line:

1. Element number or element set label.
2. Damage initiation measure for the Müschenborn and Sonne forming limit diagram based damage initiation criterion, ω_{MSFLD}.
3. Ratio of the principal strain rates, α.

Repeat this data line as often as necessary to define initial damage initiation measures in various elements or element sets.

Data lines for TYPE=ENRICHMENT:

First line:

1. Element number or element set label.
2. Relative position of the node forming the element connectivity.
3. Name of the enriched feature specified on the **ENRICHMENT* option.
4. Value of first signed distance function.
5. Value of second signed distance function. Leave this entry blank if only the first signed distance function is needed.

Repeat this data line as often as necessary to define initial signed distance functions in various elements or element sets. The signed distance functions must be defined at all nodes within an element.

Data lines for TYPE=ESDV:

First line:

1. Element number or element set label.
2. Value of first element solution-dependent variable.
3. Value of second element solution-dependent variable.
4. Etc., up to seven element solution-dependent variables.

Subsequent lines (only needed if more than seven element solution-dependent variables exist in the model):

1. Value of eighth element solution-dependent variable.
2. Etc., up to eight element solution-dependent variables per line.

The total number of variables that Abaqus expects to read for any element is based on the maximum number of element solution-dependent variables for all the elements in the model. Therefore, it might be necessary to leave blank data lines for some elements if any other element in the model has more element solution-dependent variables. These trailing initial values will be zero and will not be used in the analysis.

Repeat this set of data lines as often as necessary to define initial values of element solution-dependent variables for various elements or element sets.

Data lines for TYPE=FIELD, VARIABLE=*n*:

First line:

1. Node set or node number.
2. Initial value of this field variable at the first temperature point. For shells and beams several values (or a value and the field variable gradients across the section) can be given at each node (see *About Beam Modeling* as well as *About Shell Elements*). For heat transfer shells the field variables at each temperature point through the shell thickness must be specified. The number of values depends on the (maximum) number of points specified on the data lines associated with the **SHELL SECTION* options.
3. Initial value of this field variable at the second temperature point.
4. Etc., up to seven values.

Subsequent lines (only needed if initial values must be specified at more than seven temperature points at any node):

1. Eighth initial value of this field variable at this temperature point.
2. Etc., up to eight initial values per line.

The total number of field variables that Abaqus expects to read for any node is based on the maximum number of field variable values for all the nodes in the model. Therefore, it might be necessary to leave blank data lines for some nodes if any other node in the model has more than seven field variable points. These trailing initial values will be zero and will not be used in the analysis.

Repeat this set of data lines as often as necessary to define initial temperatures at various nodes or node sets.

Data lines for TYPE=FIELD, SECTION SPECIFICATION=UNIFORM:

First line:

1. Node set, node number, or blank.
2. Uniform field variable value.

Repeat this data line as often as necessary to define uniform field variables at various nodes and node sets. If you omit both node numbers and node sets (that is, enter a blank space), the specified uniform field variable is applied to all nodes in the model.

Data lines for TYPE=FLUID ELECTRIC POTENTIAL:

First line:

1. Node set or node number.
2. Initial value of electric potential in the electrolyte at the node.

Repeat this data line as often as necessary to define the initial electric potential in the electrolyte at various nodes or node sets.

Data lines for TYPE=FLUID PRESSURE:

First line:

1. Node set or node number of fluid cavity reference node.
2. Fluid pressure.

Repeat this data line as often as necessary to define initial fluid pressure for various fluid-filled cavities.

Data lines to prescribe initial equivalent plastic strain or backstresses using TYPE=HARDENING if the REBAR, SECTION POINTS, and USER parameters are omitted:

First line:

1. Element number or element set label.
2. Initial equivalent plastic strain, $\bar{\varepsilon}^{pl}\big|_0$.
3. First value of the initial first backstress, $\alpha 1^0_{11}$.
4. Second value of the initial first backstress, $\alpha 1^0_{22}$.
5. Etc., up to six backstress components.

Subsequent lines (only needed if the NUMBER BACKSTRESSES parameter has a value greater than one):

1. First value of the initial second backstress, $\alpha 2^0_{11}$.
2. Second value of the initial second backstress, $\alpha 2^0_{22}$.
3. Etc., backstress components for each backstress must be specified on a separate data line.

The backstress components are relevant only for the kinematic hardening models. Give the backstress components as defined for this element type in *About the Element Library*. Values given on the data lines are applied uniformly over the element. In any element for which an **ORIENTATION* option applies, backstresses must be given in the local system (*Orientations*).

Repeat this set of data lines as often as necessary to define the hardening parameters for various elements or element sets.

Data lines to prescribe initial volumetric compacting plastic strain for the crushable foam model using TYPE=HARDENING:

First line:

1. Element number or element set label.
2. Initial volumetric compacting plastic strain, $-\varepsilon^{pl}_{\text{vol}}$.

Repeat this data line as often as necessary to define the initial volumetric compacting plastic strain for various elements or element sets.

Data lines for TYPE=HARDENING, REBAR:

First line:

1. Element number or element set label.
2. Rebar name. If this field is left blank, the initial conditions will be applied to all rebars in the model.
3. Initial equivalent plastic strain, $\bar{\varepsilon}^{pl}\big|_0$.
4. Initial first backstress, $\alpha 1^0_{11}$. (Only relevant for the kinematic hardening models.)

Subsequent lines (only needed if the NUMBER BACKSTRESSES parameter has a value greater than one):

1. Initial second backstress, $\alpha 2^0_{11}$. (Only relevant for the kinematic hardening models.)

2. Etc., backstress components for each backstress must be specified on a separate data line.

Repeat this set of data lines as often as necessary to define the hardening parameters for rebars in various elements or element sets.

Data lines for TYPE=HARDENING, SECTION POINTS:

First line:

1. Element number or element set label.
2. Section point number.
3. Initial equivalent plastic strain, $\bar{\varepsilon}^{pl}\big|_0$.
4. First value of the first initial backstress, $\alpha 1^0_{11}$. (Only relevant for the kinematic hardening models.)
5. Second value of the first initial backstress, $\alpha 1^0_{22}$.
6. Third value of the first initial backstress, $\alpha 1^0_{12}$.

Subsequent lines (only needed if the NUMBER BACKSTRESSES parameter has a value greater than one):

1. First value of the initial second backstress, $\alpha 2^0_{11}$.
2. Second value of the initial second backstress, $\alpha 2^0_{22}$.
3. Etc., backstress components for each backstress must be specified on a separate data line.

The backstress components are relevant only for the kinematic hardening model. Give the backstress components as defined for this element type in *About the Element Library*. In any element for which an **ORIENTATION* option applies, the backstress components must be given in the local system (*Orientations*).

Repeat this set of data lines as often as necessary to define the hardening parameters in various elements or element sets. The hardening parameters must be defined at all section points within an element.

Data lines for TYPE=INITIAL GAP:

First line:

1. Element number or element set label.
2. The material initial damage variable, D, at the first integration point.
3. The material initial damage variable, D, at the second integration point.
4. The material initial damage variable, D, at the third integration point.
5. The material initial damage variable, D, at the fourth integration point.

Repeat this data line as often as necessary to identify various elements or element sets. Assigning the material initial damage variables at the integration points is optional. If no initial damage variables are assigned, the elements are considered fully damaged; that is, $D = 1.0$. If you assign an initial damage variable to any of the integration points and leave the other fields blank, a value of $D = 0.0$ is assigned to the integration points of the blank fields.

Data lines for TYPE=ION CONCENTRATION:

First line:

1. Node set or node number.

2. Initial ion concentration value at the node.

Repeat this data line as often as necessary to define the initial ion concentrations in the electrolyte at various nodes or node sets.

Data lines for TYPE=MASS FLOW RATE:

First line:

1. Node set or node number.
2. Initial mass flow rate per unit area in the *x*-direction or total initial mass flow rate in the cross-section for one-dimensional elements.
3. Initial mass flow rate per unit area in the *y*-direction (not needed for nodes associated with one-dimensional convective flow elements).
4. Initial mass flow rate per unit area in the *z*-direction (not needed for nodes associated with one-dimensional convective flow elements).

Repeat this data line as often as necessary to define mass flow rates at various nodes or node sets.

Data lines for TYPE=NODE REF COORDINATE:

First line:

1. Node number.
2. *X*-coordinate of the node.
3. *Y*-coordinate of the node.
4. *Z*-coordinate of the node.

Repeat this data line as often as necessary to define the initial coordinates of the mesh using nodal coordinates.

Data lines to prescibe initial species concentration using TYPE=PARTICLE SPECIES CONCENTRATION:

First line:

1. Element set or element number.
2. Initial species concentration for the first particle. You must specify a positive, nonzero value.
3. Initial species concentration for the second particle, if the second particle is defined under the material definition used for this element set or element. You can leave this data item blank if you have only a single particle. The initial particle species concentration must be a positive, nonzero value.
4. Initial species concentration for the third particle, if the second particle is defined under the material definition used for this element set or element. You can leave this data item blank if you have only a single particle or only two particles. The initial particle species concentration must be a positive, nonzero value.

Repeat this data line as often as necessary to define the initial particle species concentration at various elements or element sets.

Data lines to prescribe initial plastic strains using TYPE=PLASTIC STRAIN if the REBAR and SECTION POINTS parameters are omitted:

First line:

1. Element number or element set label.

2. Value of first plastic strain component, ε_{11}^{pl}.
3. Value of second plastic strain component, ε_{22}^{pl}.
4. Etc., up to six plastic strain components.

Give the plastic strain components as defined for this element type in *About the Element Library*. Values given on the data lines are applied uniformly over the element. In any element for which an **ORIENTATION* option applies, the plastic strains must be given in the local system (*Orientations*).

Repeat this data line as often as necessary to define initial plastic strains in various elements or element sets.

Data lines for TYPE=PLASTIC STRAIN, REBAR:

First line:

1. Element number or element set label.
2. Rebar name. If this field is left blank, the initial conditions will be applied to all rebars in the model.
3. Initial plastic strain value.

Repeat this data line as often as necessary to define the initial plastic strain in the rebars of various elements or element sets.

Data lines for TYPE=PLASTIC STRAIN, SECTION POINTS:

First line:

1. Element number or element set label.
2. Section point number.
3. Value of first plastic strain component, ε_{11}^{pl}.
4. Value of second plastic strain component, ε_{22}^{pl}.
5. Value of third plastic strain component, ε_{12}^{pl}.

Give the initial plastic strain components as defined for this element type in *About the Element Library*. In any element for which an **ORIENTATION* option applies, the plastic strain components must be given in the local system (*Orientations*).

Repeat this data line as often as necessary to define initial plastic strains in various elements or element sets. Plastic strains must be defined at all section points within an element.

Data lines for TYPE=PORE PRESSURE if the USER parameter is omitted:

First line:

1. Node set or node number.
2. First value of fluid pore pressure, u_w.
3. Vertical coordinate corresponding to the above value.
4. Second value of fluid pore pressure, u_w.
5. Vertical coordinate corresponding to the above value.

Omit the elevation values and the second pore pressure value to define a constant pore pressure distribution.

Repeat this data line as often as necessary to define the fluid pore pressure at various nodes or node sets.

Data lines for TYPE=PORE PRESSURE, FILE=*file*, STEP=*step*, INC=*inc*, INTERPOLATE, DRIVING ELSETS:

First line:

1. Element set, node set.

Repeat this data line as often as necessary. The node set identified on the data lines will be assigned values from the element set in the output database (`.odb`) file. If a duplicate node is defined on a subsequent data line, it will be removed from the subsequent void ratio mapping and printed out to the data (`.dat`) file.

Data lines for TYPE=POROSITY:

First line:

1. Element number or element set label.
2. Initial porosity.

Repeat this data line as often as necessary to define initial porosity in various elements or element sets.

Data lines for TYPE=PRESSURE STRESS:

First line:

1. Node set or node number.
2. Equivalent pressure stress, p.

Repeat this data line as often as necessary to define the pressure stress at various nodes or node sets.

Data lines for TYPE=RATIO if the USER parameter is omitted:

First line:

1. Node set or node number.
2. First value of void ratio.
3. Vertical coordinate corresponding to the above value.
4. Second value of void ratio.
5. Vertical coordinate corresponding to the above value.

Omit the elevation values and the second void ratio value to define a constant void ratio distribution.

Repeat this data line as often as necessary to define void ratios at various nodes or node sets.

Data lines for TYPE=RATIO, FILE=*file*, STEP=*step*, INC=*inc*, INTERPOLATE, DRIVING ELSETS:

First line:

1. Element set, node set.

Repeat this data line as often as necessary. The node set identified on the data lines will be assigned values from the element set in the output database (`.odb`) file. If a duplicate node is defined on a

subsequent data line, it will be removed from the subsequent void ratio mapping and printed out to the data (`.dat`) file.

Data lines for TYPE=REF COORDINATE:

First line:

1. Element number.
2. *X*-coordinate of the first node.
3. *Y*-coordinate of the first node.
4. *Z*-coordinate of the first node.
5. *X*-coordinate of the second node.
6. *Y*-coordinate of the second node.
7. *Z*-coordinate of the second node.

Second line:

1. *X*-coordinate of the third node.
2. *Y*-coordinate of the third node.
3. *Z*-coordinate of the third node.
4. *X*-coordinate of the fourth node.
5. *Y*-coordinate of the fourth node.
6. *Z*-coordinate of the fourth node.

Repeat this pair of data lines as often as necessary to define the reference mesh in various elements. The order of the nodal coordinates must be consistent with the element connectivity.

Data lines for TYPE=RELATIVE DENSITY:

First line:

1. Node set or node number.
2. Initial relative density.

Repeat this data line as often as necessary to define initial relative density at various nodes or node sets.

Data lines for TYPE=ROTATING VELOCITY, DEFINITION=COORDINATES:

First line:

1. Node set or node number.
2. Angular velocity about the axis defined from point *a* to point *b*, where the coordinates of *a* and *b* are given below.
3. Global *X*-component of translational velocity.
4. Global *Y*-component of translational velocity.
5. Global *Z*-component of translational velocity.

Second line:

1. Global *X*-component of point *a* on the axis of rotation.
2. Global *Y*-component of point *a* on the axis of rotation.
3. Global *Z*-component of point *a* on the axis of rotation.
4. Global *X*-component of point *b* on the axis of rotation.
5. Global *Y*-component of point *b* on the axis of rotation.
6. Global *Z*-component of point *b* on the axis of rotation.

Repeat this pair of data lines as often as necessary to define the angular and translational velocities at various nodes or node sets.

Data lines for TYPE=ROTATING VELOCITY, DEFINITION=NODES:

First line:

1. Node set or node number.
2. Angular velocity about the axis defined from point *a* to point *b*, where the coordinates of *a* and *b* are given below.
3. Global *X*-component of translational velocity.
4. Global *Y*-component of translational velocity.
5. Global *Z*-component of translational velocity.

Second line:

1. Node number of the node at point *a*.
2. Node number of the node at point *b*.

Repeat this pair of data lines as often as necessary to define the angular and translational velocities at various nodes or node sets.

Data lines for TYPE=SATURATION:

First line:

1. Node set or node number.
2. Saturation value, *s*. Default is 1.0.

Repeat this data line as often as necessary to define saturation at various nodes or node sets.

Data lines for TYPE=SOLIDVF:

First line:

1. Element set or element number.
2. Initial volume fraction of the solid phase.

Repeat this data line as often as necessary to define the initial volume fraction of the solid phase at various elements or element sets.

Data lines for TYPE=SLURRYVF:

First line:

1. Node set or node number.
2. Initial slurry concentration value at the node.

Repeat this data line as often as necessary to define the initial volumetric slurry concentration at various nodes or node sets.

Data lines for TYPE=SOLID ELECTRIC POTENTIAL:

First line:

1. Node set or node number.
2. Initial value of electric potential in the solid phase at the node.

Repeat this data line as often as necessary to define the initial electric potential in the solid phase at various nodes or node sets.

Data lines for TYPE=SPECIES CONCENTRATION:

First line:

1. Node set or node number.
2. Initial species concentration value at the node.

Repeat this data line as often as necessary to define the initial species concentrations at various nodes or node sets.

Data lines for TYPE=SOLUTION if the USER and REBAR parameters are omitted:

First line:

1. Element number or element set label.
2. Value of first solution-dependent state variable.
3. Value of second solution-dependent state variable.
4. Etc., up to seven solution-dependent state variables.

Subsequent lines (only needed if more than seven solution-dependent state variables exist in the model):

1. Value of eighth solution-dependent state variable.
2. Etc., up to eight solution-dependent state variables per line.

The total number of variables that Abaqus expects to read for any element is based on the maximum number of solution-dependent state variables for all the elements in the model. Therefore, it may be necessary to leave blank data lines for some elements if any other element in the model has more solution-dependent state variables. These trailing initial values will be zero and will not be used in the analysis. Values given on the data lines will be applied uniformly over the element.

Repeat this set of data lines as often as necessary to define initial values of solution-dependent state variables for various elements or element sets.

Data lines for TYPE=SOLUTION, REBAR:

First line:

1. Element number or element set label.
2. Rebar name. If this field is left blank, the solution-dependent state variables are applied to all rebars in these elements.
3. Value of first solution-dependent state variable.
4. Value of second solution-dependent state variable.
5. Etc., up to six solution-dependent state variables.

Subsequent lines (only needed if more than six solution-dependent state variables exist in the model):

1. Value of seventh solution-dependent state variable.
2. Etc., up to eight solution-dependent state variables per line.

The total number of variables that Abaqus expects to read for any element is based on the maximum number of solution-dependent state variables for all the elements in the model. Therefore, it might be necessary to leave blank data lines for some elements if any other element in the model has more solution-dependent state variables. These trailing initial values will be zero and will not be used in the analysis. Values given on the data lines will be applied uniformly over the element.

Repeat this set of data lines as often as necessary to define initial values of solution-dependent state variables for various elements or element sets.

Data lines for TYPE=SPECIFIC ENERGY:

First line:

1. Element number or element set label.
2. Initial specific energy.

Repeat this data line as often as necessary to define initial specific energy in various elements or element sets.

Data lines for TYPE=SPUD EMBEDMENT:

First line:

1. Element set or element number.
2. Spud can embedment, ν_i.

Repeat this data line as often as necessary to define initial embedment for various elements or element sets.

Data lines for TYPE=SPUD PRELOAD:

First line:

1. Element set or element number.
2. Spud can preload, $V_c^{(i)}$.

Repeat this data line as often as necessary to define initial preload for various elements or element sets.

Data lines for TYPE=STRESS if the GEOSTATIC, REBAR, SECTION POINTS, and USER parameters are omitted:

First line:

1. Element number or element set label.
2. Value of first (effective) stress component, axial force when used with the **BEAM GENERAL SECTION* or **FRAME SECTION* options, or direct membrane force per unit width in the local 1-direction when used with the **SHELL GENERAL SECTION* option.
3. Value of second stress component.
4. Etc., up to six stress components.

Give the stress components as defined for this element type in *About the Element Library*. Stress values given on data lines are applied uniformly and equally over all integration points of the element. In any element for which an **ORIENTATION* option applies, the stresses must be given in the local system (*Orientations*).

Repeat this data line as often as necessary to define initial stresses in various elements or element sets.

Data lines for TYPE=STRESS, GEOSTATIC:

First line:

1. Element number or element set label.
2. First value of vertical component of (effective) stress.
3. Vertical coordinate corresponding to the above value.
4. Second value of vertical component of (effective) stress.
5. Vertical coordinate corresponding to the above value.
6. First coefficient of lateral stress. This coefficient defines the x-direction stress components.
7. Second coefficient of lateral stress. This coefficient defines the y-direction stress component in three-dimensional cases and the thickness or hoop direction component in plane or axisymmetric cases. If this value is omitted, it is assumed to be the same as the first lateral stress coefficient given in the previous field.

Repeat this data line as often as necessary to define an initial geostatic stress state in various elements or element sets.

Data lines for TYPE=STRESS, REBAR:

First line:

1. Element number or element set label.
2. Rebar name. If this field is left blank, the stress is applied to all rebars in these elements.
3. Prestress value.

Repeat this data line as often as necessary to define initial stress in the rebars of various elements or element sets.

Data lines for TYPE=STRESS, SECTION POINTS:

First line:

1. Element number or element set label.
2. Section point number.
3. Value of first stress component.
4. Value of second stress component.
5. Etc., up to three stress components.

Give the stress components as defined for this element type in *About the Element Library*. Stress values given on data lines are applied uniformly over the element. In any element for which an **ORIENTATION* option applies, the stresses must be given in the local system (*Orientations*).

Repeat this data line as often as necessary to define initial stresses in various elements or element sets. Stresses must be defined at all section points within an element.

Data lines for TYPE=TEMPERATURE:

First line:

1. Node set or node number.
2. First initial temperature value at the node or node set. For shells and beams several values (or a value and the temperature gradients across the section) can be given at each node (see *Using a*

Beam Section Integrated during the Analysis to Define the Section Behavior, *Using a General Beam Section to Define the Section Behavior*, *Using a Shell Section Integrated during the Analysis to Define the Section Behavior*, and *Using a General Shell Section to Define the Section Behavior*). For heat transfer shells the temperature at each point through the shell thickness must be specified. The number of values depends on the (maximum) number of points specified on the data lines associated with the **SHELL SECTION* options.

3. Second initial temperature value at the node or node set.
4. Etc., up to seven initial temperature values at this node or node set.

Subsequent lines (only needed if there are more than seven temperature values at any node):

1. Eighth initial temperature value at this node or node set.
2. Etc., up to eight initial temperature values per line.

If more than seven temperature values are needed at any node, continue on the next line. The total number of temperatures that Abaqus expects to read for any node is based on the maximum number of temperature values of all the nodes in the model. Therefore, it might be necessary to leave blank data lines for some nodes if any other node in the model has more than seven temperature points. These trailing initial values will be zero and will not be used in the analysis.

Repeat this data line (or set of lines) as often as necessary to define initial temperatures at various nodes or node sets.

Data lines for TYPE=TEMPERATURE, FILE=*file*, STEP=*step*, INC=*inc*, INTERPOLATE, DRIVING ELSETS:

First line:

1. Element set, node set.

Repeat this data line as often as necessary. The node set identified on the data lines will be assigned values from the element set in the output database (`.odb`) file. If a duplicate node is defined on a subsequent data line, it will be removed from the subsequent temperature mapping and printed out to the data (`.dat`) file.

Data lines for TYPE=TEMPERATURE, SECTION SPECIFICATION=UNIFORM:

First line:

1. Node set, node number, or blank.
2. Uniform temperature value.

Repeat this data line as often as necessary to define uniform temperatures at various nodes and node sets. If you omit both node numbers and node sets (that is, enter a blank space), the specified uniform temperature is applied to all nodes in the model.

Data lines for TYPE=UNFOLD COORDINATE:

First line:

1. Node number.
2. *X*-coordinate of the first node.
3. *Y*-coordinate of the first node.
4. *Z*-coordinate of the first node.

Repeat this data line as often as necessary to define the initial coordinates at various nodes.

Data lines for TYPE=VELOCITY:

First line:

1. Node set or node number.
2. Degree of freedom.
3. Value of initial velocity.

Repeat this data line as often as necessary to define the initial velocity at various nodes or node sets.

Data lines for TYPE=VOLUME FRACTION:

First line:

1. Eulerian element number or element set label.
2. Name of the material instance as defined in the **EULERIAN SECTION*.
3. Initial volume fraction, EVF, for this material (0.0 < EVF 1.0). EVF=0.0 indicates that none of this material is present in the element, while EVF=1.0 indicates that the element is completely full of this material.

Repeat this data line as often as necessary to define the initial geometry of all Eulerian material instances. An element might appear in more than one data line if it initially contains more than one material. Elements are filled incrementally by reading the data lines in the input file from bottom to top; once the volume fraction for an element reaches one, additional volume fractions assigned to that element are ignored. If the final volume fraction for an element is less than one, the remainder of that element is filled with void; similarly, uninitialized elements are filled with void.

*INSTANCE: Begin an instance definition.

This option is used to instance a part within an assembly. It must be used in conjunction with the *ASSEMBLY and *END INSTANCE options. If the instance is not imported from a previous analysis, the *INSTANCE option must be used in conjunction with the *PART option.

When importing a part instance from a previous analysis, the *INSTANCE option must be used in conjunction with the *IMPORT option. You can define a new position for an imported instance and import an instance more than once.

When importing a part instance from a previous analysis, the *INSTANCE option can be used in conjunction with the *IMPORT SURFACE option to import surfaces selectively or to import all surfaces by default.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Assembly

Abaqus/CAE: *Assembly module* for part instances not imported from a previous analysis; *Load module* for part instances imported from a previous analysis

References:

- *Assembly Definition*
- *About Transferring Results between Abaqus Analyses*
- *END INSTANCE
- *IMPORT
- *IMPORT SURFACE

Required parameters if the instance is not imported from a previous analysis:

NAME

Set this parameter equal to a label that will be used to refer to the instance.

PART

Set this parameter equal to the name of the part being instanced.

Required parameter if the instance is to be imported once from a previous analysis:

INSTANCE

Set this parameter equal to the name of the instance to be imported from the previous analysis.

Required parameters if the instance is to be imported more than once from a previous analysis:

NAME

Set this parameter equal to a new label that will be used to refer to the instance.

INSTANCE

Set this parameter equal to the old name of the instance to be imported from the previous analysis.

Required parameter if importing instances from multiple previous analyses (Abaqus/Explicit only); optional parameter if importing instances from a single previous analysis:

LIBRARY

Set this parameter equal to a value that specifies the previous analysis from which to import the instance. You can specify the name of the previous analysis or the name including a full path. If no path is specified, all input files and result files from the previous analyses must reside in the current (working) directory.

When importing from a single previous analysis, if the LIBRARY parameter is omitted, you must specify the job name of the previous analysis on the command line using the **oldjob** option (see *Abaqus/Standard and Abaqus/Explicit Execution*). If both methods are used, the LIBRARY parameter takes precedence over the command line specification.

When importing from multiple previous analyses, set this parameter equal to the job name. You should not use the **oldjob** option on the command line.

Data line to translate an instance:

First (and only) line:

1. Value of the translation to be applied in the *X*-direction.
2. Value of the translation to be applied in the *Y*-direction.
3. Value of the translation to be applied in the *Z*-direction.

Data lines to translate and/or rotate an instance:

First line:

1. Value of the translation to be applied in the *X*-direction.
2. Value of the translation to be applied in the *Y*-direction.
3. Value of the translation to be applied in the *Z*-direction.

Enter values of zero to apply a pure rotation.

Second line:

1. *X*-coordinate of point *a* on the axis of rotation (see *Figure 1*).
2. *Y*-coordinate of point *a* on the axis of rotation.
3. *Z*-coordinate of point *a* on the axis of rotation.
4. *X*-coordinate of point *b* on the axis of rotation.
5. *Y*-coordinate of point *b* on the axis of rotation.
6. *Z*-coordinate of point *b* on the axis of rotation.
7. Angle of rotation about the axis *a*–*b*, in degrees.

If both translation and rotation are specified, translation is applied before rotation.

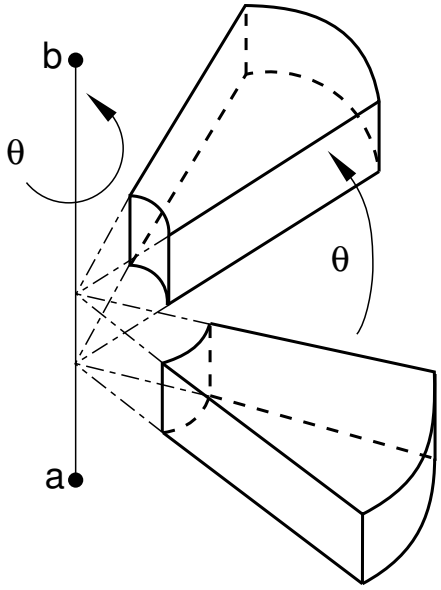


Figure 1: Rotation of an instance.

*INTEGRATED OUTPUT: Specify variables integrated over a surface to be written to the output database.

This option is used to write integrated quantities over a surface, such as the total force transmitted across a surface, to the output database. It must be used in conjunction with the *OUTPUT, HISTORY option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Output to the Output Database*
- *Integrated Output Section Definition*
- *OUTPUT
- *SURFACE
- *INTEGRATED OUTPUT SECTION

One of the following mutually exclusive parameters is required when the *INTEGRATED OUTPUT option is used in an Abaqus/Explicit analysis:

SECTION

Set this parameter equal to the name of the *INTEGRATED OUTPUT SECTION (see *Integrated Output Section Definition*) over which this output request is being made.

SURFACE

Set this parameter equal to the name of the surface (see *Element-Based Surface Definition*) over which this output request is being made.

ELSET

Set this parameter equal to the name of the element set over which this output request is being made.

One of the following mutually exclusive parameters is required when the *INTEGRATED OUTPUT option is used in an Abaqus/Standard analysis:

SECTION

Set this parameter equal to the name of the *INTEGRATED OUTPUT SECTION (see *Integrated Output Section Definition*) over which this output request is being made.

SURFACE

Set this parameter equal to the name of the surface (see *Element-Based Surface Definition*) over which this output request is being made.

Optional parameters:

VARIABLE

Set VARIABLE=ALL to indicate that all integrated output variables applicable to this procedure should be written to the output database.

Set VARIABLE=PRESELECT to indicate that the default integrated output variables for the current procedure type should be written to the output database. Additional output variables can be requested on the data lines.

If this parameter is omitted, the integrated output variables for output must be specified on the data line.

Data lines to request integrated output:

First line:

1. Specify the identifying keys for the output variables to be written to the output database. The keys are defined in *Abaqus/Standard Output Variable Identifiers* and *Abaqus/Explicit Output Variable Identifiers*.

Repeat this data line as often as necessary to define the list of variables to be written to the output database.

*INTEGRATED OUTPUT SECTION: Define an integrated output section over a surface with a local coordinate system and a reference point.

This option is used to associate a surface with a coordinate system and/or a reference node to track the average motion of the surface. It can also be used in conjunction with an integrated output request to obtain output of quantities integrated over a surface.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Step module

References:

- *Integrated Output Section Definition*
- *Output to the Output Database*
- **INTEGRATED OUTPUT*
- **SURFACE*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the integrated output section.

SURFACE

Set this parameter equal to the name of the surface (see *Element-Based Surface Definition*) to be associated with the integrated output section.

Optional parameters:

ORIENTATION

Set this parameter equal to the name of an orientation definition (*Orientations*) to define the initial coordinate system for the section. This initial system can be further modified by using the PROJECT ORIENTATION parameter.

If this parameter is omitted, the global coordinate system is used.

POSITION

This parameter is relevant only if the REF NODE parameter is included.

Set POSITION=INPUT (default) if the location of the reference node is to be defined by the user.

Set POSITION=CENTER if the reference node is to be relocated from the user-defined location to the center of the surface in the initial configuration.

PROJECT ORIENTATION

Set PROJECT ORIENTATION=NO (default) if the initial coordinate system of the section should not be projected onto the section surface. If the ORIENTATION parameter is included, this choice results in an initial coordinate system that matches the defined orientation. If an orientation is not specified, the initial coordinate system matches the global coordinate system.

Set PROJECT ORIENTATION=YES if the initial coordinate system of the section should be modified by projecting onto the section surface using the average normal to the surface. If an orientation is not specified, the global coordinate system is projected onto the section surface.

REF NODE

Set this parameter equal to either the node number of the integrated output section reference node or to the name of a node set containing the reference node. If the name of a node set is chosen, the node set must contain exactly one node.

REF NODE MOTION

This parameter is relevant only if the REF NODE parameter is included.

Set REF NODE MOTION=INDEPENDENT (default) if the motion of the reference node is not based on the average motion of the surface.

Set REF NODE MOTION=AVERAGE TRANSLATION if the reference node must translate with the average translation of the surface. This choice is relevant only if the reference node is not connected to the rest of the model.

Set REF NODE MOTION=AVERAGE if the reference node must both translate and rotate with the average motion of the surface. This choice is relevant only if the reference node is not connected to the rest of the model.

There are no data lines associated with this option.

*INTERFACE: Define properties for contact elements.

This option is used to assign element section properties to ITT-, ISL-, IRS-, and ASI-type contact elements.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: *Property module*; supported for two-dimensional, three-dimensional, and axisymmetric acoustic interface elements. Contact elements are not supported.

References:

- *Acoustic Interface Elements*
- *Tube-to-Tube Contact Elements*
- *Slide Line Contact Elements*
- *Rigid Surface Contact Elements*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the ITT-, ISL-, IRS-, and ASI-type contact elements for which properties are being defined.

Optional parameters:

NAME

Set this parameter equal to a label that will be used to refer to this interface definition. The label given can be used to identify this particular interface definition in user subroutines such as *GAPCON*.

Data line for ITT-type elements:

First (and only) line:

1. Radial clearance between the pipes.

There are no data lines for ISL21A and ISL22A elements

There are no data lines for IRS-type elements for use with axisymmetric elements

Data line for ASI1 elements:

First (and only) line:

1. Area associated with the elements.

Enter the direction cosine, in terms of the global Cartesian coordinate system, of the interface normal that points into the acoustic fluid:

5. *X*-direction cosine.
7. *Y*-direction cosine.
9. *Z*-direction cosine.

Data line for ASI-type elements for use with 2D elements:

First (and only) line:

1. Element thickness. The default is unit thickness.

There are no data lines for ASI-type elements for use with axisymmetric elements or 3D elements

*INTERFACE REACTION: Introduce an interface reaction between surfaces.

This option is used to provide an interface reaction to model the chemical kinetics in a lithium metal battery between closely adjacent surfaces. It must be used in conjunction with the *SURFACE INTERACTION option.

Products: Abaqus/Standard

Level: Part, Part instance, Assembly

References:

- *Modeling Solid Electrodes in Lithium Metal Batteries*
- *SURFACE INTERACTION

Required parameters:

TYPE

This option is valid only in a coupled thermal-electrochemical, coupled thermal-electrochemical-structural, or coupled thermal-electrochemical-structural–pore pressure analysis.

Set TYPE=BUTLER-VOLMER to define the Butler-Volmer kinetics at the interface between the solid electrode and separator surfaces.

There are no data lines associated with this option.

*INTERNAL FLUX GENERATION: This option is used to define user-defined internal source terms at the material integration point across multiple physics fields.

This option is used in a material data block to define user-defined internal source that can couple multiple physics fields (user subroutine *UINTFLUXGEN*).

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Coupled Thermal-Electrochemical Analysis*
- *UINTFLUXGEN*

Required parameters:

USER

This parameter is required to activate the usage of user subroutine *UINTFLUXGEN*.

Optional parameters:

PROPERTIES

Set this parameter equal to the number of real (floating point) property values needed as data in user subroutine *UINTFLUXGEN*. The default is PROPERTIES=0.

Data line to define properties (if the value of PROPERTIES is not set to 0):

First line:

1. Give the properties, eight per line.

Repeat this data line as often as necessary.

*ISOTROPIZATION PARAMETERS: Define isotropization parameters for a multiscale material.

This option is used to define the parameters used to compute the isotropic part of the anisotropic modulus of a constituent in a multiscale material. This option can be used only in conjunction with the **CONSTITUENT* option.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- **MEAN FIELD HOMOGENIZATION*
- *Mean-Field Homogenization*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables, in addition to temperature, on which the isotropization parameters depend. If this parameter is omitted, it is assumed that the isotropization parameters are constant or depend only on temperature.

Data lines to define isotropization parameters:

First line:

1. k_p, plastic strain multiplier. The default value is 1.0.
2. k_s, plastic strain shift. The default value is 0.0.
3. k_G, shear multiplier. The default value is 1.0.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of k_p, k_s, and k_G on temperature and other predefined field variables.

*ITS: Define properties for ITS elements.

This option is used to define the properties for ITS-type elements. At least one of the following options must immediately follow this option: **DASHPOT*, **FRICTION*, and/or **SPRING*. If either the *DASHPOT or *SPRING option is included, it must not include the ELSET parameter. The ELSET parameter should be used only on the ITS option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Unsupported; similar functionality (with the exception of friction) is available by modeling connectors.

References:

- *Rigid Surface Contact Elements*
- **DASHPOT*
- **FRICTION*
- **SPRING*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the ITS-type elements for which properties are being defined.

Data line for ITSUNI elements:

First (and only) line:

1. Tube outside diameter.
2. Distance between the parallel support plates on opposite sides of the tube.
3. *X*-direction cosine of the axis of the tube.
4. *Y*-direction cosine of the axis of the tube.
5. *Z*-direction cosine of the axis of the tube.
6. *X*-direction cosine of the normal to either one of the support plates.
7. *Y*-direction cosine of the normal to either one of the support plates.
8. *Z*-direction cosine of the normal to either one of the support plates.

Data line for ITSCYL elements:

First (and only) line:

1. Tube outside diameter.
2. Diameter of the hole in the support plate.
3. *X*-direction cosine of the axis of the tube.
4. *Y*-direction cosine of the axis of the tube.
5. *Z*-direction cosine of the axis of the tube.

J

This section describes each Abaqus input option that begins with J, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- *JOINT
- *JOINT ELASTICITY
- *JOINT PLASTICITY
- *JOINTED MATERIAL
- *JOULE HEAT FRACTION

*JOINT: Define properties for JOINTC elements.

This option is used to define the properties for JOINTC elements. The **DASHPOT* and **SPRING* options must immediately follow this option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Unsupported; similar functionality is available by modeling connectors.

References:

- *Flexible Joint Element*
- **DASHPOT*
- **SPRING*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the JOINTC elements for which properties are being defined.

Optional parameters:

ORIENTATION

Set this parameter equal to the name given to the **ORIENTATION* definition (*Orientations*) that specifies the initial orientation of the local system in the joint.

There are no data lines associated with this option; instead, include *SPRING and *DASHPOT options as needed to define the joint behavior

*JOINT ELASTICITY: Specify elastic properties for elastic-plastic joint elements.

This option is used to define linear elastic moduli for elastic-plastic joint elements. It can be used only in conjunction with the **EPJOINT* option.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- *Elastic-Plastic Joints*
- **EPJOINT*

Required parameters:

MODULI

Set MODULI=SPUD CAN to define spud can moduli. Set MODULI=GENERAL to enter a general elastic modulus.

NDIM

Set NDIM=2 to enter values for a two-dimensional problem. Set NDIM=3 to enter values for a three-dimensional problem.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the moduli. If this parameter is omitted, it is assumed that the moduli are constant or depend only on temperature. See *Material Data Definition* for more information.

Data lines for MODULI=SPUD CAN and NDIM=2:

First line:

1. G_{vv}, equivalent elastic shear modulus for vertical displacements.
2. G_{hh}, equivalent elastic shear modulus for horizontal displacements.
3. G_{rr}, equivalent elastic shear modulus for rotational displacements.
4. ν, Poisson's ratio of the soil.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

Data lines for MODULI=SPUD CAN and NDIM=3:

First line:

1. G_{vv}, equivalent elastic shear modulus for vertical displacements.
2. G_{hh}, equivalent elastic shear modulus for horizontal displacements.
3. G_{rr}, equivalent elastic shear modulus for rotational displacements.
4. ν, Poisson's ratio of the soil.
5. k_t, torsional elastic modulus.
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

Data lines for MODULI=GENERAL and NDIM=2:

First line:

1. k_{1111}.
2. k_{1122}.
3. k_{2222}.
4. k_{1112}.
5. k_{2212}.
6. k_{1212}.
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

Data lines for MODULI=GENERAL and NDIM=3:

First line:

1. k_{1111}.
2. k_{1122}.
3. k_{2222}.
4. k_{1133}.

5. k_{2233}.
6. k_{3333}.
7. k_{1112}.
8. k_{2212}.

Second line:

1. k_{3312}.
2. k_{1212}.
3. k_{1113}.
4. k_{2213}.
5. k_{3313}.
6. k_{1213}.
7. k_{1313}.
8. k_{1123}.

Third line:

1. k_{2223}.
2. k_{3323}.
3. k_{1223}.
4. k_{1323}.
5. k_{2323}.
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

*JOINT PLASTICITY: Specify plastic properties for elastic-plastic joint elements.

This option is used to define the plastic behavior for elastic-plastic joint elements. It can be used only in conjunction with the *EPJOINT option.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- *Elastic-Plastic Joints*
- *EPJOINT

Required parameters:

TYPE

Set TYPE=SAND to specify the model for interaction of spud cans and sand. Set TYPE=CLAY to specify the model for interaction of spud cans and clay. Set TYPE=MEMBER to specify the parabolic model for structural members.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the plasticity property values. If this parameter is omitted, it is assumed that the plasticity property values are constant or depend only on temperature. See *Material Data Definition* for more information.

Data lines for TYPE=SAND:

First line:

1. V_t, yield strength in pure tension.
2. Λ_1, constant for yield surface shape.
3. Λ_2, constant for yield surface shape.
4. ϕ, soil friction angle, in degrees.
5. γ, soil unit weight.
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the plastic behavior as a function of temperature and other predefined field variables.

Data lines for TYPE=CLAY:

First line:

1. s_u, undrained shear strength of the clay.
2. a, hardening parameter.
3. b, hardening parameter.
4. c, hardening parameter.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the plastic behavior as a function of temperature and other predefined field variables.

Data lines for TYPE=MEMBER:

First line:

1. V_c, compressive capacity.
2. V_t, tensile capacity.
3. H_m, horizontal capacity.
4. M_m, moment capacity.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the plastic behavior as a function of temperature and other predefined field variables.

*JOINTED MATERIAL: Specify the jointed material model.

This option is used to define a failure surface and the flow parameters for a single joint system or for bulk material failure in the elastic-plastic model of a jointed material, or it can be used to define shear retention in open joints. Up to three joint systems can be defined for each material point.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

• *Jointed Material Model*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the parameters of the model, in addition to temperature. If this parameter is omitted, it is assumed that the parameters depend only on temperature. See *Material Data Definition* for more information.

JOINT DIRECTION

Set this parameter equal to the name of the **ORIENTATION* used to define the direction of a joint system. This use of the **ORIENTATION* option does not affect the output of components of stress and strain—it only defines the joint orientation in the original configuration. Omit this parameter to give the bulk material failure parameters. The JOINT DIRECTION parameter cannot be used with the SHEAR RETENTION parameter.

NO SEPARATION

Include this parameter to prevent the joint from opening. This parameter must be used in conjunction with the JOINT DIRECTION parameter.

SHEAR RETENTION

Include this parameter to define shear retention in open joints. If this parameter is omitted, zero shear retention is assumed. The SHEAR RETENTION parameter cannot be used with the JOINT DIRECTION parameter.

Data lines defining failure surface and flow parameters (SHEAR RETENTION omitted):

First line:

1. Angle of friction, β, for this system. Give the value in degrees.
2. Dilation angle, ψ, for this system. Give the value in degrees.
3. Cohesion, d, for this system. (Units of FL^{-2}.)
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the failure and surface flow parameters on temperature and other predefined field variables.

Data lines defining the shear retention in open joints (SHEAR RETENTION included):

First line:

1. Fraction of elastic shear modulus retained when joints are open, f_{sr}. This value cannot be less than zero.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the shear retention on temperature and other predefined field variables.

*JOULE HEAT FRACTION: Define the fraction of electric energy released as heat.

This option is used to specify the fraction of dissipated electrical energy released as heat in coupled thermal-electrical and coupled thermal-electrical-structural problems.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Coupled Thermal-Electrical Analysis*
- *Fully Coupled Thermal-Electrical-Structural Analysis*

There are no parameters associated with this option.

Data line to define the Joule heat fraction:

First (and only) line:

1. Fraction of electrical energy released as heat, including any unit conversion factor. The default value is 1.0.

K

This section describes each Abaqus input option that begins with K, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- *KAPPA
- *KINEMATIC
- *KINEMATIC COUPLING

*KAPPA: Specify the material parameters for mass diffusion driven by gradients of temperature and equivalent pressure stress.

This option is used to introduce temperature- and pressure-driven mass diffusion using the material parameters κ_s and κ_p, respectively. It must appear immediately after the *DIFFUSIVITY option. For each use of the *DIFFUSIVITY option, *KAPPA can be used once with TYPE=TEMP and once with TYPE=PRESS. The *KAPPA, TYPE=TEMP and *DIFFUSIVITY, LAW=FICK options are mutually exclusive.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Diffusivity*
- *DIFFUSIVITY

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of κ_s or κ_p. If this parameter is omitted, κ_s or κ_p is assumed not to depend on any field variables but may still depend on concentration and temperature. See *Material Data Definition* for more information.

TYPE

Set TYPE=TEMP (default) to define κ_s (governing mass diffusion caused by temperature gradients).

Set TYPE=PRESS to define κ_p (governing mass diffusion caused by gradients of the equivalent pressure stress).

Data lines to define the Soret effect factor (TYPE=TEMP):

First line:

1. Soret effect factor, κ_s. (Dimensionless.)
2. Concentration, c.
3. Temperature, θ.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define κ_s as a function of concentration, temperature, and other predefined field variables.

Data lines to define the pressure stress factor (TYPE=PRESS):

First line:

1. Pressure stress factor, κ_p. (Units of $F^{-1}L^2$.)
2. Concentration, c.
3. Temperature, θ.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define κ_p as a function of concentration, temperature, and other predefined field variables.

*KINEMATIC: Define a kinematic coupling constraint.

This option is used to define a kinematic coupling constraint. It must be used in conjunction with the **COUPLING* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Interaction module

References:

- *Coupling Constraints*
- **COUPLING*

Optional parameters:

ALPHA

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the thermal expansion coefficient.

Data lines to specify the degrees of freedom to be constrained:

First line:

1. First degree of freedom constrained. See *Conventions* for a definition of the numbering of degrees of freedom in Abaqus. If this field is left blank, all degrees of freedom will be constrained.
2. Last degree of freedom constrained. If this field is left blank, the degree of freedom specified in the first field will be the only one constrained.

Repeat this data line as often as necessary to specify constraints for different degrees of freedom. When the ORIENTATION parameter is specified on the associated **COUPLING* option, the degrees of freedom are in the referenced local system in the initial configuration; otherwise, they are in the global system. In either case these directions will rotate with the reference node in large-displacement analyses (when the NLGEOM parameter is included on the **STEP* option).

*KINEMATIC COUPLING: Constrain all or specific degrees of freedom of a set of nodes to the rigid body motion of a reference node.

This option is used to impose constraints between degrees of freedom of a node or node set and the rigid body motion defined by a reference node. The preferred method of providing a kinematic constraint of this type is the **COUPLING* option used in conjunction with the **KINEMATIC* option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Unsupported; this option has been superseded by coupling constraints used in conjunction with the kinematic option.

References:

• *Kinematic Coupling Constraints*

Required parameters:

REF NODE

Set this parameter equal to either the node number of the reference node or the name of a node set containing the reference node. If the name of a node set is chosen, the node set must contain exactly one node.

Optional parameters:

ALPHA

Set this parameter equal to the thermal expansion coefficient.

ORIENTATION

Set this parameter equal to the name given to the **ORIENTATION* definition (*Orientations*) that specifies the initial orientation of the local system in which the constrained degrees of freedom are defined.

Data lines to specify the nodes and degrees of freedom to be constrained:

First line:

1. Node number or node set label.
2. First degree of freedom constrained. See *Conventions* for a definition of the numbering of degrees of freedom in Abaqus/Standard. If this field is left blank, all degrees of freedom will be constrained.
3. Last degree of freedom constrained. If this field is left blank, the degree of freedom specified in the second field will be the only one constrained.

Repeat this data line as often as necessary to specify constraints at different nodes and degrees of freedom. When the ORIENTATION parameter is specified, the degrees of freedom are in the referenced local system in the initial configuration; otherwise, they are in the global system. In either case these directions will rotate with the reference node in large-displacement analyses (when the NLGEOM parameter is included on the **STEP* option).

L

This section describes each Abaqus input option that begins with L, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **LATENT HEAT*
- **LOAD CASE*
- **LOADING DATA*
- **LOW DENSITY FOAM*

*LATENT HEAT: Specify latent heats.

This option is used to specify a material's latent heat.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Latent Heat*

Optional parameters:

PORE FLUID

This parameter applies only to Abaqus/Standard analyses.

Include this parameter if the latent heat of the pore fluid in a porous medium is being defined.

SMOOTH

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to use a smooth transition from solidus to liquidus temperature.

Data lines to define a material's latent heat:

First line:

1. Latent heat per unit mass. (Units of JM^{-1}.)
2. Solidus temperature.
3. Liquidus temperature.
4. Scale factor to extend the solidus-liquidus temperature interval. The default value is 1.0.

Repeat this data line as often as necessary to define phase changes in the material; one line per phase change. Latent heat values must be given in ascending order of temperature.

*LOAD CASE: Begin a load case definition for multiple load case analysis.

This option is used to begin each load case definition.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- *Multiple Load Case Analysis*
- **END LOAD CASE*

Required parameters:

NAME

Set this parameter equal to a label that is used to refer to the load case.

Required parameter for load cases that include nodal pressure loading:

GENERATE

Include this parameter to generate multiple load cases.

This parameter applies only to a load case definition that refers to the nodal pressure distributed load type label (NP) in the **DSLOAD* option.

Nodal pressure load cases are generated for each node of the referenced surface. The label that is used to refer to the individual load case is formatted using the specified load case name, the referenced surface name, and the node number as follows:

```
name-surface_name-node_number
```

There are no data lines associated with this option.

*LOADING DATA: Provide loading data for uniaxial behavior models in connectors or provide data from a uniaxial or a shear loading test for fabric materials.

This option is used to define the loading response data for the uniaxial behavior of connector elements when used in conjunction with the **CONNECTOR BEHAVIOR* and **CONNECTOR UNIAXIAL BEHAVIOR* options.

This option is used to define the loading response from a uniaxial or a shear test for fabric materials when used in conjunction with the **FABRIC* and **UNIAXIAL* options. A fabric uniaxial test is specified with increasing strains along the specified yarn direction. A fabric shear test is specified with increasing shear strains as the fill and the warp yarns rotate with respect to each other.

If necessary, a separate unloading response can be specified using the **UNLOADING DATA* option.

Products: Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Connector Uniaxial Behavior*
- *Fabric Material Behavior*
- **CONNECTOR BEHAVIOR*
- **CONNECTOR UNIAXIAL BEHAVIOR*
- **FABRIC*
- **UNIAXIAL*
- **UNLOADING DATA*

Defining the loading response data for uniaxial behavior in connectors

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the data, in addition to temperature. If this parameter is omitted, it is assumed that the data depend only on temperature. The DEPENDENCIES parameter also applies to any subsequent unloading data defined using the **UNLOADING DATA* option.

DIRECTION

Set DIRECTION=TENSION to define tensile behavior.

Set DIRECTION=COMPRESSION to define compressive behavior.

If this parameter is omitted, the behavior is assumed to be nonlinear elastic and the data may span both positive and negative values of the primary component. The behavior will be considered to be symmetric about the origin if the given data are limited to either positive or negative values of the primary component.

EXTRAPOLATION

Set EXTRAPOLATION=CONSTANT to use constant extrapolation of the dependent variables outside the specified range of the independent variables.

Set EXTRAPOLATION=LINEAR to use linear extrapolation of the dependent variables outside the specified range of the independent variables.

The default is EXTRAPOLATION=CONSTANT unless **CONNECTOR BEHAVIOR*, EXTRAPOLATION=LINEAR is used. The EXTRAPOLATION parameter also applies to any subsequent unloading data defined using the **UNLOADING DATA* option.

INDEPENDENT COMPONENTS

Set INDEPENDENT COMPONENTS=CONSTITUTIVE MOTION to specify dependencies on components of constitutive relative motion included in the uniaxial behavior definition. The INDEPENDENT COMPONENTS parameter should not be used if uniaxial behavior is dependent on only the component of constitutive relative motion specified with the COMPONENT parameter on the **CONNECTOR UNIAXIAL BEHAVIOR* option. The INDEPENDENT COMPONENTS parameter also applies to any subsequent unloading data defined using the **UNLOADING DATA* option.

REGULARIZE

Set REGULARIZE=ON to regularize the user-defined tabular loading data.

Set REGULARIZE=OFF to use the user-defined tabular loading data directly without regularization.

The default is REGULARIZE=ON unless **CONNECTOR BEHAVIOR*, REGULARIZE=OFF is used. The REGULARIZE parameter also applies to any subsequent unloading data defined using the **UNLOADING DATA* option.

RTOL

Set this parameter equal to the tolerance to be used to regularize the user-defined tabular loading data. If this parameter is omitted, the default is RTOL=0.03 unless the tolerance is specified on the **CONNECTOR BEHAVIOR* option.

This parameter is ignored if REGULARIZE=OFF.

The RTOL parameter also applies to any subsequent unloading data defined using the **UNLOADING DATA* option.

TYPE

Set TYPE=DAMAGE to define a damage elasticity model. TYPE=DAMAGE must be used in conjunction with the DIRECTION parameter.

Set TYPE=ELASTIC (default) to define a nonlinear elastic model with or without rate dependency.

Set TYPE=PERMANENT DEFORMATION to define models that exhibit permanent deformation (plasticity) upon unloading. TYPE=PERMANENT DEFORMATION must be used in conjunction with the DIRECTION parameter.

Rate-independent elastic models do not require the definition of unloading data. Nonelastic models and rate-dependent models require the specification of unloading behavior using the **UNLOADING DATA* option.

The following parameters are optional and can be used only with TYPE=ELASTIC:

RATE DEPENDENT

Include this parameter equal to define rate-dependent loading data. If this parameter is omitted, the data are assumed to be rate independent.

RATE INTERPOLATION

Set RATE INTERPOLATION=LINEAR (default) to use linear intervals for the relative motion rate while interpolating rate-dependent loading data.

Set RATE INTERPOLATION=LOGARITHMIC to use logarithmic intervals for the equivalent relative motion rate while interpolating rate-dependent loading data.

This parameter is ignored if the RATE DEPENDENT parameter is omitted.

The following parameter is optional and can be used only with TYPE=DAMAGE:

DAMAGE ONSET

Set this parameter equal to the displacement/strain value at which the onset of damage occurs.

The following parameters are optional, mutually exclusive, and can be used only with TYPE=PERMANENT DEFORMATION:

SLOPE DROP

Set this parameter equal to the relative drop in slope on the loading curve that defines the onset of plastic deformation. The default value is 0.1.

YIELD ONSET

Set this parameter equal to the displacement/strain value at which the onset of yield occurs.

Data lines to define uniaxial behavior that depends on the displacement/rotation in the direction of the specified component of relative motion (the RATE DEPENDENT and INDEPENDENT COMPONENTS parameters are omitted):

First line:

1. Force or moment. (Provide the absolute value if the DIRECTION parameter is defined.)
2. Constitutive relative displacement or rotation. (Provide the absolute value if the DIRECTION parameter is defined.)
3. Temperature.
4. First field variable.
5. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the loading curve data.

Data lines to define uniaxial behavior that depends on the relative positions or motions in several component directions (the RATE DEPENDENT parameter is omitted and the INDEPENDENT COMPONENTS parameter is included):

First line:

1. First independent component number (1–6).
2. Second independent component number (1–6).
3. Etc., up to N_i entries. The independent components should not include the component for which the uniaxial loading behavior is being defined.

Subsequent lines:

1. Force or moment in the direction specified by the COMPONENT parameter on the **CONNECTOR UNIAXIAL BEHAVIOR* option. (Provide the absolute value if the DIRECTION parameter is defined.)

2. Constitutive relative displacement or rotation. (Provide the absolute value if the DIRECTION parameter is defined.)
3. Constitutive relative motion in the first independent component identified on the first data line.
4. Constitutive relative motion in the second independent component identified on the first data line.
5. Etc., up to N_i entries as identified on the first data line.
6. Temperature.
7. First field variable.
8. Second field variable.

If the number of data entries exceeds the limit of eight entries per line, continue the input on the next data line.

Do not repeat the first data line. Repeat the subsequent data lines as often as necessary to define the loading curve data.

Data lines to define rate-dependent uniaxial behavior in the direction of the specified component of relative motion (the RATE DEPENDENT parameter is included and the INDEPENDENT COMPONENTS parameter is omitted):

First line:

1. μ_0.
2. μ_1.
3. α.

Second line:

1. Force or moment. (Provide the absolute value.)
2. Constitutive relative displacement or rotation. (Provide the absolute value.)
3. Relative velocity. (Provide the absolute value.)
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Do not repeat the first data line. Repeat the subsequent data lines as often as necessary to define the loading curve data.

Data lines to define rate-dependent uniaxial behavior that depends on the relative motions in several component directions (the RATE DEPENDENT and INDEPENDENT COMPONENTS parameters are included):

First line:

1. First independent component number (1–6).
2. Second independent component number (1–6).
3. Etc., up to N_i entries. The independent components should not include the component for which the uniaxial loading behavior is being defined.

Second line:

1. μ_0.
2. μ_1.
3. α.

Subsequent lines:

1. Force or moment in the direction specified by the COMPONENT parameter on the **CONNECTOR UNIAXIAL BEHAVIOR* option. (Provide the absolute value.)
2. Constitutive relative displacement or rotation. (Provide the absolute value.)
3. Relative velocity. (Provide the absolute value.)
4. Constitutive relative motion in the first independent component identified on the first data line.
5. Etc., up to N_i entries as identified on the first data line.
6. Temperature.
7. First field variable.
8. Etc., up to eight entries per line.

   If the number of data entries exceeds the limit of eight entries per line, continue the input on the next data line.

Do not repeat the first and second data lines. Repeat the subsequent data lines as often as necessary to define the loading curve data.

Defining the loading response data from uniaxial tests of fabric materials

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the data, in addition to temperature. If this parameter is omitted, it is assumed that the data depend only on temperature. The DEPENDENCIES parameter also applies to any subsequent unloading data defined using the **UNLOADING DATA* option.

DIRECTION

Set DIRECTION=TENSION for tests with the primary strain component positive.

Set DIRECTION=COMPRESSION for tests with the primary strain component negative.

If this parameter is omitted, the behavior is assumed to be nonlinear elastic and the data may span both positive and negative values of the primary strain component. The behavior will be considered to be symmetric about the origin if the given data are limited to either positive or negative values of the primary strain component.

EXTRAPOLATION

Set EXTRAPOLATION=CONSTANT to use constant extrapolation of the dependent variables outside the specified range of the independent variables.

Set EXTRAPOLATION=LINEAR (default) to use linear extrapolation of the dependent variables outside the specified range of the independent variables.

The EXTRAPOLATION parameter also applies to any subsequent unloading data defined using the **UNLOADING DATA* option.

REGULARIZE

Set REGULARIZE=ON (default) to regularize the user-defined tabular loading data.

Set REGULARIZE=OFF to use the user-defined tabular loading data directly without regularization.

The REGULARIZE parameter also applies to any subsequent unloading data defined using the **UNLOADING DATA* option.

RTOL

Set this parameter equal to the tolerance to be used to regularize the user-defined tabular loading data. If this parameter is omitted, the default is RTOL=0.03.

This parameter is ignored if REGULARIZE=OFF.

The RTOL parameter also applies to any subsequent unloading data defined using the **UNLOADING DATA* option.

TYPE

Set TYPE=DAMAGE to define a damage elasticity model. TYPE=DAMAGE must be used in conjunction with the DIRECTION parameter.

Set TYPE=ELASTIC (default) to define a nonlinear elastic model with or without rate dependency.

Set TYPE=PERMANENT DEFORMATION to define models that exhibit permanent deformation (plasticity) upon unloading. TYPE=PERMANENT DEFORMATION must be used in conjunction with the DIRECTION parameter.

Rate-independent elastic models do not require the definition of unloading data. Nonelastic models and rate-dependent models require the specification of unloading test data using the **UNLOADING DATA* option.

The following parameters are optional and can be used only with TYPE=ELASTIC:

RATE DEPENDENT

Include this parameter equal to define rate-dependent loading data. If this parameter is omitted, the data are assumed to be rate independent.

RATE INTERPOLATION

Set RATE INTERPOLATION=LINEAR (default) to use linear intervals for strain rate while interpolating rate-dependent loading data.

Set RATE INTERPOLATION=LOGARITHMIC to use logarithmic intervals for strain rate while interpolating rate-dependent loading data.

This parameter is ignored if the RATE DEPENDENT parameter is omitted.

The following parameter is optional and can be used only with TYPE=DAMAGE:

DAMAGE ONSET

Set this parameter equal to the displacement/strain value at which the onset of damage occurs.

The following parameters are optional, mutually exclusive, and can be used only with TYPE=PERMANENT DEFORMATION:

SLOPE DROP

Set this parameter equal to the relative drop in slope on the loading curve that defines the onset of plastic deformation. The default value is 0.1.

YIELD ONSET

Set this parameter equal to the strain value at which the onset of yield occurs.

Data lines to define rate-independent fabric response (the RATE DEPENDENT parameter is omitted):

First line:

1. Nominal stress (Provide the absolute value if the DIRECTION parameter is defined.)

2. Nominal strain. (Provide the absolute value if the DIRECTION parameter is defined.)
3. Temperature.
4. First field variable.
5. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the loading curve data.

Data lines to define rate-dependent fabric response (the RATE DEPENDENT parameter is included):

First line:

1. μ_0.
2. μ_1.
3. α.

Second line:

1. Nominal stress. (Provide the absolute value.)
2. Nominal strain. (Provide the absolute value.)
3. Strain rate. (Provide the absolute value.)
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Do not repeat the first data line. Repeat the second and subsequent data lines as often as necessary to define the loading curve data.

*LOW DENSITY FOAM: Specify properties for a low-density foam.

This option is used to define material coefficients for low-density foam materials. The **LOW DENSITY FOAM* option must be used in conjunction with the **UNIAXIAL TEST DATA*, DIRECTION=TENSION and the **UNIAXIAL TEST DATA*, DIRECTION=COMPRESSION options to specify the stress-strain response of the foam material in uniaxial tension and compression, respectively.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Low-Density Foams*
- **UNIAXIAL TEST DATA*

Optional parameters:

FAIL

This parameter applies only to Abaqus/Explicit and is relevant only when the TENSION CUTOFF parameter is used.

Set FAIL=NO (default) to force the maximum principal stress to remain below the tension cutoff without deleting the element.

Set FAIL=YES to allow element deletion when the tension cutoff value is reached.

LATERAL STRAIN DATA

Use this parameter to specify if lateral strain data are provided as part of the definition of the uniaxial response of the material in tension and compression.

Set LATERAL STRAIN DATA=NO (default) if no lateral strain data are specified in the definition of the uniaxial response. The lateral strains are assumed to be zero in this case (zero Poisson's ratio).

Set LATERAL STRAIN DATA=YES if lateral strain data are specified in the definition of the uniaxial response.

RATE EXTRAPOLATION

This parameter applies only to Abaqus/Explicit analyses.

Use this parameter to specify extrapolation of rate-dependent uniaxial stress-strain curves beyond the maximum specified strain rate.

Set RATE EXTRAPOLATION=NO (default) to prevent extrapolation. The curve corresponding to the maximum specified strain rate will be used for strain rates greater than the maximum.

Set RATE EXTRAPOLATION=YES to trigger extrapolation beyond the maximum specified strain rate by using the slope with respect to strain rate.

STRAIN RATE

This parameter applies only to Abaqus/Explicit analyses.

Use this parameter to define the strain rate measure used for constitutive calculations.

Set STRAIN RATE=VOLUMETRIC to use the volumetric strain rate. This value is the default if LATERAL STRAIN DATA=NO.

Set STRAIN RATE=PRINCIPAL to use the nominal strain rate along each principal direction of deformation.

Set STRAIN RATE=MAX PRINCIPAL to use the maximum nominal strain rate along the principal direction of deformation. This is the default and only option available when LATERAL STRAIN DATA=YES.

TENSION CUTOFF

This parameter applies only to Abaqus/Explicit analyses.

Include this parameter to specify a tension cutoff value for the maximum principal stress that the low-density material can sustain with tension. The tension cutoff value must be greater than zero.

Data line to specify the relaxation coefficients for low-density foams in Abaqus/Explicit:

First (and only) line:

1. μ_0. The default is =0.0001 (time units).
2. μ_1. The default is =0.005 (time units).
3. α. The default is 2.

M

This section describes each Abaqus input option that begins with M, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **M1*
- **M2*
- **MAGNETIC PERMEABILITY*
- **MAGNETOSTATIC*
- **MANIFEST*
- **MAP SOLUTION*
- **MASS*
- **MASS ADJUST*
- **MASS DIFFUSION*
- **MASS FLOW RATE*
- **MATERIAL*
- **MATRIX*
- **MATRIX ASSEMBLE*
- **MATRIX CHECK*
- **MATRIX EXPORT*
- **MATRIX GENERATE*
- **MATRIX INPUT*
- **MATRIX OUTPUT*
- **MEAN FIELD DAMAGE*
- **MEAN FIELD HOMOGENIZATION*
- **MEDIA TRANSPORT*
- **MEMBRANE SECTION*
- **MODAL DAMPING*
- **MODAL DYNAMIC*
- **MODAL FILE*
- **MODAL OUTPUT*
- **MODAL PRINT*
- **MODEL CHANGE*
- **MOHR COULOMB*
- **MOHR COULOMB HARDENING*
- **MOISTURE SWELLING*
- **MOLECULAR WEIGHT*
- **MONITOR*
- **MOTION*
- **MPC*
- **MULLINS EFFECT*

- *MULTIPHYSICS LOAD

*M1: Define the first bending moment behavior of beams.

This option is used to define the first bending moment behavior of beams. It can be used only in conjunction with the *BEAM GENERAL SECTION, SECTION=NONLINEAR GENERAL option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- *Using a General Beam Section to Define the Section Behavior*
- *BEAM GENERAL SECTION

Optional parameters (if neither ELASTIC nor LINEAR is included, elastic-plastic response is assumed):

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the moment-curvature relationship, in addition to temperature. If this parameter is omitted, it is assumed that the moment–curvature relationship is constant or depends only on temperature. See *Material Data Definition* for more information.

ELASTIC

Include this parameter if the bending moment–curvature relationship is nonlinear but elastic.

LINEAR

Include this parameter if the bending moment varies linearly with curvature.

Data lines if the LINEAR parameter is included:

First line:

1. Bending stiffness of the section about the first beam section local axis.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the bending stiffness as a function of temperature and other predefined field variables.

Data lines if the LINEAR parameter is omitted:

First line:

1. Bending moment.
2. Curvature.
3. Temperature.
4. First field variable.

5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the bending moment–curvature relationship as a function of temperature and other predefined field variables.

*M2: Define the second bending moment behavior of beams.

This option is used to define the second bending moment behavior of beams. It can be used only in conjunction with the *BEAM GENERAL SECTION, SECTION=NONLINEAR GENERAL option and is needed only for beams in space.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- *Using a General Beam Section to Define the Section Behavior*
- *BEAM GENERAL SECTION

Optional parameters (if neither ELASTIC nor LINEAR is included, elastic-plastic response is assumed):

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the moment–curvature relationship, in addition to temperature. If this parameter is omitted, it is assumed that the moment–curvature relationship is constant or depends only on temperature. See *Material Data Definition* for more information.

ELASTIC

Include this parameter if the bending moment–curvature relationship is nonlinear but elastic.

LINEAR

Include this parameter if the bending moment varies linearly with curvature.

Data lines if the LINEAR parameter is included:

First line:

1. Bending stiffness of the section about the second beam section local axis.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the bending stiffness as a function of temperature and other predefined field variables.

Data lines if the LINEAR parameter is omitted:

First line:

1. Bending moment.
2. Curvature.
3. Temperature.

4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the moment–curvature relationship as a function of temperature and other predefined field variables.

*MAGNETIC PERMEABILITY: Specify magnetic permeability.

This option is used to define magnetic permeability for electromagnetic elements in an electromagnetic or a magnetostatic analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Magnetic Permeability*
- *Eddy Current Analysis*
- *Magnetostatic Analysis*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of magnetic permeability. If this parameter is omitted, the magnetic permeability is assumed not to depend on any field variables but may still depend on temperature and frequency.

The DEPENDENCIES parameter cannot be used with the NONLINEAR parameter.

FREQUENCY

Include this parameter to specify magnetic permeability as a function of frequency.

The FREQUENCY parameter cannot be used with the NONLINEAR parameter.

NONLINEAR

Include this parameter if the magnetic behavior is nonlinear and available in tabular form (of magnetic flux density versus magnetic field values). No data line is required if this parameter is used. The tabular data for the magnetic behavior, including field variable–dependent nonlinear magnetic properties, must be provided using the **NONLINEAR BH* option.

The NONLINEAR parameter cannot be used with the FREQUENCY and DEPENDENCIES parameters.

TYPE

Set TYPE=ISOTROPIC (default) to define isotropic magnetic permeability. Set TYPE=ORTHOTROPIC to define orthotropic magnetic permeability. Set TYPE=ANISOTROPIC to define fully anisotropic magnetic permeability.

If the NONLINEAR parameter is also used, the TYPE parameter can be set only to ISOTROPIC or ORTHOTROPIC.

Data lines to define isotropic magnetic permeability (TYPE=ISOTROPIC) if the FREQUENCY and NONLINEAR parameters are omitted:

First line:

1. Magnetic permeability. (Units of FA^{-2}.)
2. Temperature.

3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define isotropic magnetic permeability as a function of temperature and field variables.

Data lines to define isotropic magnetic permeability (TYPE=ISOTROPIC) if the FREQUENCY parameter is included:

First line:

1. Magnetic permeability. (Units of FA^{-2}.)
2. Frequency, in cycles/time.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define isotropic magnetic permeability as a function of frequency, temperature, and field variables.

Data lines to define orthotropic magnetic permeability (TYPE=ORTHOTROPIC) if the FREQUENCY and NONLINEAR parameters are omitted:

First line:

1. μ_{11}. (Units of FA^{-2}.)
2. μ_{22}.
3. μ_{33}.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define orthotropic magnetic permeability as a function of temperature and field variables.

Data lines to define orthotropic magnetic permeability (TYPE=ORTHOTROPIC) if the FREQUENCY parameter is included:

First line:

1. μ_{11}. (Units of FA^{-2}.)

2. μ_{22}.
3. μ_{33}.
4. Frequency, in cycles/time.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define orthotropic magnetic permeability as a function of frequency, temperature, and field variables.

Data lines to define anisotropic magnetic permeability (TYPE=ANISOTROPIC) if the FREQUENCY and NONLINEAR parameters are omitted:

First line:

1. μ_{11}. (Units of FA^{-2}.)
2. μ_{12}.
3. μ_{22}.
4. μ_{13}.
5. μ_{23}.
6. μ_{33}.
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define anisotropic magnetic permeability as a function of temperature and field variables.

Data lines to define anisotropic magnetic permeability (TYPE=ANISOTROPIC) if the FREQUENCY parameter is included:

First line:

1. μ_{11}. (Units of FA^{-2}.)
2. μ_{12}.
3. μ_{22}.
4. μ_{13}.
5. μ_{23}.
6. μ_{33}.
7. Frequency in cycles/time.
8. Temperature.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value equal to or greater than one):

1. First field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define anisotropic magnetic permeability as a function of frequency, temperature, and field variables.

There are no data lines associated with this option if the NONLINEAR parameter is used

*MAGNETOSTATIC: Magnetostatic analysis.

This option is used to indicate that the step should be analyzed as a magnetostatic load step.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Electromagnetic Analysis Procedures*
- *Magnetostatic Analysis*

Optional parameters:

DIRECT

This parameter selects direct user control of the incrementation through the step. If this parameter is used, constant increments of the size defined on the data line are used. If it is omitted, Abaqus will choose the increments (after trying the user's initial time increment for the first attempt at the first increment).

STABILIZATION

Include this parameter to activate the stabilization scheme that may be needed in some situations to obtain a magnetostatic solution. It defines a factor that is used by Abaqus in the stabilization computations. If this parameter is included without a value, the default value is assumed to be 1.0. The parameter may be set to a higher value to increase the stabilization or to a lower value to decrease it.

Data line for a magnetostatic analysis:

First (and only) line:

1. Initial time increment. This value will be modified as required if the automatic time stepping scheme is used or will be used as the constant time increment if the DIRECT parameter is used. If this entry is zero or is not specified, a default value that is equal to the total time period of the step is assumed.
2. Time period of the step. If this entry is zero or is not specified, a default value of **1.0** is assumed.
3. Minimum time increment allowed. Only used for automatic time incrementation. If Abaqus/Standard finds it needs a smaller time increment than this value, the analysis is terminated. If this entry is zero, a default value of the smaller of the suggested initial time increment or 10^{-5} times the total time period is assumed.
4. Maximum time increment allowed. Only used for automatic time incrementation. If this value is not specified, no upper limit is imposed.

*MANIFEST: Specify the input files to execute in an evolution analysis.

This option is used to specify the input files used in an evolution analysis.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

• *Multiple Nonlinear Load Case Analysis*

Required parameters:

EVOLUTION TYPE

Set EVOLUTION TYPE=HISTORY if each input file after the first input file (which defines the model) defines the steps forming a nonlinear load case.

Optional parameters:

BASE STATE

Set BASE STATE=NO (default) if the base state used for each nonlinear load case is the initial state of the model. Steps included in the first input file form the first nonlinear load case.

Set BASE STATE=YES if steps defined in the first input file lead to the base state used for each nonlinear load case.

MODEL CHANGE

Include this parameter if element removal or reactivation occurs during any nonlinear load case.

RESULTS

Set RESULTS=APPEND (default) to create a single set of output database files that contains results for the entire analysis.

Set RESULTS=NEW to create separate output database files for each input file.

ZERO INCREMENT

Set ZERO INCREMENT=YES (default) to write output to the output database for the zero increment of the first step of each nonlinear load case.

Set ZERO INCREMENT=NO to specify that output to the output database should not be written for the zero increment of the first step of each nonlinear load case.

Data lines to define the input files associated with a multiple nonlinear load case analysis:

First line:

1. Name of the input file containing the model and optional steps defining the base state or first nonlinear load case.

Repeat this data line as often as necessary to define the input files containing each nonlinear load case. Only one file name may appear on each data line.

*MAP SOLUTION: Map a solution from an old mesh to a new mesh.

This option is used to transfer solution variables from an earlier analysis to a new mesh that occupies the same space.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

• *Mesh-to-Mesh Solution Mapping*

Optional parameters:

INC

Set this parameter equal to the increment number from which the old solution will be read. If this parameter is omitted, the last increment for which a solution is available will be read.

The STEP parameter must be specified if the INC parameter is used.

STEP

Set this parameter equal to the step number from which the old solution will be read. If this parameter is omitted, the last step and increment for which a solution is available will be read.

UNBALANCED STRESS

Set UNBALANCED STRESS=RAMP (default) if the stress unbalance is to be resolved linearly over the step.

Set UNBALANCED STRESS=STEP if the stress unbalance is to be resolved in the first increment.

Data line to translate an old-model mesh:

First (and only) line:

1. Value of the translation to be applied in the *X*-direction.
2. Value of the translation to be applied in the *Y*-direction.
3. Value of the translation to be applied in the *Z*-direction.

Data lines to translate and/or rotate an old-model mesh:

First line:

1. Value of the translation to be applied in the *X*-direction.
2. Value of the translation to be applied in the *Y*-direction.
3. Value of the translation to be applied in the *Z*-direction.

Enter values of zero to apply a pure rotation.

Second line:

1. *X*-coordinate of point *a* on the axis of rotation (see *Figure 1*).
2. *Y*-coordinate of point *a* on the axis of rotation.
3. *Z*-coordinate of point *a* on the axis of rotation.
4. *X*-coordinate of point *b* on the axis of rotation.
5. *Y*-coordinate of point *b* on the axis of rotation.
6. *Z*-coordinate of point *b* on the axis of rotation.

7. Angle of rotation about the axis *a*–*b*, in degrees.

If both translation and rotation are specified, translation is applied before rotation.

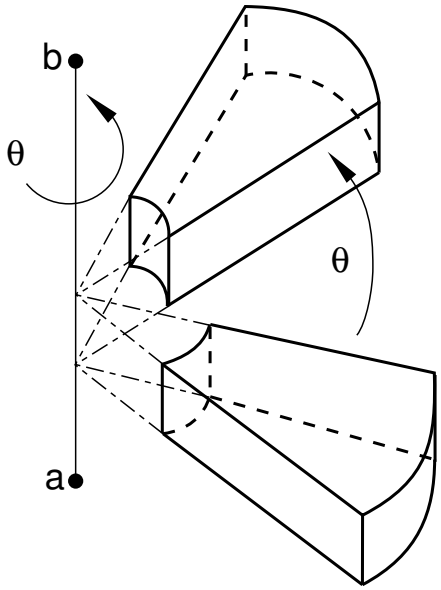


Figure 1: Rotation of an old-model mesh.

*MASS: Specify a point mass.

This option is used to define lumped mass values associated with MASS elements.

It is also used to define mass proportional damping.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: *Property module* and *Interaction module*.

References:

• *Point Masses*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the MASS elements for which the value is being given.

Optional parameters:

ALPHA

Set this parameter equal to the α_R factor to create mass proportional damping for the MASS elements when used in direct-integration dynamic analyses, in mode-based analyses that support nondiagonal damping in Abaqus/Standard, and in explicit dynamic analyses. This value is ignored in mode-based analyses that do not support nondiagonal damping. The default is 0.0.

COMPOSITE

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the fraction of critical damping to be used with the MASS elements when calculating composite damping factors for the modes when used in mode-based analyses that are not based on the SIM architecture. The default is 0.0.

For mode-based analyses that are based on the SIM architecture, the **COMPOSITE MODAL DAMPING* option should be used instead. This value is ignored in direct-integration dynamics.

ORIENTATION

This parameter applies only when TYPE=ANISOTROPIC.

Set this parameter equal to the name of an **ORIENTATION* option (*Orientations*) that is being used to define the principal directions of the anisotropic mass tensor. If the ORIENTATION parameter is omitted, it is assumed that the principal directions coincide with the global axes.

In a large-displacement analysis, the local axes of the anisotropic mass rotate with the rotation, if active, of the node to which the MASS element is attached.

TYPE

Set TYPE=ISOTROPIC (default) for an isotropic mass tensor.

Set TYPE=ANISOTROPIC for an anisotropic mass tensor with possibly three different principal values.

Data line to define the magnitude of an isotropic mass (not weight):
First (and only) line:

1. Mass magnitude.

Abaqus does not use any specific physical units, so the user's choice must be consistent.

Data line to define the three principal values of an anisotropic mass tensor (not weight):
First (and only) line:

1. The principal mass about the local 1-axis, M_{11}.
2. The principal mass about the local 2-axis, M_{22}.
3. The principal mass about the local 3-axis, M_{33}.

Abaqus does not use any specific physical units, so the user's choice must be consistent.

*MASS ADJUST: Adjust and/or redistribute the mass of an element set.

This option is used to increase or decrease the mass of the elements in an element set uniformly so that the total mass for that set matches a specified value. It can also be used to redistribute the mass among the elements in the element set to raise the minimum stable time increment for that set to a specified target value.

The total mass of an element set can also be increased or decreased using a scale factor. When using a scale factor, the total mass of the elements in the element set can be redistributed to reach the maximum time target for the set or only the added mass can be redistributed (the pre-adjusted mass of the elements is preserved).

Mass can be adjusted for multiple element sets; the mass is adjusted in the order in which the element sets are specified. This behavior influences the final outcome for element sets that share elements.

This option can appear only once in an analysis.

Products: Abaqus/Explicit

Type: Model data

Level: Assembly

References:

• *Mass Adjustment*

Optional parameters:

TARGET DT

Set this parameter equal to a value to define a default minimum element stable time increment for all of the mass-adjusted element sets. Target time increments can be specified for selected sets on the data lines, if needed. A nonzero target value causes redistribution of mass within the element sets.

Set TARGET DT=MAXIMIZE to indicate that a maximum possible time increment for the elements in the sets is desired. The mass redistribution to maximize the time increment can be done for the total mass or for only the added mass. Maximizing the time increment can be specified for selected sets on the data lines, if needed.

If this parameter is omitted, the third field in the data lines specifies the redistribution behavior.

Data lines to adjust and/or redistribute the mass of element sets:

First line:

1. Element set label.
2. Total mass of the element set. Enter the mass value, or enter CURRENT to redistribute the mass to achieve a target time increment without altering the total mass.
3. Enter the minimum element stable time increment value for the element set. If using a scale factor, enter MAXIMIZE to maximize the minimum stable time increment in the initial configuration or enter UNIFORM to distribute the mass in proportion with the pre-adjusted mass. If omitted, the default value is the setting of the TARGET DT parameter if it is used; otherwise, the default value for the minimum element stable time increment is 0.0 or, if using a scale factor, the default behavior is to distribute the mass in proportion with the pre-adjusted mass. When both the TARGET DT parameter and a data line entry for the element set are specified, the data line entry for the element set is used for that set.

4. Mass scale factor for the element set. Only relevant for redistributing mass to achieve a maximum possible time increment in the element set and distributing the mass in proportion with the pre-adjusted mass. If omitted, a default value of one is used to maximize the time increment.
5. Enter REDIST to indicate that the total element mass (the pre-adjusted mass plus the added mass) should be redistributed. Enter ADD to indicate that only the added mass should be redistributed. Only relevant for redistributing mass to achieve a maximum possible time increment in the element set. If omitted, the default behavior is to redistribute the total element mass.

Repeat this data line as often as necessary to adjust the mass for different element sets. The same element set label cannot be entered more than once. The mass is adjusted in the order the element sets are specified on the data lines.

*MASS DIFFUSION: Transient or steady-state uncoupled mass diffusion analysis.

This option is used to control uncoupled transient or steady-state mass diffusion analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

• *Mass Diffusion Analysis*

Optional parameters:

DCMAX

Set this parameter equal to the maximum normalized concentration change to be allowed in an increment. Abaqus/Standard will restrict the time step to ensure that this value will not be exceeded at any node (except nodes with boundary conditions) during any increment of the analysis. If the DCMAX parameter is omitted, fixed time increments will be used.

END

Set END=PERIOD (default) to analyze the entire time period specified on the data line. Set END=SS to end the analysis when steady state is reached.

STEADY STATE

Include this parameter to choose steady-state analysis. Transient analysis is assumed if this parameter is omitted.

Data line to define time stepping in a mass diffusion analysis:

First (and only) line:

1. Time step. If automatic stepping is used, this value should be a reasonable suggestion for the initial step and will be adjusted as necessary.
2. Time period. If END=SS is chosen, the step ends when steady state is reached or after this time period, whichever occurs first.
3. Minimum time increment allowed. If Abaqus/Standard finds it needs a smaller time increment than this value, the analysis is terminated. If no value is given, Abaqus/Standard sets the minimum increment to the minimum of 0.8 times the suggested initial time step (the first data item on this line) and 10^{-5} times the time period (the second data item on this line). If a value is given, Abaqus/Standard will use the minimum of the given value and 0.8 times the suggested initial time step.
4. Maximum time increment. If this value is omitted, no upper limit is imposed. This value is used only for automatic time incrementation.
5. Rate of change of normalized concentration (normalized concentration per time) used to define steady state; only needed if END=SS is chosen. When all nodal normalized concentrations are changing at less than this rate, the solution terminates.

*MASS FLOW RATE: Specify fluid mass flow rate in a heat transfer analysis.

This option is used to specify the mass flow rate per unit area (or through the entire section for one-dimensional elements) for forced convection/diffusion elements in a heat transfer analysis. This option cannot be used with hydrostatic fluid elements.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Uncoupled Heat Transfer Analysis*
- *UMASFL*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude versus time curve that defines the magnitude of the flow rate during the step (*Amplitude Curves*). If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (*Defining an Analysis*).

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

OP

Set OP=MOD (default) for existing **MASS FLOW RATE* values to remain, with this option modifying existing flow rates or defining additional flow rates.

Set OP=NEW if all existing **MASS FLOW RATE* values applied to the model should be removed.

USER

Include this parameter to indicate that user subroutine *UMASFL* will be used to define mass flow rate values. *UMASFL* will be called for each node given on the data lines. If values are also given on the data lines, these values will be ignored.

Data lines to define mass flow rates:

First line:

1. Node number or node set label.
2. Mass flow rate per unit area in the x-direction (units of $ML^{-2}T^{-1}$) or total mass flow rate in the cross-section (units of MT^{-1}) for one-dimensional elements.
3. Mass flow rate per unit area in the y-direction (not needed for nodes associated with one-dimensional elements).
4. Mass flow rate per unit area in the z-direction (not needed for nodes associated with one-dimensional elements).

Repeat this data line as often as necessary to define mass flow rates at different nodes.

Data lines to define mass flow rates using user subroutine `UMASFL`:

First line:

1. Node set or node number.

Repeat this data line as often as necessary. *UMASFL* will be called for each node listed.

*MATERIAL: Begin the definition of a material.

This option is used to indicate the start of a material definition.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Material Data Definition*

Required parameters:

NAME

Set this parameter equal to a label to use to refer to the material in the element property options. Material names in the same input file must be unique. Furthermore, material names should be unique from the names associated with property definitions such as **CONNECTOR BEHAVIOR* and **FLUID BEHAVIOR*. Material names adhere to the naming convention for labels (see *Input Syntax Rules*), except that they cannot begin with a number.

Optional parameters:

RTOL

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the tolerance to use for regularizing the material data. The default is RTOL=0.03.

STRAIN RATE INTERPOLATION

This parameter applies only to Abaqus/Explicit analyses and is used only to interpolate strain rate–dependent material data.

Set STRAIN RATE INTERPOLATION=LOGARITHMIC (default) to use logarithmic intervals while interpolating strain rate–dependent material data.

Set STRAIN RATE INTERPOLATION=LINEAR to use linear intervals while interpolating strain rate–dependent material data.

USER TYPE

Set this parameter equal to the name of the material user type that identifies a special-purpose material modeling technique. For more information, see *About Special-Purpose Material Modeling Techniques*.

Optional and mutually exclusive parameters in Abaqus/Explicit:

SRATE FACTOR

Set this parameter equal to the factor used to filter the equivalent plastic strain rate for the evaluation of strain rate-dependent material data.

The default value is 0.9, unless a time filter is defined with the SRATE TIME parameter.

SRATE TIME

Set this parameter equal to the time constant used to filter the equivalent plastic strain rate for the evaluation of strain rate-dependent material data.

There are no data lines associated with this option.

*MATRIX: Read in the stiffness or mass matrix for a linear user element.

This option can be used only in conjunction with the **USER ELEMENT*, LINEAR option. It is used to read in the stiffness, mass, viscous damping, or structural damping matrix for the user element. It can be used once for each of the matrix types.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance, Model

References:

- *User-Defined Elements*
- **USER ELEMENT*

Required parameters:

TYPE

Set TYPE=MASS to define the mass matrix.

Set TYPE=STIFFNESS to define the stiffness matrix.

Set TYPE=VISCOUS DAMPING to define the viscous damping matrix.

Set TYPE=STRUCTURAL DAMPING to define the structural damping matrix.

Optional parameters:

INPUT

Set this parameter equal to the name of the alternate input file from which the data lines are to be read. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

Data lines to define the matrix:

First line:

1. Matrix entries, four per line.

Repeat this data line as often as necessary.

*MATRIX ASSEMBLE: Define stiffness, mass, or damping matrices for a part of the model.

This option can be used to identify a stiffness, mass, or damping matrix that will be assembled into the corresponding global finite element matrix. This matrix must have been input previously by using the **MATRIX INPUT* option. The option also unites all the matrices from the same original model and allows you to remap their user nodes to new labels. Matrices from different original models must be specified by repeating the **MATRIX ASSEMBLE* option.

Positioning transformations are applied in the order in which they are specified. Translation, rotation, and reflection transformations are supported. There is no restriction on the number of translations, rotations, and reflections that you can apply in a single **MATRIX ASSEMBLE* option.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Using Matrices*
- **MATRIX INPUT*

At least one of the following parameters is required:

MASS

Set this parameter equal to the name of the mass matrix.

STIFFNESS

Set this parameter equal to the name of the stiffness matrix.

STRUCTURAL DAMPING

Set this parameter equal to the name of the structural damping matrix.

VISCOUS DAMPING

Set this parameter equal to the name of the viscous damping matrix.

Optional parameters:

NAME

This parameter specifies a matrix subassembly name.

If this parameter is omitted, a default name is assigned to each subassembly: `Assemble-1`, `Assemble-2`, etc.

NSET

This parameter remaps nodes of the assembled matrices at the usage level. Set this parameter equal to the name of the node set that contains the new node labels corresponding to the matrix nodes. You can map all user-defined nodes of the matrices or the interface nodes only, if defined. The size of the node set and the order of the nodes in the set must fully correspond to the combined set of nodes of all the matrices for all nodes or only interface nodes. If necessary, use an unsorted node set to obtain the correct mapping. The default matrix nodes order is in ascending order of their original labels defined at generation or specified in the matrix data.

If this parameter is omitted, all user-defined nodes retain their original labels.

POSITION TOL

Set this parameter equal to the tolerance on the distance between usage level nodes and the corresponding matrix nodes. If this parameter is omitted, the default is a tolerance of 10^{-4} times the largest estimated characteristic dimension within the matrix subassembly. If the parameter is given with a value of 0.0, the position of the matrix nodes is not checked.

Data lines to translate a matrix subassembly:

First line:

1. TRANSLATE (transformation type).
2. Value of the translation to be applied in the global *X*-direction.
3. Value of the translation to be applied in the global *Y*-direction.
4. Value of the translation to be applied in the global *Z*-direction.

Repeat this data line as often as necessary to define matrix subassembly translations.

Data lines to rotate a matrix subassembly:

First line:

1. ROTATE (transformation type).
2. Global *X*-coordinate of point *a* on the axis of rotation (see *Figure 1*).
3. Global *Y*-coordinate of point *a* on the axis of rotation.
4. Global *Z*-coordinate of point *a* on the axis of rotation.
5. Global *X*-coordinate of point *b* on the axis of rotation.
6. Global *Y*-coordinate of point *b* on the axis of rotation.
7. Global *Z*-coordinate of point *b* on the axis of rotation.
8. Angle of rotation about the *a*–*b* axis (in degrees).

Repeat this data line as often as necessary to define matrix subassembly rotations.

Data lines to reflect a matrix subassembly:

First line:

1. REFLECT (transformation type).
2. Global *X*-coordinate of point *a* in the plane of reflection (see *Figure 2*).
3. Global *Y*-coordinate of point *a* in the plane of reflection.
4. Global *Z*-coordinate of point *a* in the plane of reflection.
5. Global *X*-coordinate of point *b* in the plane of reflection.
6. Global *Y*-coordinate of point *b* in the plane of reflection.
7. Global *Z*-coordinate of point *b* in the plane of reflection.
8. Global *X*-coordinate of point *c* in the plane of reflection.
9. Global *Y*-coordinate of point *c* in the plane of reflection.
10. Global *Z*-coordinate of point *c* in the plane of reflection.

Repeat this data line as often as necessary to define matrix subassembly reflections.

Data lines to reflect a matrix subassembly using the Euler angles ϕ, θ, and ψ:

First line:

1. EULER (transformation type).
2. Precession angle, ϕ, of rotation about the z-axis (in degrees) (see *Figure 3*).

3. Nutation angle, θ, of rotation about the x'-axis (in degrees).
4. Intrinsic rotation angle, ψ, of rotation about the z'-axis (in degrees).
5. Global *X*-coordinate of the reference point, *O* (optional; default value is zero).
6. Global *Y*-coordinate of the reference point, *O* (optional; default value is zero).
7. Global *Z*-coordinate of the reference point, *O* (optional; default value is zero).

Repeat this data line as often as necessary to define matrix subassembly reflections using the Euler angles.

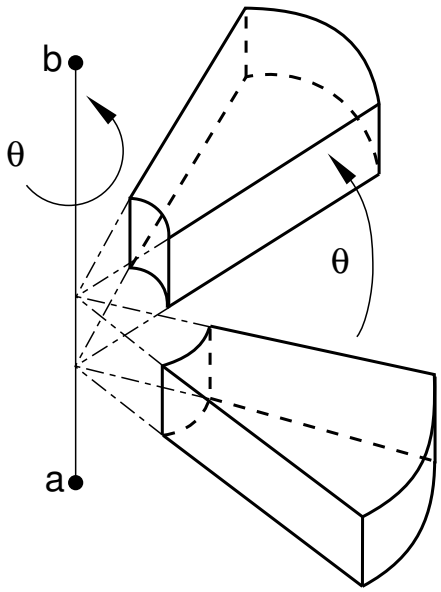


Figure 1: Matrix subassembly rotation.

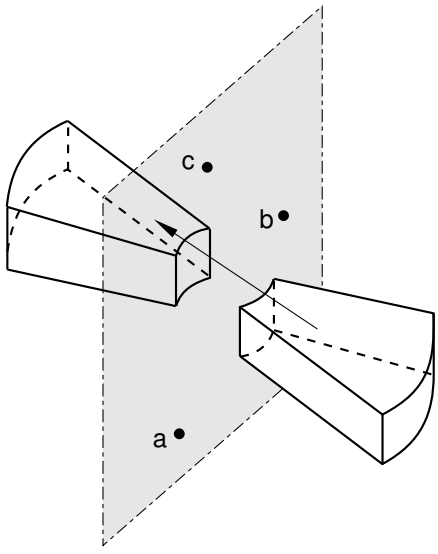


Figure 2: Matrix subassembly reflection. Points a, b, and c cannot be collinear.

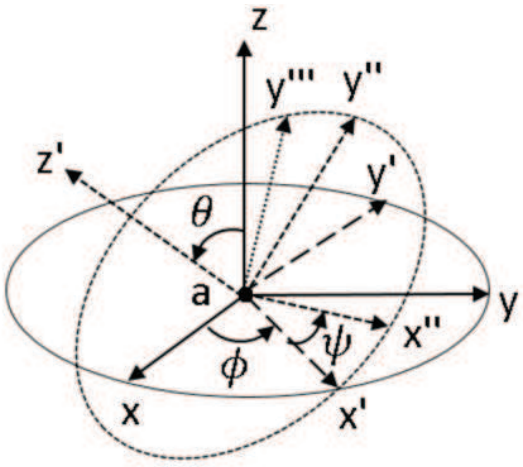


Figure 3: Euler angles.

*MATRIX CHECK: Check the quality of the generated stiffness and mass matrices.

This option is used to check the quality of the generated stiffness and mass matrices. It can be used only in matrix generation or substructure generation analyses.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Generating Matrices as a Linear Analysis Step*
- *Generating Substructures*
- **MATRIX GENERATE*
- **SUBSTRUCTURE GENERATE*

Optional parameters:

ERROR

Include this parameter to treat any problem reported by the matrix check as an error.

REFERENCE NODE

Set this parameter equal to an existing node label to define the origin of the coordinate frame to use for the matrix check.

TOLERANCE

Set TOLERANCE=ON to modify the tolerance values to use for the matrix check.

Data line to modify the matrix check tolerances:

First (and only) line:

1. Maximum acceptable ratio of the strain energy of a rigid body mode divided by the maximum element of the stiffness matrix. The default value is 1.0×10^{-10}.
2. Tolerance to check the consistency of the 3×3 upper-left submatrix of the projected 6×6 rigid body mass matrix. This submatrix must be a diagonal matrix with the mass on its main diagonal. The tolerance defines the maximum acceptable ratio of the square sum of all of the off-diagonal elements divided by the square sum of all elements. The default value is 1.0×10^{-3}.
3. Factor and divisor by which a diagonal element of the substructure stiffness matrix can deviate from the median value of its diagonal elements (only for a substructure generation analysis). The default value is 1.0×10^{7}.
4. Factor and divisor by which a diagonal element of the substructure mass matrix can deviate from the median values of its main diagonal elements (only for a substructure generation analysis). The default value is 1.0×10^{7}.
5. Minimum reciprocal 1-normalized condition number of the substructure stiffness matrix (only for a substructure generation analysis). The default value is 1.0×10^{-17}.
6. Minimum reciprocal 1-normalized condition number of the substructure mass matrix (only for a substructure generation analysis). The default value is 1.0×10^{-12}.

*MATRIX EXPORT: Write generated matrices to files in specific formats for selected workflows.

This option is used to write generated matrices to files in specific formats for selected workflows. It can be used only in a matrix generation analysis.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- **MATRIX GENERATE*
- *Generating Matrices as a Linear Analysis Step*

Required parameters:

WORKFLOW

Set WORKFLOW=ZAERO to generate the assembled mass matrix in OUTPUT4 format for aeroelastic simulations with the ZAERO™ software suite from ZONA Technology, Inc.

There are no data lines associated with this option.

*MATRIX GENERATE: Generate global or element matrices.

This option is used to generate matrices representing the stiffness, mass, viscous damping, structural damping, or load vectors in a model.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

• *Generating Matrices as a Linear Analysis Step*

At least one of the following parameters is required:

STIFFNESS

Include this parameter to generate the stiffness matrix.

MASS

Include this parameter or set MASS=CONSISTENT to generate the consistent mass matrix.

Set MASS=AT USER NODES to generate the consistent mass matrix with the mass only at the user-defined nodes. If a finite element has nonzero mass at Abaqus internal nodes, the mass is transferred to the user-defined nodes using heuristic techniques. Mass is redistributed only for modified triangle and tetrahedral elements; no mass is redistributed from internal to user-defined nodes for frame elements and substructures.

VISCOUS DAMPING

Include this parameter to generate the viscous damping matrix.

STRUCTURAL DAMPING

Include this parameter to generate the structural damping matrix.

LOAD

Include this parameter to generate the load matrix.

The following parameter is required if the model contains solid continuum infinite elements:

SOLID INFINITE FORMULATION

Set this parameter equal to STATIC to select the static formulation for solid infinite elements.

Set this parameter equal to DYNAMIC to select the dynamic formulation for solid infinite elements.

Optional parameters:

ELEMENT BY ELEMENT

Include this parameter to generate local element matrices. By default, global assembled matrices are generated.

ELSET

Use this parameter to generate matrices for a part of a model. Set this parameter equal to the name of an element set that contains all the elements in the selected part of a model. By default, matrices are generated for the whole model.

FIELD

Set FIELD=ALL (default) to indicate that matrices are generated for the structural and acoustic parts of the model.

Set FIELD=MECHANICAL to indicate that matrices are generated only for the structural part of the model.

Set FIELD=ACOUSTIC to indicate that matrices are generated only for the acoustic part of the model.

FRICTION DAMPING

Set FRICTION DAMPING=NO (default) to ignore friction-induced viscous damping effects.

Set FRICTION DAMPING=YES to include friction-induced viscous damping effects.

INTERFACE NODES

This parameter specifies some of the nodes as "interface nodes." By specifying interface nodes, you can reduce the number of nodes required for including matrices in the matrix usage analysis. Set this parameter equal to the name of the node set that contains all of the interface nodes in the desired order.

MPC

Set MPC=YES (default) to generate the matrices with applied multipoint constraints. The generated matrices will include entries only for the independent degrees of freedom.

Set MPC=NO to skip applying the multipoint constraints during the matrix generation. The matrices will include entries for all active degrees of freedom in the model.

PROPERTY EVALUATION

Set this parameter equal to the frequency at which to evaluate frequency-dependent properties for viscoelasticity, springs, and dashpots during the matrix generation. If this parameter is omitted, Abaqus/Standard evaluates the stiffness associated with frequency-dependent springs and dashpots at zero frequency and does not consider the stiffness contributions from frequency-domain viscoelasticity.

SOURCE

Set SOURCE=ALL (default) to generate matrices including contributions from the finite elements and from the matrix input.

Set SOURCE=ELEMENTS to generate matrices including only contributions from the finite elements.

Set SOURCE=MATRIX INPUT to generate matrices including only contributions from the matrix input. The ELSET and SOURCE=MATRIX INPUT parameters are mutually exclusive.

There are no data lines associated with this option.

*MATRIX INPUT: Read in a matrix for a part of the model.

This option can be used to input a matrix in sparse format.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Using Matrices*
- **MATRIX ASSEMBLE*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to this matrix.

Optional parameters:

INPUT

Set this parameter equal to the name of the alternate input file from which the data lines are to be read in text format. See *Input Syntax Rules* for the syntax of such file names.

Set this parameter equal to the name of the `.sim` file to read a matrix from the SIM database. The MATRIX parameter is also required in this case.

If this parameter is omitted, it is assumed that the data follow the keyword line.

INTERNAL DOFS

This parameter defines the internal degree of freedom type for all internal nodes associated with the matrix entries.

Set INTERNAL DOFS=LAGRANGE if the internal degrees of freedom represent Lagrange multipliers.

Set INTERNAL DOFS=MODAL if the internal degrees of freedom represent substructure generalized displacements associated with dynamic modes.

MATRIX

This parameter defines the matrix to be read from the SIM database. It must be used together with the INPUT parameter defining the `.sim` file.

Set MATRIX=STIFFNESS to read the stiffness matrix.

Set MATRIX=MASS to read the mass matrix.

Set MATRIX=VISCOUS DAMPING to read the viscous damping matrix.

Set MATRIX=STRUCTURAL DAMPING to read the structural damping matrix.

SCALE FACTOR

Set this parameter equal to a nonzero real number s by which all matrix entries will be multiplied.

The default value is $s = 1.0$.

TYPE

This parameter defines the shape of the matrix. It is ignored for matrix input from the SIM database because the shape is internally set up according to the matrix data.

Set TYPE=SYMMETRIC (default) to read the upper or lower triangular portion of a square symmetric matrix. If a full matrix is specified, corresponding terms above and below the diagonal must be equal.

Set TYPE=UNSYMMETRIC to read a square unsymmetric matrix.

Data lines to define the matrix in sparse format (only nonzero terms):

First line:

1. Row node number.
2. Degree of freedom number for row node.
3. Column node number.
4. Degree of freedom number for column node.
5. Matrix entry.

Give data to define a symmetric matrix in lower triangular, upper triangular, or square format. For a square matrix to be symmetric, corresponding entries above and below the diagonal must have exactly the same values.

Repeat this data line as often as necessary.

*MATRIX OUTPUT: Output generated matrices in various forms.

This option is used to write generated global matrices to files in assembled or element-by-element form. It can be used only in a matrix generation analysis.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Generating Matrices as a Linear Analysis Step*
- **MATRIX GENERATE*

At least one of the following parameters is required:

STIFFNESS

Include this parameter to output the stiffness matrix.

MASS

Include this parameter to output the mass matrix.

VISCOUS DAMPING

Include this parameter to output the viscous damping matrix.

STRUCTURAL DAMPING

Include this parameter to output the structural damping matrix.

LOAD

Include this parameter to output the load matrix.

Optional parameters:

FORMAT

Set FORMAT=MATRIX INPUT (default) to specify that the output use the matrix input text format that is consistent with the format used by the matrix definition technique in Abaqus/Standard.

Set FORMAT=LABELS to specify that the output use the standard labeling format.

Set FORMAT=COORDINATE to specify that the output use the common mathematical coordinate format.

Set FORMAT=DMIG to specify that the output use the NASTRAN DMIG-s format.

Set FORMAT=USER ELEMENT to specify the output format for use under the **MATRIX* option for symmetric or unsymmetric linear user elements.

There are no data lines associated with this option.

*MEAN FIELD DAMAGE: Define the impact of microlevel damage on the behavior of the composite material at the macro level.

This option is used to define the impact of microlevel damage on the behavior of the composite material at the macro level.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Assembly

References:

- **MEAN FIELD HOMOGENIZATION*

Required parameters:

DEGRADATION

Set DEGRADATION=MAXIMUM to specify that the overall damage variable of the composite is determined using the maximum of the overall damage variable in all the constituents.

Set DEGRADATION=MULTIPLICATIVE to specify that the overall damage variable of each constituent contributes to the overall damage variable of the composite in a multiplicative sense.

Set DEGRADATION=USER to specify a user-defined overall damage behavior for the composite in a user subroutine. User-defined damage is supported only in Abaqus/Standard.

Optional parameters:

PROPERTIES

This parameter can be used only if DEGRADATION=USER.

Set this parameter equal to the number of property values required as data in user subroutine *UDAMAGEMF*. The default is 0.

Data line for a user-defined overall damage variable (DEGRADATION=USER):

First line:

1. Material properties, eight per line.

Repeat this data line as often as necessary to define material properties.

No data lines are needed if the PROPERTIES parameter is omitted or set to 0.

*MEAN FIELD HOMOGENIZATION: Begin the definition of a multiscale material modeled with mean-field homogenization.

This option is used to indicate the start of a multiscale material definition. If this option is used, the **CONSTITUENT* option is required.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- **MATERIAL*
- **CONSTITUENT*
- *Mean-Field Homogenization*

Optional parameters:

ANGLE SUBDIVISIONS

This parameter applies only to Abaqus/Standard analyses.

This parameter is meaningful only when a second-order orientation tensor is used to specify the orientation of the inclusions in the aggregate.

Set this parameter equal to the number of angle increments used for the discretization of the orientation space.

CLOSURE

This parameter is used to select the closure approximation to compute the orientation distribution function using the second-order orientation tensor. This parameter is meaningful only when a second-order orientation tensor is used to specify the orientation of the inclusions in the aggregate.

Set CLOSURE=FITTED to use the orthotropic fitted (ORF) closure approximation.

Set CLOSURE=HYBRID (default) to use the hybrid closure approximation.

Set CLOSURE=ORE to use the orthotropic equivalent of the natural closure approximation.

Set CLOSURE=QUADRATIC to use the quadratic closure approximation.

Set CLOSURE=SMOOTH to use the orthotropic smooth closure approximation.

DIRECTION UPDATE

This parameter is used to indicate whether the direction of the inclusion is allowed to rotate relative to the local coordinate system. It is used only for 2D and 3D continuum elements in Abaqus/Standard and is relevant only when geometric nonlinearity is accounted for during the step (NLGEOM=YES).

Set DIRECTION UPDATE=YES (default) to indicate that the direction of the inclusion follows the deformation of the material point and not the averaged rigid body motion.

Set DIRECTION UPDATE=NO to indicate that the direction of the inclusion remains constant in the local coordinate system.

If you omit this parameter, the direction of the inclusion remains constant relative to the local coordinate system.

FORMULATION

Set FORMULATION=MT (default) to use the Mori-Tanaka model as the homogenization method. This setting is the only formulation available in Abaqus/Explicit.

Set FORMULATION=VOIGT to use the Voigt model as the homogenization method.

Set FORMULATION=REUSS to use the Reuss model as the homogenization method.

Set FORMULATION=INVERSED MT to use the inverse Mori-Tanaka model as the homogenization method.

Set FORMULATION=BALANCED to use the interpolative double inclusion model as the homogenization method.

Set FORMULATION=CHAMIS to use the Chamis model as the homogenization method.

Set FORMULATION=CCA to use the composite cylinder assemblage model as the homogenization method.

This parameter is ignored if the **CONCENTRATION TENSOR* option is used in the same material definition.

ISOTROPIZATION

Set ISOTROPIZATION=ALLISO (default) to use the isotropic projection of the stiffness to compute the entire concentration tensor. This setting is the only isotropization method available in Abaqus/Explicit.

Set ISOTROPIZATION=E-ISO to use the isotropic projection of the matrix stiffness to compute only the Eshelby tensor part of the concentration tensor.

Set ISOTROPIZATION=P-ISO to use the isotropic projection of the matrix stiffness to compute only the Hill's tensor part of the concentration tensor.

ODF ADJUST

Set ODF ADJUST=YES (default) to indicate that the orientation distribution function (ODF) reconstructed from the orientation tensor is adjusted to remove any negative values.

Set ODF ADJUST=NO to indicate that the orientation distribution function (ODF) reconstructed from the orientation tensor is used directly without adjustment.

ORIENTATION AVERAGING

Set ORIENTATION AVERAGING=DISCRETE (default in Abaqus/Standard) to compute integrals numerically during homogenization calculations of composites containing short fibers with specified fiber dispersion.

Set ORIENTATION AVERAGING=DIRECT (default in Abaqus/Explicit) to compute integrals directly during homogenization calculations of composites containing short fibers with specified fiber dispersion. ORIENTATION AVERAGING=DIRECT can be used only when UNIFORM MATRIX STRAIN=YES.

UNIFORM MATRIX STRAIN

This parameter is meaningful only for multiphase composites with multiple inclusions or void constituents. See *Multistep Homogenization*.

Set UNIFORM MATRIX STRAIN=NO (default) to indicate that Voigt homogenization is performed between pseudo-grains in the second homogenization step.

Set UNIFORM MATRIX STRAIN=YES to indicate that a direct Mori-Tanaka homogenization is used for the composite. This approach assumes that the average strain in the matrix is uniform across all pseudo-grains.

In Abaqus/Explicit the matrix strain is always assumed to be uniform across all pseudo-grains.

There are no data lines associated with this option.

*MEDIA TRANSPORT: Activate or deactivate a periodic media.

This option is used to activate or deactivate a periodic media.

Products: Abaqus/Explicit

Type: History data

Level: Step

References:

• *Periodic Media Analysis*

There are no parameters associated with this option.

Data lines to define a periodic media:

First line:

1. Periodic media name.
2. The word “ACTIVE”' or “INACTIVE”.

Repeat this data line as often as necessary.

*MEMBRANE SECTION: Specify section properties for membrane elements.

This option is used to assign section properties to a set of membrane elements. Section properties include thickness, thickness change behavior, material definition, and material orientation.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- *Membrane Elements*
- **DISTRIBUTION*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the membrane elements for which the section properties are being defined.

MATERIAL

Set this parameter equal to the name of the material to be used with these elements.

Optional parameters:

CONTROLS

In an Abaqus/Explicit analysis, set this parameter equal to the name of a section controls definition (see *Section Controls*) to be used to specify a nondefault hourglass control formulation option or scale factors.

In an Abaqus/Standard analysis, set this parameter equal to the name of a section controls definition to be used to specify the enhanced hourglass control formulation (see *Section Controls*) or to be used in a subsequent Abaqus/Explicit import analysis.

DENSITY

Set this parameter equal to a mass per unit surface area of the membrane.

If this parameter is used, the mass of the membrane includes a contribution from this parameter in addition to any contribution from the material definition.

MEMBRANE THICKNESS

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the name of a distribution (*Distribution Definition*) to define spatially varying thickness.

The distribution used to define membrane thickness must have a default value. The default thickness is used by any membrane element assigned to the membrane section that is not specifically assigned a value in the distribution.

The NODAL THICKNESS and MEMBRANE THICKNESS parameters are mutually exclusive.

NODAL THICKNESS

Include this parameter to indicate that the membrane thickness should not be read from the data lines but should be interpolated from the thickness specified at the nodes with the **NODAL THICKNESS* option.

The NODAL THICKNESS and MEMBRANE THICKNESS parameters are mutually exclusive.

ORIENTATION

Set this parameter equal to the name given for the **ORIENTATION* option to be used to define a local coordinate system for material calculations in the elements in this set.

POISSON

This parameter is relevant only in a large-deformation analysis. Set it equal to a nonzero value to cause the thickness to change as a function of membrane strains. The value of the POISSON parameter must be between −1.0 and 0.5. A value of 0.5 will enforce incompressible behavior of the element. POISSON=0.0 means that the thickness will not change.

Set this parameter equal to MATERIAL in an Abaqus/Explicit analysis to cause the thickness to change based on the element material definition.

The default is POISSON=0.5 in Abaqus/Standard and POISSON=MATERIAL in Abaqus/Explicit.

Data line for a constant thickness membrane:

First (and only) line:

1. Section thickness.

To define a continuously varying thickness membrane:

No data lines are used with this option when the NODAL THICKNESS or MEMBRANE THICKNESS parameters are specified; any value input on the data line will be ignored. Instead, the **NODAL THICKNESS* or MEMBRANE THICKNESS option is used to define the section thickness.

*MODAL DAMPING: Specify damping for modal dynamic analysis.

This option is used to specify damping for mode-based procedures. It is usually used in conjunction with the *SELECT EIGENMODES option for selecting eigenmodes for modal superposition. If the *SELECT EIGENMODES option is not used, all eigenmodes extracted in the prior *FREQUENCY step will be used with the damping values specified under the *MODAL DAMPING option. If the *MODAL DAMPING option is not used, zero damping values are assumed.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Material Damping*
- *About Dynamic Analysis Procedures*

Optional, mutually exclusive parameters (if no parameter is specified, Abaqus assumes that the modal damping coefficients are provided on the data lines):

STRUCTURAL

Include this parameter to select structural damping, which means that the damping is proportional to the internal forces but opposite in direction to the velocity. This parameter can be used only with the *STEADY STATE DYNAMICS, *RANDOM RESPONSE, or SIM-based *MODAL DYNAMIC or *COMPLEX FREQUENCY procedures (see *Mode-Based Steady-State Dynamic Analysis*, *Random Response Analysis*, *Transient Modal Dynamic Analysis*, and *Complex Eigenvalue Extraction*). The value of the damping constant, *s*, that multiplies the internal forces is entered on the data line.

VISCOUS

Set VISCOUS=FRACTION OF CRITICAL DAMPING to select modal damping using the damping coefficients given in this option. The data lines after the keyword line specify the modal damping values to be used in the analysis.

Set VISCOUS=COMPOSITE to select composite modal damping using the damping coefficients that have been calculated in the *FREQUENCY step (*Natural Frequency Extraction*). These coefficients are calculated from the material damping factors given on the *DAMPING material definition option for procedures that use the traditional architecture and from the composite modal damping factors provided on the *COMPOSITE MODAL DAMPING option for SIM-based analyses that use the Lanczos eigensolver (*Material Damping*). Composite modal damping can be used only with DEFINITION=MODE NUMBERS.

Set VISCOUS=RAYLEIGH to indicate that the damping for a particular mode is defined as $\alpha_M m_M + \beta_M k_M$, where α_M and β_M are factors defined on the first data line of the option and m_M is the modal mass and k_M is the modal stiffness for mode *M*.

This parameter supersedes the previously used MODAL and RAYLEIGH parameters.

Optional parameters:

DEFINITION

Set DEFINITION=MODE NUMBERS (default) to indicate that the damping values are given for the specified mode numbers.

Set DEFINITION=FREQUENCY RANGE to indicate that the damping values are given for the specified frequency ranges. Frequency ranges can be discontinuous.

If both the **MODAL DAMPING* and **SELECT EIGENMODES* options are used in the same step, the DEFINITION parameter must be set equal to the same value in both options.

FIELD

Set FIELD=ALL (default) to indicate that the damping values are to be applied to both structural and acoustic modes.

Set FIELD=MECHANICAL to indicate that the damping values are to be applied only to structural modes.

Set FIELD=ACOUSTIC to indicate that the damping values are to be applied only to acoustic modes.

This option can be used only with VISCOUS=FRACTION OF CRITICAL DAMPING and DEFINITION=FREQUENCY RANGE for uncoupled structural and acoustic modes.

Data lines to define a fraction of critical damping by specifying mode numbers (if no parameters are specified or if VISCOUS=FRACTION OF CRITICAL DAMPING and DEFINITION=MODE NUMBERS):

First line:

1. Mode number of the lowest mode of a range.
2. Mode number of the highest mode of a range. (If this entry is left blank, it is assumed to be the same as the previous entry so that values are being given for one mode only.)
3. Fraction of critical damping, ξ.

Repeat this data line as often as necessary to define modal damping for different modes.

Data lines to define Rayleigh damping by specifying mode numbers (VISCOUS=RAYLEIGH and DEFINITION=MODE NUMBERS):

First line:

1. Mode number of the lowest mode of a range.
2. Mode number of the highest mode of a range. (If this entry is left blank, it is assumed to be the same as the previous entry so that values are being given for one mode only.)
3. Mass proportional damping, α_M.
4. Stiffness proportional damping, β_M.

Repeat this data line as often as necessary to define modal damping for different modes.

Data lines to define composite modal damping (VISCOUS=COMPOSITE):

First line:

1. Mode number of the lowest mode of a range.
2. Mode number of the highest mode of a range. (If this entry is left blank, it is assumed to be the same as the previous entry so that values are being given for one mode only.)
3. Scaling factor for the mass weighted fraction of composite critical damping calculated in the frequency analysis. If omitted, the default value is 1.0. If the mass weighted fraction is excluded from the analysis, enter a value of 0.0. Only relevant in SIM-based analyses.

4. Scaling factor for the stiffness weighted fraction of composite critical damping calculated in the frequency analysis. If omitted, the default value is 1.0. If the stiffness weighted fraction is excluded from the analysis, enter a value of 0.0. Only relevant in SIM-based analyses.

Repeat this data line as often as necessary to define modal damping for different modes.

Data lines to define structural damping by specifying mode numbers (STRUCTURAL and DEFINITION=MODE NUMBERS):

First line:

1. Mode number of the lowest mode of a range.
2. Mode number of the highest mode of a range. (If this entry is left blank, it is assumed to be the same as the previous entry so that values are being given for one mode only.)
3. Damping factor, s.

Repeat this data line as often as necessary to define modal damping for different modes.

Data lines to define a fraction of critical damping by specifying frequency ranges (VISCOUS=FRACTION OF CRITICAL DAMPING and DEFINITION=FREQUENCY RANGE):

First line:

1. Frequency value (in cycles/time).
2. Fraction of critical damping, ξ.

Repeat this data line as often as necessary to define modal damping for different frequencies. Abaqus will interpolate linearly between frequencies and keep the damping value constant and equal to the closest specified value outside the frequency range.

Data lines to define Rayleigh damping by specifying frequency ranges (VISCOUS=RAYLEIGH and DEFINITION=FREQUENCY RANGE):

First line:

1. Frequency value (in cycles/time).
2. Mass proportional damping, α_M.
3. Stiffness proportional damping, β_M.

Repeat this data line as often as necessary to define modal damping for different frequencies. Abaqus will interpolate linearly between frequencies and keep the damping value constant and equal to the closest specified value outside the frequency range.

Data lines to define structural damping by specifying frequency ranges (STRUCTURAL and DEFINITION=FREQUENCY RANGE):

First line:

1. Frequency value (in cycles/time).
2. Damping factor, s.

Repeat this data line as often as necessary to define modal damping for different frequencies. Abaqus will interpolate linearly between frequencies and keep the damping value constant and equal to the closest specified value outside the frequency range.

*MODAL DYNAMIC: Dynamic time history analysis using modal superposition.

This option is used to provide dynamic time history response as a linear perturbation procedure using modal superposition.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

• *Transient Modal Dynamic Analysis*

Optional parameters:

CONTINUE

Set CONTINUE=NO (default) to specify that this step is not to carry over the initial conditions from the results of the preceding step. In this case the initial displacements are zero, and the initial velocities are taken from the **INITIAL CONDITIONS*, TYPE=VELOCITY option if it is used; otherwise, they are zero. Step time begins at zero.

Set CONTINUE=YES to specify that this step is to carry over the initial conditions from the end of the immediately preceding **MODAL DYNAMIC* step or static perturbation step. If this preceding step is a **MODAL DYNAMIC* step, both the velocities and the displacements from the end of this step are used as the initial conditions for the current step. If this preceding step is a static perturbation step, the displacements from this step are used as the initial displacements for the current step and the initial velocities are taken from the **INITIAL CONDITIONS*, TYPE=VELOCITY option if it is used; otherwise, they are zero. Step time is continued from the immediately preceding **MODAL DYNAMIC* or static perturbation step.

SOLVER

Set SOLVER=STATE SPACE (default) to use the state space solver.

Set SOLVER=CONVENTIONAL to use the conventional solver.

Data line for a transient modal dynamic analysis:

First (and only) line:

1. Time increment to be used.
2. Time period.

*MODAL FILE: Write generalized coordinate (modal amplitude) data or eigendata to the results file during a mode-based dynamic or eigenvalue extraction procedure.

This option is used during mode-based dynamic or eigenvalue extraction procedures to control the writing of generalized coordinate (modal amplitude and phase) values or eigendata to the Abaqus/Standard results file.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Unsupported; Abaqus/CAE reads output from the output database file only.

References:

• *Output to the Data and Results Files*

Optional parameters:

FREQUENCY

This parameter is valid only in mode-based dynamic procedures.

Set this parameter equal to the output frequency, in increments. The output will always be written to the results file at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

Data lines to request modal output in the results file during mode-based dynamic procedures:

First line:

1. Give the identifying keys for the variables to be written to the results file. The keys are defined in the “Modal variables” section of *Abaqus/Standard Output Variable Identifiers*.

Repeat this data line as often as necessary.

To write eigendata during an eigenvalue extraction procedure:

No data lines are required; the eigendata are written automatically.

*MODAL OUTPUT: Write generalized coordinate (modal amplitude) data to the output database during a mode-based dynamic or complex eigenvalue extraction procedure.

This option is used during a mode-based dynamic or complex eigenvalue extraction procedure to write generalized coordinate (modal amplitude and phase) values to the Abaqus/Standard output database. It must be used in conjunction with the **OUTPUT*, HISTORY option.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Output to the Output Database*
- **OUTPUT*

Optional parameters:

VARIABLE

Set VARIABLE=ALL to indicate that all modal variables applicable to this procedure and material type should be written to the output database.

If this parameter is omitted, the modal variables requested for output must be specified on the data lines.

Data lines to request modal output:

First line:

1. Give the identifying keys for the variables to be written to the output database. The keys are defined in the "Modal variables" section of *Using Abaqus/Standard Output Variable Identifiers*.

Repeat this data line as often as necessary to define the modal variables to be written to the output database.

*MODAL PRINT: Print generalized coordinate (modal amplitude) data during a mode-based dynamic procedure.

This option is used during mode-based dynamic procedures to control the printed output of generalized coordinate (modal amplitude and phase) values.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

• *Output to the Data and Results Files*

Optional parameters:

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

Data lines to request modal output in the data file:

First line:

1. Give the identifying keys for the variables to be printed. The keys are defined in the “Modal variables” section of *Abaqus/Standard Output Variable Identifiers*.

Repeat this data line as often as necessary: each line defines a table.

*MODEL CHANGE: Remove or reactivate elements and contact pairs.

This option is used to remove or reactivate elements or contact pairs during an analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *Element and Contact Pair Removal and Reactivation*
- *Removing and Reactivating Contact Pairs*

Required, mutually exclusive parameters:

ACTIVATE

Include this parameter in any step during an analysis to indicate that elements or contact pairs may need to be removed or added during a subsequent restart analysis.

ADD

Include this parameter to indicate that the elements or contact pairs involved are being reactivated during the step.

Set ADD=STRAIN FREE (or include the ADD parameter without a value) to specify strain-free reactivation for stress/displacement elements or to reactivate other elements or contact pairs.

Set ADD=WITH STRAIN to specify that stress/displacement elements are reactivated with strain. This option is not relevant for contact pairs.

REMOVE

Include this parameter to indicate that the elements or contact pairs involved are being removed during the step.

Optional parameters:

TYPE

This parameter can be used only with the parameters ADD or REMOVE.

Set TYPE=ELEMENT (default) to remove or reactivate elements. Set TYPE=CONTACT PAIR to remove or reactivate contact pairs.

Data lines to remove/reactivate elements (TYPE=ELEMENT):

First line:

1. Give a list of element numbers and/or element set names that are involved in the removal or reactivation.

Repeat this data line as often as necessary.

Data lines to remove/reactivate contact pairs (TYPE=CONTACT PAIR):

First line:

1. Secondary surface name used in the contact pair being removed or reactivated.
2. Main surface name used in the contact pair being removed or reactivated. For self-contact the main surface name is omitted or is the same as the secondary surface name.

Repeat this data line as often as necessary.

No data lines are used with this option when the ACTIVATE parameter is included

*MOHR COULOMB: Specify the Mohr-Coulomb plasticity model.

This option is used to define the yield surface and flow potential parameters for elastic-plastic materials that use the Mohr-Coulomb plasticity model. It must be used in conjunction with the **MOHR COULOMB HARDENING* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Mohr-Coulomb Plasticity*
- **MOHR COULOMB HARDENING*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the material parameters other than temperature. If this parameter is omitted, it is assumed that the material properties are constant or depend only on temperature. See *Material Data Definition* for more information.

DEVIATORIC ECCENTRICITY

Set this parameter equal to the flow potential eccentricity in the deviatoric plane, e. This feature allows the shape of flow potential in the deviatoric stress space to be controlled independently of the angle of friction. If this parameter is omitted, the deviatoric eccentricity is calculated by default as $e = (3 - \sin\phi) / (3 + \sin\phi)$, where ϕ is the Mohr-Coulomb angle of friction defined on the data lines. The range of values e can have is $1/2 < e \leq 1$.

ECCENTRICITY

Set this parameter equal to the flow potential eccentricity in the meridional plane, ϵ. The meridional eccentricity is a small positive number that defines the rate at which the flow potential approaches its asymptote. The default is $\epsilon = 0.1$.

Data lines to define a Mohr-Coulomb plasticity model:

First line:

1. Friction angle, ϕ, at high confining pressure in the p–$R_{mc}q$ plane. Give the value in degrees. The friction angle can range from $0° \leq \phi \leq 89.9°$.
2. Dilation angle, ψ, at high confining pressure in the p–$R_{mw}q$ plane. Give the value in degrees. The dilation angle can range from $0° \leq \psi \leq 89.9°$. Abaqus will set ψ to $0.1°$ when $\phi > 0$ and the dilation angle is not specified or is specified as zero.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the material parameters on temperature and other predefined field variables.

*MOHR COULOMB HARDENING: Specify hardening for the Mohr-Coulomb plasticity model.

This option is used to define piecewise linear hardening/softening behavior for a material defined by the Mohr-Coulomb plasticity model. It must be used in conjunction with the **MOHR COULOMB* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Mohr-Coulomb Plasticity*
- **MOHR COULOMB*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the cohesion yield stress, in addition to temperature. If this parameter is omitted, it is assumed that the cohesion yield stress depends only on the plastic strain and, possibly, on temperature. See *Material Data Definition* for more information.

Data lines to define Mohr-Coulomb hardening:

First line:

1. Cohesion yield stress.
2. Absolute value of the corresponding plastic strain. (The first tabular value entered must always be zero.)
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the cohesion yield stress on plastic strain and, if needed, on temperature and other predefined field variables.

*MOISTURE SWELLING: Define moisture-driven swelling.

This option is used to define the moisture-driven swelling of the solid skeleton in a partially saturated porous medium. It can be used in the analysis of coupled wetting liquid flow and porous medium stress.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Moisture Swelling*

There are no parameters associated with this option.

Data lines to define moisture-driven swelling:

First line:

1. Volumetric moisture swelling strain, ε^{ms}.
2. Saturation, s. This value must lie in the range $0.0 \leq s \leq 1.0$.

Repeat this data line as often as necessary to define the ε^{ms}–s relationship from $s = 0.0$ to $s = 1.0$ in increasing values of s.

*MOLECULAR WEIGHT: Define the molecular weight of an ideal gas species.

This option is used to define the molecular weight of an ideal gas species. It can be used only in conjunction with the *FLUID BEHAVIOR option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Fluid Cavity Definition*
- *Inflator Definition*
- *FLUID BEHAVIOR
- *FLUID CAVITY

There are no parameters associated with this option.

Data line to define the molecular weight:

First (and only) line:

1. Molecular weight of the ideal gas species. (Units of $MMOLE^{-1}$.)

*MONITOR: Define a degree of freedom to monitor.

This option is used to choose a node and degree of freedom to monitor the progress of the solution in the status file. In Abaqus/Standard the information will also be written to the message file.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

• *About Output*

Required parameters:

DOF

Set this parameter equal to the degree of freedom to be monitored at the node. In an Abaqus/Explicit analysis the degree of freedom will be in the global coordinate system. If the **TRANSFORM* option is used at the node in an Abaqus/Standard analysis, the degree of freedom is in the local, transformed, system.

NODE

Set this parameter equal to either the node number to be monitored or the name of a node set containing the node to be monitored. If the name of a node set is chosen, the node set must contain exactly one node.

Optional parameters:

FREQUENCY

This parameter applies only to Abaqus/Standard analyses.

This parameter will only affect output to the message file. Set this parameter equal to the output frequency in increments. The output will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

There are no data lines associated with this option.

*MOTION: Specify motions as a predefined field.

This option is used to specify motions of node sets or individual nodes during cavity radiation heat transfer analysis, to define the motion of a reference frame in steady-state transport analysis, or to define the velocity of the material transported through the mesh during a static analysis. It can also be used to specify the velocity of an element set representing a conductor transported through the mesh in an eddy current analysis.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Cavity Radiation in Abaqus/Standard*
- *Steady-State Transport Analysis*
- *Static Stress Analysis*
- *Eddy Current Analysis*
- *UMOTION*

Specifying motion during cavity radiation heat transfer analysis, steady-state transport analysis, or static analysis

Optional, mutually exclusive parameters:

ROTATION

Include this parameter to define a rigid body rotation about an axis.

TRANSLATION

Include this parameter to give the *x*-, *y*-, and *z*-components of translation in the global coordinate system or in the local coordinate system if **TRANSFORM* was used at these nodes. Translational motion is the default.

USER

Include this parameter to indicate that magnitudes of motion will be defined in user subroutine *UMOTION*. If this parameter is used, any magnitudes defined by the data lines can be redefined in the user subroutine.

The value of the TYPE parameter is not relevant when this parameter is used.

This parameter cannot be used for steady-state transport analysis.

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve (defined in the **AMPLITUDE* option) that gives the time variation of the motion throughout the step (*Amplitude Curves*).

If this parameter is omitted and the translational or rotational motion is given with TYPE=DISPLACEMENT, the default is a RAMP function. If the translational or rotational motion

is given with TYPE=VELOCITY, the default is a STEP function for cavity radiation analysis and a RAMP function for steady-state transport analysis.

DEFINITION

This parameter is used to indicate how the rotational axis is defined.

Set DEFINITION=COORDINATES (default) to define the rotational axis by giving the coordinates of two points a and b.

Set DEFINITION=NODES to define the rotational axis by giving node numbers for points a and b. The locations of these nodes at the beginning of a step determine the position of the rotational axis.

TYPE

This parameter is used to specify whether the magnitude is in the form of a displacement or a velocity.

Set TYPE=DISPLACEMENT (default for cavity radiation analysis) to give translational or rotational displacement values.

Set TYPE=VELOCITY (only type available for steady-state transport and static analysis) to give translational or rotational velocities. Velocity histories for cavity radiation problems can be specified as illustrated in the discussion on prescribing large rotations in *Cavity Radiation in Abaqus/Standard*.

Data lines to define translational motion (TRANSLATION):

First line:

1. Node set label or node number.
2. First translational component of motion prescribed (only degrees of freedom 1, 2, or 3 can be entered). See *Conventions* for a definition of the numbering of degrees of freedom in Abaqus.
3. Last translational component of motion prescribed (only degrees of freedom 1, 2, or 3 can be entered). This field can be left blank if motion for only one component is being prescribed.
4. Magnitude of the translational displacement or velocity. This magnitude will be modified by the **AMPLITUDE* specification if the AMPLITUDE parameter is used.

Repeat this data line as often as necessary to define translational motion for different nodes and degrees of freedom.

Data lines to define rotational motion by specifying coordinates (ROTATION, DEFINITION=COORDINATES):

First line:

1. Node set label or node number.
2. Magnitude of the rotation (in radians) or rotational velocity (in radians/time). This magnitude will be modified by the **AMPLITUDE* specification if the AMPLITUDE parameter is used. The rotation is about the axis defined from point a to point b, where the coordinates of a and b are given next. In steady-state transport analysis, the position and orientation of the rotation axis are applied at the beginning of the step and remain fixed during the step.
3. Global x-component of point a on the axis of rotation.
4. Global y-component of point a on the axis of rotation.

The following data are required only for three-dimensional cases:

11. Global z-component of point a on the axis of rotation.
13. Global x-component of point b on the axis of rotation.
15. Global y-component of point b on the axis of rotation.
17. Global z-component of point b on the axis of rotation.

Repeat this data line as often as necessary to define rotational motion for different nodes.

Data lines to define rotational motion by specifying nodes (ROTATION, DEFINITION=NODES):

First line:

1. Node set label or node number.
2. Magnitude of the rotation (in radians) or rotational velocity (in radians/time). This magnitude will be modified by the **AMPLITUDE* specification if the AMPLITUDE parameter is used. The rotation is about the axis defined from point *a* to point *b*, where the coordinates of *a* and *b* are given next. In static or steady-state transport analysis, the rotational axis is defined by the position of these two nodes at the beginning of the step and remains fixed during the step.
3. Node number of the node at point *a*.

The following data are required only for three-dimensional cases:

9. Node number of the node at point *b*.

Repeat this data line as often as necessary to define rotational motion for different nodes.

Data lines to define motion in user subroutine `UMOTION` (USER):

First line:

1. Node set label or node number.
2. First translational component of motion prescribed (only degrees of freedom 1, 2, or 3 can be entered). See *Conventions* for a definition of the numbering of degrees of freedom in Abaqus.
3. Last translational component of motion prescribed (only degrees of freedom 1, 2, or 3 can be entered). This field can be left blank if motion for only one component is being prescribed.
4. Magnitude of the translational displacement or rotation. This magnitude can be redefined in user subroutine *UMOTION*.

Repeat this data line as often as necessary to define the nodes and degrees of freedom that will have their motion prescribed by user subroutine *UMOTION*.

Specifying motion during an eddy current analysis

Required parameters:

ELEMENT

Include this parameter to specify the motion of an element set.

Optional, mutually exclusive parameters:

ROTATION

Include this parameter to define a rigid body rotation about an axis.

TRANSLATION

Include this parameter to give the *x*-, *y*-, and *z*-components of translation in the global coordinate system or in the local coordinate system if **TRANSFORM* was used at these nodes. Translational motion is the default.

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve (defined in the **AMPLITUDE* option) that gives the time variation of the motion throughout the step (*Amplitude Curves*).

If this parameter is omitted, the default is a STEP function.

Data lines to define translational velocity of motion (TRANSLATION):

First (and only) line:

1. Element set label or element number.
2. First component of the direction cosine of the translational velocity vector.
3. Second component of the direction cosine of the translational velocity vector.
4. Third component of the direction cosine of the translational velocity vector.
5. Magnitude of the translational velocity. This magnitude will be modified by the **AMPLITUDE* specification if the AMPLITUDE parameter is used.

Data line to define rotational velocity of motion (ROTATION):

First (and only) line:

1. Element set label or element number.
2. Magnitude of the rotational velocity (in radians/time). This magnitude will be modified by the **AMPLITUDE* specification if the AMPLITUDE parameter is used. The rotation is about the axis defined from point a to point b, where the coordinates of a and b are given next. The position and orientation of the rotation axis are applied at the beginning of the step and remain fixed during the step.
3. Global x-component of point a on the axis of rotation.
4. Global y-component of point a on the axis of rotation.

The following data are required only for three-dimensional cases:

11. Global z-component of point a on the axis of rotation.
13. Global x-component of point b on the axis of rotation.
15. Global y-component of point b on the axis of rotation.
17. Global z-component of point b on the axis of rotation.

*MPC: Define multi-point constraints.

This option is used to impose constraints between different degrees of freedom of the model.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Interaction module

References:

- *General Multi-Point Constraints*
- *MPC*

Optional parameters:

ALPHA

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the thermal expansion coefficient.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

MODE

This parameter applies only to Abaqus/Standard analyses.

This parameter is used only if the USER parameter is included.

Set MODE=DOF (default) for user subroutine *MPC* to operate in a degree of freedom mode.

Set MODE=NODE for user subroutine *MPC* to operate in a nodal mode.

USER

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to indicate that the constraint is defined in user subroutine *MPC*.

Data lines to define multi-point constraints:

First line:

1. MPC type from *General Multi-Point Constraints* or, if the USER parameter is included, an integer key to be used in user subroutine *MPC* to distinguish between different constraint types.
2. Node numbers or node sets involved in the constraint.

The first 15 nodes or node sets of an MPC must be entered on the first line. If the MPC contains more than 15 nodes, enter 0 on the next line to indicate that it is a continuation line and then continue to enter the following nodes on this line. Any number of continuation lines are allowed. Exactly 15 nodes or node sets must be given on each line except the last line.

*MULLINS EFFECT: Specify Mullins effect material parameters for elastomers.

This option is used to define material constants for the Mullins effect in filled rubber elastomers or for modeling energy dissipation in elastomeric foams. It can be used only with the *HYPERELASTIC or the *HYPERFOAM options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Hyperelastic Behavior of Rubberlike Materials*
- *Hyperelastic Behavior in Elastomeric Foams*
- *Stress Softening in Rubberlike Materials*
- *Stress Softening in Elastomeric Foams*
- *UMULLINS*
- *VUMULLINS*
- *BIAXIAL TEST DATA
- *PLANAR TEST DATA
- *UNIAXIAL TEST DATA

Required parameters:

DEFINITION

Set DEFINITION=OGDEN-ROXBURGH to choose the Ogden-Roxburgh model.

Set DEFINITION=EXPONENTIAL to choose that the damage variable is defined using the exponential form.

Set DEFINITION=SLOW DECAY to choose that the damage variable is defined using the slow decay form.

Set DEFINITION=USER to choose that the damage variable is defined in user subroutine *UMULLINS* in Abaqus/Standard or *VUMULLINS* in Abaqus/Explicit.

Optional parameters:

BETA

This parameter can be used only when the TEST DATA INPUT parameter is used; it defines the value of β while the other coefficients of the Mullins effect model are fitted from the test data. It cannot be specified if both the R and M parameters are also specified (use the data line instead to specify all three parameters). If this parameter is omitted, β will be determined from a nonlinear, least-squares fit of the test data. Allowable values of BETA are $\beta \geq 0$. The M and BETA parameters cannot both be zero.

DEPENDENCIES

Set this parameter equal to the number of field variables, in addition to temperature, on which the material parameters depend. If this parameter is omitted, it is assumed that the material parameters are constant or depend only on temperature. See *Material Data Definition* for more information.

This parameter is not relevant if the DEFINITION=USER or the TEST DATA INPUT parameter is included.

M

This parameter can be used only when the TEST DATA INPUT parameter is used; it defines the value of m while the other coefficients of the Mullins effect model are fitted from the test data. It cannot be specified if both the R and BETA parameters are also specified (use the data line instead to specify all three parameters). If this parameter is omitted, m will be determined from a nonlinear, least-squares fit of the test data. Allowable values of M are $m \geq 0$. The M and BETA parameters cannot both be zero.

PROPERTIES

This parameter can be used only if the DEFINITION=USER parameter is specified. Set this parameter equal to the number of property values needed as data in user subroutine *UMULLINS* in Abaqus/Standard or *VUMULLINS* in Abaqus/Explicit. The default value is 0.

R

This parameter can be used only when the TEST DATA INPUT parameter is used; it defines the value of r while the other coefficients of the Mullins effect model are fitted from the test data. It cannot be specified if both the M and BETA parameters are also specified (use the data line instead to specify all three parameters). If this parameter is omitted, r will be determined from a nonlinear, least-squares fit of the test data. Allowable values of R are $r > 1$.

TEST DATA INPUT

This parameter is relevant only for DEFINITION=OGDEN-ROXBURGH.

Include this parameter if the material constants are to be computed by Abaqus from data taken from simple tests on a material specimen. If this parameter is omitted, the material constants can be given directly on the data lines.

To define the material behavior for the Ogden-Roxburgh model by giving test data:

No data lines are used with this option when the TEST DATA INPUT parameter is specified. The data are given instead under the **BIAXIAL TEST DATA*, **PLANAR TEST DATA*, and **UNIAXIAL TEST DATA* options. These options are applicable except for the case where the damage variable is defined by the user.

Data lines to define the material constants if the DEFINITION=OGDEN-ROXBURGH parameter is specified, and the TEST DATA INPUT parameter is omitted:

First line:

1. r.
2. m. (Units of FL^{-2}.)
3. β (If this entry is left blank, the default value is taken to be 0.0 in Abaqus/Standard and 0.1 in Abaqus/Explicit.)
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the material constants as a function of temperature and other predefined field variables.

Data lines to define the material properties if the DEFINITION=USER parameter is specified:

No data lines are needed if the PROPERTIES parameter is omitted or set to 0. Otherwise, first line:

1. Material properties, eight per line.

Repeat this data line as often as necessary to define the material properties.

Data lines to define the material constants if the DEFINITION=EXPONENTIAL parameter is specified:

Data lines to define the material constants if the DEFINITION=EXPONENTIAL parameter is specified:

First line:

1. η_F.
2. U_h. (Units of FL^{-2}.)
3. α.
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the material constants as a function of temperature and other predefined field variables.

Data lines to define the material constants if the DEFINITION=SLOW DECAY parameter is specified:

First line:

1. η_F.
2. U_h. (Units of FL^{-2}.)
3. α.
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the material constants as a function of temperature and other predefined field variables.

*MULTIPHYSICS LOAD: Specify a Butler-Volmer surface load to model solid electrodes.

This option is used to specify a Butler-Volmer load in coupled thermal-electrochemical, coupled thermal-electrochemical-structural, and coupled thermal-electrochemical-structural–pore pressure analyses to model solid electrodes.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

• *Modeling Solid Electrodes in Lithium Metal Batteries*

Required parameters:

DEFINITION

Set DEFINITION=BUTLER-VOLMER to define Butler-Volmer kinetics on the surface.

TYPE

Set TYPE=SURFACE to define the load on an element surface.

Data lines to define surface-based loading using an element or element set:

First line:

1. Element number or element set label.
2. Distributed multiphysics load type label (see *Three-Dimensional Solid Element Library*).
3. Blank.
4. Table collection name to use for this element or element set.

Repeat this data line as often as necessary to define loading for various elements or element sets.

Data lines to define surface-based loading using a surface definition:

First line:

1. Surface name.
2. Multiphysics load type label QS (uniform).
3. Blank.
4. Table collection name to use for this surface.

Repeat this data line as often as necessary to define loading for various surfaces.

N

This section describes each Abaqus input option that begins with N, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **NCOPY*
- **NETWORK STIFFNESS RATIO*
- **NFILL*
- **NGEN*
- **NMAP*
- **NO COMPRESSION*
- **NODAL ENERGY RATE*
- **NODAL THICKNESS*
- **NODE*
- **NODE FILE*
- **NODE OUTPUT*
- **NODE PRINT*
- **NODE RESPONSE*
- **NONLINEAR BH*
- **NONSTRUCTURAL MASS*
- **NORMAL*
- **NO TENSION*
- **NSET*

*NCOPY: Create nodes by copying.

This option is used to copy a node set to create a new node set.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Not applicable; copying portions of sketches and instancing of parts serve similar purposes.

References:

• *Node Definition*

Required parameters:

CHANGE NUMBER

Set this parameter equal to an integer that will be added to each of the existing node numbers to define the node numbers of the nodes being created.

OLD SET

Set this parameter equal to the name of the node set being copied. This set will be used for the copy operation with the nodes that belong to it at the time this **NCOPY* option appears in the input file.

Required, mutually exclusive parameters:

POLE

Include this parameter if the new nodes are created by projecting the nodes in the old set from the pole node. Each new node will be located such that the corresponding old node is equidistant between the pole node and the new node.

This parameter is particularly useful for creating nodes associated with infinite elements (*Infinite Elements*).

REFLECT

Set REFLECT=LINE to create the new nodes by reflection through a line.

Set REFLECT=MIRROR to create the new nodes by reflection through a plane.

Set REFLECT=POINT to create the new nodes by reflection through a point.

SHIFT

Include this parameter if the new nodes are to be created by translation and/or rotation of the nodes in the old node set. If both translation and rotation are specified, the translation is applied once before the rotation.

Optional parameters:

MULTIPLE

This parameter is used with the SHIFT parameter to define the number of times the rotation should be applied. The default is MULTIPLE=1.

NEW SET

Set this parameter equal to the name of the node set to which the nodes created by the operation will be assigned. This new node set will be unsorted if the OLD SET was unsorted and if the NEW SET does not already exist. Otherwise, this new node set will be a sorted set.

If this parameter is omitted, the newly created nodes are not assigned to a node set.

Data lines if the SHIFT parameter is included:

First line:

1. Value of the translation to be applied in the *X*-direction.
2. Value of the translation to be applied in the *Y*-direction.
3. Value of the translation to be applied in the *Z*-direction.

Second line:

1. *X*-coordinate of the first point defining the rotation axis (point *a* in *Figure 1*).
2. *Y*-coordinate of the first point defining the rotation axis.
3. *Z*-coordinate of the first point defining the rotation axis.
4. *X*-coordinate of the second point defining the rotation axis (point *b* in *Figure 1*).
5. *Y*-coordinate of the second point defining the rotation axis.
6. *Z*-coordinate of the second point defining the rotation axis.
7. Angle of rotation about the axis *a*–*b*, in degrees.

Data line if REFLECT=LINE:

First (and only) line:

1. *X*-coordinate of the first point defining the reflection line (point *a* in *Figure 2*).
2. *Y*-coordinate of the first point defining the reflection line.
3. *Z*-coordinate of the first point defining the reflection line.
4. *X*-coordinate of the second point defining the reflection line (point *b* in *Figure 2*).
5. *Y*-coordinate of the second point defining the reflection line.
6. *Z*-coordinate of the second point defining the reflection line.

Data lines if REFLECT=MIRROR:

First line:

1. *X*-coordinate of the first point defining the reflection plane (point *a* in *Figure 3*).
2. *Y*-coordinate of the first point defining the reflection plane.
3. *Z*-coordinate of the first point defining the reflection plane.
4. *X*-coordinate of the second point defining the reflection plane (point *b* in *Figure 3*).
5. *Y*-coordinate of the second point defining the reflection plane.
6. *Z*-coordinate of the second point defining the reflection plane.

Second line:

1. *X*-coordinate of the third point defining the reflection plane (point *c* in *Figure 3*).
2. *Y*-coordinate of the third point defining the reflection plane.
3. *Z*-coordinate of the third point defining the reflection plane.

Data line if REFLECT=POINT:

First (and only) line:

1. *X*-coordinate of the reflection point (point *a* in *Figure 4*).
2. *Y*-coordinate of the reflection point.
3. *Z*-coordinate of the reflection point.

Data line if the POLE parameter is included:

First (and only) line:

1. Number of the pole node (optional: it must have been defined already).
2. *X*-coordinate of the pole node (point *a* in *Figure 5* only required if the pole node number was not entered).
3. *Y*-coordinate of the pole node (only required if the pole node number was not entered).
4. *Z*-coordinate of the pole node (only required if the pole node number was not entered).

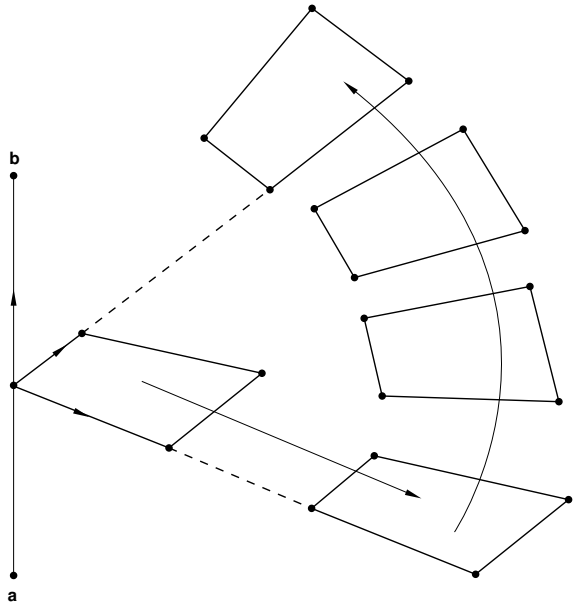


Figure 1: *NCOPY, SHIFT option.

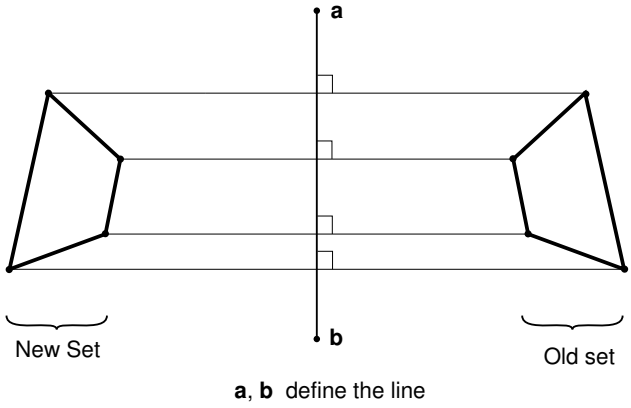
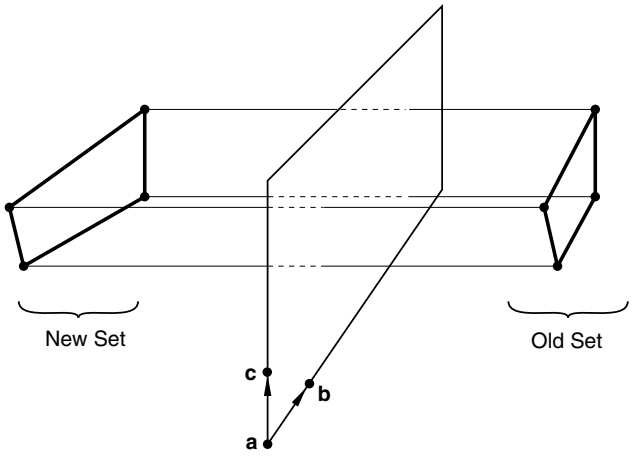
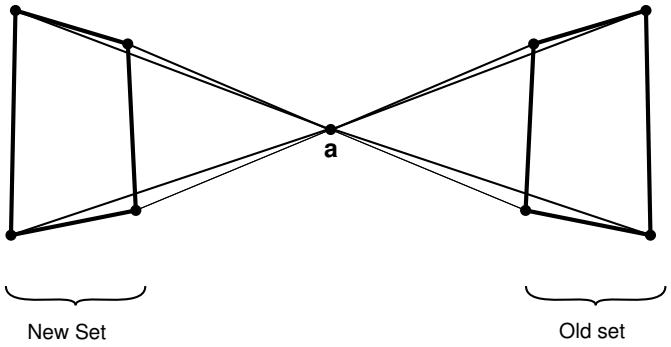


Figure 2: *NCOPY, REFLECT=LINE option.



a, **b**, **c** define the mirror plane

Figure 3: *NCOPY, REFLECT=MIRROR option.



a is the point through which the nodes are reflected

Figure 4: *NCOPY, REFLECT=POINT option.

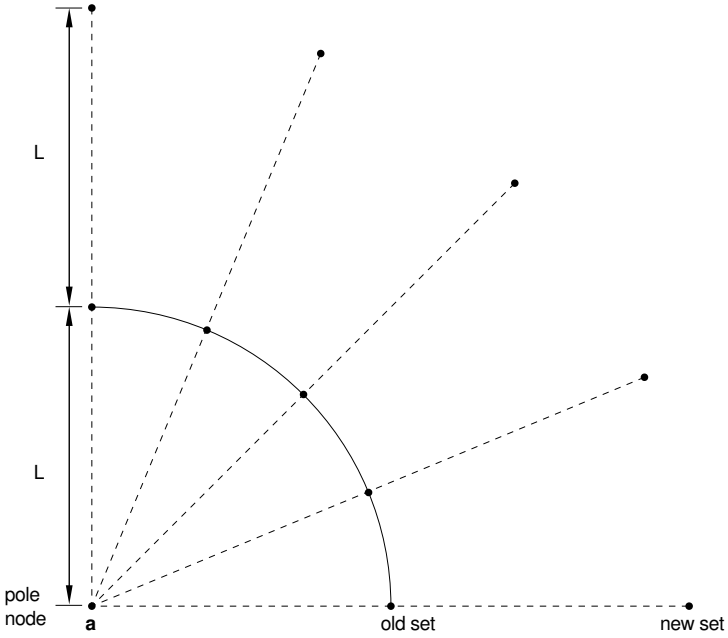


Figure 5: *NCOPY, POLE option.

*NETWORK STIFFNESS RATIO: Specify stiffness ratios for viscoelastic networks.

This option is used to specify the stiffness ratios for viscoelastic networks for a material model defined using the parallel rheological framework.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Parallel Rheological Framework*
- **VISCOELASTIC*

Required parameters:

N

Set this parameter equal to the number of viscoelastic networks in the model.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the stiffness ratios, in addition to temperature. If this parameter is omitted, the hardening behavior does not depend on field variables. See *Material Data Definition* for more information.

Data lines to define the stiffness ratio if N=1:

First line:

1. Stiffness ratio for the viscoelastic network 1, r_1.
2. Temperature.
3. First field variable.
4. Etc., up to eight entries per line.

Subsequent lines (only needed if the number of entries is greater than eight):

1. Seventh field variable.
2. Etc., up to eight entries per line.

Repeat this set of data lines as often as necessary to define the dependence of the stiffness ratios on temperature and other predefined field variables.

Data lines to define the stiffness ratio if N=2:

First line:

1. Stiffness ratio for the viscoelastic network 1, r_1.
2. Stiffness ratio for the viscoelastic network 2, r_2.
3. Temperature.
4. First field variable.
5. Etc., up to eight entries per line.

Subsequent lines (only needed if the number of entries is greater than eight):

1. Sixth field variable.
2. Etc., up to eight entries per line.

Repeat this set of data lines as often as necessary to define the dependence of the stiffness ratios on temperature and other predefined field variables.

Data lines to define the stiffness ratio for higher values of N:

Data lines:

1. The data lines for higher values of N follow the same pattern. First, give r_i for i from 1 to N. Then, give the temperature. Finally, give the values of the field variables. Exactly eight data values should be given on each line.

*NFILL: Fill in nodes in a region.

This option is used to generate nodes for a region of a mesh by filling in nodes between two bounds.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Not applicable; nodes are generated when you *mesh* the model.

References:

• *Node Definition*

Optional parameters:

BIAS

Include this parameter to bias the spacing of the nodes being generated toward one end of the line of nodes being generated. Set this parameter equal to the ratio of adjacent distances between nodes along each line of nodes generated, as the nodes go from the first bounding node set to the second. If the BIAS parameter is less than one, the nodes are concentrated toward the first bounding node set; if it is greater than one, the nodes are concentrated toward the second bounding set. The value of the parameter must be positive.

The BIAS and SINGULAR parameters are mutually exclusive.

NSET

Set this parameter equal to the name of the node set to which the nodes created by this operation (including the bounding nodes) will be assigned. Node sets generated by this option are always sorted node sets.

SINGULAR

This parameter applies only to Abaqus/Standard analyses.

Include this parameter for fracture mechanics calculations with second-order isoparametric elements to create a “quarter point” crack-tip element and bias the remaining elements. Set SINGULAR=1 or 2, depending on whether the first or the second bounding node set represents the crack tip.

The BIAS and SINGULAR parameters are mutually exclusive.

TWO STEP

This parameter is meaningful only if the BIAS parameter is used. When this parameter is included, the BIAS value is applied only at each second interval along the line; therefore, the midside nodes of second-order elements will be at the middle of the two adjacent intervals.

Data lines to fill in nodes between two bounds:

First line:

1. Name of the node set defining the first bound of the region.
2. Name of the node set defining the second bound of the region.
3. Number of intervals along each line between bounding nodes.
4. Increment in node numbers from the node number at the first bound set end. The default is 1.

Repeat this data line as often as necessary, one line per region to be filled by this option.

*NGEN: Generate incremental nodes.

This option is used to generate nodes incrementally.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Not applicable; nodes are generated when you *mesh*the model.

References:

• *Node Definition*

Optional parameters:

LINE

Set LINE=P to generate the nodes along a parabola. In this case the user must define an extra point, the midpoint between the two end points.

Set LINE=C to generate the nodes along a circular arc. In this case the user must define an extra point, the center of the circle.

If this parameter is omitted, the nodes will be generated along a straight line.

NSET

Set this parameter equal to the name of a node set to which the nodes will be assigned. The two end nodes will also be included in the node set. Node sets created or modified with this option will always be sorted.

SYSTEM

Set SYSTEM=RC (default) to define the extra node in a Cartesian coordinate system. Set SYSTEM=C to define the extra node in a cylindrical coordinate system. Set SYSTEM=S to define the extra node in a spherical coordinate system. See *Figure 1*.

Data lines to generate nodes incrementally:

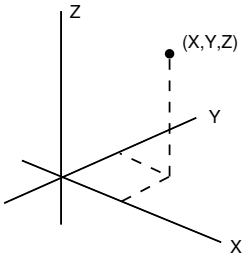
First line:

1. Number of the first end node (it must have been previously defined).
2. Number of the second end node (it must have been previously defined).
3. Increment in the numbers between each node along the line. The default is 1.
4. Number of the third node giving the extra point (if required; it must have been previously defined).
5. First coordinate of the extra point (if required).
6. Second coordinate of the extra point (if required).
7. Third coordinate of the extra point (if required).

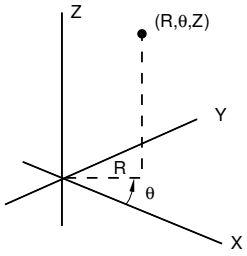
The following entries are used only for a circular arc equal to or larger than 180°:

17. First component of a normal to the circular arc.
19. Second component of a normal to the circular arc.
21. Third component of a normal to the circular arc.

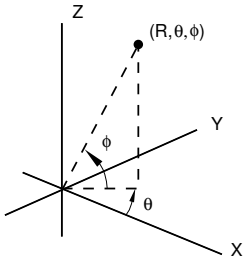
Repeat this data line as often as necessary.



Rectangular Cartesian
(default)



Cylindrical
(θ and ϕ are given in degrees)



Spherical

Figure 1: Coordinate systems.

*NMAP: Map nodes from one coordinate system to another and rotate, translate, or scale the nodal coordinates.

Map nodes from one coordinate system to another and rotate, translate, or scale the nodal coordinates.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

Abaqus/CAE: Unsupported; meshing techniques in the *Mesh module* are usually preferable.

References:

• *Node Definition*

Required parameters:

NSET

Set this parameter equal to the name of the node set containing the nodes to be mapped. The nodes that are mapped are those that belong to this set at the time this option is encountered.

TYPE

Set TYPE=ROTATION to introduce a rotation of a specified angle about a given axis defined by two points *a* and *b* (or by the coordinates of these points). The origin of rotation is given by a third point *c* (or by the coordinates of this point).

Set TYPE=TRANSLATION to introduce a translation along a given axis defined by two nodes *a* and *b* (or by the coordinates of these points).

Set TYPE=SCALE to scale each axis with respect to one node *a* (or by the coordinates of this point).

Set TYPE=RECTANGULAR to introduce a simple shift or rotation. Point *a* in *Figure 1* defines the origin of the local rectangular coordinate system defining the map. The local $\hat{\boldsymbol{x}}$-axis is defined by the line joining points *a* and *b*. The local $\hat{\boldsymbol{x}}$–$\hat{\boldsymbol{y}}$ plane is defined by the plane passing through points *a*, *b*, and *c*.

Set TYPE=CYLINDRICAL to map from cylindrical coordinates. Point *a* in *Figure 1* defines the origin of the local cylindrical coordinate system defining the map. The line going through point *a* and point *b* defines the $\hat{\boldsymbol{z}}$-axis of the local cylindrical coordinate system. The local $\hat{\boldsymbol{r}}$–$\hat{\boldsymbol{z}}$ plane for $\boldsymbol{\theta = 0}$ is defined by the plane passing through points *a*, *b*, and *c*.

Set TYPE=DIAMOND to map from skewed Cartesian coordinates. Point *a* in *Figure 1* defines the origin of the local diamond coordinate system defining the map. The line going through point *a* and point *b* defines the $\hat{\boldsymbol{x}}$-axis of the local coordinate system. The line going through point *a* and point *c* defines the $\hat{\boldsymbol{y}}$-axis of the local coordinate system. The line going through point *a* and point *d* defines the $\hat{\boldsymbol{z}}$-axis of the local coordinate system.

Set TYPE=SPHERICAL to map from spherical coordinates. Point *a* in *Figure 1* defines the origin of the local spherical coordinate system defining the map. The line going through point *a* and point *b* defines the polar axis of the local spherical coordinate system. The plane passing through point *a* and perpendicular to the polar axis defines the $\boldsymbol{\phi = 0}$ plane. The plane passing through points *a*, *b*, and *c* defines the local $\boldsymbol{\theta = 0}$ plane.

Set TYPE=TOROIDAL to map from toroidal coordinates. Point *a* in *Figure 1* defines the origin of the local toroidal coordinate system defining the map. The axis of the local toroidal system lies in the plane defined by points *a*, *b*, and *c*. The *R*-coordinate of the toroidal system is defined by the distance between points *a* and *b*. The line between points *a* and *b* defines the $\phi = 0$ position. For every value of ϕ the θ-coordinate is defined in a plane perpendicular to the plane defined by the points *a*, *b*, and *c* and perpendicular to the axis of the toroidal system. $\theta = 0$ lies in the plane defined by the points by *a*, *b*, and *c*.

Set TYPE=BLENDED to map via blended quadratics in an Abaqus/Standard analysis.

Optional parameters:

DEFINITION

Set DEFINITION =COORDINATES (default) to define the local system, the axis of rotation, the origin of rotation, or the axis of translation by giving the coordinates of the points *a*, *b*, *c*, and *d* whichever appropriate for the chosen type.

Set DEFINITION=NODES to define the local system, the axis of rotation, the origin of rotation, or the axis of translation by giving global node numbers for points *a*, *b*, *c*, and *d* depending on the type. This option cannot be used with TYPE=BLENDED.

Data lines for TYPE=ROTATION, DEFINITION=COORDINATES:

First line:

1. *X*-coordinate of point *a*.
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.
4. *X*-coordinate of point *b*.
5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.

Second line:

1. *X*-coordinate of point *c*.
2. *Y*-coordinate of point *c*.
3. *Z*-coordinate of point *c*.

Third line:

1. The rotation angle in degrees.

Data lines for TYPE=ROTATION, DEFINITION=NODES:

First line:

1. Local node number of point *a*.
2. Local node number of point *b*.

Second line:

1. Local node number of point *c*.

Third line:

1. The rotation angle in degrees.

Data lines for TYPE=TRANSLATION, DEFINITION=COORDINATES:

First line:

1. *X*-coordinate of point *a*.
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.

4. *X*-coordinate of point *b*.
5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.

Second line:

1. The translation magnitude.

Data lines for TYPE=TRANSLATION, DEFINITION=NODES:

First line:

1. Local node number of point *a*.
2. Local node number of point *b*.

Second line:

1. The translation magnitude.

Data lines for TYPE=SCALE, DEFINITION=COORDINATES:

First line:

1. *X*-coordinate of point *a*.
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.

Second line:

1. Scale factor to be applied to the first local coordinate.
2. Scale factor to be applied to the second local coordinate.
3. Scale factor to be applied to the third local coordinate.

Data lines for TYPE=SCALE, DEFINITION=NODES:

First line:

1. Local node number of point *a*.

Second line:

1. Scale factor to be applied to the first local coordinate before mapping.
2. Scale factor to be applied to the second local coordinate before mapping.
3. Scale factor to be applied to the third local coordinate before mapping.

Data lines for TYPE=RECTANGULAR, CYLINDRICAL, DIAMOND, SPHERICAL, or TOROIDAL with DEFINITION=COORDINATES:

First line:

1. *X*-coordinate of point *a* (see *Figure 1*).
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.
4. *X*-coordinate of point *b*.
5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.

Second line:

1. *X*-coordinate of point *c*.
2. *Y*-coordinate of point *c*.
3. *Z*-coordinate of point *c*.

The following fields are needed only for TYPE=DIAMOND:

9. *X*-coordinate of point *d*.
11. *Y*-coordinate of point *d*.
13. *Z*-coordinate of point *d*.

If TYPE=RECTANGULAR is specified and only point *a* is given, the coordinates of the nodes in the set are simply shifted by X_a, Y_a, and Z_a.

Third line:

1. Scale factor to be applied to the first local coordinate before mapping. If the value entered is zero or blank, a scale factor of 1.0 is assumed.
2. Scale factor to be applied to the second local coordinate before mapping. If the value entered is zero or blank, a scale factor of 1.0 is assumed.
3. Scale factor to be applied to the third local coordinate before mapping. If the value entered is zero or blank, a scale factor of 1.0 is assumed.

Data lines for TYPE=RECTANGULAR, CYLINDRICAL, DIAMOND, SPHERICAL, or TOROIDAL with DEFINITION=NODES:

First line:

1. Local node number of point *a*.
2. Local node number of point *b*.

Second line:

1. Local node number of point *c*.

The following field is needed only for TYPE=DIAMOND:

5. Local node number of point *d*.

If TYPE=RECTANGULAR is specified and only point *a* is given, the coordinates of the nodes in the set are simply shifted by X_a, Y_a, and Z_a.

Third line:

1. Scale factor to be applied to the first local coordinate before mapping. If the value entered is zero or blank, a scale factor of 1.0 is assumed.
2. Scale factor to be applied to the second local coordinate before mapping. If the value entered is zero or blank, a scale factor of 1.0 is assumed.
3. Scale factor to be applied to the third local coordinate before mapping. If the value entered is zero or blank, a scale factor of 1.0 is assumed.

Data lines for TYPE=BLENDED:

First line:

1. Node number of the first control node.
2. *X*-coordinate of the point to which this control node is to be mapped.
3. *Y*-coordinate of the point to which this control node is to be mapped.
4. *Z*-coordinate of the point to which this control node is to be mapped.

Second line:

1. Node number of the second control node.
2. *X*-coordinate of the point to which this control node is to be mapped.
3. *Y*-coordinate of the point to which this control node is to be mapped.
4. *Z*-coordinate of the point to which this control node is to be mapped.

Continue, giving up to 20 control nodes, but giving at least the eight corner nodes. If an edge of the blended mapping is to be mapped linearly, the corresponding mid-edge control node can be omitted from the list. This is done by inserting a line with node number 0 only (a blank line) in place of the definition of the control node and its mapped coordinates. The control nodes do not have to be nodes

in the finite element model—they can be nodes used just for mesh generation. Abaqus eliminates any nodes that are not used in the analysis model before doing the analysis.

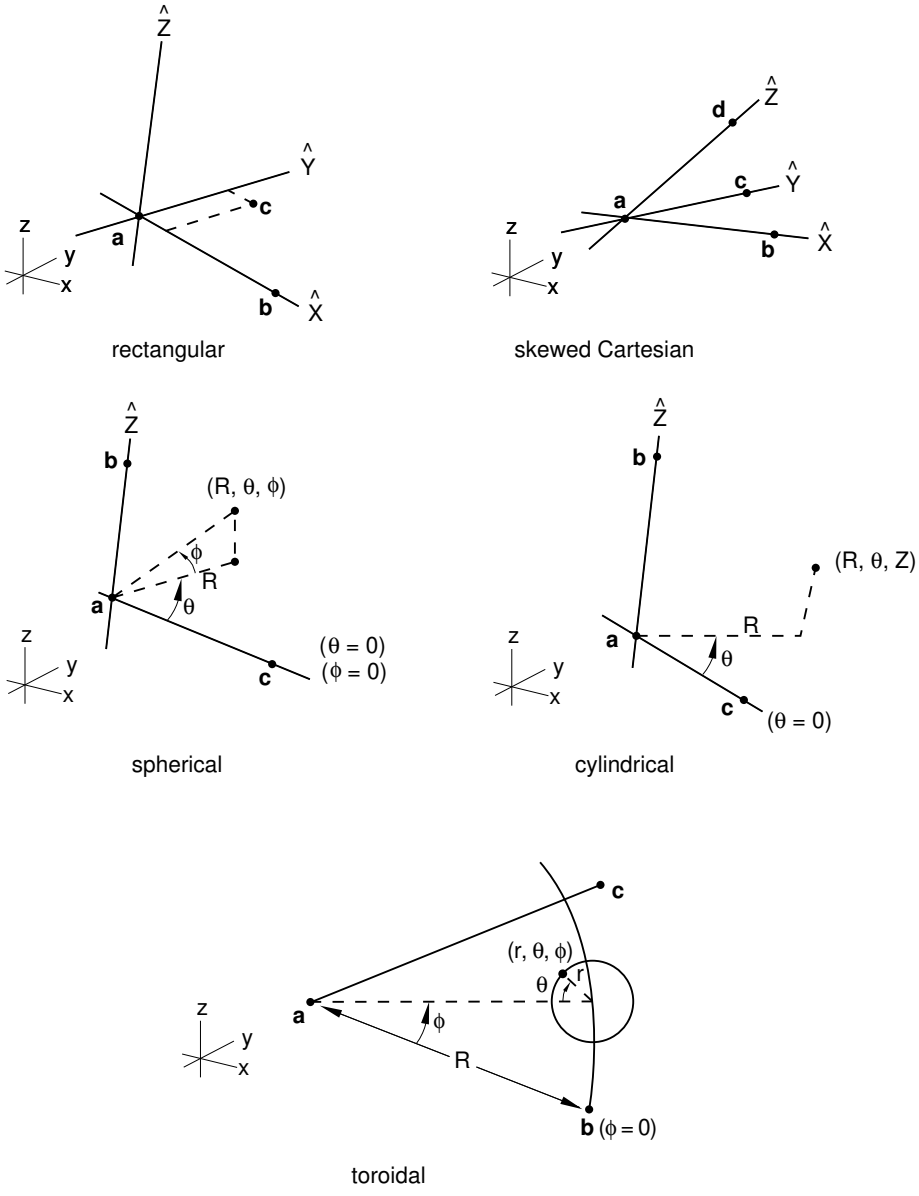


Figure 1: Coordinate systems; angles are in degrees.

*NO COMPRESSION: Introduce a compressive failure theory (tension only materials).

This option is used to modify the elasticity definition so that no compressive stress is allowed. It can be used only in conjunction with the *ELASTIC option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *No Compression or No Tension*
- *ELASTIC

Optional parameters:

CUT OFF STRESS

This parameter is available only in Abaqus/Explicit.

Set this parameter equal to the value of the cutoff stress, σ_c^{cutoff}. The default value is based on a small fraction (10^{-4}) of the dilatational wave modulus of the material.

There are no data lines associated with this option.

*NODAL ENERGY RATE: Define critical energy release rates at nodes.

This option is used to define variable critical energy release rates on a nodal basis. The critical energy release rate data defined with this option is ignored unless the NODAL ENERGY RATE parameter is included on the **FRACTURE CRITERION*, TYPE=VCCT option or the **FRACTURE CRITERION*, TYPE=FATIGUE option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- *Crack Propagation Analysis*
- **FRACTURE CRITERION*

Optional parameters:

GENERATE

Include this parameter to interpolate the critical energy release rates between two bounding nodes or node sets. The critical energy release rates for the bounding nodes or node sets must have been defined earlier. If the node sets do not have the same number of nodes, the extra nodes in the longer set are ignored.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

Data lines when the GENERATE parameter is omitted:

First line:

1. Node set label or node number.
2. G_{IC}, Mode I critical energy release rate.
3. G_{IIC}, Mode II critical energy release rate.
4. G_{IIIC}, Mode III critical energy release rate.

Repeat this data line as often as necessary to define the variation in critical energy release rates.

Data lines when the GENERATE parameter is included:

First line:

1. Node number or node set label that defines the first bound for the generate operation.
2. Node number or node set label that defines the second bound for the generate operation.
3. Number of intervals between the bounding nodes or node sets.
4. Increment in node numbers from the first bounding node or node set.

Repeat this data line as often as necessary to define the variation in critical energy release rates.

*NODAL THICKNESS: Define shell or membrane thickness at nodes.

This option is used to define variable shell or membrane thicknesses on a nodal basis. The thickness data defined with this option will be ignored unless the NODAL THICKNESS parameter is included on either the **SHELL GENERAL SECTION* or the **SHELL SECTION* options for shell elements or on the **MEMBRANE SECTION* option for membrane elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- *Nodal Thicknesses*
- *Membrane Elements*
- *Using a Shell Section Integrated during the Analysis to Define the Section Behavior*
- *Using a General Shell Section to Define the Section Behavior*
- *Line Spring Elements for Modeling Part-through Cracks in Shells*
- **MEMBRANE SECTION*
- **SHELL GENERAL SECTION*
- **SHELL SECTION*

Optional parameters:

GENERATE

Include this parameter to interpolate the thickness between two bounding nodes or node sets. The thickness for the bounding nodes or node sets must have been defined earlier. If the node sets do not have the same number of nodes, the extra nodes in the longer set are ignored.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

Data lines when the GENERATE parameter is omitted:

First line:

1. Node set label or node number.
2. Thickness.

Repeat this data line as often as necessary to define the variation in shell or membrane thickness.

Data lines when the GENERATE parameter is included:

First line:

1. Node number or node set label that defines the first bound for the generate operation.
2. Node number or node set label that defines the second bound for the generate operation.
3. Number of intervals between the bounding nodes or node sets.

4. Increment in node numbers from the first bounding node or node set.

Repeat this data line as often as necessary to define the variation in thickness.

*NODE: Specify nodal coordinates.

This option is used to define a node directly by specifying its coordinates. Nodal coordinates given in this option are in a local system if the **SYSTEM* option is in effect when this option is used.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Mesh module

References:

• *Node Definition*

Optional parameters:

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

NSET

Set this parameter equal to the name of the node set to which these nodes will be assigned. Node sets created or modified with this option will always be sorted.

SYSTEM

Set SYSTEM=R (default) to give coordinates in a rectangular Cartesian coordinate system. Set SYSTEM=C to give coordinates in a cylindrical system. Set SYSTEM=S to give coordinates in a spherical system. See *Figure 1*.

The SYSTEM parameter is entirely local to this option. As the data lines are read, the coordinates given are transformed to rectangular Cartesian coordinates immediately. If the **SYSTEM* option is also in effect, these are local rectangular Cartesian coordinates, which are then immediately transformed to global Cartesian coordinates.

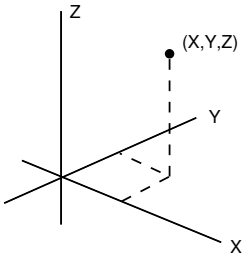
Data lines to define the node:

First line:

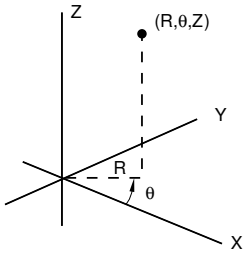
1. Node number.
2. First coordinate of the node.
3. Second coordinate of the node.
4. Third coordinate of the node.
5. First direction cosine of the normal at the node (optional).
6. Second direction cosine of the normal at the node (optional). For nodes entered in a cylindrical or spherical system, this entry is an angle given in degrees.
7. Third direction cosine of the normal at the node (optional). For nodes entered in a spherical system, this entry is an angle given in degrees.

The normal will be used only for element types with rotational degrees of freedom. See *About the Element Library*.

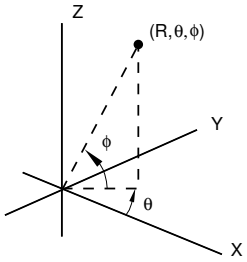
Repeat this data line as often as necessary.



Rectangular Cartesian
(default)



Cylindrical
(θ and ϕ are given in degrees)



Spherical

Figure 1: Coordinate systems.

*NODE FILE: Define results file requests for nodal data.

This option is used to choose the nodal variables that will be written to the results (`.fil`) file in an Abaqus/Standard analysis or to the selected results (`.sel`) file in an Abaqus/Explicit analysis. In an Abaqus/Explicit analysis it must be used in conjunction with the **FILE OUTPUT* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Unsupported; Abaqus/CAE reads output from the output database file only.

References:

- *Output to the Data and Results Files*
- **FILE OUTPUT*

Optional parameters:

FREQUENCY

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the output frequency, in increments. The output will always be written to the results file at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

GLOBAL

This parameter applies only to Abaqus/Standard analyses.

This parameter is relevant only at nodes where the **TRANSFORM* option has been used to define a local coordinate system.

Set GLOBAL=NO to write vector-valued nodal variables in the local directions.

Set GLOBAL=YES (default) to write vector-valued nodal variables in the global directions. This default is the opposite of the default on the **NODE PRINT* option and is used because most postprocessors assume that components are given in the global system.

LAST MODE

This parameter applies only to Abaqus/Standard analyses.

This parameter is useful only during eigenvalue extraction for natural frequencies (*Natural Frequency Extraction*) and for eigenvalue buckling estimation (*Eigenvalue Buckling Prediction*).

Set this parameter equal to the highest mode number for which output is required.

The default value is LAST MODE=*N*, where *N* is the number of modes extracted. If the MODE parameter is used, the default value is LAST MODE=*M*, where *M* is the value of the MODE parameter.

MODE

This parameter applies only to Abaqus/Standard analyses.

This parameter is useful only during eigenvalue extraction for natural frequencies and for eigenvalue buckling estimation. Set this parameter equal to the first mode number for which output is required. The default is MODE=1. See also the LAST MODE parameter. When performing a **FREQUENCY*

analysis, the normalization will follow the format set by the NORMALIZATION parameter. Otherwise, the normalization is such that the largest displacement component in the mode has a magnitude of 1.0.

NSET

Set this parameter equal to the name of the node set for which the output is being written to the results file. If this parameter is omitted, the output will be written for all nodes in the model.

Data lines to request nodal output in the results or selected results file:

First line:

1. Give the identifying keys for the variables to be written to the results or selected results file. The keys are defined in the "Nodal variables" section of *Abaqus/Standard Output Variable Identifiers* and *Abaqus/Explicit Output Variable Identifiers*.

Repeat this data line as often as necessary to define the nodal variables to be written to the results or selected results file.

*NODE OUTPUT: Define output database requests for nodal data.

This option is used to write nodal variables to the output database. It must be used in conjunction with the *OUTPUT option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Output to the Output Database*
- *OUTPUT

One of the following mutually exclusive parameters is required when the *NODE OUTPUT option is used in conjunction with the *OUTPUT, HISTORY option for Abaqus/Standard or Abaqus/Explicit:

NSET

Set this parameter equal to the name of the node set for which this output request is being made.

MONITOR

This parameter applies only to Abaqus/Standard analyses. It is relevant only at the nodes used for monitoring the solution at selected nodes within substructures.

Include this parameter to obtain history output of solution variables at the substructure monitor nodes in the substructure usage analysis.

This parameter cannot be used with any other parameters. No data lines are required when this parameter is used.

SURFACE

Set this parameter equal to the name of the surface for which this output request is being made.

TRACER SET

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the name of the tracer set for which this output request is being made.

Optional parameters when the *NODE OUTPUT option is used in conjunction with the *OUTPUT, FIELD option:

DISPLAY

This parameter applies only to Abaqus/Standard analyses.

This parameter is relevant only at the nodes of the display elements used for the coarse visualization of substructures.

Include this parameter to obtain output of displacements at the nodes of display elements representing substructures in the substructure usage analysis. If you request output of nodal displacements for all the nodes in the model (the NSET and EXTERIOR parameters are omitted), you do not need to use this parameter.

This parameter cannot be used with any other parameters. No data lines are required when this parameter is used.

EXTERIOR

This parameter applies only to Abaqus/Standard and Abaqus/Explicit analyses.

Include this parameter to restrict output to only nodes that belong to the exterior three-dimensional elements.

If this parameter and the NSET parameter are omitted, output will be written for all the nodes in the model.

NSET

Set this parameter equal to the name of the node set for which this output request is being made.

If this parameter and the EXTERIOR parameter are omitted, output will be written for all the nodes in the model.

SURFACE

Set this parameter equal to the name of the surface for which this output request is being made.

TRACER SET

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the name of the tracer set for which this output request is being made.

This parameter is valid only for displacement output requests.

Optional parameter when the *NODE OUTPUT option is used in conjunction with the *OUTPUT, HISTORY option:

GLOBAL

This parameter applies only to Abaqus/Standard and Abaqus/Explicit analyses.

This parameter is relevant only at nodes where the **TRANSFORM* option has been used to define a local coordinate system.

Set GLOBAL=NO to write vector-valued nodal variables in the local directions.

Set GLOBAL=YES (default) to write vector-valued nodal variables in the global directions. This default is the opposite of the default on the **NODE PRINT* option and is used because most postprocessors assume that components are given in the global system.

Optional parameters:

VARIABLE

Set VARIABLE=ALL to indicate that all nodal variables applicable to this procedure and material type should be written to the output database.

Set VARIABLE=PRESELECT to indicate that the default nodal output variables for the current procedure type should be written to the output database. Additional output variables can be requested on the data lines.

If this parameter is omitted, the nodal variables requested for output must be specified on the data lines.

Data lines to request nodal output:

First line:

1. Specify the identifying keys for the variables to be written to the output database. The keys are defined in *Abaqus/Standard Output Variable Identifiers* and *Abaqus/Explicit Output Variable Identifiers*.

Repeat this data line as often as necessary to define the nodal variables to be written to the output database.

*NODE PRINT: Define print requests for nodal variables.

This option is used to provide tabular printed output of nodal variables (displacements, reaction forces, etc.) in the data file.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

• *Output to the Data and Results Files*

Optional parameters:

FREQUENCY

Set this parameter equal to the output frequency in increments. The output will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

GLOBAL

This parameter is relevant only at nodes where the **TRANSFORM* option has been used to define a local coordinate system.

Set GLOBAL=NO (default) to obtain printout of vector-valued nodal variables in the local directions.

Set GLOBAL=YES to obtain printout of vector-valued nodal variables in the global directions.

LAST MODE

This parameter is useful only during eigenvalue extraction for natural frequencies (*Natural Frequency Extraction*), complex eigenvalue extraction (*Complex Eigenvalue Extraction*), and eigenvalue buckling estimation (*Eigenvalue Buckling Prediction*). Set this parameter equal to the highest mode number for which output is required.

The default value is LAST MODE=*N*, where *N* is the number of modes extracted. If the MODE parameter is used, the default value is LAST MODE=*M*, where *M* is the value of the MODE parameter.

MODE

This parameter is useful only during natural frequency extraction, complex eigenvalue extraction, and eigenvalue buckling estimation. Set this parameter equal to the first mode number for which output is required. The default is MODE=1. See also the LAST MODE parameter. When performing a **FREQUENCY* analysis, the normalization will follow the format set by the NORMALIZATION parameter. Otherwise, the normalization is such that the largest displacement component in the mode has a magnitude of 1.0.

NSET

Set this parameter equal to the name of the node set for which this output request is being made. If this parameter is omitted, the output will be printed for all of the nodes in the model.

SUMMARY

Set SUMMARY=YES (default) to obtain a summary and the locations of the maximum and minimum values in each column of the table. Set SUMMARY=NO to suppress this summary.

TOTALS

Set TOTALS=YES to print the total of each column in the table. This is useful, for example, to sum reaction forces in a particular direction. The default is TOTALS=NO.

Data lines to request nodal output in the data file:

First line:

1. Give the identifying keys for the variables to be printed in a table for this node set. The keys are defined in the “Nodal variables” portion of *Abaqus/Standard Output Variable Identifiers*.

Repeat this data line as often as necessary: each line defines a table. If this line is omitted, no nodal output will be printed to the data file.

*NODE RESPONSE: Define nodal responses for design sensitivity analysis.

This option is used in a direct design sensitivity analysis to write nodal response sensitivities to the output database. It is also used in an adjoint design sensitivity analysis to write a nodal response and its corresponding sensitivities with respect to the appropriate design variables to the output database. It must be used in conjunction with the **DESIGN RESPONSE* option.

Products: Abaqus/Design

Type: History data

Level: Step

References:

- *Direct Design Sensitivity Analysis*
- *Adjoint Design Sensitivity Analysis*
- **DESIGN RESPONSE*

Optional parameters:

LOWER BOUND

This parameter is meaningful only in an adjoint design sensitivity analysis.

This parameter is used to specify a lower bound on the design response value below which adjoint sensitivities are not calculated. That is, if the value of the design response value is below the specified lower bound, then only the design response value is written to the output database, but not the corresponding adjoint sensitivities.

NSET

Set this parameter equal to the name of the node set for which the nodal design response value and its sensitivity output are calculated.

ORIENTATION

This parameter is meaningful only in an adjoint design sensitivity analysis.

Set this parameter equal to the name of an orientation definition to be used to define a local coordinate system for the node response.

OPERATOR

This parameter is meaningful only in an adjoint design sensitivity analysis.

Set OPERATOR=MAX to indicate that the maximum value of the design response across the node set is calculated.

Set OPERATOR= MAXABS to indicate that the maximum absolute value of the design response across the node set is calculated. This parameter is not available for design responses in a steady-state dynamic analysis.

Set OPERATOR=MIN to indicate that the minimum value of the design response across the node set is calculated. This value is not supported for direct steady-state dynamic adjoint sensitivity analysis.

Set OPERATOR=MINABS to indicate that the minimum absolute value of the design response across the node set is calculated. This parameter is not available for design responses in a steady-state dynamic analysis.

Set OPERATOR=SUB to indicate that the difference between the maximum and the minimum values of the design response across the node set is calculated. This value is not supported for direct steady-state dynamic adjoint sensitivity analysis.

Set OPERATOR=AVE to calculate the average value of the design response over all the nodes in the node set.

Set OPERATOR=SUM (default) to calculate the sum of the design response over all the nodes in the node set.

RANGE OPERATOR

This parameter is meaningful only in direct steady-state dynamic adjoint design sensitivity analysis.

Set RANGE OPERATOR=AVE to calculate the average value of the design response over the range of frequencies specified for the node response.

Set RANGE OPERATOR=MAX (default) to calculate the maximum value of the design response over the range of frequencies specified for the node response.

Set RANGE OPERATOR=SUM to calculate the sum of the design response over the range of frequencies specified for the node response.

UPPER BOUND

This parameter is meaningful only in an adjoint design sensitivity analysis.

This parameter is used to specify an upper bound on the design response value above which adjoint sensitivities are not calculated. In other words if the value of the design response value is above the specified upper bound, then only the design response value is written to the output database, but not the corresponding adjoint sensitivities.

Data lines to request nodal response value and sensitivity output:

First line (only line in a static or a transient dynamic adjoint design sensitivity analysis):

1. Specify the identifying keys for the responses whose values and sensitivities are to be written to the output database. Only one response key can be specified in an adjoint design sensitivity analysis. The valid keys are listed in *Direct Design Sensitivity Analysis* and *Adjoint Design Sensitivity Analysis*.

In a direct design sensitivity analysis, repeat this data line as often as necessary to define the nodal responses whose values and sensitivities are to be written to the output database.

Second line (required only for a direct steady-state dynamic adjoint sensitivity analysis):

1. Lower limit of the frequency range for the node response, in cycles/time.
2. Higher limit of the frequency range for the node response, in cycles/time.

*NONLINEAR BH: Specify nonlinear magnetic behavior of a soft magnetic material.

This option is used to specify nonlinear magnetic behavior of a soft magnetic material.

Products: Abaqus/Standard

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Magnetic Permeability*

Required parameters:

DIR

Set this parameter equal to the local direction (1, 2, or 3) for which the material behavior is defined in the data lines to follow; set DIR=0 to define material behavior that is inactive in the current model but is available for future use. The **NONLINEAR BH* option must be repeated with different values of the DIR parameter to define independent magnetic behavior in three orthogonal directions.

For isotropic nonlinear magnetic behavior, include the **NONLINEAR BH* option only once (DIR=1, 2, or 3).

For orthotropic nonlinear magnetic behavior, the **NONLINEAR BH* option must be included three times (DIR=1, DIR=2, and DIR=3).

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the nonlinear magnetic behavior, in addition to temperature. If this parameter is omitted, the nonlinear magnetic behavior does not depend on field variables. See *Material Data Definition* for more information.

Data lines to define nonlinear magnetic behavior:

First line:

1. Magnitude of the magnetic flux density vector in the direction specified by the DIR parameter.
2. Magnitude of the magnetic field vector in the direction specified by the DIR parameter.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the magnetic flux density on the magnetic field and, if needed, on temperature and other predefined field variables.

*NONSTRUCTURAL MASS: Specify mass contribution to the model from nonstructural features.

This option is used to include the mass contribution from nonstructural features in the model. The nonstructural mass can be applied over an element set that contains solid, shell, membrane, surface, beam, or truss elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: *Property module* and *Interaction module*.

References:

• *Nonstructural Mass Definition*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the elements over which a given nonstructural mass is to be distributed.

UNITS

Set UNITS=TOTAL MASS to specify the nonstructural mass in the units of "mass."

Set UNITS=MASS PER VOLUME to specify the nonstructural mass in the units of "mass per unit volume."

Set UNITS=MASS PER AREA to specify the nonstructural mass in the units of "mass per unit area." This value is valid only for an element set that contains conventional shells, membranes, and/or surface elements.

Set UNITS=MASS PER LENGTH to specify the nonstructural mass in the units of "mass per unit length." This value is valid only for an element set that contains beam and/or truss elements.

Optional parameters:

DISTRIBUTION

This parameter is relevant only when UNITS=TOTAL MASS.

Set DISTRIBUTION=MASS PROPORTIONAL (default) to distribute the total nonstructural mass among the members of the element set region in proportion to the element structural mass. The underlying structural density over the element set region is scaled uniformly; therefore, the center of mass for the element set region is not altered.

Set DISTRIBUTION=VOLUME PROPORTIONAL to distribute the total nonstructural mass among the members of the element set region in proportion to the element volume in the initial configuration. A uniform value is added to the underlying structural density over the element set region; therefore, if the region has nonuniform structural density, the center of mass for the element set region may be altered.

Data line for UNITS=TOTAL MASS:

First (and only) line:

1. Mass magnitude of the nonstructural feature for distribution over the element set region. Mass, not weight, should be given.

Abaqus does not use any specific physical units, so the user's choice must be consistent.

Data line for UNITS=MASS PER VOLUME:

First (and only) line:

1. Mass per unit volume of the nonstructural feature for application over the element set region.

Abaqus does not use any specific physical units, so the user's choice must be consistent.

Data line for UNITS=MASS PER AREA:

First (and only) line:

1. Mass per unit area of the nonstructural feature for application over the element set region.

Abaqus does not use any specific physical units, so the user's choice must be consistent.

Data line for UNITS=MASS PER LENGTH:

First (and only) line:

1. Mass per unit length of the nonstructural feature for application over the element set region.

Abaqus does not use any specific physical units, so the user's choice must be consistent.

*NORMAL: Specify a particular normal direction.

This option is used to define alternative nodal normals for elements. In an Abaqus/Standard analysis it can also be used to define alternative normals for contact surfaces.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance, Assembly

References:

- *Normal Definitions at Nodes*
- *Beam Element Cross-Section Orientation*

Optional parameters:

TYPE

Set TYPE=ELEMENT (default) to allow the alternative normal definition of elements.

Set TYPE=CONTACT SURFACE to allow the alternative normal definition of contact surfaces in an Abaqus/Standard analysis.

BEAM NORMAL

This parameter applies to beam elements only, and it can be used only with TYPE=ELEMENT.

Set BEAM NORMAL=RULE BASED (default) to use a rule-based approach to define nodal normals. In some situations, the specified nodal normals might be reversed.

Set BEAM NORMAL=FIXED to specify that nodal normals will be used as they are specified, without any rule-based reversal.

Data lines to define normals for elements (TYPE=ELEMENT):

First line:

1. Element number or element set label.
2. Node number or node set label.
3. First component of normal in global coordinates.
4. Second component of normal in global coordinates.
5. Third component of normal in global coordinates.

Repeat this data line as often as necessary to define the normals.

Data lines to define normals for contact surfaces (TYPE=CONTACT SURFACE):

First line:

1. Main surface name.
2. Node number or node set label (on main surface).
3. First component of normal in global coordinates.
4. Second component of normal in global coordinates.
5. Third component of normal in global coordinates.

Repeat this data line as often as necessary to define the normals.

*NO TENSION: Introduce a tension failure theory (compression only material).

This option is used to modify the elasticity definition so that no tensile stress is allowed. It can be used only in conjunction with the **ELASTIC* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *No Compression or No Tension*
- **ELASTIC*

Optional parameters:

CUT OFF STRESS

This parameter is available only in Abaqus/Explicit.

Set this parameter equal to the value of the cutoff stress, σ_t^{cutoff}. The default value is based on a small fraction (10^{-4}) of the dilatational wave modulus of the material.

There are no data lines associated with this option.

*NSET: Assign nodes to a node set.

This option assigns nodes to a node set.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Part, Part instance, Assembly, Model, Step

Abaqus/CAE: Set toolset

References:

• *Node Definition*

Required parameters:

NSET

Set this parameter equal to the name of the node set to which the nodes will be assigned.

Optional parameters:

ELSET

Set this parameter equal to the name of a previously defined element set. The nodes that define the elements that belong to this element set at the time this option is encountered will be assigned to the node set specified. The UNSORTED parameter cannot be used with this parameter.

The ELSET, GENERATE, and SURFACE parameters are mutually exclusive.

GENERATE

If this parameter is included, each data line should give a first node, n_1; a last node; n_2; and the increment in node numbers between these nodes, *i*. Then, all nodes going from n_1 to n_2 in steps of *i* will be added to the set. *i* must be an integer such that $(n_2 - n_1)/i$ is a whole number (not a fraction).

The ELSET, GENERATE, and SURFACE parameters are mutually exclusive.

INSTANCE

Set this parameter equal to the name of the part instance that contains the nodes listed on the data line. This parameter can be used only at the assembly level and is intended to be used as a shortcut to the naming convention. It can be used only in a model defined in terms of an assembly of part instances.

INTERNAL

Abaqus/CAE uses the INTERNAL parameter to identify sets that are created internally. The INTERNAL parameter is used only in models defined in terms of an assembly of part instances. The default is to omit the INTERNAL parameter.

SURFACE

Set this parameter equal to the name of a previously defined surface. The nodes that are on the surface at the time this option is encountered will be assigned to the specified node set. The UNSORTED parameter cannot be used with this parameter.

The ELSET, GENERATE, and SURFACE parameters are mutually exclusive.

UNSORTED

If this parameter is included, the nodes in this node set will be assigned to the set (or added to the set if it already exists) in the order in which they are given. This parameter is ignored if the ELSET or the SURFACE parameter is used.

If this parameter is omitted, the nodes in the set are sorted into ascending order of their node numbers, with duplicates eliminated.

Data lines if the GENERATE parameter is omitted:

First line:

1. List of nodes or node set labels to be assigned to this node set. Only previously defined node sets can be assigned to another node set.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

Data lines if the GENERATE parameter is included:

First line:

1. First node in the set.
2. Last node in the set.
3. Increment in node numbers between nodes in the set. The default is 1.

Repeat this data line as often as necessary.

There are no data lines when the ELSET or the SURFACE parameter is specified

O

This section describes each Abaqus input option that begins with O, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **ONE-STEP INVERSE*
- **OPERATOR OUTPUT*
- **ORIENTATION*
- **ORNL*
- **OUTPUT*

*ONE-STEP INVERSE: Perform a one-step inverse analysis of a sheet metal part.

This option is used to perform a one-step inverse analysis of a sheet metal part.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- **STEP*
- *One-Step Inverse Analysis*

Required parameters:

UNFOLD

This parameter is used to perform a one-step inverse analysis to obtain the initial shape of a sheet metal part given its final deformed configuration and a blank surface.

There are no data lines associated with this option.

*OPERATOR OUTPUT: Write operator output to a SIM document.

This option is used to write operator output to a SIM document in element-by-element or assembled form. It can be used only in a static or in an uncoupled heat transfer analyses.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

• *Generating Matrices from Nonlinear Procedures*

Required parameters:

OPERATOR

Set OPERATOR=ASSEMBLED to write assembled operators.

Set OPERATOR=ELEMENT BY ELEMENT to write element operators.

At least one of the following parameters is required:

FREE BODY RESPONSE

Include this parameter to output the free-body force and moment vectors at convergence.

LOAD

Set LOAD=EXTERNAL (default) to output the external load vector.

Set LOAD=NET to output the net flux vector. This option can be used only in an uncoupled heat transfer analysis.

MASS

Include this parameter to output the mass operator at convergence.

This parameter can be used only in static analyses.

SOLUTION

Include this parameter to output the total solution vector.

In static analyses, the total solution vector includes displacements, rotations, and Lagrange multipliers.

This parameter can be used only in static analyses and with OPERATOR=ASSEMBLED.

STIFFNESS

Include this parameter to output the stiffness matrix at convergence.

This parameter can be used only in static analyses.

Optional parameters:

BOUNDARY CONDITIONS

Set BOUNDARY CONDITIONS=YES (default) to output the STIFFNESS parameter operators with boundary penalty values.

Set BOUNDARY CONDITIONS=NO to output the STIFFNESS parameter operators without boundary penalty values.

ELSET

Use this parameter to write operators for a part of the model.

Set this parameter equal to the name of an element set that contains all the elements in the selected part of the model. By default, operators are generated for all supported elements in the whole model, including internal elements.

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output is always printed at the last increment of each step unless FREQUENCY=0.

The default is FREQUENCY=1.

Set FREQUENCY=0 to suppress the output.

If this parameter and the NUMBER INTERVAL parameter are omitted, output is written at every increment of the analysis.

The FREQUENCY and NUMBER INTERVAL parameters are mutually exclusive.

NSET

Use this parameter to write operators for a part of the model.

Set this parameter equal to the name of a node set that contains all the nodes in the selected part of the model. Generated operators get contributions from all elements that are connected to the nodes of the selected node set. By default, operators are generated for all supported nodes in the whole model, including internal element nodes.

This parameter can be used only with OPERATOR=ASSEMBLED.

NUMBER INTERVAL

Set this parameter equal to the number of intervals during the step at which to write the output database states.

If this parameter and the FREQUENCY parameter are omitted, output is written at every increment of the analysis.

The FREQUENCY and NUMBER INTERVAL parameters are mutually exclusive.

There are no data lines associated with this option.

*ORIENTATION: Define a local axis system for material or element property definition, for kinematic coupling constraints, for free directions for inertia relief loads, or for connectors.

This option is used to define a local coordinate system for definition of material properties; for material calculations at integration points; for element property definitions (for example, connector elements); for output of components of stress, strain, and element section forces; and for kinematic and distributing coupling constraints. A spatially varying local coordinate system can be defined for solid continuum elements and shell elements using distributions (*Distribution Definition*).

The **ORIENTATION* option can also be used to specify local material directions for anisotropic hyperelastic materials with invariant-based formulation (*Invariant-Based Formulation*). The local directions are defined with respect to the local coordinate system.

In Abaqus/Explicit the **ORIENTATION* option can be used to:

- specify fiber dispersion for multiscale materials (*Specifying a Second-Order Orientation Tensor in Abaqus/Explicit*) or linear short-fiber reinforced composite materials (*Defining the Elasticity of a Short-Fiber Reinforced Composite*), and
- initialize the directions of the fill and the warp yarns of a fabric material in the reference configuration. The yarn directions lying in the plane of the fabric are defined with respect to the two in-plane axes of the orthogonal coordinate system.

In Abaqus/Standard the **ORIENTATION* option can be used to define local directions for contact pair interaction properties and spring, dashpot, and JOINTC elements; for definition of local free directions for inertia relief loads; and for output of components of surface variables.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: *Property module*, *Interaction module*, and *Load module*

References:

- *Orientations*
- *ORIENT*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the orientation definition. Orientation names in the same input file must be unique.

Optional parameters:

DEFINITION

Set DEFINITION=COORDINATES (default) to define the local system by giving the coordinates of the three points *a*, *b*, and, optionally, *c* (the origin) appropriate to the SYSTEM choice from *Figure 1*. A spatially varying local coordinate system can be created for this parameter value by using a distribution to define the spatially varying coordinates for points *a* and *b*. Using a distribution to define the coordinates of the optional point *c* is not supported. See *Distribution Definition*.

Set DEFINITION=NODES to define the local system by giving global node numbers for points *a*, *b*, and, optionally, *c* (the origin).

Set DEFINITION=OFFSET TO NODES to define the local system by giving local node numbers (on the element where the orientation is being used) to define the points *a*, *b*, and, optionally, *c* (the origin) in *Figure 1*. This parameter value cannot be used with spring, dashpot, or JOINTC elements. In addition, it cannot be used with the **KINEMATIC COUPLING*, **INERTIA RELIEF*, or **CONTACT PAIR* options.

For all DEFINITION parameter values a spatially varying local coordinate system can be created for solid continuum elements and shell elements by using a distribution to define a spatially varying additional rotation angle α. See *Distribution Definition*.

DISPERSION

This parameter is relevant only for multiscale materials with dispersed fiber directions or linear elastic materials with fiber dispersion specified.

Include this parameter to specify the second-order tensor for the fiber dispersion. The dispersed fiber direction is specified in the material definition.

If no dispersion is specified as part of the orientation definition, the orientation tensor specified in the material definition is used. Otherwise, the orientation tensor specified in the orientation definition overrides the values specified in the material definition.

LOCAL DIRECTIONS

This parameter is relevant only for anisotropic materials with preferred material directions (or fiber directions), such as anisotropic hyperelastic materials or, in Abaqus/Explicit, fabric materials.

Set this parameter equal to the number of local directions that are applicable to the material model (for example, two for a fabric material). The local directions are specified with respect to the orthonormal system at the material point resulting from the current orientation definition. Up to three local directions can be specified as part of the definition of a local orientation system.

For the fabric material in Abaqus/Explicit, the two yarn directions are given with respect to the in-plane axes of the orthonormal system. If no local directions are specified as part of the orientation definition, the local material directions are assumed to match the in-plane axes of the orthonormal system in the reference configuration.

SYSTEM

Set SYSTEM=RECTANGULAR (default) to define a rectangular Cartesian system by the three points *a*, *b*, and *c* shown in *Figure 1*. Point *c* is the origin of the system, point *a* must lie on the X'-axis, and point *b* must lie on the X'-Y' plane. Although not required, it is intuitive to select point *b* such that it is on or near the local Y'-axis.

Set SYSTEM=CYLINDRICAL to define a cylindrical system by giving the two points *a* and *b* on the polar axis of the cylindrical system (*Figure 1*). The local axes are 1=radial, 2=circumferential, 3=axial.

Set SYSTEM=SPHERICAL to define a spherical system by giving the center of the sphere, *a*, and point *b* on the polar axis (*Figure 1*). The local axes are 1=radial, 2=circumferential, 3=meridional.

Set SYSTEM=Z RECTANGULAR to define a rectangular Cartesian system by the three points *a*, *b*, and *c* shown in *Figure 1*. Point *c* is the origin of the system, point *a* must lie on the Z'-axis, and point *b* must lie on the X'-Z' plane. Although not necessary, it is intuitive to select point *b* such that it is on or near the local X'-axis.

Set SYSTEM=USER in an Abaqus/Standard analysis to define the local coordinate system in user subroutine *ORIENT*. The DEFINITION parameter and any data lines associated with the option are ignored if SYSTEM=USER.

Data lines to define an orientation using DEFINITION=COORDINATES:

First line:

1. *X*-coordinate of point *a*.
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.
4. *X*-coordinate of point *b*.
5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.

The following items, the coordinates of point *c* (the origin), are optional and relevant only for SYSTEM=RECTANGULAR and SYSTEM=Z RECTANGULAR. The default location of the origin, *c*, is the global origin.

15. *X*-coordinate of point *c*.
17. *Y*-coordinate of point *c*.
19. *Z*-coordinate of point *c*.

Second line:

1. Local direction about which the additional rotation or rotations are given. The default is the local 1-direction. For shell, membrane, and cohesive elements this direction should have a nonzero component in the direction of the normal to the surface.
2. Additional rotation α defined by either a single scalar value or by a distribution. An orientation defined with a distribution can be used only for solid continuum elements and shell elements. The additional rotation (in degrees) is applied to both directions orthogonal to the specified local direction. The default is zero degrees.

Third line when the LOCAL DIRECTIONS parameter is included and the first local material direction is specified directly:

1. *X*-component of the first local material direction with respect to the orthonormal system at the material point.
2. *Y*-component of the first local material direction with respect to the orthonormal system at the material point.
3. *Z*-component of the first local material direction with respect to the orthonormal system at the material point.

Repeat the above data line to define additional local directions, as needed, with each direction on a separate line.

Third line when the LOCAL DIRECTIONS parameter is included and the first local material direction is defined using a distribution:

1. Distribution name. The data in the distribution define a spatially varying first local material direction with respect to the orthonormal system at the material point.

Repeat the above data line to define additional local directions, as needed, with each direction on a separate line.

Third line when the DISPERSION parameter is included and the components of the second-order tensor are specified directly:

1. *XX*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.
2. *YY*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.

3. *ZZ*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.
4. *XY*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.
5. *XZ*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.
6. *YZ*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.

Third line when the DISPERSION parameter is included and the components of the second-order tensor are defined using a distribution:

1. Distribution name. The data in the distribution define spatially varying components of the second-order orientation tensor with respect to the orthonormal system at the material point.

Data lines to define a spatially varying orientation using a distribution when DEFINITION=COORDINATES:

First line:

1. Distribution name. The data in the distribution define the spatially varying coordinates for points *a* and *b*.

Second line:

1. Local direction about which the additional rotation or rotations are given. The default is the local 1-direction. For shell, membrane, and cohesive elements this direction should have a nonzero component in the direction of the normal to the surface.
2. Additional rotation α defined by either a single scalar value or by a distribution. A local coordinate system defined with a distribution can be used only for solid continuum elements and shell elements. The additional rotation (in degrees) is applied to both directions orthogonal to the specified local direction. The default is zero degrees.

Third line when the LOCAL DIRECTIONS parameter is included and the first local material direction is specified directly:

1. *X*-component of the first local material direction with respect to the orthonormal system at the material point.
2. *Y*-component of the first local material direction with respect to the orthonormal system at the material point.
3. *Z*-component of the first local material direction with respect to the orthonormal system at the material point.

Repeat the above data line to define additional local directions, as needed, with each direction on a separate line.

Third line when the LOCAL DIRECTIONS parameter is included and the first local material direction is defined using a distribution:

1. Distribution name. The data in the distribution define a spatially varying first local material direction with respect to the orthonormal system at the material point.

Repeat the above data line to define additional local directions, as needed, with each direction on a separate line.

Third line when the DISPERSION parameter is included and the components of the second-order tensor are specified directly:

1. *XX*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.
2. *YY*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.

3. *ZZ*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.
4. *XY*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.
5. *XZ*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.
6. *YZ*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.

Third line when the DISPERSION parameter is included and the components of the second-order tensor are defined using a distribution:

1. Distribution name. The data in the distribution define spatially varying components of the second-order orientation tensor with respect to the orthonormal system at the material point.

Data lines to define an orientation using DEFINITION=NODES:

First line:

1. Node number of the node at point *a*.
2. Node number of the node at point *b*.

The next item, specification of point *c* (the origin), is optional and relevant only for SYSTEM=RECTANGULAR and SYSTEM=Z RECTANGULAR. The default location of the origin, *c*, is the global origin.

7. Node number of the node at point *c*.

Second line:

1. Local direction about which the additional rotation or rotations are given. The default is the local 1-direction. For shell, membrane, and cohesive elements this direction should have a nonzero component in the direction of the normal to the surface.
2. Additional rotation α defined by either a single scalar value or by a distribution. A local coordinate system defined with a distribution can be used only for solid continuum elements and shell elements. The additional rotation (in degrees) is applied to both directions orthogonal to the specified local direction. The default is zero degrees.

Third line when the LOCAL DIRECTIONS parameter is included and the first local material direction is specified directly:

1. *X*-component of the first local material direction with respect to the orthonormal system at the material point.
2. *Y*-component of the first local material direction with respect to the orthonormal system at the material point.
3. *Z*-component of the first local material direction with respect to the orthonormal system at the material point.

Repeat the above data line to define additional local directions, as needed, with each direction on a separate line.

Third line when the LOCAL DIRECTIONS parameter is included and the first local material direction is defined using a distribution:

1. Distribution name. The data in the distribution define a spatially varying first local material direction with respect to the orthonormal system at the material point.

Repeat the above data line to define additional local directions, as needed, with each direction on a separate line.

Third line when the DISPERSION parameter is included and the components of the second-order tensor are specified directly:

1. *XX*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.
2. *YY*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.
3. *ZZ*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.
4. *XY*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.
5. *XZ*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.
6. *YZ*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.

Third line when the DISPERSION parameter is included and the components of the second-order tensor are defined using a distribution:

1. Distribution name. The data in the distribution define spatially varying components of the second-order orientation tensor with respect to the orthonormal system at the material point.

Data lines to define an orientation using DEFINITION=OFFSET TO NODES:

First line:

1. Local node number of point *a*.
2. Local node number of point *b*.

The next item, specification of point *c* (the origin), is optional and relevant only for SYSTEM=RECTANGULAR and SYSTEM=Z RECTANGULAR. The default location of the origin, *c*, is the first node of the element (local node number 1).

7. Local node number of point *c*.

Second line:

1. Local direction about which the additional rotation or rotations are given. The default is the local 1-direction. For shell, membrane, and cohesive elements this direction should have a nonzero component in the direction of the normal to the surface.
2. Additional rotation α defined by either a single scalar value or by a distribution. A local coordinate system defined with a distribution can only be used for solid continuum elements and shell elements. The additional rotation (in degrees) is applied to both directions orthogonal to the specified local direction. The default is zero degrees.

Third line when the LOCAL DIRECTIONS parameter is included and the first local material direction is specified directly:

1. *X*-component of the first local material direction with respect to the orthonormal system at the material point.
2. *Y*-component of the first local material direction with respect to the orthonormal system at the material point.
3. *Z*-component of the first local material direction with respect to the orthonormal system at the material point.

Repeat the above data line to define additional local directions, as needed, with each direction on a separate line.

Third line when the LOCAL DIRECTIONS parameter is included and the first local material direction is defined using a distribution:

1. Distribution name. The data in the distribution define a spatially varying first local material direction with respect to the orthonormal system at the material point.

Repeat the above data line to define additional local directions, as needed, with each direction on a separate line.

Third line when the DISPERSION parameter is included and the components of the second-order tensor are specified directly:

1. *XX*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.
2. *YY*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.
3. *ZZ*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.
4. *XY*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.
5. *XZ*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.
6. *YZ*-component of the second-order orientation tensor with respect to the orthonormal system at the material point.

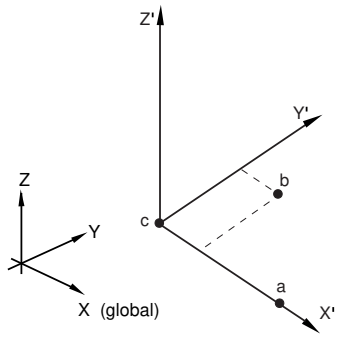
Third line when the DISPERSION parameter is included and the components of the second-order tensor are defined using a distribution:

1. Distribution name. The data in the distribution define spatially varying components of the second-order orientation tensor with respect to the orthonormal system at the material point.

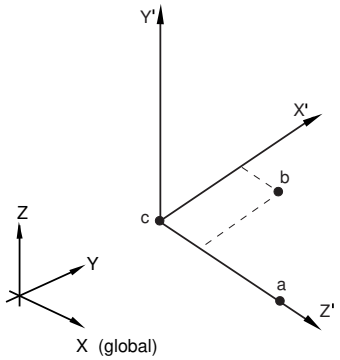
To define an orientation using a user subroutine (SYSTEM=USER):

No data lines are used with this option when SYSTEM=USER is specified. Instead, user subroutine *ORIENT* must be used to define the orientation.

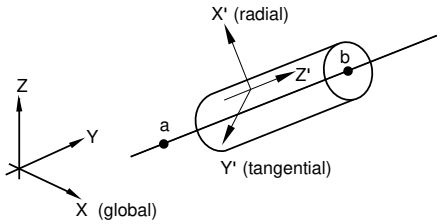
SYSTEM = RECTANGULAR



SYSTEM = Z RECTANGULAR



SYSTEM = CYLINDRICAL



SYSTEM = SPHERICAL

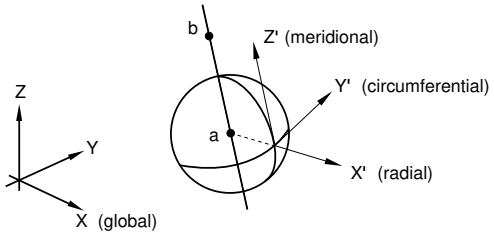


Figure 1: Orientation systems.

*ORNL: Specify constitutive model developed by Oak Ridge National Laboratory.

This option is used to provide plasticity and creep calculations for type 304 and 316 stainless steel according to the specification in Nuclear Standard NEF 9–5 T, “Guidelines and Procedures for Design of Class I Elevated Temperature Nuclear System Components.” It can be used only with the **PLASTIC* option and/or the **CREEP*, LAW=STRAIN option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *ORNL – Oak Ridge National Laboratory Constitutive Model*

Optional parameters:

A

Set this parameter equal to the saturation rates for kinematic shift caused by creep strain, as defined by Equation (15) of Section 4.3.3–3 of the Nuclear Standard. The default value is 0.3, as per that section of the Standard. Set A=0.0 to use the 1986 revision of the Standard.

H

Set this parameter equal to the rate of kinematic shift with respect to creep strain [Equation (7) of Section 4.3.2–1 of the Nuclear Standard]. Set H=0.0 to use the 1986 revision of the Standard. If this parameter is omitted, the value of H is determined according to Section 4.3.3–3 of the 1981 revision of the Standard.

MATERIAL

Set MATERIAL=SS to use the hardening law appropriate to either type 304 or type 316 stainless steel. This is the only option presently available and, thus, the default.

RESET

Include this parameter to invoke the optional $\boldsymbol{\alpha}$ reset procedure described in Section 4.3.5 of the Nuclear Standard. If this parameter is omitted, the $\boldsymbol{\alpha}$ reset procedure is not used.

There are no data lines associated with this option.

*OUTPUT: Define output requests to the output database.

This option is used to write contact, element, energy, nodal, or diagnostic output to the output database. In an Abaqus/Standard analysis it is also used to write modal or radiation output to the output database. In an Abaqus/Explicit analysis it is also used to write incrementation output to the output database. The **CONTACT OUTPUT*, **ELEMENT OUTPUT*, **ENERGY OUTPUT*, **INCREMENTATION OUTPUT*, **MODAL OUTPUT*, **NODE OUTPUT*, and/or **RADIATION OUTPUT* options can be used in conjunction with this option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Output to the Output Database*
- *Abaqus/Standard Output Variable Identifiers*
- *Abaqus/Explicit Output Variable Identifiers*
- *Understanding job diagnostics*
- **CONTACT OUTPUT*
- **ELEMENT OUTPUT*
- **ENERGY OUTPUT*
- **INCREMENTATION OUTPUT*
- **MODAL OUTPUT*
- **NODE OUTPUT*
- **RADIATION OUTPUT*
- **TIME POINTS*

Defining output requests in an Abaqus/Standard analysis

One of the following mutually exclusive parameters is required:

DIAGNOSTICS

Set DIAGNOSTICS=YES (default) to indicate that detailed diagnostic information should be written to the output database. Set DIAGNOSTICS=NO to suppress the output.

FIELD

Include this parameter to indicate that the output requests used in conjunction with the **OUTPUT* option will be written to the output database as field-type output.

HISTORY

Include this parameter to indicate that the output requests used in conjunction with the **OUTPUT* option will be written to the output database as history-type output.

Optional parameters:

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be written to the output database at the last increment of each step. Set FREQUENCY=0 to suppress the output.

If this parameter and the NUMBER INTERVAL, TIME INTERVAL, and TIME POINTS parameters are omitted, output will be written at every increment of the analysis for all procedure types except **DYNAMIC* and **MODAL DYNAMIC*; output will be written every 10 increments for these procedure types.

The FREQUENCY, NUMBER INTERVAL, TIME INTERVAL, and TIME POINTS parameters are mutually exclusive.

MODE LIST

Include this parameter to indicate that a list of eigenmodes for which output is desired will be listed on the data lines. This parameter is valid only in a **FREQUENCY*, **COMPLEX FREQUENCY*, or **BUCKLE* procedure, and if the FIELD parameter is included.

NAME

Set this parameter equal to the name associated with this output definition.

NUMBER INTERVAL

Set this parameter equal to the number of intervals during the step at which the output database states are to be written.

If this parameter and the FREQUENCY, TIME INTERVAL, and TIME POINTS parameters are omitted, output will be written at every increment of the analysis for all procedure types except **DYNAMIC* and **MODAL DYNAMIC*; output will be written every 10 increments for these procedure types.

The FREQUENCY, NUMBER INTERVAL, TIME INTERVAL, and TIME POINTS parameters are mutually exclusive.

TIME MARKS

Set TIME MARKS=YES (default) to write results at the exact times dictated by the NUMBER INTERVAL, TIME INTERVAL, or TIME POINTS parameter.

Set TIME MARKS=NO to write results to the output database at the increment ending immediately after the time dictated by the NUMBER INTERVAL, TIME INTERVAL, or TIME POINTS parameter.

TIME POINTS

Set this parameter equal to the name of the **TIME POINTS* option that defines the time points at which output is to be written.

If this parameter and the FREQUENCY, NUMBER INTERVAL, and TIME INTERVAL parameters are omitted, output will be written at every increment of the analysis for all procedure types except **DYNAMIC* and **MODAL DYNAMIC*; output will be written every 10 increments for these procedure types.

The FREQUENCY, NUMBER INTERVAL, TIME INTERVAL, and TIME POINTS parameters are mutually exclusive.

The following parameters are optional and valid only if the FIELD or HISTORY parameter is included:

OP

Set OP=NEW (default) to indicate that all output database requests defined in previous steps should be removed. New output database requests can be defined.

Set OP=ADD to indicate that the output request being defined should be added to the output requests defined in previous steps.

Set OP=REPLACE to indicate that this output request should replace an output request of the same type (e.g., FIELD) and frequency defined in a previous step. If there is no matching request, this output request will be interpreted as OP=ADD.

TIME INTERVAL

Set this parameter equal to the time interval at which the output states are to be written.

If this parameter and the FREQUENCY, NUMBER INTERVAL, and TIME POINTS parameters are omitted, output will be written at every increment of the analysis for all procedure types except **DYNAMIC* and **MODAL DYNAMIC*; output will be written every 10 increments for these procedure types.

The FREQUENCY, NUMBER INTERVAL, TIME INTERVAL, and TIME POINTS parameters are mutually exclusive.

VARIABLE

Set VARIABLE=ALL to indicate that all variables applicable to this procedure and material type should be written to the output database.

Set VARIABLE=PRESELECT (default) to indicate that the default output variables for the current procedure type should be written to the output database. Additional output requests can be defined with the output options used in conjunction with the **OUTPUT* option, listed previously. Preselected output is ignored in SIM-based dynamic analysis procedures.

The output behavior of the variables included in ALL or PRESELECT cannot be controlled by the parameters or data lines of the individual output options. To get the desired output behavior of a particular variable in an individual output option, the particular variable must be listed in that individual output option.

The following parameter is optional and valid only if the HISTORY parameter is included:

SENSOR

Include this parameter to associate this history output request with a sensor definition. The name of the associated sensor is given by the NAME parameter.

Data lines to list desired eigenmodes if the MODE LIST parameter is included:

First line:

1. Specify a list of desired eigenmodes.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

Defining output requests in an Abaqus/Explicit analysis

One of the following mutually exclusive parameters is required:

DIAGNOSTICS

Set DIAGNOSTICS=YES (default) to indicate that detailed diagnostic information should be written to the output database. Set DIAGNOSTICS=NO to suppress the output.

FIELD

Include this parameter to indicate that the output requests used in conjunction with the **OUTPUT* option will be written to the output database as field-type output.

HISTORY

Include this parameter to indicate that the output requests used in conjunction with the **OUTPUT* option will be written to the output database as history-type output.

Optional parameters:

NAME

Set this parameter equal to the name associated with this output definition.

The following parameters are optional and valid only if the FIELD parameter is included:

NUMBER INTERVAL

Set this parameter equal to the number of intervals during the step at which the output database states are to be written. Abaqus/Explicit will always write the results at the beginning of the step. For example, if NUMBER INTERVAL=10, Abaqus/Explicit will write 11 output database states consisting of the values at the beginning of the step and the values at the end of 10 intervals throughout the step. The value of this parameter must be a positive integer or zero. A value of zero suppresses all output. If this parameter is omitted, its value will be set to 20.

The NUMBER INTERVAL and TIME POINTS parameters are mutually exclusive.

TIME MARKS

Set TIME MARKS=NO (default) to write results to the output database at the increment ending immediately after the time dictated by the NUMBER INTERVAL or TIME POINTS parameter.

Set TIME MARKS=YES to write results at the exact times dictated by the NUMBER INTERVAL or TIME POINTS parameter.

TIME MARKS=YES cannot be used when either the FIXED TIME INCREMENTATION or DIRECT USER CONTROL parameter is included on the **DYNAMIC* option.

TIME POINTS

Set this parameter equal to the name of the **TIME POINTS* option that defines the time points at which output is to be written. If this parameter and the NUMBER INTERVAL parameter are omitted, field output will be written at 20 equally spaced intervals throughout the step.

The NUMBER INTERVAL and TIME POINTS parameters are mutually exclusive.

The following parameters are optional and valid only if the HISTORY parameter is included:

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be written to the output database at the last increment of each step. Set FREQUENCY=0 to suppress the output. If both this parameter and the TIME INTERVAL parameter are omitted, history output will be written at 200 equally spaced intervals throughout the step.

The FREQUENCY and TIME INTERVAL parameters are mutually exclusive.

SENSOR

Include this parameter to associate this history output request with a sensor definition. The name of the associated sensor is given by the NAME parameter.

The following parameters are optional and valid only if the FIELD or HISTORY parameter is included:

FILTER

Set this parameter equal to the name of the **FILTER* option to be used to filter the element and nodal field output or the element, nodal, contact, fastener interaction, or integrated history output.

Set FILTER=ANTIALIASING to filter the data based on the time interval that is specified; in this case the filter does not need to be defined in the model data. The antialiasing filter cannot be used with the FREQUENCY parameter for history output.

OP

Set OP=NEW (default) to indicate that all output database requests defined in previous steps should be removed. New output database requests can be defined.

Set OP=ADD to indicate that the output request being defined should be added to the output requests defined in previous steps.

Set OP=REPLACE to indicate that this output request should replace an output request of the same type (e.g., FIELD) and frequency defined in a previous step. If there is no matching request, this output request will be interpreted as OP=ADD.

TIME INTERVAL

Set this parameter equal to the time interval at which the output states are to be written.

For field output Abaqus/Explicit will always write the output at the beginning of the step. If both this parameter and the NUMBER INTERVAL parameter are omitted, field output will be written at 20 equally spaced intervals throughout the step. The NUMBER INTERVAL, TIME INTERVAL, and TIME POINTS parameters are mutually exclusive for field data.

For history output Abaqus/Explicit will always write the data values at the beginning and end of the step. If both this parameter and the FREQUENCY parameter are omitted, history output will be written at 200 equally spaced intervals throughout the step. The FREQUENCY and TIME INTERVAL parameters are mutually exclusive for history data.

VARIABLE

Set VARIABLE=ALL to indicate that all variables applicable to this procedure and material type should be written to the output database.

Set VARIABLE=PRESELECT to indicate that the default output variables for the current procedure type should be written to the output database. Additional output requests can be defined with the output options used in conjunction with the **OUTPUT* option, listed previously.

If this parameter is omitted, only the variables requested for output with the individual output options will be written to the output database.

The output behavior of the variables included in ALL or PRESELECT cannot be controlled by the parameters or data lines of the individual output options. To get the desired output behavior of a particular variable in an individual output option, the particular variable must be listed in that individual output option.

There are no data lines associated with this option in Abaqus/Explicit

P, Q

This section describes each Abaqus input option that begins with P or Q, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **PAPERBOARD HARDENING*
- **PAPERBOARD PLASTICITY*
- **PAPERBOARD THICKNESS COMPRESSION ELASTIC*
- **PAPERBOARD THICKNESS COMPRESSION PLASTICITY*
- **PAPERBOARD TRANSVERSE SHEAR PLASTICITY*
- **PARAMETER*
- **PARAMETER DEPENDENCE*
- **PARAMETER SHAPE VARIATION*
- **PARAMETER TABLE*
- **PARAMETER TABLE TYPE*
- **PART*
- **PARTICLE GENERATOR*
- **PARTICLE GENERATOR FLOW*
- **PARTICLE GENERATOR INLET*
- **PARTICLE GENERATOR MIXTURE*
- **PARTICLE OUTLET*
- **PARTICLE OUTLET FLOW*
- **PERFECTLY MATCHED LAYER*
- **PERIODIC*
- **PERIODIC MEDIA*
- **PERMANENT MAGNETIZATION*
- **PERMEABILITY*
- **PHYSICAL CONSTANTS*
- **PIEZOELECTRIC*
- **PIEZOELECTRIC DAMPING*
- **PIEZORESISTIVITY*
- **PIPE-SOIL INTERACTION*
- **PIPE-SOIL STIFFNESS*
- **PLANAR TEST DATA*
- **PLASTIC*
- **PLASTIC AXIAL*
- **PLASTICITY CORRECTION*
- **PLASTIC M1*
- **PLASTIC M2*
- **PLASTIC TORQUE*
- **PLY FABRIC FAILURE*

- **PLY FABRIC HARDENING*
- **PML COEFFICIENT*
- **PORE FLUID PRESSURE*
- **POROUS BULK MODULI*
- **POROUS ELASTIC*
- **POROUS ELECTRODE THEORY*
- **POROUS FAILURE CRITERIA*
- **POROUS METAL PLASTICITY*
- **POST OUTPUT*
- **POTENTIAL*
- **PREPRINT*
- **PRESSURE PENETRATION*
- **PRESSURE STRESS*
- **PRESTRESS HOLD*
- **PRE-TENSION SECTION*
- **PRINT*
- **PROBABILITY DENSITY FUNCTION*
- **PROPERTY TABLE*
- **PROPERTY TABLE TYPE*
- **PSD-DEFINITION*

*PAPERBOARD HARDENING: Specify the hardening data of the in-plane paperboard plasticity model.

This option is used to specify the hardening data for the in-plane paperboard plasticity model defined using the **PAPERBOARD PLASTICITY* option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- **PAPERBOARD PLASTICITY*
- *Hardening*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the paperboard in-plane plasticity hardening data. If this parameter is omitted, the hardening data are assumed not to depend on any field variables but might still depend on temperature. See *Material Data Definition* for more information.

Data lines to define the paperboard in-plane hardening:

First line:

1. σ_1^0. (Units of FL^{-2}.)
2. A_1. (Units of FL^{-2}.)
3. B_1.
4. C_1. (Units of FL^{-2}.)
5. σ_2^0. (Units of FL^{-2}.)
6. A_2. (Units of FL^{-2}.)
7. B_2.
8. C_2. (Units of FL^{-2}.)

Second line:

1. σ_3^0. (Units of FL^{-2}.)
2. A_3. (Units of FL^{-2}.)
3. B_3.
4. C_3. (Units of FL^{-2}.)
5. σ_4^0. (Units of FL^{-2}.)
6. A_4. (Units of FL^{-2}.)
7. B_4.

8. C_4. (Units of FL^{-2}.)

Third line:

1. σ_5^0. (Units of FL^{-2}.)
2. A_5. (Units of FL^{-2}.)
3. B_5.
4. C_5. (Units of FL^{-2}.)
5. Temperature, θ.
6. Etc., up to three field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this data line as often as necessary.

*PAPERBOARD PLASTICITY: Specify the in-plane paperboard plasticity model.

This option is used to define the exponent and plastic Poisson's ratios used in the paperboard in-plane yield function. It must be used in conjunction with the **PAPERBOARD HARDENING* option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- **PAPERBOARD HARDENING*
- *Yield Criteria*

Required parameters:

EXPONENT

Set this parameter equal to the exponent in the yield function. This value must be an even positive number.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the paperboard plasticity properties. If this parameter is omitted, the paperboard plasticity properties are assumed not to depend on any field variables but might still depend on temperature. See *Material Data Definition* for more information.

Data line to define the plastic Poisson's ratio:

First line:

1. Tensile plastic Poisson's ratio in 1-direction, v_{1p}.
2. Tensile plastic Poisson's ratio in 2-direction, v_{2p}.
3. Compressive plastic Poisson's ratio in 1-direction, v_{4p}.
4. Compressive plastic Poisson's ratio in 2-direction, v_{5p}.
5. Temperature, θ.
6. Etc., up to three field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this data line as often as necessary.

*PAPERBOARD THICKNESS COMPRESSION ELASTIC: Define the out-of-plane compression elasticity for the paperboard model.

This option is used to define the out-of-plane compression elasticity of the paperboard model.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

• *ZD Response*

Required parameters:

VOID RATIO

Set this parameter equal to the original void to solid ratio, r_{orig}.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the out-of-plane compression elasticity. If this parameter is omitted, the out-of-plane compression elasticity is assumed not to depend on any field variables but might still depend on temperature. See *Material Data Definition* for more information.

Data line to define the out-of-plane compression elasticity:

First line:

1. Elasticity parameter μ.
2. Elasticity parameter σ_Z^t. (Units of FL^{-2}.)
3. Temperature, θ.
4. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this data line as often as necessary.

*PAPERBOARD THICKNESS COMPRESSION PLASTICITY: Define the out-of-plane compression yield surface for the paperboard model.

This option is used to define the out-of-plane compression yield surface of the paperboard model. It must be used in conjunction with the **PAPERBOARD THICKNESS COMPRESSION ELASTIC* option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- **PAPERBOARD THICKNESS COMPRESSION ELASTIC*
- *ZD Response*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the out-of-plane compression yield surface. If this parameter is omitted, the out-of-plane compression yield surface is assumed not to depend on any field variables but might still depend on temperature. See *Material Data Definition* for more information.

Data line to define the out-of-plane compression yield surface:

First line:

1. A_σ. (Units of FL^{-2}.)
2. B_σ. (Units of FL^{-2}.)
3. C_σ.
4. Temperature, θ.
5. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this data line as often as necessary.

*PAPERBOARD TRANSVERSE SHEAR PLASTICITY: Define the transverse shear yield surface for the paperboard model.

This option is used to define the transverse shear yield surface of the paperboard model.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

• *Out-of-Plane Shear Response*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the transverse shear yield surface. If this parameter is omitted, the transverse shear yield surface is assumed not to depend on any field variables but might still depend on temperature. See *Material Data Definition* for more information.

Data line to define the transverse shear yield surface:

First line:

1. A_τ. (Units of FL^{-2}.)
2. B_τ. (Units of FL^{-2}.)
3. C_τ.
4. Temperature, θ.
5. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this data line as often as necessary.

*PARAMETER: Define parameters for input parametrization.

This option is used to define parameters that can be used in place of Abaqus input quantities.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance, Assembly, Model, Step

References:

- *Parametric Modeling*
- *Parametric Shape Variation*

Optional keyword parameters to define tabularly dependent parameters (if any is specified, they must all be specified):

DEPENDENT

Set this keyword parameter equal to the list of dependent parameters being defined in this option. The list must be given inside parentheses as parameter names separated by commas; for example, `(depPar1, depPar2, depPar3)`.

INDEPENDENT

Set this keyword parameter equal to the list of independent parameters used in this option. The list must be given inside parentheses as parameter names separated by commas; for example, `(indPar1, indPar2, indPar3)`.

TABLE

Set this keyword parameter equal to the name of the parameter dependence table, defined by the **PARAMETER DEPENDENCE* option, that defines the relationship between the dependent and independent parameters in this option.

Data lines to define independent or expressionally dependent parameters if the DEPENDENT, INDEPENDENT, and TABLE keyword parameters are omitted:

First line:

1. Python expression assigning a value to a parameter.

Repeat this data line as often as necessary to define independent and expressionally dependent parameters. The data given on this data line cannot be parameterized.

*PARAMETER DEPENDENCE: Define dependence table for tabularly dependent parameters.

This option is used to define the dependence table that specifies the relationship between tabularly dependent and independent parameters.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance, Assembly, Model, Step

References:

- *Parametric Modeling*
- *Parametric Shape Variation*

Required parameters:

NUMBER VALUES

Set this keyword parameter equal to the number of values in each line of the parameter dependence table. This number must be equal to the sum of the number of dependent and the number of independent parameters for which this table is used.

TABLE

Set this keyword parameter equal to the name of the table being defined in this option.

Data lines to define the parameter dependence table:

First line:

1. Values in the first row of the dependence table. Give the dependent and independent parameter data values that are to be interpolated. The dependent parameter values must precede the independent parameter values. The total number of entries in the row must equal the value of the NUMBER VALUES parameter.

Repeat this data line as often as necessary to define additional rows of the parameter dependence table. The data given on this data line cannot be parameterized.

*PARAMETER SHAPE VARIATION: Define parametric shape variations.

This option is used to define parametric shape variations.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- *Parametric Shape Variation*
- *Direct Design Sensitivity Analysis*

Required parameters:

PARAMETER

Set this keyword parameter equal to the name of the parameter to which the shape variation data refer. If this parameter is also a design parameter, the shape variations are used to define the design gradients of the nodal coordinates for design sensitivity analysis.

Optional parameters (mutually exclusive—if neither parameter is specified, Abaqus assumes that the shape variation data will be entered directly on the data lines):

FILE

Set this parameter equal to the name of the results file from a previous Abaqus/Standard analysis containing either the mode shapes from a **BUCKLE* or **FREQUENCY* analysis or the nodal displacements from a **STATIC* analysis. This option cannot be used for models defined in terms of an assembly of part instances.

INPUT

Set this parameter equal to the name of the alternate input file containing the shape variation data. See *Input Syntax Rules* for the syntax of such file names.

Required parameter if the FILE parameter is used:

STEP

Set this parameter equal to the step number (in the analysis whose results file is being used as input to this option) from which the modal or displacement data are to be read.

Optional parameters if the FILE parameter is used:

INC

Set this parameter equal to the increment number (in the analysis whose results file is being used as input to this option) from which the displacement data are to be read. If this parameter is omitted, Abaqus will read the data from the last increment available for the specified step on the results file.

MODE

Set this parameter equal to the mode number (in the analysis whose results file is being used as input to this option) from which the modal data are to be read. If this parameter is omitted, Abaqus will read the data from the first mode available for the specified step on the results file.

NSET

Set this parameter equal to the node set to which the shape variation values are to be applied. If this parameter is omitted, the shape variation will be applied to all nodes in the model.

Optional parameter if the FILE parameter is omitted:

SYSTEM

Set SYSTEM=R (default) to specify the shape variation as values of Cartesian coordinates. Set SYSTEM=C to specify the shape variation as values of cylindrical coordinates. Set SYSTEM=S to specify the shape variation as values of spherical coordinates. See *Figure 1*.

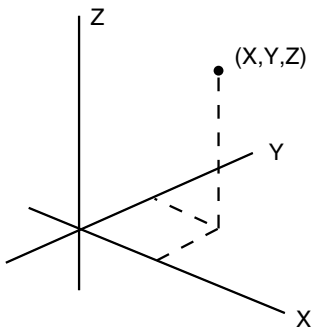
The SYSTEM parameter is entirely local to this option and should not be confused with the **SYSTEM* option. As the data lines are read, the shape variation values specified are transformed to the global rectangular Cartesian coordinate system. This transformation requires that the object be centered about the origin of the global coordinate system; i.e., the **SYSTEM* option should be off when specifying shape variations as values using either cylindrical or spherical coordinates. The details of how the shape variation is computed in particular coordinate systems are given in *Parametric Shape Variation*.

Data lines to define the shape variation if the FILE and INPUT parameters are omitted:

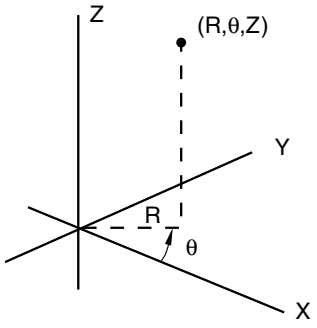
First line:

1. Node number or node set.
2. Component of shape variation in the first coordinate direction.
3. Component of shape variation in the second coordinate direction.
4. Component of shape variation in the third coordinate direction.
5. Shape variation in the first normal component.
6. Shape variation in the second normal component.
7. Shape variation in the third normal component.

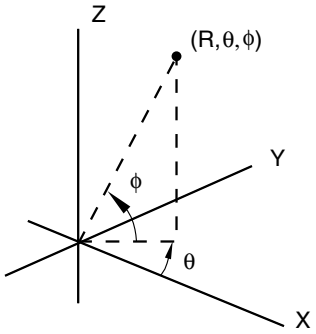
Repeat this data line as often as necessary to define the shape variation. The data given on this data line cannot be parameterized.



Rectangular Cartesian
(SYSTEM=R)
(default)



Cylindrical
(SYSTEM=C)



Spherical
(SYSTEM=S)

(θ and ϕ are given in degrees)

Figure 1: Coordinate systems.

*PARAMETER TABLE: Define a parameter table.

This option can be used only as a suboption of the **TABLE COLLECTION* or **MATERIAL* option to define a parameter table.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- *About Table Collections, Parameter Tables, and Property Tables*
- *Accessing Abaqus Table Collections*
- **PARAMETER TABLE TYPE*
- **PROPERTY TABLE TYPE*
- **TABLE COLLECTION*
- **PROPERTY TABLE*

Required parameters:

TYPE

Set this parameter equal to the name of the parameter table type that defines the data types of this parameter table.

Optional parameters:

LABEL

Set this parameter equal to the label to be used as the reference to the parameter table. If this parameter is omitted, the label is set equal to the name of the parameter table type.

Data lines to define the values of parameters:

First line:

1. First parameter. This field can be left blank if only the first parameter has a default value defined.
2. Second parameter. This field can be left blank if only the first parameter has a default value defined.
3. Etc., up to sixteen parameters.

Repeat this data line as often as necessary to define all parameters. The number of parameters must be a multiple of the parameter number declared for the parameter table type.

*PARAMETER TABLE TYPE: Define the type of parameter table.

This option is used to define the type of parameter table.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Accessing Abaqus Table Collections*
- **PROPERTY TABLE TYPE*
- **TABLE COLLECTION*
- **PARAMETER TABLE*
- **PROPERTY TABLE*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the parameter table type.

PARAMETERS

Set this parameter equal to the number of parameters.

Data lines to define the data types (INTEGER, FLOAT, or STRING) of the parameters:

First line:

1. Data type of the first parameter. Enter INTEGER, FLOAT, or STRING.
2. Default value of the first parameter. This field can be left blank if the parameter does not have a default value.
3. Description of the first parameter. This field can be left blank and is unused in the analysis.
4. Unit of the first parameter. This field can be left blank and is unused.
5. Allowed values of the first parameter. This field can be left blank and is unused in the analysis.

Repeat this data line as often as necessary to define all of the parameter data types as indicated by the value of the PARAMETERS parameter.

*PART: Begin a part definition.

This option is used to begin a part definition. It must be used in conjunction with the *ASSEMBLY, *END PART, and *INSTANCE options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Part module

References:

- *Assembly Definition*
- *END PART*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the part.

There are no data lines associated with this option.

*PARTICLE GENERATOR: Specify a particle generator.

This option is used to define a particle generator.

Products: Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

• *Particle Generator*

Required parameters:

CAVITY VOLUME

This parameter applies only to LKM analyses.

Set this parameter equal to the estimated maximum volume of the associated fluid cavity.

FLUID CAVITY REFNODE

This parameter applies only to LKM analyses.

Set this parameter equal to the name of the node set containing the associated fluid cavity reference node.

MAXIMUM NUMBER OF PARTICLES

Set this parameter equal to an upper bound of the maximum number of particles expected to be generated during the analysis.

NAME

Set this parameter equal to the name of the particle generator.

TYPE

Set this parameter equal to the type of particle to be generated. PD3D and PC3D elements are supported.

Optional parameters:

MAX ATTEMPTS PER INCREMENT

Set this parameter equal to the maximum number of particle insertion attempt cycles the generator makes before halting generation for an increment. The default value is 10.

MAX INSERTION ATTEMPTS

Set this parameter equal to the maximum number of attempts the generator makes to place a particle of a given size before rejecting that particle size. The default value is 15.

RLDP

This parameter applies only to LKM analyses.

Set this parameter equal to the estimated ratio of the mean free path to the lumped molecule particle diameter. The default value is 500.

SOLID FRACTION

Set this parameter equal to a threshold value that will temporarily halt the particle generator for an increment if the region near the inlet has a high volume of particles. The threshold value is a particle

volume fraction over a volume that extends $2.0R_{max}$ in front of the inlet surface, where R_{max} is the maximum particle radius for the generator. The default value is 0.4.

There are no data lines associated with this option.

*PARTICLE GENERATOR FLOW: Specify flow speed and mass flow rate per unit inlet area for a particle species.

This option is used to define particle flow speed and particle species mass flow rate per unit inlet area.

Products: Abaqus/Explicit

Type: History data

Level: Step

References:

- *Particle Generator*
- **PARTICLE GENERATOR*
- **PARTICLE GENERATOR MIXTURE*

Required parameters:

FLUID INFLATOR PROPERTY

This parameter applies only to LKM analyses.

Set this parameter equal to the name of the associated fluid inflator property.

GENERATOR

Set this parameter equal to the name of the particle generator for which the flow speed and mass flow rate per unit inlet area are being specified.

Optional parameters:

DEFINITION

Set DEFINITION=SPEED (default) for DEM and SPH analyses.

Set DEFINITION=NATURAL to apply the natural boundary condition for SPH analysis.

If this parameter is omitted, the first amplitude defined on the first data line is the speed of the generated particles along the normal to the inlet facet for DEM and SPH analyses. For LKM analyses it is the temperature of generated particles.

FLOW AMPLITUDE

This parameter applies only to LKM analyses.

Set this parameter equal to the name of the amplitude used to scale the generated mass of the lumped gas particles. If this parameter is omitted, the default scaling factor is 1.0.

GENERATION TIME AMPLITUDE

This parameter applies only to LKM analyses.

Set this parameter equal to the name of the amplitude used to compute the time for generating lumped gas particles.

MASS FLOW RATE TYPE

This parameter applies only to DEM and LKM analyses.

Set MASS FLOW RATE TYPE=TOTAL if the particle mass generated from a facet is the mass flow rate multiplied by the ratio of the current area of the facet to the current combined area of all facets.

Set MASS FLOW RATE TYPE=PER UNIT AREA (default) if the particle mass generated from a facet is the mass flow rate multiplied by the current area of the facet.

Data lines for particle generator flow:

This data line is ignored if a FLUID INFLATOR PROPERTY is specified.

First line:

1. Enter a single amplitude name for the particle flow speed at the generator inlet.
2. Enter the list of amplitude names for the mass flow rate per unit inlet area for each species for the particle generator mixture. The order is consistent with the species specified on the **PARTICLE GENERATOR MIXTURE* option.

Repeat this data line as often as necessary to add more mass flow rates per unit inlet area amplitude names. Up to 16 entries are allowed per line.

*PARTICLE GENERATOR INLET: Specify a particle generator inlet surface.

This option is used to specify an inlet surface to be used by a particle generator.

Products: Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- *Particle Generator*
- **PARTICLE GENERATOR*

Required parameters:

SURFACE

Set this parameter equal to the name of the inlet surface for the particle generator.

There are no data lines associated with this option.

*PARTICLE GENERATOR MIXTURE: Specify particle generator species mixture.

This option is used to specify the particle species to be used by a particle generator.

Products: Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- *Particle Generator*
- **PARTICLE GENERATOR*

There are no parameters associated with this option.

Data lines to define a particle generator mixture:

First line:

1. Enter the list of element set names for the respective particle species. The element sets listed here must also be associated with a **DISCRETE SECTION* definition.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

*PARTICLE OUTLET: Specify a particle outlet.

This option is used to define a particle outlet.

Products: Abaqus/Explicit

Type: History data

Level: Step

References:

• *Smoothed Particle Hydrodynamics*

Required parameters:

NAME

Set this parameter equal to the name of the particle outlet.

SURFACE

Set this parameter equal to the surface that defines the outflow boundary conditions.

TYPE

Set TYPE=SURFACE to define a surface outlet.

Data line to define a particle outlet:

First (and only) line:

1. Enter the name for the PC3D element set. The element set listed here can be associated with a **SOLID SECTION* definition or a **PARTICLE GENERATOR* definition.

*PARTICLE OUTLET FLOW: Specify nonreflecting or pressure boundary conditions.

This option is used to define nonreflecting or pressure boundary conditions on outlet surfaces.

Products: Abaqus/Explicit

Type: History data

Level: Step

References:

- **PARTICLE OUTLET*
- *Smoothed Particle Hydrodynamics*

Required parameters:

OUTLET

Set this parameter equal to the name of the particle outlet for which nonreflecting or pressure boundary conditions can be specified.

Optional parameters:

BOUNDARY CONDITION

Set BOUNDARY CONDITION=NONREFLECT (default) to specify a nonreflecting boundary condition.

Set BOUNDARY CONDITION=PRESSURE to specify a pressure boundary condition.

Data line for particle outlet flow:

First (and only) line:

1. Enter a single amplitude name for the flow pressure at the particle outlet.

*PERFECTLY MATCHED LAYER: Specify perfectly matched layer properties.

This option is used to specify the perfectly matched layer element behavior for acoustic elements.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- *Coupled Acoustic-Structural Analysis*
- *Acoustic and Shock Loads*

Required parameters:

ELSET

Set this parameter equal to the name of the element set to which the perfectly matched layer property is applied.

NAME

Set this parameter equal to the label that will be used to identify the perfectly matched layer property.

TYPE

Set TYPE=CARTESIAN to specify coordinates of two points to identify the start of the perfectly matched layer domain. These two points define the diagonal of the square domain in 2D and a cube in 3D.

Data lines to define the start of the perfectly matched layer domain:

First line:

1. *X*-coordinate of first point in the global coordinates.
2. *Y*-coordinate of first point in the global coordinates.
3. *Z*-coordinate of first point in the global coordinates.

Second line:

1. *X*-coordinate of second point in the global coordinates.
2. *Y*-coordinate of second point in the global coordinates.
3. *Z*-coordinate of second point in the global coordinates.

*PERIODIC: Define periodic symmetry for a cavity radiation heat transfer analysis.

This option is used to define cavity symmetry by periodic repetition in a given direction. It can be used only following the *RADIATION SYMMETRY option.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *Cavity Radiation in Abaqus/Standard*
- *RADIATION SYMMETRY

Required parameters:

TYPE

Set TYPE=2D to create a cavity composed of the cavity surface defined in the model and a series of similar images generated by its repetition according to a two-dimensional distance vector. The repeated images are bounded by lines parallel to line ***ab*** (see *Figure 1*). The distance vector must be defined so that it points away from line ***ab*** and into the domain of the model. This option can be used only for two-dimensional cases.

Set TYPE=3D to create a cavity composed of the cavity surface defined in the model and a series of similar images generated by its repetition according to a three-dimensional distance vector. The repeated images are bounded by planes parallel to plane ***abc*** (see *Figure 2*). The distance vector must be defined so that it points away from plane ***abc*** and into the domain of the model. This option can be used only for three-dimensional cases.

Set TYPE=ZDIR to create a cavity composed of the cavity surface defined in the model and a series of similar images generated by its repetition in the z-direction. The repeated images are bounded by lines of constant z-coordinate (see *Figure 3*). The z-distance vector must be defined so that it points away from the z-constant periodic symmetry reference line and into the domain of the model. This option can be used only for axisymmetric cases.

Optional parameters:

NR

Set this parameter equal to the number of repetitions used in the numerical calculation of the cavity view factors resulting from the periodic symmetry. The result of this symmetry is a cavity composed of the cavity surface defined in the model plus twice NR similar images, since the periodic symmetry is assumed to apply both in the positive and negative directions of the distance vector. The default value is NR=2.

Data line to define periodic symmetry of a two-dimensional cavity (TYPE=2D):

First (and only) line:

1. x-coordinate of point a (see *Figure 1*).
2. y-coordinate of point a.

3. x-coordinate of point b.
4. y-coordinate of point b.
5. x-component of periodic distance vector.
6. y-component of periodic distance vector.

Data lines to define periodic symmetry of a three-dimensional cavity (TYPE=3D):

First line:

1. x-coordinate of point a (see *Figure 2*).
2. y-coordinate of point a.
3. z-coordinate of point a.
4. x-coordinate of point b.
5. y-coordinate of point b.
6. z-coordinate of point b.

Second line:

1. x-coordinate of point c.
2. y-coordinate of point c.
3. z-coordinate of point c.
4. x-component of periodic distance vector.
5. y-component of periodic distance vector.
6. z-component of periodic distance vector.

Data line to define periodic symmetry of an axisymmetric cavity (TYPE=ZDIR):

First (and only) line:

1. z-coordinate of periodic symmetry reference line (see *Figure 3*).
2. z-component of periodic distance vector.

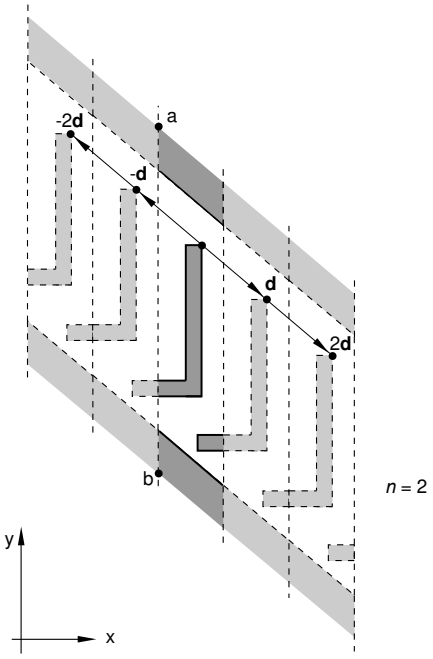


Figure 1: *PERIODIC, TYPE=2D option.

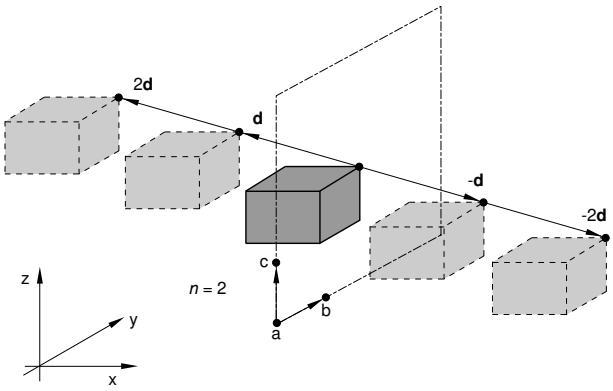


Figure 2: *PERIODIC, TYPE=3D option.

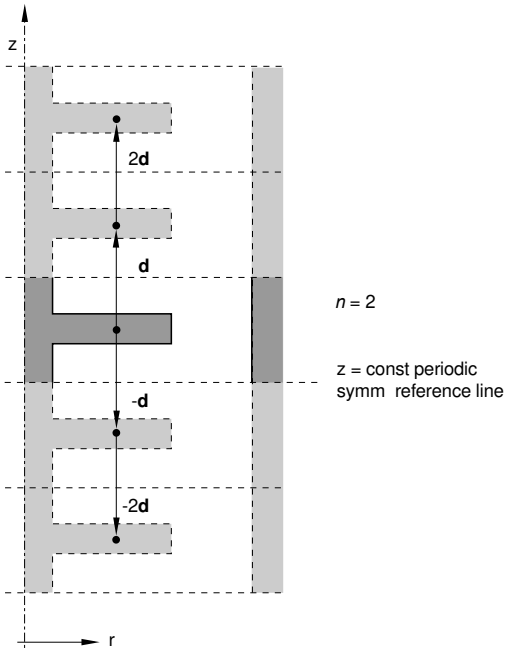


Figure 3: *PERIODIC, TYPE=ZDIR option.

*PERIODIC MEDIA: Specify a periodic media.

This option is used to specify a periodic media.

Products: Abaqus/Explicit

Type: Model data

Level: Model

References:

• *Periodic Media Analysis*

Required parameters:

INLET CONTROL NODE

Set this parameter equal to either the node number of the inlet control node or the name of a node set containing the inlet control node. If a node set name is chosen, the node set must contain exactly one node.

NAME

Set this parameter equal to the name of the periodic media.

ORIENTATION

Set this parameter equal to the name of an orientation definition (*Orientations*) used to define the trigger plane for periodic media block shuffling. The local *z*-axis is perpendicular to the trigger plane and points in the same direction as the media transport.

OUTLET CONTROL NODE

Set this parameter equal to either the node number of the outlet control node or the name of a node set containing the outlet control node. If a node set name is chosen, the node set must contain exactly one node.

TRIGGER NODE

Set this parameter equal to either the node number of the trigger node or the name of a node set containing the trigger node. If a node set name is chosen, the node set must contain exactly one node. The trigger plane is defined via a combination of trigger node location and the specified orientation.

Data lines to define a periodic media:

First line:

1. Element set label for all elements in the first block. (The element set used must be unsorted.)
2. Node set label for all nodes in the first block. (The node set used must be unsorted.)
3. Surface name defining the front edge nodes of the first block. The surface must be node-based, and the node set used in the surface definition must be unsorted.
4. Surface name defining the back edge nodes of the first block. The surface must be node-based, and the node set used in the surface definition must be unsorted.

Repeat this data line as often as necessary. Use one data line for each section of periodic media. The order of the periodic section is from inlet to outlet.

*PERMANENT MAGNETIZATION: Specify permanent magnetization.

This option is used to define permanent magnetization through coercivity of a permanent magnet for electromagnetic elements in a transient electromagnetic or magnetostatic analysis. This option can be used only in conjunction with the **MAGNETIC PERMEABILITY* option and, optionally, with the **NONLINEAR BH* option.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Magnetic Permeability*
- *Eddy Current Analysis*
- *Magnetostatic Analysis*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of coercivity of a permanent magnet. If this parameter is omitted, the coercivity is assumed not to depend on any field variables but may still depend on temperature.

Data lines to define the coercivity vector:

First line:

1. Global *X*-component of the direction of the coercivity vector.
2. Global *Y*-component of the direction of the coercivity vector.
3. Global *Z*-component of the direction of the coercivity vector.

Second line:

1. Magnitude of coercivity. (Units of $CL^{-1}T^{-1}$)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Do not repeat the first data line. Repeat the second and subsequent data lines as often as necessary to define the magnitude of coercivity as a function of temperature and field variables.

*PERMEABILITY: Define permeability for pore fluid flow.

This option is used to define permeability for pore fluid flow in problems involving seepage and porous media.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Permeability*

Defining permeability in Abaqus/Standard analyses

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the permeability. If this parameter is omitted, it is assumed that the permeability is independent of field variables.

This parameter can be used only in conjunction with TYPE=ISOTROPIC, ORTHOTROPIC, or ANISOTROPIC.

STABILIZE

This parameter applies only to first-order solid continuum elements with full integration.

Include this parameter to add a stabilization term to the element operator and the right-hand-side term by projecting the pressure into the strain space to eliminate the pressure oscillations for materials with very low permeability.

Set this parameter equal to a value by which Abaqus/Standard multiplies the shear modulus and the time increment to form the stabilization term in the element operator.

If this parameter is included without a value specified, the default stabilization coefficient value is 100.

TYPE

Set TYPE=ISOTROPIC (default) to define fully saturated isotropic permeability. Set TYPE=ORTHOTROPIC to define fully saturated orthotropic permeability. Set TYPE=ANISOTROPIC to define fully saturated anisotropic permeability.

Set TYPE=SATURATION to define $k_s\,(s)$; this must be a repeated use of the option for the same material and must follow the definition of fully saturated permeability. The definition must give k_s for $0.0 < s \leq 1.0$, with $k_s = 1.0$ at $s = 1.0$.

Set TYPE=VELOCITY to define $\beta\,(e)$; this must be a repeated use of the option for the same material and must follow the definition of fully saturated permeability.

Required parameter when fully saturated material properties are defined:

SPECIFIC

Set this parameter equal to the specific weight of the wetting liquid, γ_w (units of FL^{-3}). The actual specific weight must be given as a nonzero positive value, and the GRAV distributed load type must be used to apply the gravitational loading if a total pressure solution is required (see *Coupled Pore Fluid Diffusion and Stress Analysis* for a discussion of total and excess pressure solutions).

Data lines to define fully saturated isotropic permeability (TYPE=ISOTROPIC):

First line:

1. k. (Units of LT^{-1}.)
2. Void ratio, e.
3. Temperature, θ.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the variation.

Data lines to define fully saturated orthotropic permeability (TYPE=ORTHOTROPIC):

First line:

1. k_{11}. (Units of LT^{-1}.)
2. k_{22}.
3. k_{33}.
4. Void ratio, e.
5. Temperature, θ.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the variation.

Data lines to define fully saturated anisotropic permeability (TYPE=ANISOTROPIC):

First line:

1. k_{11}. (Units of LT^{-1}.)
2. k_{12}.
3. k_{22}.
4. k_{13}.
5. k_{23}.

6. k_{33}.
7. Void ratio, e.
8. Temperature, θ.

Subsequent lines (only needed if the DEPENDENCIES parameter is specified):

1. First field variable.
2. Second field variable.
3. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the variation.

Data lines to define the dependence of permeability on saturation of the wetting liquid, $k_s(s)$ (TYPE=SATURATION):

First line:

1. k_s. (Dimensionless.)
2. Saturation, s. (Dimensionless.)

Repeat this data line as often as necessary to define the variation. The table must provide $k_s = 1.0$ at $s = 1.0$.

Data lines to define the velocity coefficient (TYPE=VELOCITY):

First line:

1. β. (Units of TL^{-1}.) Only $\beta > 0.0$ is allowed.
2. Void ratio, e.

Repeat this data line as often as necessary to define the variation.

*PHYSICAL CONSTANTS: Specify physical constants.

This option is used to define physical constants necessary for an analysis; because Abaqus has no built-in units, no default values are provided. If a physical constant required for the analysis is not given, Abaqus issues a fatal error message. The units used for the constants must be consistent with the remaining input data.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Model attribute

References:

- *Uncoupled Heat Transfer Analysis*
- *Mass Diffusion Analysis*
- *Fluid Cavity Definition*
- *Rate-Dependent Plasticity: Creep and Swelling*
- *Diffusivity*
- *Solubility*
- *Thermal Contact Properties*
- *Cavity Radiation in Abaqus/Standard*
- *Obtaining Physical Constants in an Abaqus/Standard Analysis*

Optional parameters:

ABSOLUTE ZERO

Set this parameter equal to the absolute zero on the temperature scale chosen. For example, if the analysis uses temperature in degrees Celsius, set ABSOLUTE ZERO=−273.15.

AVOGADRO NUMBER

Set this parameter equal to the Avogadro number. For example, AVOGADRO NUMBER=$6.02214086 \times 10^{23}$ mol^{-1} in SI units.

BOLTZMANN

Set this parameter equal to the Ludwig Boltzmann constant. For example, BOLTZMANN=1.380662×10^{-23} joule per kelvin molecule in SI units.

ELEMENTARY CHARGE

Set this parameter equal to the elementary charge number. For example, ELEMENTARY CHARGE=$1.60217662 \times 10^{-19}$ Coulombs in SI units.

FARADAY CONSTANT

Set this parameter equal to the Faraday constant. For example, FARADAY CONSTANT=96485.3329 second Ampere mol^{-1} in SI units.

SPL REFERENCE PRESSURE

Set this parameter equal to the reference pressure used for computing the sound pressure level. For example, SPL REFERENCE PRESSURE=20 micropascals for air in SI units.

STEFAN BOLTZMANN

Set this parameter equal to the Stefan Boltzmann constant. For example, STEFAN BOLTZMANN=5.669×10^{-8} joule per sec m^2 $kelvin^4$ in SI units.

UNIVERSAL GAS CONSTANT

Set this parameter equal to the universal gas constant. For example, UNIVERSAL GAS CONSTANT=8.31434 joule per mole kelvin in SI units.

There are no data lines associated with this option.

*PIEZOELECTRIC: Specify piezoelectric material properties.

This option is used to define the piezoelectric properties of a material.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Piezoelectric Behavior*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the piezoelectric properties. If this parameter is omitted, the piezoelectric properties are assumed not to depend on any field variables but may still depend on temperature. See *Material Data Definition* for more information.

TYPE

Set TYPE=S (default) to specify stress material coefficients for the piezoelectric property. Set TYPE=E to specify strain material coefficients for the piezoelectric property.

Data lines to define the piezoelectric stress coefficient matrix (TYPE=S):

First line:

1. $e^{\varphi}_{1\,11}$. (Units of $FL^{-1}\varphi^{-1}$.)
2. $e^{\varphi}_{1\,22}$.
3. $e^{\varphi}_{1\,33}$.
4. $e^{\varphi}_{1\,12}$.
5. $e^{\varphi}_{1\,13}$.
6. $e^{\varphi}_{1\,23}$.
7. $e^{\varphi}_{2\,11}$.
8. $e^{\varphi}_{2\,22}$.

Second line:

1. $e^{\varphi}_{2\,33}$.
2. $e^{\varphi}_{2\,12}$.
3. $e^{\varphi}_{2\,13}$.
4. $e^{\varphi}_{2\,23}$.
5. $e^{\varphi}_{3\,11}$.

6. $e^{\varphi}_{3\,22}$.
7. $e^{\varphi}_{3\,33}$.
8. $e^{\varphi}_{3\,12}$.

Third line:

1. $e^{\varphi}_{3\,13}$.
2. $e^{\varphi}_{3\,23}$.
3. Temperature, θ.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the piezoelectric properties as a function of temperature and other predefined field variables.

Data lines to define the piezoelectric strain coefficient matrix (TYPE=E; for the shear components these coefficients relate the engineering, and not the tensorial, shear strain components to the components of the potential gradient vector):

First line:

1. $d^{\varphi}_{1\,11}$. (Units of $\mathrm{L}\varphi^{-1}$.)
2. $d^{\varphi}_{1\,22}$.
3. $d^{\varphi}_{1\,33}$.
4. $d^{\varphi}_{1\,12}$.
5. $d^{\varphi}_{1\,13}$.
6. $d^{\varphi}_{1\,23}$.
7. $d^{\varphi}_{2\,11}$.
8. $d^{\varphi}_{2\,22}$.

Second line:

1. $d^{\varphi}_{2\,33}$.
2. $d^{\varphi}_{2\,12}$.
3. $d^{\varphi}_{2\,13}$.
4. $d^{\varphi}_{2\,23}$.
5. $d^{\varphi}_{3\,11}$.
6. $d^{\varphi}_{3\,22}$.
7. $d^{\varphi}_{3\,33}$.

8. $d^{\varphi}_{3\,12}$.

Third line:

1. $d^{\varphi}_{3\,13}$.
2. $d^{\varphi}_{3\,23}$.
3. Temperature, θ.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the piezoelectric properties as a function of temperature and other predefined field variables.

*PIEZOELECTRIC DAMPING: Specify piezoelectric material damping for a piezoelectric material.

This option is used to provide piezoelectric material damping for mode-based analyses and for direct-integration steady-state analysis in Abaqus/Standard.

It is relevant only for piezoelectric elements. You can provide damping coefficients for the displacement (mechanical), piezoelectric coupling, and dielectric parts of the damping operator.

This option is ignored in mode-based procedures that use Lanczos or subspace iteration eigenvalue extraction that does not use the SIM architecture (*FREQUENCY, EIGENSOLVER=SUBSPACE, SIM=NO).

Products: Abaqus/Standard

Type: Model data

Level: Part instance

References:

- *Material Damping*
- *About Dynamic Analysis Procedures*

Required and mutually exclusive parameters:

BETA

Use this parameter to indicate that four $\beta_i \ (i = 1, 4)$ factors will be read from the data line to create Rayleigh stiffness proportional damping for the displacement, piezoelectric coupling, and dielectric parts of the damping operator in the following procedures:

- *COMPLEX FREQUENCY
- *MATRIX GENERATE
- *SUBSTRUCTURE GENERATE
- *STEADY STATE DYNAMICS, DIRECT
- *STEADY STATE DYNAMICS, SUBSPACE PROJECTION
- *STEADY STATE DYNAMICS that allows nondiagonal damping

STRUCTURAL

Use this parameter to indicate that four $s_i \ (i = 1, 4)$ factors will be read from the data line to create stiffness proportional structural damping for the displacement, piezoelectric coupling, and dielectric parts of the damping operator in the following procedures:

- *COMPLEX FREQUENCY
- *MATRIX GENERATE
- *SUBSTRUCTURE GENERATE
- *STEADY STATE DYNAMICS, DIRECT
- *STEADY STATE DYNAMICS, SUBSPACE PROJECTION
- *STEADY STATE DYNAMICS that allows nondiagonal damping

Data lines to define temperature-dependent stiffness proportional Rayleigh damping:
First line:

1. β_1. Damping coefficient for the displacement part of the damping operator. (Units of T.)
2. β_2. Damping coefficient for the dielectric part of the damping operator. (Units of T.)
3. β_3. Damping coefficient for the piezoelectric coupling (displacement/dielectric) part of the damping operator. (Units of T.)
4. β_4. Damping coefficient for the piezoelectric coupling (dielectric/displacement) part of the damping operator. (Units of T.)
5. Temperature.

Repeat this data line as often as necessary.

Data lines to define temperature-dependent stiffness proportional structural damping:
First line:

1. s_1. Damping coefficient for the displacement part of the damping operator.
2. s_2. Damping coefficient for the dielectric part of the damping operator.
3. s_3. Damping coefficient for the piezoelectric coupling (displacement/dielectric) part of the damping operator.
4. s_4. Damping coefficient for the piezoelectric coupling (dielectric/displacement) part of the damping operator.
5. Temperature.

Repeat this data line as often as necessary.

*PIEZORESISTIVITY: Specify piezoresistivity coefficients.

This option is used to define the piezoresistivity coefficients for a piezoresistive material. It can be used only in conjunction with **ELECTRICAL RESISTIVITY*.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Piezoresistive Effect*
- *Electrical Resistivity*
- **ELECTRICAL RESISTIVITY*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of piezoresistivity. If this parameter is omitted, the piezoresistivity is assumed not to depend on any field variables but may still depend on temperature. See *Material Data Definition* for more information.

TYPE

Set TYPE=ISOTROPIC (default) to define isotropic behavior.

Set TYPE=CUBIC to define cubic behavior.

Set TYPE=ORTHOTROPIC to define orthotropic behavior.

Set TYPE=ANISOTROPIC to define fully anisotropic behavior.

Data lines to define isotropic piezoresistivity (TYPE=ISOTROPIC):

First line:

1. π_{1111}. (Units of $F^{-1}L^2$.)
2. π_{1122}.
3. Temperature, θ.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the piezoresistivity behavior as a function of temperature and other predefined field variables.

Data lines to define cubic piezoresistivity (TYPE=CUBIC):

First line:

1. π_{1111}. (Units of $F^{-1}L^2$.)

2. π_{1122}.
3. π_{1212}.
4. Temperature, θ.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the piezoresistivity behavior as a function of temperature and other predefined field variables.

Data lines to define fully anisotropic piezoresistivity directly (TYPE=ANISOTROPIC):

First line:

1. π_{1111}. (Units of $F^{-1}L^2$.)
2. π_{1122}.
3. π_{2222}.
4. π_{1133}.
5. π_{2233}.
6. π_{3333}.
7. π_{1112}.
8. π_{2212}.

Second line:

1. π_{3312}.
2. π_{1212}.
3. π_{1113}.
4. π_{2213}.
5. π_{3313}.
6. π_{1213}.
7. π_{1313}.
8. π_{1123}.

Third line:

1. π_{2223}.
2. π_{3323}.
3. π_{1223}.
4. π_{1323}.
5. π_{2323}.
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the piezoresistivity behavior as a function of temperature and other predefined field variables.

Data lines to define orthotropic piezoresistivity directly (TYPE=ORTHOTROPIC):

First line:

1. π_{1111}. (Units of $F^{-1}L^2$.)
2. π_{1122}.
3. π_{2222}.
4. π_{1133}.
5. π_{2233}.
6. π_{3333}.
7. π_{1212}.
8. π_{1313}.

Second line:

1. π_{2323}.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the piezoresistivity behavior as a function of temperature and other predefined field variables.

*PIPE-SOIL INTERACTION: Specify element properties for pipe-soil interaction elements.

This option is used to define properties for pipe-soil interaction elements. The **PIPE-SOIL STIFFNESS* option must follow immediately after this option.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- *Pipe-Soil Interaction Elements*
- *Pipe-Soil Interaction Element Library*
- **PIPE-SOIL STIFFNESS*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the pipe-soil interaction elements for which properties are being defined.

Optional parameters:

ORIENTATION

Set this parameter equal to the name of the **ORIENTATION* definition (*Orientations*) that gives the orientation of the local system used for material calculations.

There are no data lines associated with this option.

*PIPE-SOIL STIFFNESS: Define constitutive behavior for pipe-soil interaction elements.

This option is used to define the constitutive behavior for pipe-soil interaction elements. It can be used only in conjunction with the **PIPE-SOIL INTERACTION* option. Repeat the option as needed to define behavior in the different local directions.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- *Pipe-Soil Interaction Elements*
- *Pipe-Soil Interaction Element Library*
- *UMAT*
- **PIPE-SOIL INTERACTION*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the material property values. If this parameter is omitted, it is assumed that the properties are independent of field variables. See "Using the DEPENDENCIES parameters to define field variable dependence" in *Material Data Definition* for more information.

DIRECTION

Set this parameter equal to the direction in the local orientation system for which the behavior is defined. The DIRECTION parameter can be set equal to a label or to a numerical value. Omit the DIRECTION parameter to define an isotropic model. The DIRECTION parameter must be used to define a constitutive model using the ASCE formulae.

Set DIRECTION=AXIAL (or DIRECTION=1) to specify behavior along the first local direction.

Set DIRECTION=VERTICAL (or DIRECTION=2) to specify behavior along the second local direction.

Set DIRECTION=HORIZONTAL (or DIRECTION=3) to specify behavior along the third local direction.

TYPE

Set TYPE=LINEAR (default) to define a linear constitutive model.

Set TYPE=NONLINEAR to define a nonlinear constitutive model.

Set TYPE=CLAY to define a constitutive model using the ASCE formulae for clay. This parameter must be used in conjunction with the DIRECTION parameter.

Set TYPE=SAND to define a constitutive model using the ASCE formulae for sand. This parameter must be used in conjunction with the DIRECTION parameter.

Set TYPE=USER to indicate the constitutive behavior is defined in user subroutine *UMAT*.

The following optional parameters can be used only in combination with TYPE=USER:

PROPERTIES

Set this parameter equal to the number of property values needed as data in user subroutine *UMAT*. The default is PROPERTIES=0.

VARIABLES

Set this parameter equal to the number of solution-dependent variables that must be stored for the material calculations in user subroutine *UMAT*. The default is VARIABLES=1.

Data lines to define linear constitutive behavior (TYPE=LINEAR):

First line:

1. Interaction stiffness (force per relative displacement) corresponding to positive relative displacements, K_p.
2. Interaction stiffness (force per relative displacement) corresponding to negative relative displacements, K_n. The default is $K_n = K_p$.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the stiffness as a function of temperature and other predefined field variables.

Data lines to define nonlinear constitutive behavior (TYPE=NONLINEAR):

First line:

1. Force per unit length along pipeline.
2. Relative displacement.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the force per unit length as a function of relative displacement, temperature, and other predefined field variables.

Data lines to define constitutive behavior using the ASCE formula for sand in the axial direction (TYPE=SAND, DIRECTION=AXIAL):

First line:

1. Effective unit weight of soil, $\overline{\gamma}$.
2. Coefficient of soil pressure at rest, K_0.
3. Interface angle of friction, δ. Give the value in degrees.
4. External pipe diameter, D.

5. Ultimate relative displacement, $\bar{\varepsilon}_a$.
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the parameters for the ASCE formulae as a function of temperature and other predefined field variables.

Data lines to define constitutive behavior using the ASCE formula for clay in the axial direction (TYPE=CLAY, DIRECTION=AXIAL):

First line:

1. Undrained soil shear strength, S.
2. Empirical adhesion factor, α.
3. External pipe diameter, D.
4. Ultimate relative displacement, $\bar{\varepsilon}_a$.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the parameters for the ASCE formulae as a function of temperature and other predefined field variables.

Data lines to define constitutive behavior using the ASCE formula for sand in the vertical direction (TYPE=SAND, DIRECTION=VERTICAL):

First line:

1. Total unit weight of soil, γ.
2. Bearing capacity factor, N_q.
3. Bearing capacity factor, N_γ.
4. Vertical uplift factor, N_{qv}.
5. Ultimate relative displacement for downward motion, $\bar{\varepsilon}_{vp}$.
6. Ultimate relative displacement for upward motion, $\bar{\varepsilon}_{vn}$.
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the parameters for the ASCE formulae as a function of temperature and other predefined field variables.

Data lines to define constitutive behavior using the ASCE formula for clay in the vertical direction (TYPE=CLAY, DIRECTION=VERTICAL):

First line:

1. Bearing capacity factor, N_c.
2. Vertical uplift factor, N_{cv}.
3. Ultimate relative displacement for downward motion, $\overline{\varepsilon}_{vp}$.
4. Ultimate relative displacement for upward motion, $\overline{\varepsilon}_{vn}$.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the parameters for the ASCE formulae as a function of temperature and other predefined field variables.

Data lines to define constitutive behavior using the ASCE formula for sand in the horizontal direction (TYPE=SAND, DIRECTION=HORIZONTAL):

First line:

1. Horizontal bearing capacity factor, N_{qh}.
2. Ultimate relative displacement, $\overline{\varepsilon}_h$.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables per line.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the parameters for the ASCE formulae as a function of temperature and other predefined field variables.

Data lines to define constitutive behavior using the ASCE formula for clay in the horizontal direction (TYPE=CLAY, DIRECTION=HORIZONTAL):

First line:

1. Horizontal bearing capacity factor, N_{ch}.
2. Ultimate relative displacement, $\overline{\varepsilon}_h$.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables per line.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define parameters for the ASCE formulae as a function of temperature and other predefined field variables.

Data lines if the constitutive behavior is defined in user subroutine `UMAT` (TYPE=USER):

First line:

1. Enter the data to be used as properties in user subroutine *UMAT*.

Repeat this data line as often as necessary to define properties required in *UMAT*. Enter eight values per line.

*PLANAR TEST DATA: Used to provide planar test (or pure shear) data (compression and/or tension).

This option is used to provide planar test (or pure shear) data. It can be used only in conjunction with the *HYPERELASTIC option, the *HYPERFOAM option, and the *MULLINS EFFECT option. This type of test does not define the hyperelastic material constants fully; at the least, uniaxial or biaxial test data should also be given.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Hyperelastic Behavior of Rubberlike Materials*
- *Hyperelastic Behavior in Elastomeric Foams*
- *Stress Softening in Rubberlike Materials*
- *Stress Softening in Elastomeric Foams*
- *HYPERELASTIC
- *HYPERFOAM
- *MULLINS EFFECT

Using planar test data to define a hyperelastic material

Optional parameters:

SMOOTH

Include this parameter to apply a smoothing filter to the stress-strain data. If the parameter is omitted, no smoothing is performed.

Set this parameter equal to the number n such that $2n + 1$ is equal to the total number of data points in the moving window through which a cubic polynomial is fit using the least-squares method. n should be larger than 1. The default is SMOOTH=3.

Optional parameter when the *PLANAR TEST DATA option is used in conjunction with the *HYPERELASTIC, MARLOW option:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the test data. If this parameter is omitted, it is assumed that the test data depend only on temperature. See *Material Data Definition* for more information.

Data lines to specify planar test data for hyperelasticity other than the Marlow model (the nominal strains must be arranged in either ascending or descending order if the SMOOTH parameter is used):

First line:

1. Nominal stress, T_S.
2. Nominal strain in the direction of loading, ϵ_S.

Repeat this data line as often as necessary to give the stress-strain data.

Data lines to specify planar test data for the Marlow model (the nominal strains must be arranged in ascending order if the SMOOTH parameter is used):

First line:

1. Nominal stress, T_S.
2. Nominal strain, ϵ_S.
3. Nominal lateral strain, ϵ_3. Not needed if the POISSON parameter is specified on the **HYPERELASTIC* option or if the **VOLUMETRIC TEST DATA* option is used.
4. Temperature, θ.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the test data as a function of temperature and other predefined field variables. Nominal strains and nominal stresses must be given in ascending order.

Using planar test data to define an elastomeric foam

There are no parameters associated with this option.

Data lines to specify planar test data for a hyperfoam:

First line:

1. Nominal stress, T_L.
2. Nominal strain in the direction of loading, ϵ_P.
3. Nominal transverse strain, ϵ_3. Default is zero. Not needed if the POISSON parameter is specified on the **HYPERFOAM* option.

Repeat this data line as often as necessary to give the stress-strain data.

Using planar test data to define the Mullins effect material model

There are no parameters associated with this option.

Data lines to specify planar test data for defining the unloading-reloading response of the Mullins effect material model:

First line:

1. Nominal stress, T_L.
2. Nominal strain in the direction of loading, ϵ_P.

Repeat this data line as often as necessary to give the stress-strain data.

*PLASTIC: Specify a metal plasticity model.

This option is used to specify the plastic part of the material model for elastic-plastic materials that use the Mises or Hill yield surface.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Classical Metal Plasticity*
- *Models for Metals Subjected to Cyclic Loading*
- *Johnson-Cook Plasticity*
- *Permanent Set in Rubberlike Materials*
- *UHARD*
- *VUHARD*

Optional parameters:

HARDENING

Set HARDENING=ISOTROPIC (default) to specify isotropic hardening.

Set HARDENING=KINEMATIC to specify linear kinematic hardening.

Set HARDENING=MULTILINEAR KINEMATIC to specify multilinear kinematic hardening.

Set HARDENING=COMBINED to specify nonlinear isotropic/kinematic hardening.

Set HARDENING=JOHNSON COOK to specify Johnson-Cook hardening.

Set HARDENING=USER to define isotropic hardening in user subroutine *UHARD* in an Abaqus/Standard analysis or user subroutine *VUHARD* in an Abaqus/Explicit analysis.

SCALESTRESS

This parameter cannot be used with the HARDENING parameter.

Set this parameter equal to the factor by which you want the yield stress to be scaled.

Optional parameter for use with HARDENING=ISOTROPIC or HARDENING=COMBINED:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of hardening behavior, in addition to temperature and possibly strain range. If this parameter is omitted, the hardening behavior does not depend on field variables. See *Material Data Definition* for more information.

Optional parameters for use with HARDENING=ISOTROPIC:

EXTRAPOLATION

This parameter specifies the extrapolation method for the yield stress with respect to the equivalent plastic strain.

If you use the **PLASTIC* option multiple times to specify test data at different equivalent plastic strain rates, you can use the EXTRAPOLATION parameter only one time, when you specify data at a zero equivalent plastic strain rate (**PLASTIC*, RATE=0.0). On the subsequent options, you must either omit the EXTRAPOLATION parameter or specify the same method as that used at a zero equivalent plastic strain rate.

Set EXTRAPOLATION=CONSTANT (default) to use a constant value of the yield stress outside the specified range of the equivalent plastic strains.

Set EXTRAPOLATION=LINEAR to use linear extrapolation (based on the slope) of the yield stress outside the specified range of the equivalent plastic strains.

RATE

Set this parameter equal to the equivalent plastic strain rate, $\dot{\bar{\varepsilon}}^{pl}$, for which this stress-strain curve applies.

Optional parameters for use with HARDENING=COMBINED:

DATA TYPE

Set DATA TYPE=HALF CYCLE (default) to specify stress versus plastic strain values of the first half-cycle for calibrating the kinematic hardening parameters.

Set DATA TYPE=PARAMETERS to specify the calibrated kinematic hardening material parameters directly.

Set DATA TYPE=STABILIZED to specify stress versus plastic strain values of a stabilized cycle for calibrating the kinematic hardening parameters.

NUMBER BACKSTRESSES

Set this parameter equal to the number of backstresses. The default number of backstresses is 1, and the maximum allowed is 10.

STATIC RECOVERY

This parameter can be used only in conjunction with DATA TYPE=PARAMETERS.

Include this parameter to indicate that the static recovery term must be included with the nonlinear isotropic/kinematic hardening model.

Optional parameter for use with HARDENING=USER:

PROPERTIES

Set this parameter equal to the number of property values needed as data in user subroutine *UHARD* in Abaqus/Standard analyses and user subroutine *VUHARD* in Abaqus/Explicit analyses. The default is PROPERTIES=0.

Data lines for HARDENING=ISOTROPIC or HARDENING=COMBINED with DATA TYPE=HALF CYCLE:

First line:

1. Yield stress.
2. Plastic strain.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of yield stress on plastic strain and, if needed, on temperature and other predefined field variables.

Data lines for HARDENING=COMBINED with DATA TYPE=STABILIZED:

First line:

1. Yield stress.
2. Plastic strain.
3. Strain range.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of yield stress on plastic strain and, if needed, on strain range, temperature, and other predefined field variables.

Data lines for HARDENING=COMBINED with DATA TYPE=PARAMETERS:

First line:

1. Yield stress at zero plastic strain.
2. Kinematic hardening parameter, C_1.
3. Kinematic hardening parameter, γ_1.
4. Etc., specify kinematic hardening parameters C_k and γ_k for each backstress k. (Only needed if the NUMBER BACKSTRESSES parameter has a value greater than 1.)
5. Temperature.
6. First field variable.
7. Second field variable.
8. Etc., up to eight entries per line.

Subsequent lines (only needed if the number of entries is greater than eight):

1. Etc., up to eight entries per line.

Repeat this set of data lines as often as necessary to define the dependence of yield stress and kinematic hardening parameters C_k and γ_k on temperature and other predefined field variables.

Data lines for HARDENING=COMBINED with DATA TYPE=PARAMETERS and STATIC RECOVERY:

First line:

1. Yield stress at zero plastic strain.
2. Kinematic hardening parameter, C_1. (Units of FL^{-2}.)
3. Kinematic hardening parameter, γ_1.

4. Kinematic hardening parameter, ξ_1. (Units of T^{-1}.)
5. Kinematic hardening parameter, $\overline{R}_1$. (Units of FL^{-2}.)
6. Kinematic hardening parameter, m_1.
7. Etc., specify kinematic hardening parameters C_k, γ_k, ξ_k, $\overline{R}_k$, and m_k for each backstress k. (Only needed if the NUMBER BACKSTRESSES parameter has a value greater than 1.)
8. Temperature.

Subsequent lines (only needed if the number of entries is greater than eight):

1. First field variable.
2. Second field variable.
3. Etc., up to eight entries per line.

Repeat this set of data lines as often as necessary to define the dependence of yield stress and kinematic hardening parameters on temperature and other predefined field variables.

Data lines for HARDENING=KINEMATIC:

First line:

1. Yield stress.
2. Plastic strain.
3. Temperature, if temperature dependent.

Repeat this data line a maximum of two times to define linear kinematic hardening independent of temperature. Repeat this set of data lines as often as necessary to define a variation of the linear kinematic hardening modulus with respect to temperature.

Data lines for HARDENING=MULTILINEAR KINEMATIC:

First line:

1. Yield stress.
2. Plastic strain.
3. Temperature, if temperature dependent.
4. First field variable.
5. Second field variable.
6. Etc., up to eight entries per line.

Subsequent lines (only needed if the number of entries is greater than eight):

1. Etc., up to eight entries per line.

Repeat this set of data lines as often as necessary to define the dependence of yield stress on plastic strain and, if needed, on temperature and other predefined field variables.

Data line for HARDENING=JOHNSON COOK:

First (and only) line:

1. A. (Units of FL^{-2}.)
2. B. (Units of FL^{-2}.)
3. n. (Dimensionless.)
4. m. (Dimensionless.) If this field is left blank or a zero value is specified, the yield stress does not depend on temperature.
5. Melting temperature, θ_{melt}. (Only required if a nonzero value of m is specified.)

6. Transition temperature, $\theta_{\text{transition}}$. (Only required if a nonzero value of m is specified.)

Data lines for HARDENING=USER with PROPERTIES:

First line:

1. Give the hardening properties, eight per line.

Repeat this data line as often as necessary to define all hardening properties.

*PLASTIC AXIAL: Define plastic axial force for frame elements.

This option can be used only in conjunction with the *FRAME SECTION option. It describes the axial force in a frame element as a function of the axial plastic displacement.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- *Frame Section Behavior*
- *FRAME SECTION

There are no parameters associated with this option.

Data lines to define the variation of axial yield force with plastic displacement:

First line:

1. Axial yield force.
2. Plastic axial displacement.

Repeat this data line as often as necessary to define the relationship between the axial force and the axial plastic displacement. At least three pairs of data are required.

*PLASTICITY CORRECTION: Specify the plastic response used to compute Neuber and Glinka plasticity corrections.

This option is used to specify the plastic response that is used to compute Neuber and Glinka plasticity corrections.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Plasticity Corrections*

Required parameters:

DEFINITION

Set DEFINITION=RAMBERG-OSGOOD to specify the hardening behavior using the Ramberg-Osgood formula.

Set DEFINITION=TABULAR to specify the hardening curve in the tabular form.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of hardening behavior, in addition to temperature. If this parameter is omitted, the hardening behavior does not depend on field variables.

Data lines for DEFINITION=RAMBERG-OSGOOD:

First line:

1. K. (Units of FL^{-2}.)
2. n. (Dimensionless.)
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the Ramberg-Osgood model parameters on temperature and other predefined field variables.

Data lines for DEFINITION=TABULAR:

First line:

1. Yield stress.
2. Plastic strain.

3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of yield stress on plastic strain and, if needed, on temperature and other predefined field variables.

*PLASTIC M1: Define the first plastic bending moment behavior for frame elements.

This option can be used only in conjunction with the **FRAME SECTION* option. It describes the bending moment in a frame element as a function of the plastic rotation about the first cross-section direction.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- *Frame Section Behavior*
- **FRAME SECTION*

There are no parameters associated with this option.

Data lines to define the variation of yield moment M_1^0 with plastic rotation:

First line:

1. Bending yield moment M_1^0 about the first cross-section direction.
2. Plastic rotation about the first cross-section direction.

Repeat this data line as often as necessary to define the relationship between the bending moment and the plastic rotation about the first cross-section axis. At least three pairs of data are required.

*PLASTIC M2: Define the second plastic bending moment behavior for frame elements.

This option can be used only in conjunction with the **FRAME SECTION* option and is available only for FRAME3D elements. It describes the bending moment in a frame element as a function of the plastic rotation about the second cross-section direction.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- *Frame Section Behavior*
- **FRAME SECTION*

There are no parameters associated with this option.

Data lines to define the variation of yield moment M_2^0 with plastic rotation:

First line:

1. Bending yield moment M_2^0 about the second cross-section direction.
2. Plastic rotation about the second cross-section direction.

Repeat this data line as often as necessary to define the relationship between the bending moment and the plastic rotation about the second cross-section axis. At least three pairs of data are required.

*PLASTIC TORQUE: Define the plastic torsional moment behavior for frame elements.

This option can be used only in conjunction with the **FRAME SECTION* option and is available only for FRAME3D elements. It describes the torsional moment in a frame element as a function of the plastic rotation about the element's axis.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- *Frame Section Behavior*
- **FRAME SECTION*

There are no parameters associated with this option.

Data lines to define the variation of torsional yield moment with plastic rotation:

First line:

1. Torsional yield moment, T^0.
2. Plastic rotation about the element's axis.

Repeat this data line as often as necessary to define the relationship between the torsional moment and the plastic rotation. At least three pairs of data are required.

*PLY FABRIC FAILURE: Defines the conditions that trigger material failure for the ply fabric damage model for bidirectional fabric-reinforced composites.

This option defines the conditions that trigger material failure and is used in combination with the ply fabric damage model for bidirectional fabric-reinforced composites.

Products: Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Ply Fabric Criterion*
- *Damage Evolution for the Ply Fabric Model*
- *Maximum Degradation and Choice of Element Removal*

Optional parameters:

NUM FIBERS

This parameter allows you to define a material failure criterion based on the number of local fiber directions that have failed.

Set NUM FIBERS=1 (default) to fail the material point when either fiber direction fails ($d_1 = d_{max}$ or $d_2 = d_{max}$).

Set NUM FIBERS=2 to fail the material point when both fiber directions fail ($d_1 = d_{max}$ and $d_2 = d_{max}$).

Data line to define failure strains:

First (and only) line:

1. Maximum value of equivalent plastic strain, $\bar{\varepsilon}^{pl}_{\max}$. The material point fails if $\bar{\varepsilon}^{pl} \geq \bar{\varepsilon}^{pl}_{\max}$. If this value is omitted or a value of zero is specified, $\bar{\varepsilon}^{pl}_{\max}$ is not used in ply fabric damage initiation and evolution.
2. Maximum (positive) principal logarithmic strain, $\hat{\varepsilon}_{\max}$. The material point fails if the maximum principal strain is greater than or equal to $\hat{\varepsilon}_{\max}$. If this value is omitted or a value of zero is specified, $\hat{\varepsilon}_{\max}$ is not used as a criterion for element deletion.
3. Minimum (negative) principal logarithmic strain, $\hat{\varepsilon}_{\min}$. The material point fails if the minimum principal strain is less than or equal to $\hat{\varepsilon}_{\min}$. If this value is omitted or a value of zero is specified, $\hat{\varepsilon}_{\min}$ is not used as a criterion for element deletion.

*PLY FABRIC HARDENING: Specify hardening for the in-plane shear plasticity response of bidirectional fabric-reinforced composite materials.

This option is used to specify the hardening behavior of the shear plasticity model for bidirectional fabric-reinforced composite materials with or without damage.

Products: Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Defining Orthotropic Elasticity in Plane Stress with Different Moduli in Tension and Compression*
- *Plasticity Model for Bidirectional Fabric-Reinforced Composite Materials*
- *Ply Fabric Criterion*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the hardening behavior, in addition to temperature and possibly plastic strain. If this parameter is omitted, the hardening behavior does not depend on field variables. See *Material Data Definition* for more information.

TYPE

Set TYPE=JOHNSON COOK (default) to define Johnson Cook hardening.

Set TYPE=TABULAR to define the hardening behavior by giving the yield stress as a function of the yield strain.

Data lines to define ply fabric shear plasticity (TYPE=JOHNSON COOK):

First line:

1. Initial yield stress, $\sigma|_0$. (Units of FL^{-2}.)
2. C. (Units of FL^{-2}.)
3. n. (Dimensionless.)
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of $\sigma|_0$, C, and n on temperature and other predefined field variables.

Data lines to define ply fabric shear plasticity (TYPE=TABULAR):

First line:

1. Yield stress.
2. Absolute value of the corresponding plastic strain. (The first tabular value entered must always be zero.)
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of yield stress on plastic strain and, if needed, on temperature and other predefined field variables.

*PML COEFFICIENT: Specify perfectly matched layer coefficients.

This option can be used only as a suboption of the **PERFECTLY MATCHED LAYER* option. It is used to specify the coefficients to be used in the perfectly matched layer (PML) to model infinite domains.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- *Coupled Acoustic-Structural Analysis*
- *Acoustic and Shock Loads*

Required parameters:

VARIATION

Set VARIATION=LINEAR to define a linear variation of the PML coefficients from zero at the interface of the acoustic and perfectly matched layer domains to the prescribed value at the outer surface of the perfectly matched layer domain.

Data line to define coefficients:

First (and only) line:

1. PML coefficient in the *X*-direction.
2. PML coefficient in the *Y*-direction.
3. PML coefficient in the *Z*-direction.

*PORE FLUID PRESSURE: Specify a known pore fluid pressure field in a static or in an explicit dynamic stress analysis.

This option is used to specify a known pore fluid pressure field in a **STATIC* analysis or in a **DYNAMIC*, EXPLICIT analysis.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Pore Fluid Pressure*

Required parameters:

FIELD

Set this parameter equal to the predefined field variable number that corresponds to the pore fluid pressure.

There are no data lines associated with this option.

*POROUS BULK MODULI: Define bulk moduli for soils and rocks.

This option is used to define the bulk moduli of solid grains and a permeating fluid such that their compressibility can be considered in the analysis of a porous medium. The **POROUS BULK MODULI* option cannot be used with the porous metal plasticity material model. For cohesive elements, this option can be used in conjunction with the **ELASTIC*, TYPE=TRACTION option to define Biot's coefficient. If this option is omitted, the bulk moduli are assumed to be infinite.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Porous Bulk Moduli*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the porous bulk moduli. If this parameter is omitted, it is assumed that the porous bulk moduli are constant or depend only on temperature.

This parameter is not active for cohesive elements.

Data lines to define bulk moduli:

First line:

1. Bulk modulus of solid grains. (Units of FL^{-2}.) If this entry is blank or zero, this value is assumed to be infinite.
2. Bulk modulus of permeating fluid. (Units of FL^{-2}.) If this entry is blank or zero, this value is assumed to be infinite.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the porous bulk moduli on temperature and other predefined field variables.

Data line for cohesive elements:

First (and only) data line:

1. E_{so}. (Units of FL^{-2}.)

Biot's coefficient then follows from the identity $\alpha_0 = 1 - E_{nn}/E_{so}$, where E_{nn} is given by the *ELASTIC, TYPE=TRACTION option.

*POROUS ELASTIC: Specify elastic material properties for porous materials.

This option is used to define the elastic parameters for porous materials.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Elastic Behavior of Porous Materials*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the parameters, in addition to temperature. If this parameter is omitted, it is assumed that the parameters depend only on temperature.

SHEAR

This parameter is relevant only when TYPE=LOGARITHMIC.

Set SHEAR=G to define the deviatoric behavior with a constant shear modulus G.

Set SHEAR=POISSON (default) to compute the instantaneous shear modulus from the bulk modulus and Poisson's ratio. The Poisson's ratio should be given on the data lines.

STIFFNESS

This parameter is relevant only when TYPE=POWER LAW.

Set STIFFNESS=TANGENT (default) to indicate that the porous elastic material constants define the tangent modulus.

Set STIFFNESS=SECANT to indicate that the porous elastic material constants define the secant modulus.

TYPE

Set TYPE=LOGARITHMIC (default) to define a logarithmic relationship between the pressure and the volumetric elastic strain.

Set TYPE=POWER LAW to define a power-law relationship between the pressure and the Young's modulus.

Data lines to define the deviatoric behavior with a constant shear modulus G for SHEAR=G and TYPE=LOGARITHMIC:

First line:

1. Value of the logarithmic bulk modulus, κ. (Dimensionless.)
2. Value of the shear modulus, G.
3. Value of the elastic tensile limit, p_t^{el}. (This value cannot be negative.)
4. Temperature, θ.
5. First field variable.

6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the material parameters κ, G, and p_t^{el} on temperature and field variables.

Data lines to define the instantaneous shear modulus from the bulk modulus and Poisson's ratio for SHEAR=POISSON and TYPE=LOGARITHMIC:

First line:

1. Value of the logarithmic bulk modulus, κ. (Dimensionless.)
2. Value of Poisson's ratio, ν.
3. Value of the elastic tensile limit, p_t^{el}. (This value cannot be negative.)
4. Temperature, θ.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the material parameters κ, ν, and p_t^{el} on temperature and field variables.

Data lines for TYPE=POWER LAW:

First line:

1. Reference Young's modulus, E_{ref}.
2. Reference pressure, p_{ref}.
3. Material parameter, p_0 (units of pressure). This value must satisfy $p_0 > 0$ and $(p_{ref} + p_0) > 0$.
4. Material parameter, n. This value must be greater than or equal to zero.
5. Poisson's ratio, ν_∞, corresponding to the limit $p \to \infty$.
6. Poisson's ratio, ν_0, corresponding to $p = 0$.
7. Material parameter, m. This value must be greater than zero.
8. Temperature, θ.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than zero):

1. First field variable.
2. Second field variable.
3. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the material parameters on temperature and field variables.

*POROUS ELECTRODE THEORY: Activate use of the electrochemistry framework to model a porous electrode.

This option is used in a material data block to activate the usage of the electrochemistry framework to model a porous electrode.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Coupled Thermal-Electrochemical Analysis*

There are no parameters or data lines associated with this option.

*POROUS FAILURE CRITERIA: Define porous material failure criteria for a POROUS METAL PLASTICITY model.

This option is used to specify the material failure criteria in a porous metal.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- **POROUS METAL PLASTICITY*
- *Porous Metal Plasticity*

There are no parameters associated with this option.

Data line to define material failure criteria:
First (and only) line:

1. Void volume fraction at total failure, $f_F > 0$. The default is 1.
2. Critical void volume fraction (threshold of rapid loss of stress carrying capacity), $f_c \geq 0$. The default is f_F.

*POROUS METAL PLASTICITY: Specify a porous metal plasticity model.

This option is used to specify the porous part of the porous metal plasticity model.

The **POROUS METAL PLASTICITY* option can be used in conjunction with the **VOID NUCLEATION* option to define the nucleation of voids. In an Abaqus/Explicit analysis it can also be used in conjunction with the **POROUS FAILURE CRITERIA* option to specify the material failure criteria.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Porous Metal Plasticity*
- **VOID NUCLEATION*
- **POROUS FAILURE CRITERIA*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies, in addition to temperature, included in the definition of the q_1, q_2, and q_3 parameters. If this parameter is omitted, q_1, q_2, and q_3 may depend only on temperature. See *Material Data Definition* for more information.

RELATIVE DENSITY

Set this parameter equal to r_0, the initial relative density of the material. If this parameter is omitted, the initial relative density is interpolated from the values given in the **INITIAL CONDITIONS*, TYPE=RELATIVE DENSITY option.

Data lines to define porous metal plasticity:

First line:

1. q_1.
2. q_2.
3. q_3.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of q_1, q_2, and q_3 on temperature and other predefined field variables.

*POST OUTPUT: Postprocess for output from the restart file.

This option can be used only for postprocessing to recover additional printed (`.dat`), output database (`.odb`), and results file (`.fil`) output from the restart file of a previous analysis.

Products: Abaqus/Standard

Type: History data

Level: Model

References:

- *About Output*
- *Restarting an Analysis*

Required parameters:

STEP

Set this parameter equal to the step number from which output is required.

Optional parameters:

CYCLE

This parameter applies only to postprocessing a low-cycle fatigue analysis (see *Low-Cycle Fatigue Analysis Using the Direct Cyclic Approach*).

Set this parameter equal to the cycle number in a low-cycle fatigue analysis from which additional output is being requested.

ITERATION

This parameter applies only to postprocessing a direct cyclic analysis (see *Direct Cyclic Analysis*).

Set this parameter equal to the iteration number in a direct cyclic analysis from which additional output is being requested.

Data lines to request output from specified increments if both the ITERATION and the CYCLE parameters are omitted:

First line:

1. First increment number at which additional output is being requested.
2. Second increment number at which additional output is being requested.
3. Etc., up to eight increments per line.

Repeat this data line as often as necessary to define the increments at which output is required.

To recover additional output from a previous direct cyclic analysis when the ITERATION parameter is included:

No data lines are needed when the ITERATION parameter is specified.

To recover additional output from a previous low-cycle fatigue analysis when the CYCLE parameter is included:

No data lines are needed when the CYCLE parameter is specified.

*POTENTIAL: Define the stress potential of a yield/creep model.

This option is used to define the stress potential for yield and creep behavior. It can be used only in conjunction with material models defined by the following options:

- **CLAY PLASTICITY*
- **CREEP*
- **CRUSHABLE FOAM*
- **DRUCKER PRAGER*
- **PLASTIC* (HARDENING=ISOTROPIC, KINEMATIC, or COMBINED; the **POTENTIAL* option can be used in conjunction with COMBINED hardening only in Abaqus/Explicit)
- **VISCOUS*

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Classical Metal Plasticity*
- *Models for Metals Subjected to Cyclic Loading*
- *Rate-Dependent Plasticity: Creep and Swelling*
- *Hill Anisotropic Yield/Creep*
- *Nonquadratic Yield*
- *Two-Layer Viscoplasticity*
- *Critical State (Clay) Plasticity Model*
- *Crushable Foam Plasticity Models*
- *Extended Drucker-Prager Models*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the stress potential coefficients in addition to temperature. If this parameter is omitted, it is assumed that the coefficients are constant or depend only on temperature. See *Material Data Definition* for more information.

POWER

This parameter applies only to Abaqus/Explicit analyses. It can be used only in conjunction with TYPE=BARLAT, BARLAT91, or HOSFORD.

Set this parameter equal to the exponent in the potential function.

TYPE

This parameter is used to define the stress potential function.

Set TYPE=BARLAT to specify nonquadratic anisotropic yield using the Barlat Yld2004-18p stress potential. This parameter setting is available only in Abaqus/Explicit.

Set TYPE=BARLAT91 to specify nonquadratic anisotropic yield using the Barlat Yld91 stress potential. This parameter setting is available only in Abaqus/Explicit.

Set TYPE=HILL (default) to specify quadratic anisotropic yield using the Hill stress potential.

Set TYPE=HOSFORD to specify nonquadratic isotropic yield using the Hosford criterion. This parameter setting is available only in Abaqus/Explicit.

Set TYPE=TRESCA to specify nonquadratic isotropic yield using the Tresca criterion. This parameter setting is available only in Abaqus/Explicit.

Data lines used to define stress ratios for the Hill stress potential (TYPE=HILL):

First line:

1. R_{11}.
2. R_{22}.
3. R_{33}.
4. R_{12}.
5. R_{13}.
6. R_{23}.
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of R_{ij} on temperature and other field variables.

Data lines used to define coefficients for the Barlat Yld2004-18p stress potential (TYPE=BARLAT):

First line:

1. c'_{1122}.
2. c'_{1133}.
3. c'_{2211}.
4. c'_{2233}.
5. c'_{3311}.
6. c'_{3322}.
7. c'_{1212}.
8. c'_{1313}.

Second line:

1. c'_{2323}.
2. c''_{1122}.
3. c''_{1133}.
4. c''_{2211}.

5. c''_{2233}.
6. c''_{3311}.
7. c''_{3322}.
8. c''_{1212}.

Third line:

1. c''_{1313}.
2. c''_{2323}.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of c'_{ijkl} and c''_{ijkl} on temperature and other field variables.

Data lines used to define coefficients for the Barlat Yld91 stress potential (TYPE=BARLAT91):

First line:

1. $\overline{a}$.
2. $\overline{b}$.
3. $\overline{c}$.
4. $\overline{f}$.
5. $\overline{g}$.
6. $\overline{h}$.
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of $\overline{a}$, $\overline{b}$, $\overline{c}$, $\overline{f}$, $\overline{g}$, and $\overline{h}$ on temperature and other field variables.

There are no data lines associated with the Hosford stress potential (TYPE=HOSFORD) or the Tresca stress potential (TYPE=TRESCA)

*PREPRINT: Select printout for the analysis input file processor.

This option is used to select the printout that will be obtained from the analysis input file processor.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Job module

References:

• *About Output*

Optional parameters:

CONTACT

This parameter applies only to Abaqus/Standard analyses.

Set CONTACT=YES to print detailed information about the contact constraints generated by the contact pair definition data. Set CONTACT=NO (default) to suppress this printout.

ECHO

Set ECHO=YES to print an echo of the input data. Set ECHO=NO (default) to suppress this printout.

HISTORY

Set HISTORY=YES to print history data. Set HISTORY=NO (default) to suppress this printout.

MODEL

Set MODEL=YES to print model definition data. Set MODEL=NO (default) to suppress this printout.

In Abaqus/Explicit, setting MODEL=YES automatically sets MASS PROPERTY=YES.

PARSUBSTITUTION

Set PARSUBSTITUTION=YES to print the modified version of the original input file that is free of input model parametrization. Set PARSUBSTITUTION=NO (default) to suppress this printout.

PARVALUES

Set PARVALUES=YES to print a modified version of the original input file showing the parameters used for model parametrization and their values. Set PARVALUES=NO (default) to suppress this printout.

MASS PROPERTY

This parameter applies only to Abaqus/Explicit analyses.

Set MASS PROPERTY=YES to print a mass property table, which includes the original mass, initial mass scaling, non-structural mass, center of mass, and rotary inertia of each user-defined element set. Set MASS PROPERTY=NO (default) to suppress this printout.

If MODEL=YES, the mass property table will be printed, even if MASS PROPERTY= NO.

There are no data lines associated with this option.

*PRESSURE PENETRATION: Specify pressure penetration loads with surface-based contact.

This option is used to prescribe pressure penetration loading simulated with surface-based contact.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

• *Fluid Pressure Penetration Loads*

Required parameters:

MAIN

Set this parameter equal to the name of the main surface of the contact pair used in the pressure penetration analysis.

SECONDARY

Set this parameter equal to the name of the secondary surface of the contact pair used in the pressure penetration analysis.

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the variation of the fluid pressure during the step. If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step or ramped up linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (see *Defining an Analysis*).

OP

Set OP=MOD (default) for existing pressure penetration loads to remain, with this option modifying existing pressure penetration loads or defining additional pressure penetration loads.

Set OP=NEW if all existing pressure penetration loads applied to the model should be removed. New pressure penetration loads can be defined.

PENETRATION TIME

Set this parameter equal to a time period over which the fluid pressure on newly penetrated contact surface segments is ramped up to the current magnitude. The default penetration time period is chosen to be 0.001 of the current step time. This parameter will be neglected in a linear perturbation analysis, in which case the current fluid pressure will be applied immediately once the pressure penetration criterion is satisfied.

WETTED FRONT

This parameter only applies to two dimensional elements.

Set WETTED FRONT=NODE (default) to indicate the fluid pressure loading is applied only to the wetted region until the front penetrated node.

Set WETTED FRONT=MID ELEMENT to indicate that the fluid pressure loading is ramped down and also applied to the unwetted front element region beyond the front penetrated node.

Optional, mutually exclusive parameters for matrix generation and steady-state dynamics analysis (direct or modal):

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the loading.

REAL

Include this parameter (default) to define the real (in-phase) part of the loading.

Data lines to define the pressure penetration loads:

First line:

1. A node or node set on the secondary surface that is exposed to the fluid. If a node set label is given, it can contain only one node for two dimensional models; while for three dimensional models, it can contain any number of nodes.
2. A node or node set on the main surface that is exposed to the fluid. If a node set label is given, it can contain only one node for two dimensional models; while for three dimensional models, it can contain any number of nodes. This field should be left blank if the main surface is an analytical rigid surface. This field can be also left blank for a three-dimensional model even if the main surface is not an analytical rigid surface.
3. Fluid pressure magnitude.
4. Enter the critical contact pressure below which fluid penetration starts to occur. The higher this value, the easier the fluid penetrates. The default is zero, in which case fluid penetration occurs only if contact is lost.

Repeat this data line as often as necessary to define fluid penetrations from different locations on the surface, possibly with different fluid pressure magnitudes. Alternatively, the **PRESSURE PENETRATION* option can be repeated (if, for example, a different amplitude reference is needed).

*PRESSURE STRESS: Specify equivalent pressure stress as a predefined field for a mass diffusion analysis.

This option can be used only in a *MASS DIFFUSION analysis to specify pressure as a predefined field. The user defines equivalent pressure stresses at the nodes, and Abaqus/Standard interpolates the pressure to the material points.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Mass Diffusion Analysis*
- *Predefined Fields*

Optional parameters for using the data line format:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that gives the time variation of the pressure throughout the step. If the AMPLITUDE parameter is omitted, the reference magnitude is applied either immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the *STEP option (see *Defining an Analysis*).

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

OP

Set OP=MOD (default) for existing *PRESSURE STRESS values to remain, with this option modifying existing values or defining additional values.

Set OP=NEW if all existing *PRESSURE STRESS values should be removed. New pressure stress values can be defined.

For a general analysis step, a pressure that is removed via OP=NEW is reset to the value given on the *INITIAL CONDITIONS option or to zero if no initial pressure was defined. For a linear perturbation step, a pressure that is removed via OP=NEW is always reset to zero. If pressures are being returned to their initial condition values, the AMPLITUDE parameter described above does not apply. Rather, the AMPLITUDE parameter given on the *STEP option governs the behavior. If pressures are being reset to new values (not to initial conditions) via OP=NEW, the AMPLITUDE parameter described above applies.

Required parameter for reading equivalent pressure stresses from the results file:

FILE

Set this parameter equal to the name of the results file (including the optional `.fil` extension) from which the data are read. See *Input Syntax Rules* for the syntax of such file names.

Optional parameters for reading equivalent pressure stresses from the results file:

BSTEP

Set this parameter equal to the step number (of the analysis whose results file is being used as input to this option) that begins the history data to be read. If no value is supplied, Abaqus/Standard will begin reading pressure data from the first step available on the results file.

BINC

Set this parameter equal to the increment number (of the analysis whose results file is being used as input to this option) that begins the history data to be read. If no value is supplied, Abaqus/Standard will begin reading pressure data from the first increment available (excluding any zero increments) for step BSTEP on the results file.

ESTEP

Set this parameter equal to the step number (of the analysis whose results file is being used as input to this option) that ends the history data to be read. If no value is supplied, ESTEP is taken as equal to BSTEP.

EINC

Set this parameter equal to the increment number (of the analysis whose results file is being used as input to this option) that ends the history data to be read. If no value is supplied, EINC is taken as the last available increment of step ESTEP on the results file.

Required parameter for defining data in user subroutine `UPRESS`:

USER

Include this parameter to indicate that user subroutine *UPRESS* will be used to define equivalent pressure stress values. *UPRESS* will be called for each node given on the data lines. If values are also given on the data lines, these values will be ignored. If a results file has been specified in addition to user subroutine *UPRESS*, values read from the results file will be passed into *UPRESS* for possible modification.

Data lines to define pressures using the data line format:

First line:

1. Node set or node number. If a node set label is given, all nodes in this set must have identical initial pressures.
2. Reference pressure value (positive in compression). If the AMPLITUDE parameter is present, this value will be modified by the AMPLITUDE specification.

Repeat this line as often as necessary to define the pressure at different nodes or node sets.

To read pressures from an Abaqus/Standard results file (FILE):

No data lines are used when pressure data are read from a results file.

Data lines to define equivalent pressure stresses using user subroutine `UPRESS`:

First line:

1. Node set or node number. If a node set label is given, all nodes in this set must have identical initial equivalent pressure stress values.

Repeat this line as often as necessary. *UPRESS* will be called for each node listed.

*PRESTRESS HOLD: Keep rebar prestress constant during initial equilibrium solution.

This option is used within a *STATIC step (*Static Stress Analysis*) to keep the stress in some or all of the rebar constant during the initial equilibrium solution.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

• *Defining Reinforcement*

There are no parameters associated with this option.

Data lines to hold the prestress constant:

First line:

1. Element set name.
2. Rebar name. The stress in all rebar included in the above element set will be held fixed throughout the step.
3. Etc.

Repeat this data line as often as necessary. Give four pairs of data per line.

*PRE-TENSION SECTION: Associate a pre-tension node with a pre-tension section.

This option is used to associate a pre-tension node with a pre-tension section.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Load module

References:

• *Prescribed Assembly Loads*

Required parameters:

NODE

Set this parameter equal to either the pre-tension node number or the name of a node set containing the pre-tension node. If the name of a node set is chosen, the node set must contain exactly one node.

Required, mutually exclusive parameters:

ELEMENT

Set this parameter equal to either the element number of the truss or beam element used to define the pre-tension section or the name of an element set containing the truss or beam element used to define the pre-tension section. If the name of an element set is chosen, the element set must contain exactly one element.

SURFACE

Set this parameter equal to the name of the surface definition (**SURFACE*) that defines the pre-tension section (when continuum elements are used).

Optional parameters:

FOLLOWER

Omit this parameter or set FOLLOWER=NO to indicate that the normal of the pre-tension section is fixed during the analysis.

Include this parameter or set FOLLOWER=YES to indicate that the normal of the pre-tension section is updated when performing large-displacement analysis. In this case Abaqus/Standard ignores the optional data line used to define a fixed normal to the pre-tension section.

Data line to define the normal to the section (optional):

First (and only) data line:

1. First component of the normal.
2. Second component of the normal.
3. Third component of the normal.

If the data line is omitted, Abaqus/Standard will compute an average normal to the pre-tension section for continuum elements. For truss or beam elements the default normal points from the first to the last node in the element connectivity.

*PRINT: Request or suppress output to the message file in an Abaqus/Standard analysis or to the status file in an Abaqus/Explicit analysis.

This option is used to obtain or suppress detailed printout in the message (`.msg`) file in an Abaqus/Standard analysis or in the status (`.sta`) file in an Abaqus/Explicit analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *About Output*
- *Output to the Output Database*

Optional parameters in Abaqus/Standard analyses:

ADAPTIVE MESH

Set ADAPTIVE MESH=YES to request detailed output during adaptive mesh smoothing. The default is ADAPTIVE MESH=NO.

CONTACT

Set CONTACT=YES to request detailed output of points that are contacting or separating in interface and gap problems. This output is useful in difficult contact problems to track the development of the solution during iteration within an increment. The output is printed for every increment unless FREQUENCY=0. The default is CONTACT=NO.

FREQUENCY

Set this parameter equal to the output frequency, in increments. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output. Unless you set FREQUENCY=0, the output will always print at the last increment of each step. The output for the last increment may occur several times due to unsuccessful attempts (nonconvergence of global Newton iterations) or may not occur at all if the step ends prematurely.

MODEL CHANGE

Set MODEL CHANGE=YES to request detailed output of which elements are being removed or reactivated in the step. This output includes the new original coordinates and normals of elements being reactivated strain free in a large-displacement analysis. The default is MODEL CHANGE=NO.

PLASTICITY

Set PLASTICITY=YES to request detailed output of element and integration point numbers for which the plasticity algorithms have failed to converge in the material routines. This output is useful to determine the location in the mesh and the plasticity model for which Abaqus/Standard is encountering material model difficulties. This information may help in identifying modeling problems as well as material parameter specification problems. The default is PLASTICITY=NO.

RESIDUAL

Set RESIDUAL=YES (default) if the output of equilibrium residuals is to be given during the equilibrium iterations. Set RESIDUAL=NO to suppress the output.

SOLVE

Set SOLVE=YES (default) to request information regarding the actual number of equations and the memory requirement in each iteration. Set SOLVE=NO to suppress the output.

Optional parameters in Abaqus/Explicit analyses:

ALLKE

Set ALLKE=YES to request that a column containing the total kinetic energy be printed in the status file. Set ALLKE=NO to suppress this printout. The default is ALLKE=YES.

CRITICAL ELEMENT

Set CRITICAL ELEMENT=YES to request that a column listing the element that has the smallest stable time increment be printed in the status file. Set CRITICAL ELEMENT=NO to suppress this printout. The default is CRITICAL ELEMENT=YES.

DMASS

Set DMASS=YES to request that a column containing the percent change in total mass of the model due to mass scaling be printed in the status file. Set DMASS=NO to suppress this printout. The default is DMASS=NO unless active mass scaling is present in the step. **FIXED MASS SCALING* propagated from a previous step is not considered to be active mass scaling.

ETOTAL

Set ETOTAL=YES to request that a column containing the energy balance of the model be printed in the status file. Set ETOTAL=NO to suppress this printout. The default is ETOTAL=YES.

Other energy variables can also be printed. Please refer to *Total Energy Output* for energy variables available in Abaqus/Explicit.

MASS

Set MASS=YES to request that a column containing the total mass of the model be printed in the status file. Set MASS=NO to suppress this printout. The default is MASS=NO.

There are no data lines associated with this option.

*PROBABILITY DENSITY FUNCTION: Specify a probability density function.

This option is used to define a probability density function.

Products: Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- *Probability Density Function*
- *Particle Generator*

Required parameters:

NAME

Set this parameter equal to the name of the probability density function.

TYPE

Set TYPE=UNIFORM to define a uniform probability density function.

Set TYPE=NORMAL to define a normal probability density function.

Set TYPE=LOGNORMAL to define a log-normal probability density function.

Set TYPE=PIECEWISE LINEAR to specify a histogram.

Set TYPE=DISCRETE to specify a discrete probability density function.

Data line to define a uniform probability density function (TYPE=UNIFORM):

First (and only) line:

1. Minimum value of random variable.
2. Maximum value of random variable.

Data line to define a normal probability density function (TYPE=NORMAL or LOGNORMAL):

First (and only) line:

1. Minimum value of random variable.
2. Maximum value of random variable.
3. Mean value of random variable.
4. Standard deviation of random variable.

Data lines to specify a histogram (TYPE=PIECEWISE LINEAR):

First line:

1. Enter the value of the random variable (midpoint of a range of values for a histogram).
2. Enter the percentage of the total count that fall in this range of values.

Repeat this data line as often as necessary. Up to eight pairs of entries are allowed per line.

Data line to define a constant probability density function (TYPE=DISCRETE):

First line:

1. Enter a discrete value of the random variable.
2. Enter the probability of occurrence of this discrete value.

Repeat this data line as often as necessary. Up to eight pairs of entries are allowed per line.

*PROPERTY TABLE: Define a property table.

This option can be used only as a suboption of the *TABLE COLLECTION* or *MATERIAL* option to define a property table.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- *About Table Collections, Parameter Tables, and Property Tables*
- *Accessing Abaqus Table Collections*
- *PARAMETER TABLE TYPE*
- *PROPERTY TABLE TYPE*
- *TABLE COLLECTION*
- *PARAMETER TABLE*

Required parameters:

TYPE

Set this parameter equal to the name of the property table type that defines the type of this property table.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the properties. If this parameter is omitted, it is assumed that the properties do not depend on field variables.

EXTRAPOLATION

Set EXTRAPOLATION=CONSTANT (default) to use constant extrapolation of the dependent variables outside the specified range of the independent variables.

Set EXTRAPOLATION=LINEAR to use linear extrapolation of the dependent variables outside the specified range of the independent variables.

LABEL

Set this parameter equal to the label to be used as the reference to the property table. If this parameter is omitted, the label is set equal to the name of the property table type.

REGULARIZE

This parameter applies only to Abaqus/Explicit analyses.

Set REGULARIZE=ON (default) to regularize the user-defined property table data.

Set REGULARIZE=OFF to use the user-defined property table data directly without regularization.

Set REGULARIZE=ORIGIN to regularize the user-defined property table data and preserve property values at point 0 of the first independent variable.

RTOL

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the tolerance to be used to regularize the property table data. The default is RTOL=0.03.

TEMPERATURE

Include this parameter if the properties depend on temperature. If this parameter is omitted, it is assumed that the properties do not depend on temperature.

Data lines to define properties as a function of independent variables, temperature, and/or field variables:

First line:

1. First property.
2. Second property or first independent variable, temperature, or field variable.
3. Etc., up to 8 data line items.

Subsequent lines (only needed if the sum of the entries for the PROPERTIES and INDEPENDENT VARIABLES parameters on the **PROPERTY TABLE TYPE* option and the TEMPERATURE and DEPENDENCIES parameters on the **PROPERTY TABLE* option is greater than eight):

1. Property, independent variable, temperature, or field variable.
2. Etc., up to 8 data line items.

Repeat this set of data lines as often as necessary to define properties as a function of independent variables, temperature, and other predefined field variables.

*PROPERTY TABLE TYPE: Define the type of property table.

This option is used to define the type of property table.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Accessing Abaqus Table Collections*
- **PARAMETER TABLE TYPE*
- **TABLE COLLECTION*
- **PARAMETER TABLE*
- **PROPERTY TABLE*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the property table type.

PROPERTIES

Set this parameter equal to the number of properties.

Optional parameters:

INDEPENDENT VARIABLES

Set this parameter equal to the number of independent variables.

Optional data lines:

First line:

1. Description of the first property. This field can be left blank and is unused in the analysis.

Repeat this data line as often as necessary to define all of the property descriptions as indicated by the value of the PROPERTIES parameter.

*PSD-DEFINITION: Define a cross-spectral density frequency function for random response loading.

This option is used to define a frequency function for reference in the **CORRELATION* option to define the frequency dependence of the random loading in the **RANDOM RESPONSE* analysis procedure.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Random Response Analysis*
- *UPSD*
- **CORRELATION*
- **RANDOM RESPONSE*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to this frequency function.

Optional parameters:

DB REFERENCE

Set this parameter equal to the reference power value, in $(\text{load units})^2$. This parameter is required when the frequency function is given in decibel units (TYPE=DB).

G

Set this parameter equal to the reference gravity acceleration; for example, 9.81 m/s^2. The default is G=1.0. This parameter can be used only with TYPE=BASE.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

TYPE

Set TYPE=BASE if this frequency function will be used to define a base motion.

Set TYPE=FORCE (default) if this frequency function is given directly in power units.

Set TYPE=DB if this frequency function is defined in decibel units (see below). This option cannot be used with the USER parameter.

USER

Include this parameter if the frequency function is defined in user subroutine *UPSD*. If this parameter is included, no data lines are needed.

Data lines for TYPE=BASE or TYPE=FORCE:

First line:

1. Real part of the frequency function, in units^2 per frequency.

2. Imaginary part of the frequency function, in units^2 per frequency.
3. Frequency, in cycles/time.

Repeat this data line as often as necessary to define the frequency function.

Data lines for TYPE=DB:

First line:

1. Real part of the frequency function, in decibels.
2. Imaginary part of the frequency function, in decibels.
3. Frequency band number (1–15). Up to 15 standard frequency bands can be used.

Repeat this data line as often as necessary to define the frequency function in decibels.

To define the frequency function by a user subroutine (USER parameter included):

No data lines are used with this option if the USER parameter is specified. Instead, user subroutine *UPSD* must be used to define the frequency function.

R

This section describes each Abaqus input option that begins with R, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **RADIATE*
- **RADIATION FILE*
- **RADIATION OUTPUT*
- **RADIATION PRINT*
- **RADIATION SYMMETRY*
- **RADIATION VIEW FACTOR*
- **RANDOM RESPONSE*
- **RATE DEPENDENT*
- **RATIOS*
- **REACTION RATE*
- **REBAR*
- **REBAR LAYER*
- **REBAR LINE*
- **REDUCED BASIS GENERATE*
- **REFLECTION*
- **RELEASE*
- **RESPONSE SPECTRUM*
- **RESTART*
- **RETAINED NODAL DOFS*
- **RIGID BODY*
- **RIGID SURFACE*
- **ROTARY INERTIA*

*RADIATE: Specify radiation conditions in heat transfer analyses.

This option is used to apply radiation boundary conditions between a nonconcave surface and a nonreflecting environment in fully coupled thermal-stress analysis. In Abaqus/Standard it is also used for all procedures that solve for nodal temperature degrees of freedom.

It must be used in conjunction with the **PHYSICAL CONSTANTS* option, which is used to define the Stefan-Boltzmann constant.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

• *Thermal Loads*

Optional parameters:

AMBIENT AMPLITUDE

Set this parameter equal to the name of the **AMPLITUDE* option that gives the variation of the ambient temperature with time.

If this parameter is omitted in an Abaqus/Standard analysis, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (see *Defining an Analysis*). If this parameter is omitted in an Abaqus/Explicit analysis, the reference magnitude given on the data lines is applied throughout the step.

OP

Set OP=MOD (default) for existing **RADIATE* definitions to remain, with this option modifying existing radiation conditions or defining additional radiation conditions.

Set OP=NEW if all existing **RADIATE* definitions applied to the model should be removed.

RADIATION AMPLITUDE

Set this parameter equal to the name of the **AMPLITUDE* option that gives the variation of the total radiation flux with time.

If this parameter is omitted in an Abaqus/Standard analysis, the total radiation flux, accounting for any AMBIENT AMPLITUDE, is applied immediately, independent of the value assigned to the AMPLITUDE parameter on the **STEP* option. If this parameter is omitted in an Abaqus/Explicit analysis, the total radiation flux, accounting for any AMBIENT AMPLITUDE, is applied immediately at the beginning of the step.

REGION TYPE

This parameter applies only to Abaqus/Explicit analyses.

This parameter is relevant only for radiation conditions applied to the boundary of an adaptive mesh domain. If a radiation condition is applied to a surface in the interior of an adaptive mesh domain, the nodes on the surface will move with the material in all directions (they will be nonadaptive). Abaqus/Explicit will create a boundary region automatically on the surface subjected to the defined radiation condition.

Set REGION TYPE=LAGRANGIAN (default) to apply the radiation condition to a Lagrangian boundary region. The edge of a Lagrangian boundary region will follow the material while allowing adaptive meshing along the edge and within the interior of the region.

Set REGION TYPE=SLIDING to apply the radiation condition to a sliding boundary region. The edge of a sliding boundary region will slide over the material. Adaptive meshing will occur along the edge and in the interior of the region. Mesh constraints are typically applied on the edge of a sliding boundary region to fix it spatially.

Set REGION TYPE=EULERIAN to apply the radiation condition to an Eulerian boundary region. This option is used to create a boundary region across which material can flow. Mesh constraints must be used normal to an Eulerian boundary region to allow material to flow through the region. If no mesh constraints are applied, an Eulerian boundary region will behave in the same way as a sliding boundary region.

Data lines to define radiation conditions:

First line:

1. Element number or element set name.
2. Radiation type (see *About the Element Library*).
3. Reference ambient temperature value, θ^0. (Units of θ.)
4. Emissivity, ϵ.

Repeat this data line as often as necessary to define radiation conditions.

*RADIATION FILE: Define results file requests for cavity radiation heat transfer.

This option is used to write cavity radiation variables to the Abaqus/Standard results file.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Unsupported; Abaqus/CAE reads output from the output database file only.

References:

- *Cavity Radiation in Abaqus/Standard*
- *About Output*

Optional, mutually exclusive parameters (if not specified, output will be provided for all cavities in the model):

CAVITY

Set this parameter equal to the name of the cavity for which this output request is being made.

ELSET

Set this parameter equal to the name of the element set for which this output request is being made.

SURFACE

Set this parameter equal to the name of the surface for which this output request is being made.

Optional parameters:

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be written to the results file at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

Data lines to request results file output:

First line:

1. Give the identifying keys for the variables to be written to the results file for this request. The keys are defined in the “Surface variables” section of *Abaqus/Standard Output Variable Identifiers*.

Repeat this data line as often as necessary to define the surface variables to be written to the results file for the specified cavity, surface, or element set. If this line is omitted, the default variables will be used.

*RADIATION OUTPUT: Define output database requests for cavity radiation variables.

This option is used to write cavity radiation variables to the output database. It must be used in conjunction with the *OUTPUT option.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Output to the Output Database*
- *OUTPUT

Optional, mutually exclusive parameters (if not specified, output will be provided for all cavities in the model):

CAVITY

Set this parameter equal to the name of the cavity for which this output request is being made.

ELSET

Set this parameter equal to the name of the element set for which this output request is being made.

SURFACE

Set this parameter equal to the name of the surface for which this output request is being made.

Optional parameters:

VARIABLE

Set VARIABLE=ALL to indicate that all cavity radiation variables applicable to this procedure and material type should be written to the output database.

If this parameter is omitted, the cavity radiation variables requested for output must be specified on the data lines.

Data lines to request cavity radiation output:

First line:

1. Specify the identifying keys for the variables to be written to the output database. The keys are defined in *Abaqus/Standard Output Variable Identifiers*.

Repeat this data line as often as necessary to define the cavity radiation variables to be written to the output database.

*RADIATION PRINT: Define print requests for cavity radiation heat transfer.

This option is used to print tabular output of cavity radiation variables (radiation fluxes, view factor totals, and facet temperatures).

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Cavity Radiation in Abaqus/Standard*
- *About Output*

Optional, mutually exclusive parameters (if not specified, output will be provided for all cavities in the model):

CAVITY

Set this parameter equal to the name of the cavity for which this output request is being made.

ELSET

Set this parameter equal to the name of the element set for which this output request is being made.

SURFACE

Set this parameter equal to the name of the surface for which this output request is being made.

Optional parameters:

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

SUMMARY

Set SUMMARY=YES (default) to obtain a summary of the maximum and minimum values in each column of the table and their locations.

Set SUMMARY=NO to suppress this summary.

TOTALS

Set TOTALS=YES to print the total of each column in the table. This is useful, for example, to sum radiation fluxes over all facets composing a radiation surface. The default is TOTALS=NO.

Data lines to request printed output:

First line:

1. Give the identifying keys for the variables to be printed in a table for this request. The keys are defined in the “Surface variables” section of *Abaqus/Standard Output Variable Identifiers*.

Repeat this data line as often as necessary: each line defines a table (or more than one table if the request is for a cavity made up of more than one surface). If this line is omitted, the default variables will be used.

*RADIATION SYMMETRY: Define cavity symmetries for radiation heat transfer analysis.

This option must precede the *CYCLIC, *PERIODIC, and/or *REFLECTION options to specify symmetries in cavities used for cavity radiation heat transfer analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *Cavity Radiation in Abaqus/Standard*
- *CYCLIC
- *PERIODIC
- *REFLECTION

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the symmetry reference appearing in the *RADIATION VIEW FACTOR option.

There are no data lines associated with this option.

*RADIATION VIEW FACTOR: Control cavity radiation and view factor calculations.

This option is used to control the calculation of view factors during a cavity radiation analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

• *Cavity Radiation in Abaqus/Standard*

Optional parameters:

BLOCKING

Set BLOCKING=ALL (default) to specify that full blocking checks be performed in the view factor calculations.

Set BLOCKING=NO to specify that no blocking checks be performed in the view factor calculations.

Set BLOCKING=PARTIAL to specify that partial blocking checks be performed in the view factor calculations. The data lines are then used to specify the potential blocking surfaces.

CAVITY

Set this parameter equal to the name of the cavity for which radiation view factor control is being specified. If this parameter is omitted, the specification applies to all cavities defined in the model.

INFINITESIMAL

Set this parameter equal to the facet area ratio above which the infinitesimal-to-finite area approximation is used for view factor calculations. The default value is 64.0.

INTEGRATION

Set this parameter equal to the number of Gauss integration points to be used along each edge when the numerical integration of contour integrals is used for view factor calculations. One to five integration points are allowed. The default value is three integration points.

LUMPED AREA

Set this parameter equal to the nondimensional distance-square value above which the lumped area approximation is used for view factor calculations. The default value is 5.0.

OFF

Include this parameter to switch off cavity radiation effects. Cavity radiation effects can be switched on again by a subsequent use of this option without the OFF parameter. If this parameter is omitted, cavity radiation is active.

RANGE

Set this parameter equal to a distance beyond which view factors need not be calculated because surfaces are judged to be too far apart to “see” each other (due to blocking by other surfaces).

REFLECTION

Set REFLECTION=YES (default) to indicate that reflection must be included in the cavity radiation calculations.

Set REFLECTION=NO to indicate that reflection is to be ignored in the cavity radiation calculations. No reflection corresponds to the special case of *black body* radiation (see *Cavity radiation*).

SYMMETRY

Include this parameter to indicate the existence of radiation symmetries in the model. This parameter must be set equal to the name appearing in the **RADIATION SYMMETRY* option where the symmetries are defined. If this parameter is omitted, it is assumed that there are no radiation symmetries in the cavity.

VTOL

Set this parameter equal to the acceptable tolerance for the view factor calculations. If this parameter is omitted, the default view factor tolerance is 0.05.

Optional parameters (for use with the *MOTION option):

MDISP

Set this parameter equal to the maximum allowable motion of any node in the monitored node set before recalculation of radiation view factors. This parameter is relevant only when the **MOTION* option appears in conjunction with the **RADIATION VIEW FACTOR* option. This parameter must be used with the NSET parameter.

NSET

Set this parameter equal to the name of the node set whose displacements are to be monitored to decide when geometrical changes are significant enough to trigger recalculation of the radiation view factors during the step. This parameter is relevant only when the **MOTION* option appears in conjunction with the **RADIATION VIEW FACTOR* option. This parameter must be used with the MDISP parameter.

Data lines to define blocking surfaces (BLOCKING=PARTIAL):

First line:

1. List of surfaces that provide blocking inside the cavity, up to eight surfaces per line.

Repeat this data line as often as necessary to define partial blocking.

There are no data lines associated with this option if BLOCKING=PARTIAL is not specified

*RANDOM RESPONSE: Calculate response to random loading.

This option is used to give the linearized response of a model to random excitation.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Random Response Analysis*
- **CORRELATION*
- **PSD-DEFINITION*

There are no parameters associated with this option.

Data lines for a random response analysis:

First line:

1. Lower limit of frequency range, in cycles/time.
2. Upper limit of frequency range, in cycles/time. If this value is given as zero, it is assumed that results are required at only one frequency.
3. Number of points between eigenfrequencies at which the response should be calculated, including the end points, from the lower limit of the frequency range to the first eigenfrequency in the range; in each interval from eigenfrequency to eigenfrequency; and from the highest eigenfrequency in the range to the high limit of the frequency range. If the value given is less than two (or omitted), the default value of 20 points is assumed. Accurate RMS values can be obtained only if enough points are used so that Abaqus/Standard can integrate accurately over the frequency range.
4. Bias parameter. This parameter is useful only if results are requested at four or more frequency points. It is used to bias the results points toward the ends of the intervals so that better resolution is obtained there, since the ends of each interval are the eigenfrequencies where the response amplitudes vary most rapidly. The default bias parameter is 3.0. The bias formula is defined in *Random Response Analysis*.
5. Frequency scale choice. Enter 1 to use a linear frequency scale. If this field is left blank or is 0, a logarithmic frequency scale is used.

Repeat this data line as often as necessary to define the frequency ranges in which results are required.

*RATE DEPENDENT: Define a rate-dependent viscoplastic model.

This option can be used only as a suboption of the *PLASTIC, HARDENING=ISOTROPIC option; the *PLASTIC, HARDENING=COMBINED option; the *PLASTIC, HARDENING=JOHNSON COOK option; the *DRUCKER PRAGER HARDENING option; or the *CRUSHABLE FOAM HARDENING option to introduce strain rate dependence in the material models.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Rate-Dependent Yield*
- *Classical Metal Plasticity*
- *Johnson-Cook Plasticity*
- *Extended Drucker-Prager Models*
- *Crushable Foam Plasticity Models*

Optional parameters:

DEPENDENCIES

In an Abaqus/Explicit analysis this parameter is relevant only for TYPE=POWER LAW or TYPE=YIELD RATIO.

Set this parameter equal to the number of field variable dependencies in the definition of material parameters, in addition to temperature. If this parameter is omitted, it is assumed that the rate-dependent material behavior depends only on temperature. See *Material Data Definition* for more information.

TYPE

Set TYPE=POWER LAW (default) to specify the Cowper-Symonds overstress power law.

Set TYPE=CHABOCHE to specify Chaboche rate dependence.

Set TYPE=JOHNSON COOK to specify Johnson-Cook rate dependence (this option cannot be used with the crushable foam plasticity model).

Set TYPE=YIELD RATIO to enter yield stress ratios.

If this option is used in conjunction with the *CRUSHABLE FOAM option, the static hardening relation must be defined on the *CRUSHABLE FOAM HARDENING option.

Data lines to define the overstress power law parameters (TYPE=POWER LAW):

First line:

1. D. (Units of T^{-1}.)
2. n.
3. Temperature.
4. First field variable.
5. Second field variable.

6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the material parameters on temperature and other field variables.

Data lines to define the overstress power law parameters (TYPE=CHABOCHE):

First line:

1. $\dot{\varepsilon}_0$. The default is 1.0. (Units of T^{-1}.)
2. K. (Units of FL^{-2}.)
3. n.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the material parameters on temperature and other field variables.

Data line to define the Johnson-Cook rate parameters (TYPE=JOHNSON COOK):

First (and only) line:

1. C.
2. $\dot{\varepsilon}_0$. (Units of T^{-1}.)

Data lines to define yield stress ratios (TYPE=YIELD RATIO):

First line:

1. Yield stress ratio, $R = \overline{\sigma}/\sigma^0$.
2. Equivalent plastic strain rate, $\dot{\bar{\varepsilon}}^{pl}$ (or $\left|\dot{\varepsilon}^{pl}_{axial}\right|$, the absolute value of the axial plastic strain rate in uniaxial compression, for the crushable foam model).
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the yield stress ratio on equivalent plastic strain and, if needed, on temperature and other predefined field variables.

*RATIOS: Define anisotropic swelling.

This option is used to specify ratios that define anisotropic swelling. The **RATIOS* option can be used only in conjunction with the **MOISTURE SWELLING* option or the **SWELLING* option, and it should appear immediately after either one.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Moisture Swelling*
- *Rate-Dependent Plasticity: Creep and Swelling*
- **MOISTURE SWELLING*
- **SWELLING*

Optional parameter when the *RATIOS option is used with the *SWELLING option:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the anisotropy ratios in addition to temperature. If this parameter is omitted, it is assumed that the ratios depend only on temperature. See *Material Data Definition* for more information.

Data lines to define anisotropic swelling ratios:

First line:

1. r_{11}.
2. r_{22}.
3. r_{33}.
4. Temperature. This value and all of the following field variables can be defined only if the **RATIOS* option is used with the **SWELLING* option.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the anisotropic swelling ratios on temperature and other field variables.

*REACTION RATE: Define the reaction rate for an ignition and growth equation of state.

This option is used to define the reaction rate for an ignition and growth equation of state. It is required when the *EOS, TYPE=IGNITION AND GROWTH option is used. The *REACTION RATE option should appear immediately after the *EOS or the *GAS SPECIFIC HEAT option.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Equation of State*

There are no parameters associated with this option.

Data lines to define the reaction rate:

First line:

1. Constant, I. (Units of T^{-1}.)
2. Product covolume, a. (Dimensionless.)
3. Exponent on the unreacted fraction (ignition term), b. (Dimensionless.)
4. Exponent (ignition term), x. (Dimensionless.)
5. First burn rate coefficient, G_1. (Units of T^{-1}.)
6. Exponent on the unreacted fraction (growth term), c. (Dimensionless.)
7. Exponent on the reacted fraction (growth term), d. (Dimensionless.)
8. Pressure exponent (growth term), y. (Dimensionless.)

Second line:

1. Second burn rate coefficient, G_2. (Units of T^{-1}.)
2. Exponent on the unreacted fraction (completion term), e. (Dimensionless.)
3. Exponent on the reacted fraction (completion term), g. (Dimensionless.)
4. Pressure exponent (completion term), z. (Dimensionless.)
5. Initial reacted fraction, F_{ig}^{max}. (Dimensionless.)
6. Maximum reacted fraction for the growth term, F_{G1}^{max}. (Dimensionless.)
7. Minimum reacted fraction for the completion term, F_{G2}^{min}. (Dimensionless.)
8. Reference pressure, p_{ref}. (Units of FL^{-2}.) If this field is left blank, a default value of 1.0 is used.

*REBAR: Define rebar as an element property.

This option is used as an alternative method to define rebar as an element property in shells, membranes, and solid (continuum) elements. It must be used to define rebar in beams in Abaqus/Standard analyses. The preferred option for defining rebar in shells, membranes, and surface elements is the **REBAR LAYER* option, which must be used in conjunction with the **SHELL SECTION*, the **MEMBRANE SECTION*, or the **SURFACE SECTION* options. The preferred method for defining rebar in solids is to embed reinforced surface or membrane elements in "host" solid elements using the **EMBEDDED ELEMENT* option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- *Defining Rebar as an Element Property*
- *Defining Reinforcement*

Required parameters:

ELEMENT

Set ELEMENT=BEAM to define rebar in beam elements in an Abaqus/Standard analysis.

Set ELEMENT=SHELL to define rebar in three-dimensional shell elements. Rebar cannot be used with triangular shell elements.

Set ELEMENT=AXISHELL to define rebar in axisymmetric shell elements.

Set ELEMENT=MEMBRANE to define rebar in three-dimensional membrane elements. Rebar cannot be used with triangular membrane elements.

Set ELEMENT=AXIMEMBRANE to define rebar in axisymmetric membrane elements in an Abaqus/Standard analysis.

Set ELEMENT=CONTINUUM to define rebar in continuum (solid) elements. Rebar cannot be used with any plane triangular, triangular prism, tetrahedral, or infinite elements.

MATERIAL

Set this parameter equal to the name of the material of which these rebar are made.

NAME

Set this parameter equal to a label that will be used to refer to this rebar set. This label can be used in defining rebar prestress and output requests. Each layer of rebar must be assigned a separate name in a particular element or element set.

Optional parameters:

GEOMETRY

This parameter is not meaningful for rebar in beams, axisymmetric shells, or axisymmetric membranes, or for single rebar in continuum elements.

Set GEOMETRY=ISOPARAMETRIC (default) to indicate that the layer of rebar is parallel to a direction of the element local (isoparametric) coordinate system.

Set GEOMETRY=SKEW to indicate that the rebar layer is in a skew direction with respect to the element faces.

ISODIRECTION

Set this parameter equal to the isoparametric direction from which the rebar angle output will be measured. The default is 1.

ORIENTATION

This parameter is meaningful only for skew rebar in shell and membrane elements. Set this parameter equal to the name of an orientation definition that defines the angular orientation of the rebar. This parameter is not permitted with axisymmetric shell and axisymmetric membrane elements.

SINGLE

This parameter is meaningful only for continuum elements. Include this parameter if a single rebar is being defined by each data line. If this parameter is omitted, each line defines a layer of uniformly spaced rebar in the element isoparametric space.

Data lines to define rebar in beam elements:

First line:

1. Element number or name of the element set that contains these rebar.
2. Cross-sectional area of the rebar.
3. Distance $\boldsymbol{X_1}$ (see *Figure 1*).
4. Distance $\boldsymbol{X_2}$.

Repeat this data line as often as necessary. Each line defines a single rebar.

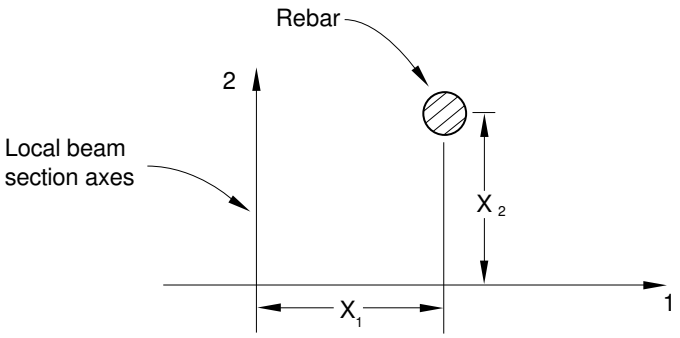


Figure 1: Rebar location in a beam section.

Data lines to define isoparametric rebar in three-dimensional shell elements:

First line:

1. Element number or name of the element set that contains these rebar.
2. Cross-sectional area of each rebar.
3. Spacing of the rebar in the plane of the shell. The default is 1.0.
4. Position of the rebar in the shell section thickness direction. This value is given as the distance of the rebar from the middle surface of the shell, positive in the direction of the positive normal to the shell. This value is modified if the NODAL THICKNESS parameter is included with the **SHELL SECTION* option of the underlying shell element.
5. Edge number to which the rebar are parallel in the element's local (isoparametric) coordinate system. See *Figure 2*.

Repeat this data line as often as necessary. Each line defines a layer of rebar.

Data lines to define isoparametric rebar in three-dimensional membrane elements:

First line:

1. Element number or name of the element set that contains these rebar.
2. Cross-sectional area of each rebar.
3. Spacing of the rebar in the plane of the membrane. The default is 1.0.
4. Edge number to which the rebar are parallel in the element's local (isoparametric) coordinate system. See *Figure 2*.

Repeat this data line as often as necessary. Each line defines a layer of rebar.

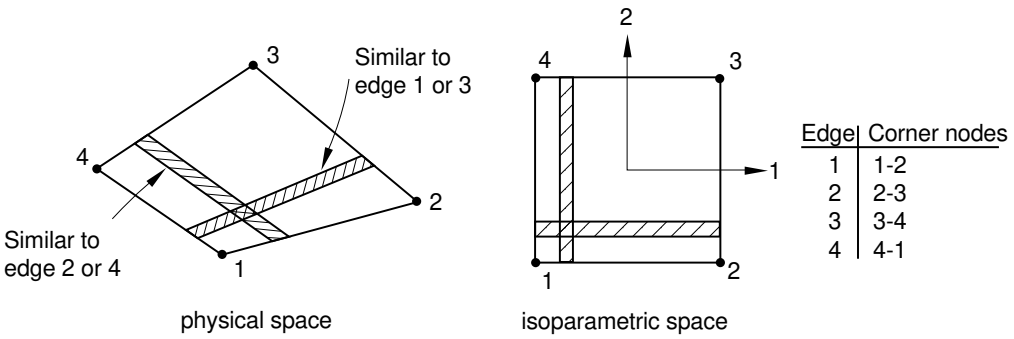


Figure 2: "Isoparametric" rebar in a three-dimensional shell or membrane.

Data lines to define skew rebar in three-dimensional shell elements:

First line:

1. Element number or name of the element set that contains these rebar.
2. Cross-sectional area of each rebar.
3. Spacing of rebar in the plane of the shell. The default is 1.0.
4. Position of the rebar in the shell section thickness direction. This value is given as the distance of the rebar from the middle surface of the shell, positive in the direction of the positive normal to the shell. This value is modified if the NODAL THICKNESS parameter is included with the **SHELL SECTION* option of the underlying shell element.
5. Angular orientation of rebar (in degrees) between the positive local 1-direction and the rebar. The optional ORIENTATION parameter given on the **SHELL SECTION* option should have no influence on the rebar angular orientation.

Repeat this data line as often as necessary. Each line defines a layer of rebar.

Data lines to define skew rebar in three-dimensional membrane elements:

First line:

1. Element number or name of the element set that contains these rebar.
2. Cross-sectional area of each rebar.
3. Spacing of rebar in the plane of the membrane. The default is 1.0.
4. Angular orientation of rebar (in degrees) between the positive local 1-direction and the rebar. The optional ORIENTATION parameter given on the **MEMBRANE SECTION* option should have no influence on the rebar angular orientation.

Repeat this data line as often as necessary. Each line defines a layer of rebar.

Data lines to define rebar in axisymmetric shell elements:

First line:

1. Element number or name of the element set that contains these rebar.
2. Cross-sectional area of each rebar.
3. Spacing of rebar in this rebar layer. The default is 1.0.
4. Position of the rebar in the shell section thickness direction. This value is given as the distance of the rebar from the middle surface of the shell, positive in the direction of the positive normal to the shell. This value is modified if the NODAL THICKNESS parameter is included with the **SHELL SECTION* option of the underlying shell element.
5. Angular orientation of rebar from the meridional plane in degrees (0° is meridional, 90° is circumferential). Positive rotation is about the positive normal to the shell.
6. Radial position at which the spacing of the rebar is measured. If this entry is nonzero, it is assumed that the rebar spacing varies linearly with radial position. If this entry is zero or blank, the rebar spacing does not vary with position. This entry has no meaning for circumferential rebar.

Repeat this data line as often as necessary. Each line defines a layer of rebar.

Data lines to define rebar in axisymmetric membrane elements:

First line:

1. Element number or name of the element set that contains these rebar.
2. Cross-sectional area of each rebar.
3. Spacing of rebar in this rebar layer. The default is 1.0.
4. Angular orientation of rebar from the meridional plane in degrees (0° is meridional, 90° is circumferential). Positive rotation is about the positive normal to the membrane.
5. Radial position at which the spacing of the rebar is measured. If this entry is nonzero, it is assumed that the rebar spacing varies linearly with radial position. If this entry is zero or blank, the rebar spacing does not vary with position. This entry has no meaning for circumferential rebar.

Repeat this data line as often as necessary. Each line defines a layer of rebar.

Data lines to define a layer of uniformly spaced rebar in continuum elements (SINGLE parameter omitted) when the layer is parallel to two isoparametric directions in the element's local (isoparametric) coordinate system (GEOMETRY=ISOPARAMETRIC):

First line:

1. Element number or name of the element set that contains these rebar.
2. Cross-sectional area of each rebar.
3. Spacing of rebar. The default is 1.0.
4. Orientation of rebar in degrees. See *Figure 3*.
5. Fractional distance from the edge given below, *f* (ratio of the distance between the edge and the rebar to the distance across the element).
6. Edge number from which the rebar are defined. See *Figure 4* or *Figure 7*.
7. Isoparametric direction (for three-dimensional elements only).
8. For axisymmetric elements only, the radial position at which the spacing of the rebar is measured. If this entry is nonzero, the rebar spacing is assumed to vary linearly with radial position. If this entry is zero or blank, the rebar spacing does not vary with position.

Repeat this data line as often as necessary. Each line defines a layer of rebar.

Data lines to define a layer of uniformly spaced rebar in continuum elements (SINGLE parameter omitted) when the layer is parallel to only one isoparametric direction in the element's local (isoparametric) coordinate system (GEOMETRY=SKEW):

First line:

1. Element number or name of the element set that contains these rebar.
2. Cross-sectional area of each rebar.
3. Spacing of rebar. The default is 1.0.
4. Orientation of rebar in degrees. See *Figure 3*.
5. For axisymmetric elements only, give the radial position at which the spacing of the rebar is measured. If this entry is nonzero, the rebar spacing is assumed to vary linearly with radial position. If this entry is zero or blank, the rebar spacing does not vary with position.
6. Isoparametric direction (for three-dimensional elements only).

Second line:

1. Fractional distance along edge 1, f_1 (see *Figure 5*).
2. Fractional distance along edge 2, f_2.
3. Fractional distance along edge 3, f_3.
4. Fractional distance along edge 4, f_4.

Only the two values corresponding to the two edges that the rebar intersects can be nonzero.

Repeat the first and second data lines as often as necessary. Each pair of lines defines a layer of rebar.

Data lines to define a single rebar in continuum elements (SINGLE parameter included):

First line:

1. Element number or name of the element set that contains the rebar.
2. Cross-sectional area of the rebar.
3. Fractional distance, f_1, locating the rebar's position in the element (see *Figure 6*).
4. Fractional distance, f_2, locating the rebar's position in the element.
5. Isoparametric direction (for three-dimensional elements only).

In three-dimensional cases the fractional distances f_1, and f_2 are given along the first two edges of the face identified in *Figure 7* for the isoparametric direction chosen.

Repeat this data line as often as necessary. Each line defines a single rebar.

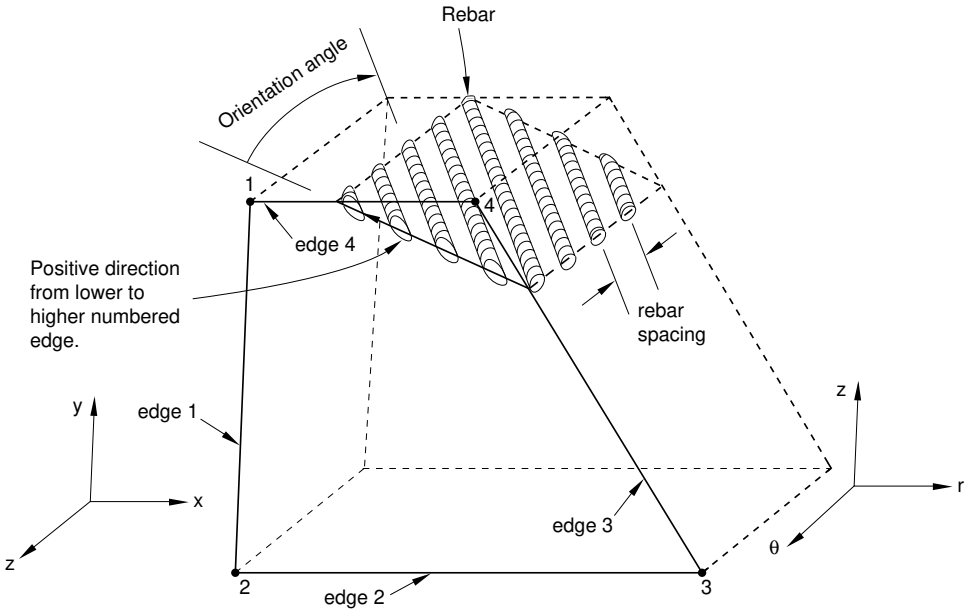


Figure 3: Orientation of rebar in plane and axisymmetric solid elements.

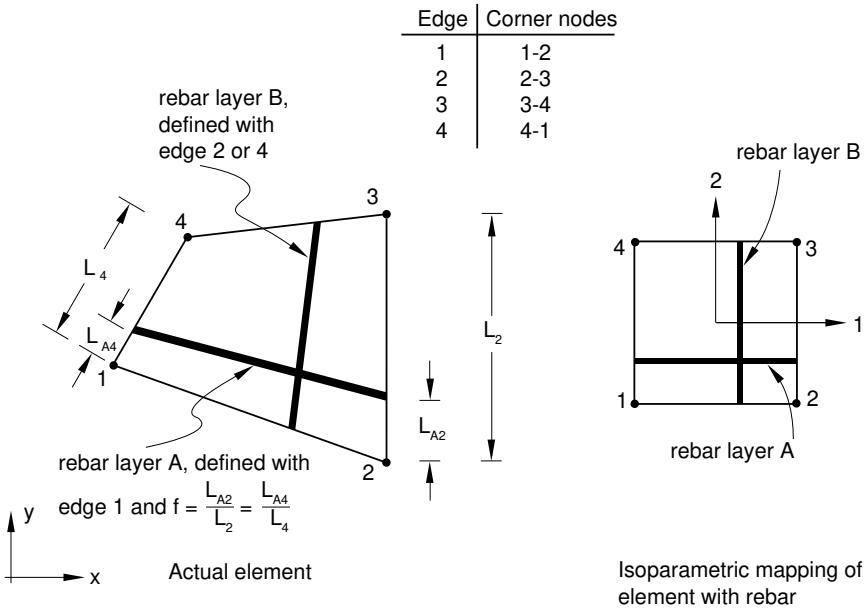


Figure 4: Rebar layer definition in solid elements with GEOMETRY=ISOPARAMETRIC.

Edge	Corner nodes
1	1-2
2	2-3
3	3-4
4	4-1

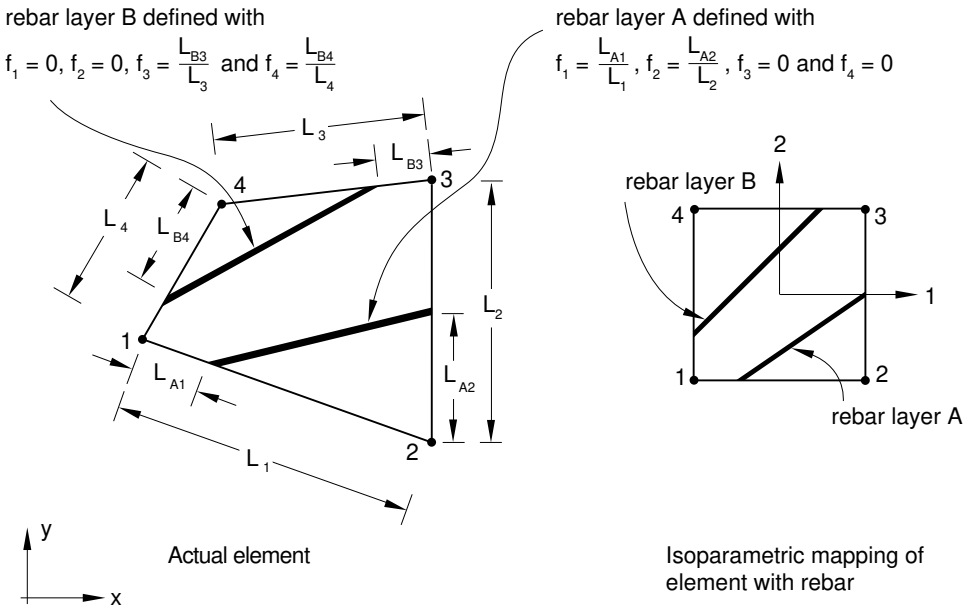


Figure 5: Rebar layer definition in solid elements with GEOMETRY=SKEW.

Edge	Corner nodes
1	1-2
2	2-3

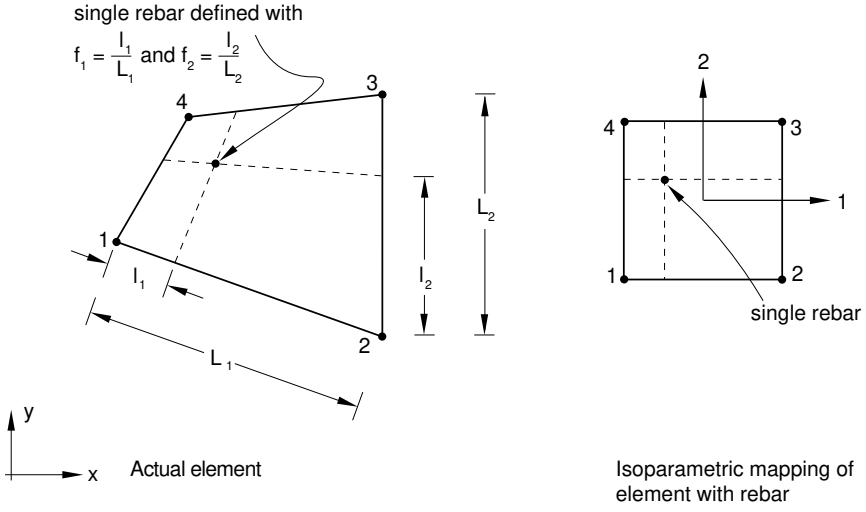
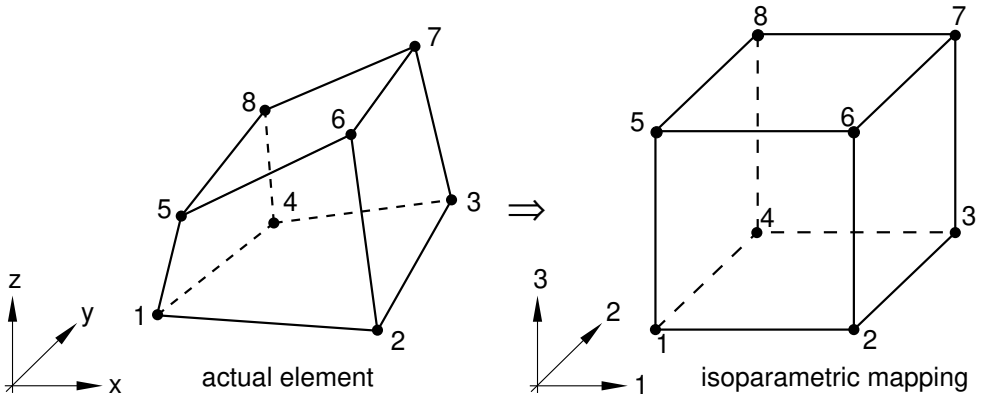


Figure 6: SINGLE rebar in a solid element.



Isoparametric direction: 1 (parallel to the 1-2 edge of the element and intersecting face 1-4-8-5)

Edge	Corner nodes
1	1-4
2	4-8
3	8-5
4	5-1

Isoparametric direction: 2 (parallel to the 1-4 edge of the element and intersecting face 1-5-6-2)

Edge	Corner nodes
1	1-5
2	5-6
3	6-2
4	2-1

Isoparametric direction: 3 (parallel to the 1-5 edge of the element and intersecting face 1-2-3-4)

Edge	Corner nodes
1	1-2
2	2-3
3	3-4
4	4-1

Figure 7: Isoparametric direction and edge definitions for three-dimensional elements.

*REBAR LAYER: Define layers of reinforcement in membrane, shell, surface, and continuum elements.

This option is used to define one or multiple rebar layers in membrane, shell, and surface elements. It must be used in conjunction with the *MEMBRANE SECTION, the *SHELL SECTION, or the *SURFACE SECTION option. Rebar layers in solid (continuum) elements can be defined by embedding a set of surface or membrane elements with rebar layers in a set of host continuum elements through the use of the *EMBEDDED ELEMENT option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: *Property module*; supported only for membrane and shell elements.

References:

- *Defining Reinforcement*
- *Embedded Elements*
- *EMBEDDED ELEMENT
- *MEMBRANE SECTION
- *SHELL SECTION
- *SURFACE SECTION

Optional parameters:

GEOMETRY

Use this parameter to specify the type of rebar geometry.

Set GEOMETRY=CONSTANT (default) if the rebar spacing in the element is constant. The spacing is given as a length measure on the data lines.

Set GEOMETRY=ANGULAR if the rebar spacing increases linearly as a function of the distance measured from the axis of revolution in a cylindrical coordinate system. The spacing is given as an angular spacing in degrees on the data lines. A cylindrical orientation system must be defined if this option is used with three-dimensional membrane, shell, or surface elements.

Set GEOMETRY=LIFT EQUATION if the rebar spacing and orientation within the element is determined by the tire “lift” equation. The rebar parameters are defined with respect to the uncured or “green” tire configuration, and the lift equation maps the rebar parameters to the cured tire configuration. The spacing is given as a length measure on the data lines. A cylindrical orientation system must be defined if this option is used with three-dimensional membrane, shell, or surface elements.

ORIENTATION

This parameter is meaningful only for rebar in general shell, membrane, and surface elements. Set this parameter equal to the name of an orientation definition that defines the angular orientation of the rebar on the data lines. If this parameter is omitted, the angular orientation of rebar on the data lines is specified relative to the default projected local surface directions. This parameter is not permitted with axisymmetric shell, axisymmetric membrane, and axisymmetric surface elements.

Data lines to define rebar layers:

First line:

1. A name that will be used to identify this rebar layer. This name can be used in defining rebar prestress and output requests. Each layer of rebar must be given a separate name in a particular element or element set.
2. Cross-sectional area of the rebar.
3. Spacing of rebar in the plane of the membrane, shell, or surface element. The value is given as a length measure unless the GEOMETRY=ANGULAR parameter is specified, in which case the value should be given in terms of angular spacing in degrees. Specify the spacing in the uncured geometry if GEOMETRY=LIFT EQUATION.
4. Position of the rebar in the shell section thickness direction. This value is given as the distance of the rebar from the middle surface of the shell, positive in the direction of the positive normal to the shell. This value is modified if the NODAL THICKNESS parameter is included with the **SHELL SECTION* option for the underlying shell element. This entry has no meaning for rebar in either membrane or surface elements.
5. Name of the material forming this rebar layer.
6. For three-dimensional shell, membrane, and surface elements, specify the angular orientation of rebar (in degrees) between the positive local 1-direction and the rebar. Specify the angle in the uncured geometry if GEOMETRY=LIFT EQUATION.

   The local 1-direction can be defined by using the **ORIENTATION* option and setting the ORIENTATION parameter equal to the orientation name. If the ORIENTATION parameter is omitted, the local 1-direction is defined by the default projected local surface directions. The optional ORIENTATION parameter given on the **MEMBRANE SECTION* and the **SHELL SECTION* options has no influence on the rebar angular orientation.

   For axisymmetric shell, axisymmetric membrane, and axisymmetric surface elements, specify the angular orientation of rebar from the meridional plane in degrees (0° is meridional, 90° is circumferential). Positive rotation is about the positive normal to the shell, membrane, or surface elements.
7. The isoparametric direction from which the rebar angle output will be measured. The default is 1.
8. Extension ratio, e, for rebar defined with GEOMETRY=LIFT EQUATION. In a tire e represents the pre-strain that occurs during the curing process; e =1 means a 100% extension. This entry has no meaning for rebar defined using GEOMETRY=CONSTANT or GEOMETRY=ANGULAR.
9. Radius, r_0, of the rebar defined with GEOMETRY=LIFT EQUATION. The value is the position of the rebar in the uncured geometry, measured with respect to the axis of rotation in a cylindrical coordinate system. This entry has no meaning for rebar defined using GEOMETRY=CONSTANT or GEOMETRY=ANGULAR.

Repeat the data line as often as necessary. Each data line defines a layer of rebar.

*REBAR LINE: Define rebar reinforcement in beam elements.

This option is used to define one or multiple rebar lines in beam elements. It must be used in conjunction with the *BEAM SECTION* option, and it cannot be used in conjunction with the *BEAM GENERAL SECTION* option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- **BEAM SECTION*
- *Defining Reinforcement*

Data lines to define rebar lines:

First line:

1. Name to be used to identify this rebar line. This name can be used in defining rebar prestress and output requests. Each rebar line must be given a separate name in a particular element or element set.
2. Cross-sectional area of the rebar.
3. Position of the rebar in the local X_1-direction.
4. Position of the rebar in the local X_2-direction.
5. Name of the material forming this rebar line.

Repeat the data line as often as necessary. Each data line defines a rebar line.

*REDUCED BASIS GENERATE: Reduced basis generation analysis.

This option is used to indicate that the step should be analyzed as a reduced basis generation step.

Products: Abaqus/Standard

Type: History data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

References:

• *Reduced Basis Generation*

Optional parameters:

NAME

Set this parameter equal to the name of the output SIM document to which the reduced basis data are written. See *Input Syntax Rules* for the syntax of such names.

The default name is *jobname*_RB*n*, where *n* is the current step number of the job.

Data line to define the level of reduction:

First (and only) line:

1. Level of reduction of the nonlinear deformation space given by a dimensionless value between 0 (the number of basis vectors and the number of nonlinear deformation fields from preceding operator output requests are the same) and 1 (no basis vectors are kept). The default is 10^{-4}.
2. Level of reduction of the auxiliary linear deformation space given by a dimensionless value between 0 (the number of residual basis vectors and the number of natural frequency modes from preceding analysis steps are the same) and 1 (no residual basis vectors are kept). If no value is specified, the same level of reduction is used as for the nonlinear deformation space.

*REFLECTION: Define reflection symmetries for a cavity radiation heat transfer analysis.

This option is used to define a cavity symmetry by reflection through a line or a plane. It can be used only following the **RADIATION SYMMETRY* option.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *Cavity Radiation in Abaqus/Standard*
- **RADIATION SYMMETRY*

Required parameters:

TYPE

Set TYPE=LINE to create a cavity composed of the cavity surface defined in the model and its reflection through a line. See *Figure 1*. This option can be used only for two-dimensional cases.

Set TYPE=PLANE to create a cavity composed of the cavity surface defined in the model and its reflection through a plane. See *Figure 2*. This option can be used only for three-dimensional cases.

Set TYPE=ZCONST to create a cavity composed by the cavity surface defined in the model and its reflection through a line of constant z-coordinate. See *Figure 3*. This option can be used only for axisymmetric cases.

Data line to define reflection of a two-dimensional cavity (TYPE=LINE):

First (and only) line:

1. X-coordinate of point a (see *Figure 1*).
2. Y-coordinate of point a.
3. X-coordinate of point b.
4. Y-coordinate of point b.

Data lines to define reflection of a three-dimensional cavity (TYPE=PLANE):

First line:

1. X-coordinate of point a (see *Figure 2*).
2. Y-coordinate of point a.
3. Z-coordinate of point a.
4. X-coordinate of point b.
5. Y-coordinate of point b.
6. Z-coordinate of point b.

Second line:

1. X-coordinate of point c.
2. Y-coordinate of point c.
3. Z-coordinate of point c.

Data line to define reflection of an axisymmetric cavity (TYPE=ZCONST):

First (and only) line:

1. z-coordinate of symmetry line (see *Figure 3*).

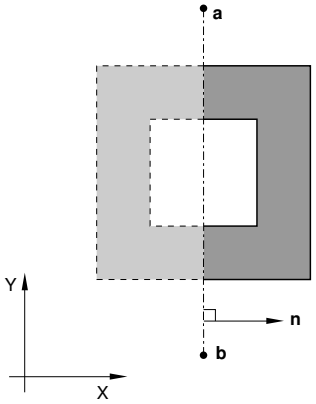


Figure 1: *REFLECTION, TYPE=LINE option.

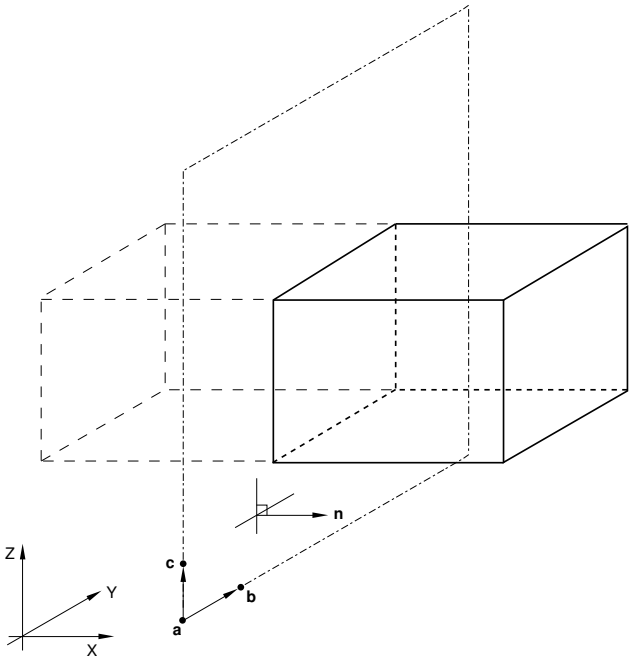


Figure 2: *REFLECTION, TYPE=PLANE option.

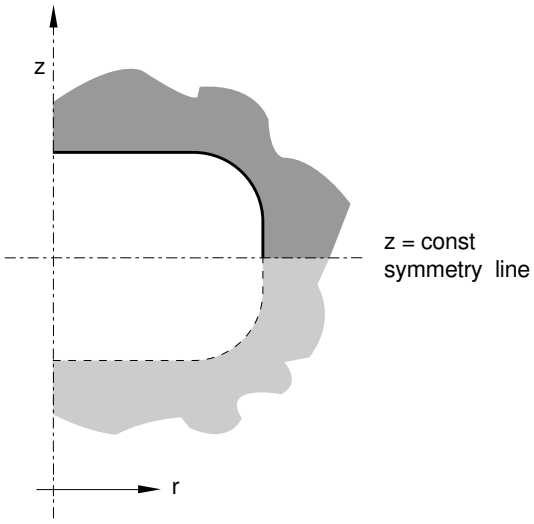


Figure 3: *REFLECTION, TYPE=ZCONST option.

*RELEASE: Release rotational degrees of freedom at one or both ends of a beam element.

This option is used to release a rotational degree of freedom or a combination of rotational degrees of freedom at one or both ends of a beam element.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- *Kinematic Coupling Constraints*
- *Element End Release*

Optional parameters:

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

Data lines to prescribe the released degrees of freedom:

First line:

1. Element number or element set label.
2. Element end identifier (S1 or S2).
3. Release combination code at the end of the element (M1, M2, T, M1-M2, M1-T, M2-T, ALLM). See *Element End Release* for a definition of the release combination codes in Abaqus/Standard. This field can be left blank if none of the rotational degrees of freedom are released at this end of the element.

Repeat this data line as often as necessary to specify the rotational degrees of freedom to be released for different elements and element ends.

*RESPONSE SPECTRUM: Calculate the response based on user-supplied response spectra.

This option is used to calculate estimates of peak values of nodal and element responses based on user-supplied response spectra (defined using the **SPECTRUM* option) and on the natural modes of the system.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Response Spectrum Analysis*
- **SPECTRUM*

Optional parameters:

COMP

Set COMP=ALGEBRAIC to sum the directional excitation components algebraically. If this parameter is used, the directional excitation components are summed first, followed by the modal summation.

Set COMP=SRSS (default) to use the square root of the sum of the squares. If this parameter is used, the modal summation is performed first, followed by the summation of the directional excitation components.

Set COMP=R40 to use the 40% rule as recommended by the ASCE 4–98 Guide that assumes that for the maximum response from one component, the responses from the other two components are 40% of their maximum value.

Set COMP=R30 to use the 30% rule. This rule assumes that for the maximum response from one component, the responses from the other two components are 30% of their maximum value. It follows the expressions for the 40% rule as described in the ASCE 4–98 Guide.

MISSING MASS METHOD

Include this parameter to achieve greater response accuracy by augmenting the missing inertia in the dynamic response. Quasistatic residual modes in the direction of applied spectra are substituted for the truncated higher frequency modes that belong to the in-phase (rigid) response.

RIGID RESPONSE

Set RIGID RESPONSE=GUPTA to use the Gupta method for modal summation of periodic and rigid regions in the response spectrum. If this parameter is omitted, no rigid response is included.

Set RIGID RESPONSE=LINDLEY-YOW to use the Lindley-Yow method for modal summation of periodic and rigid regions in the response spectrum. If this parameter is omitted, no rigid response is included.

SUM

Set SUM=ABS (default) to sum the absolute values of the responses in each natural mode.

Set SUM=CQC to use the complete quadratic combination method to sum the responses in each natural mode.

Set SUM=DSC to use the double sum combination. This method is the first attempt to evaluate modal correlation based on random vibration theory. It utilizes the time duration t_D of strong earthquake motion.

Set SUM=GRP to use the grouping method as described in USNRC Regulatory Guide 1.92, February 1976.

Set SUM=NRL to use the Naval Research Laboratory method.

Set SUM=SRSS to use the square root of the sum of squares summation.

Set SUM=TENP to use the Ten Percent Method.

Data lines for a response spectrum analysis:

First line:

1. Name of the response spectrum to be used.
2. *X*-direction cosine of this direction.
3. *Y*-direction cosine of this direction.
4. *Z*-direction cosine of this direction.
5. Factor multiplying the magnitudes in the response spectrum. Default is 1.0.
6. Time duration of the dynamic event, from which this spectrum was created. This value is required for the DSC modal summation rule.
7. Frequency of the zero period acceleration or cut-off-frequency value in cycles per time. Default is 33 Hz. This value is used for the Lindley-Yow rigid response method. If the Gupta rigid response method is used, this is the *F1* frequency value in cycles/time.
8. For the Gupta rigid response method, enter the *F2* frequency value in cycles/time.

Second line (optional):

1. Name of the response spectrum to be used in the second direction.
2. *X*-direction cosine of this direction. This direction must be at a right angle to the direction defined above.
3. *Y*-direction cosine of this direction. This direction must be at a right angle to the direction defined above.
4. *Z*-direction cosine of this direction. This direction must be at a right angle to the direction defined above.
5. Factor multiplying the magnitudes in the response spectrum. Default is 1.0.
6. Time duration of the dynamic event, from which this spectrum was created. This value is required for the DSC modal summation rule.
7. Frequency of the zero period acceleration or cut-off-frequency value in cycles per time. Default is 33 Hz. This value is used for the Lindley-Yow rigid response method. If the Gupta rigid response method is used, this is the *F1* frequency value in cycles/time.
8. For the Gupta rigid response method, enter the *F2* frequency value in cycles/time.

Third line (optional):

1. Name of the response spectrum to be used in the third direction.
2. *X*-direction cosine of this direction. This direction must be at right angles to the two directions defined above.
3. *Y*-direction cosine of this direction. This direction must be at right angles to the two directions defined above.
4. *Z*-direction cosine of this direction. This direction must be at right angles to the two directions defined above.
5. Factor multiplying the magnitudes in the response spectrum. Default is 1.0.
6. Time duration of the dynamic event from which this spectrum was created. This value is required for the DSC modal summation rule.

7. Frequency of the zero period acceleration or cut-off-frequency value in cycles per time. Default is 33 Hz. This value is used for the Lindley-Yow rigid response method. If the Gupta rigid response method is used, this is the *F1* frequency value in cycles/time.
8. For the Gupta rigid response method, enter the *F2* frequency value in cycles/time.

*RESTART: Save and reuse data and analysis results.

 **Warning:** This option can create a very large amount of data.

This option is used to control the writing and reading of restart data.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Model, Step

Abaqus/CAE: *Step module* for saving restart data; *Job module* for performing a restart analysis

References:

• *Restarting an Analysis*

Controlling the writing and reading of restart data in an Abaqus/Standard analysis

At least one of the following parameters is required:

READ

Include this parameter to specify that this analysis is a restart of a previous analysis. The basic model definition data (elements, materials, nodes) cannot be changed at such a restart. However, element sets, node sets, and **AMPLITUDE* tables can be added, and history data subsequent to that part of the history already analyzed can be changed.

WRITE

Include this parameter to specify that restart data are to be written during the analysis.

Optional parameters if the READ parameter is used:

CYCLE

If the new analysis is restarted from a previous low-cycle fatigue analysis, set this parameter equal to the cycle number within the specified step of the low-cycle fatigue analysis after which the analysis will resume. Since the new analysis can be restarted only from the end of a loading cycle in the previous low-cycle fatigue analysis, the INC parameter is irrelevant and is ignored if the CYCLE parameter is specified.

If this parameter is omitted, the restart will begin at the end of the last cycle for the specified step.

END STEP

This parameter specifies that the user wishes to terminate the current step in the analysis from which the restart is being made.

This parameter is useful when the user wishes to redefine the loading history, output options, or tolerance controls, etc. If this parameter is included, the data must contain further step definitions to define how the analysis will continue.

If this parameter is omitted, Abaqus/Standard will continue the analysis to complete the current step as it is defined in the run from which the restart is being made.

INC

Set this parameter equal to the increment number within the step specified by the STEP parameter, after which the analysis will resume.

If this parameter is omitted, the restart will begin at the end of the step specified on the STEP parameter.

ITERATION

If the new analysis is restarted from a previous direct cyclic analysis, set this parameter equal to the iteration number within the specified step of the direct cyclic analysis after which the analysis will resume. Since the new analysis can be restarted only from the end of a loading cycle in the previous direct cyclic analysis, the INC parameter is irrelevant and is ignored if the ITERATION parameter is specified.

If this parameter is omitted, the restart will begin at the end of the last iteration for the specified step.

STEP

Set this parameter equal to the step number at which the restart will be made.

If this parameter is omitted, the analysis will restart at the last available step found.

Optional parameters if the WRITE parameter is used:

FREQUENCY

This parameter specifies the increments at which restart information will be written. For example, FREQUENCY=2 will write restart information at increments 2, 4, 6, etc.

For a direct cyclic analysis this parameter specifies the iteration numbers at which restart information will be written; restart information will be written only at the end of an iteration.

For a low-cycle fatigue analysis this parameter specifies the cycle numbers at which restart information will be written; restart information will be written only at the end of a cycle.

When restart is requested in an analysis, restart information is always written at the end of a step. Set FREQUENCY=0 to discontinue the writing of restart data.

The FREQUENCY and NUMBER INTERVAL parameters are mutually exclusive. The default is FREQUENCY=1.

NUMBER INTERVAL

Set this parameter equal to the number of intervals during the step at which the **RESTART* data are to be written. The value of this parameter must be a positive integer.

Specifying the NUMBER INTERVAL parameter is the recommended method of synchronizing restart information when performing a co-simulation. In this case Abaqus/Standard will write restart information at the co-simulation target time immediately after the time dictated by the NUMBER INTERVAL parameter.

The FREQUENCY and NUMBER INTERVAL parameters are mutually exclusive. The default is FREQUENCY=1.

TIME MARKS

Set TIME MARKS=YES (default) to write results at the exact times dictated by the NUMBER INTERVAL parameter.

Set TIME MARKS=NO to write the restart at the increment ending immediately after the time dictated by the NUMBER INTERVAL parameter.

When used with the NUMBER INTERVAL parameter in a co-simulation, the TIME MARKS parameter is always set to NO to write restart information at the co-simulation target time after the time dictated by the NUMBER INTERVAL parameter.

OVERLAY

This parameter specifies that only one increment should be retained per step, thus minimizing the storage space needed.

When the OVERLAY parameter is included, each increment written overlays the previous increment, if any, written for the same step. If this parameter is omitted, data are retained for every increment. In either case the last increment of every step is retained.

There are no data lines associated with this option.

Controlling the writing and reading of restart data in an Abaqus/Explicit analysis

At least one of the following parameters is required:

READ

Include this parameter to specify that this analysis is a restart of a previous analysis. The basic model definition data (elements, materials, nodes) cannot be changed at such a restart. However, element sets, node sets, and **AMPLITUDE* tables can be added, and history data subsequent to that part of the history already analyzed can be changed.

WRITE

Include this parameter to specify that restart data are to be written during the analysis.

Required parameter if the READ parameter is used:

STEP

Set this parameter equal to the step number at which the restart will be made. If the END STEP and INTERVAL parameters are omitted, the analysis will resume after the last interval available in this step.

Optional parameters if the READ parameter is used:

END STEP

Include this parameter to specify that the current step should be terminated at this point. The INTERVAL parameter is required when this parameter is used.

This parameter is useful when the user wishes to redefine the loading history, contact surface definitions, etc.

If this parameter is omitted, Abaqus/Explicit will continue the analysis from the last available restart interval to complete the current step as it is defined in the run from which the restart is being made.

INTERVAL

Set this parameter equal to the interval number within the step specified by the STEP parameter after which the analysis will resume. The END STEP parameter is required when this parameter is used.

Optional parameters if the WRITE parameter is used:

NUMBER INTERVAL

Set this parameter equal to the number of intervals during the step at which the **RESTART* data are to be written. The value of this parameter must be a positive integer or zero. A value of zero turns off all restart frame output, which can reduce disk usage if there is no continuation after the current analysis. The default is NUMBER INTERVAL=1.

Abaqus/Explicit will divide the step into the specified number of time intervals, n, and write the results at the beginning of the step and at the end of each interval, for a total of n+1 points for the step, with the last point matching the end of the step. For example, if NUMBER INTERVAL=10, Abaqus/Explicit will write 11 restart states, consisting of the values at the beginning of the step and the values at 10 evenly spaced intervals throughout the step.

For a co-simulation Abaqus/Explicit will write restart information at the co-simulation target time immediately after the time dictated by the NUMBER INTERVAL parameter.

TIME MARKS

Set TIME MARKS=NO (default) to write the restart at the increment ending immediately after the time dictated by the NUMBER INTERVAL parameter.

Set TIME MARKS=YES to write results at the exact times dictated by the NUMBER INTERVAL parameter. TIME MARKS=YES cannot be used when either the FIXED TIME INCREMENTATION or DIRECT USER CONTROL parameter is included on the **DYNAMIC* option (*Explicit Dynamic Analysis*).

For a co-simulation the TIME MARKS parameter is always set to NO to write restart information at the co-simulation target time after the time dictated by the NUMBER INTERVAL parameter.

OVERLAY

This parameter specifies that only one increment should be retained per step, thus minimizing the storage space needed.

When the OVERLAY parameter is included, each increment written overlays the previous increment, if any, written for the same step. If this parameter is omitted, data are retained for every increment. In either case the last increment of every step is retained.

SINGLE

This parameter specifies that only single precision restart data are to be written when the double precision executable is run.

When the single precision executable is run, this parameter is ignored and only single precision restart data will be written. If this parameter is omitted, double precision restart data will be written when the double precision executable is run.

There are no data lines associated with this option.

*RETAINED NODAL DOFS: Specify the degrees of freedom to retain as external to a substructure.

This option is used to list degrees of freedom to retain as external degrees of freedom on the substructure. It can be used only in a **SUBSTRUCTURE GENERATE* analysis to define a conventional substructure or in a **STEADY STATE DYNAMICS*, DIRECT analysis to define a frequency-based substructure.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- *Generating Substructures*
- *Generating Frequency-Based Substructures*

Optional parameters:

SORTED

Set SORTED=NO to prevent the retained nodes from being sorted. The ordering of the nodes when using a substructure is then the same as the ordering used when specifying the retained nodes. The default is SORTED=YES, the retained nodes are sorted into ascending numerical order.

Data lines to define the retained degrees of freedom:

First line:

1. Node number or node set label.
2. First degree of freedom to be retained.
3. Last degree of freedom to be retained.

If only the node number or node set label is given, all degrees of freedom are retained.

Repeat this data line as often as necessary to list all degrees of freedom to be retained.

*RIGID BODY: Define a set of elements as a rigid body and define rigid element properties.

This option is used to bind a set of elements and/or a set of nodes and/or an analytical surface into a rigid body and assign a reference node to the rigid body, which can optionally be declared as an isothermal rigid body for fully coupled thermal-stress analysis. It is also used to specify density, thickness, and offset for rigid elements that are part of a rigid body in an Abaqus/Explicit analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: *Part module* and *Interaction module*

References:

- *Rigid Elements*
- *Analytical Rigid Surface Definition*
- *Rigid Body Definition*

Required parameters:

REF NODE

Set this parameter equal to either the node number of the rigid body reference node or the name of a node set containing the rigid body reference node. If the name of a node set is chosen, the node set must contain exactly one node.

At least one of the following parameters is required:

ANALYTICAL SURFACE

Set this parameter equal to the name of the analytical surface to be assigned to the rigid body.

ELSET

Set this parameter equal to the name of the element set containing the elements to be assigned to the rigid body. An element cannot appear in more than one rigid body.

PIN NSET

Set this parameter equal to the name of a node set containing pin-type nodes to be assigned to the rigid body. This parameter can be used to add nodes to a rigid body or to redefine node types of nodes on elements included in the rigid body by the ELSET parameter. Pin-type nodes have only their translational degrees of freedom associated with the rigid body. A node cannot appear in more than one rigid body definition.

TIE NSET

Set this parameter equal to the name of a node set containing tie-type nodes to be assigned to the rigid body. This parameter can be used to add nodes to a rigid body or to redefine node types of nodes on elements included in the rigid body by the ELSET parameter. Tie-type nodes have both their translational and rotational degrees of freedom associated with the rigid body. A node cannot appear in more than one rigid body definition.

Optional parameters:

ALPHA

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the thermal expansion coefficient.

ISOTHERMAL

This parameter is used only for fully coupled thermal-stress analysis.

Set this parameter equal to YES to specify an isothermal rigid body. The default is ISOTHERMAL=NO.

POSITION

Set POSITION=INPUT (default) if the location of the reference node is to be defined by the user.

Set POSITION=CENTER OF MASS if the reference node is to be located at the center of mass of the rigid body.

Optional parameters (use only when the element set specified contains rigid elements):

DENSITY

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the density of all of the rigid elements in the element set.

NODAL THICKNESS

This parameter applies only to Abaqus/Explicit analyses.

Include this parameter to indicate that the thickness of the rigid elements should not be read from the data line but should be interpolated from the thickness specified at the nodes with the **NODAL THICKNESS* option.

OFFSET

This parameter applies only to Abaqus/Explicit analyses.

Include this parameter to define the distance (as a fraction of the rigid element's thickness) from the element's midsurface to the reference surface containing the element's nodes. Since no element level calculations are performed for rigid elements, a specified offset affects only the handling of contact pairs with rigid surfaces formed by rigid elements.

This parameter accepts positive or negative values or the labels SPOS or SNEG. The positive values of the offset are in the direction of the element normal. When OFFSET=0.5 (or SPOS), the top surface of the rigid element is the reference surface. When OFFSET=−0.5 (or SNEG), the bottom surface of the rigid element is the reference surface. The default is OFFSET=0, which indicates that the middle surface of the rigid element is the reference surface.

There are no data lines associated with this option in an Abaqus/Standard analysis

Data line for R2D2, RB2D2, and RB3D2 elements in an Abaqus/Explicit analysis:

First (and only) line:

1. Cross-sectional area of the element. The default is 0.

Data line for RAX2, R3D3, and R3D4 elements in an Abaqus/Explicit analysis:

First (and only) line:

1. Thickness of the element. The default is 0.

*RIGID SURFACE: Define an analytical rigid surface.

This option must be used when defining the seabed for three-dimensional drag chain elements in Abaqus/Standard analyses. For all other cases the preferred options for defining analytical rigid surfaces are the **SURFACE* and the **RIGID BODY* options.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Part module

References:

- *About Surfaces*
- *Analytical Rigid Surface Definition*
- *Drag Chains*
- *RSURFU*
- **SURFACE*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the IRS-type elements or the three-dimensional drag chain elements that may interact with the rigid surface being defined.

The ELSET and NAME parameters are mutually exclusive.

NAME

Set this parameter equal to a label that will be used to refer to the rigid surface being created. This surface name is used to define contact interaction with another surface through the **CONTACT PAIR* option.

The ELSET and NAME parameters are mutually exclusive.

REF NODE

Set this parameter equal to either the node number of the rigid body reference node or the name of a node set containing the rigid body reference node. If the name of a node set is chosen, the node set must contain exactly one node.

This parameter is relevant only when the NAME parameter is used.

TYPE

Set TYPE=SEGMENTS to create a two-dimensional rigid surface in the (X, Y) plane for planar models or in the (r, z) plane for axisymmetric models by defining connected line segments.

Set TYPE=CYLINDER to define a three-dimensional rigid surface by providing connected line segments and then sweeping them along a specified generator vector.

Set TYPE=REVOLUTION to define a three-dimensional rigid surface by providing connected line segments, which are given in an (r, z) plane and are rotated about an axis.

Set TYPE=USER to define a rigid surface via user subroutine *RSURFU*.

Optional parameters:

FILLET RADIUS

This parameter can be used with TYPE=SEGMENTS, TYPE=CYLINDER, or TYPE=REVOLUTION to define a radius of curvature to smooth discontinuities between adjoining straight-line segments, adjoining circular-arc segments, and adjoining straight-line and circular-arc segments. It has no effect on rigid surfaces defined with TYPE=USER.

No data lines are needed for TYPE=USER

Data lines to define surfaces created with TYPE=SEGMENTS:

First line:

1. The "word" START.
2. Global *X*-coordinate or *r*-coordinate of the starting point of the line segments.
3. Global *Y*-coordinate or *z*-coordinate of the starting point of the line segments.

Data lines to define surfaces created with TYPE=CYLINDER:

First line:

1. Global *X*-coordinate of point *a*, the origin of the local (x, y) system (see *Figure 1*).
2. Global *Y*-coordinate of point *a*, the origin of the local (x, y) system.
3. Global *Z*-coordinate of point *a*, the origin of the local (x, y) system.
4. Global *X*-coordinate of point *b* on the local *x*-axis.
5. Global *Y*-coordinate of point *b* on the local *x*-axis.
6. Global *Z*-coordinate of point *b* on the local *x*-axis.

Second line:

1. Global *X*-coordinate of point *c* on the local cylinder generator vector.
2. Global *Y*-coordinate of point *c* on the local cylinder generator vector.
3. Global *Z*-coordinate of point *c* on the local cylinder generator vector.

Third line:

1. The "word" START.
2. Local *x*-coordinate of the starting point of the line segments.
3. Local *y*-coordinate of the starting point of the line segments.

Data lines to define surfaces created with TYPE=REVOLUTION:

First line:

1. Global *X*-coordinate of point *a*, the origin of the local (r, z) system (see *Figure 2*).
2. Global *Y*-coordinate of point *a*, the origin of the local (r, z) system.
3. Global *Z*-coordinate of point *a*, the origin of the local (r, z) system.
4. Global *X*-coordinate of point *b* on the symmetry axis (the local *z*-axis).
5. Global *Y*-coordinate of point *b* on the symmetry axis (the local *z*-axis).
6. Global *Z*-coordinate of point *b* on the symmetry axis (the local *z*-axis).

Second line:

1. The "word" START.
2. Local *r*-coordinate of the starting point of the line segments.
3. Local *z*-coordinate of the starting point of the line segments.

Data lines that define the line segments that form the rigid surface for TYPE=SEGMENTS, TYPE=CYLINDER, and TYPE=REVOLUTION:

Data line to define a straight line segment:

1. The "word" LINE.
2. x-coordinate of the endpoint of the line.
3. y-coordinate of the endpoint of the line.

Data line to define a circular arc segment (the arc must be less than 180°):

1. The "word" CIRCL.
2. x-coordinate of the endpoint of the circular arc.
3. y-coordinate of the endpoint of the circular arc.
4. x-coordinate of the center (origin) of the circular arc.
5. y-coordinate of the center (origin) of the circular arc.

Data line to define a parabolic arc segment:

1. The "word" PARAB.
2. x-coordinate of the middle point along the parabolic arc.
3. y-coordinate of the middle point along the parabolic arc.
4. x-coordinate of the endpoint of the parabolic arc.
5. y-coordinate of the endpoint of the parabolic arc.

For rigid surfaces created with TYPE=SEGMENTS, the x- and y-coordinates are the global X- and Y-coordinates or r- and z-coordinates. For rigid surfaces created with TYPE=CYLINDER, the x- and y-coordinates are the local x- and y-coordinates. For rigid surfaces created with TYPE=REVOLUTION, the x- and y-coordinates are the local r- and z-coordinates.

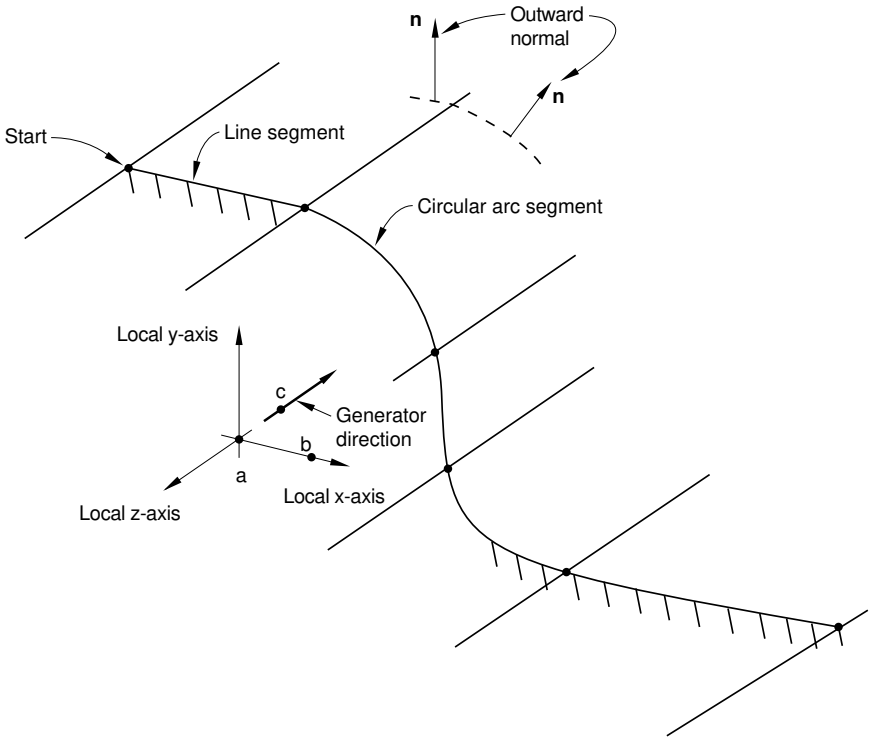


Figure 1: *RIGID SURFACE, TYPE=CYLINDER.

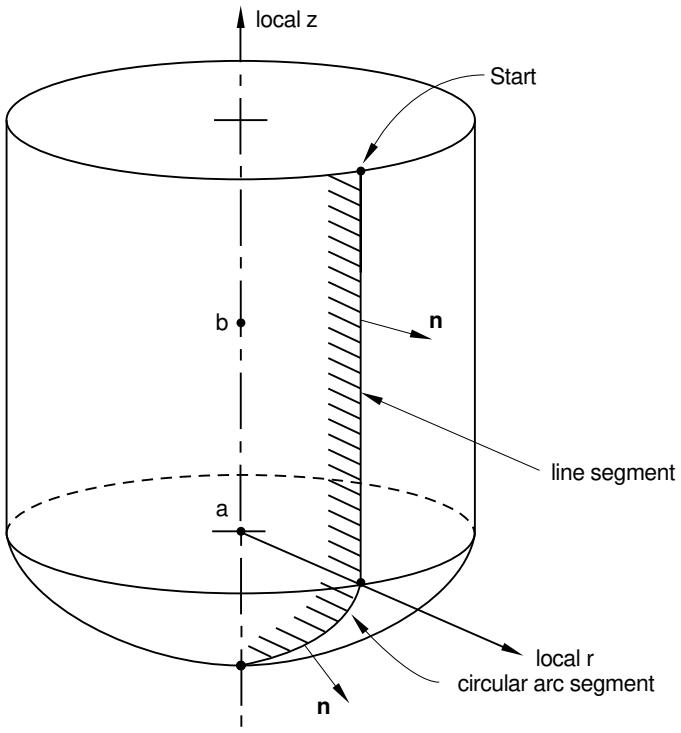


Figure 2: *RIGID SURFACE, TYPE=REVOLUTION.

*ROTARY INERTIA: Define rigid body rotary inertia.

This option is used to define rigid body rotary inertia values associated with ROTARYI elements.

It is also used to define mass proportional damping (for direct-integration dynamic analysis and explicit dynamic analysis) and composite damping (for modal dynamic analysis) associated with ROTARYI elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: *Property module* and *Interaction module*.

References:

• *Rotary Inertia*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the ROTARYI elements for which the value is being given.

Optional parameters:

ALPHA

Set this parameter equal to the α_R factor to create inertia proportional damping for the ROTARYI elements when used in direct-integration dynamics or explicit dynamics. This value is ignored in modal dynamics. The default is 0.0.

COMPOSITE

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the fraction of critical damping to be used with the ROTARYI elements when calculating composite damping factors for the modes when used in modal dynamics. The default is 0.0.

This value is ignored in direct-integration dynamics. It is also ignored in mode-based analyses that are based on the SIM architecture, where the **COMPOSITE MODAL DAMPING* option should be used instead.

ORIENTATION

Set this parameter equal to the name of an **ORIENTATION* option (*Orientations*) that is being used to define the directions of the local axes for which the rotary inertia values are being given. If the ORIENTATION parameter is omitted, it is assumed that the components of the inertia tensor are being given with respect to the global axes; i.e., the global and local inertia axes coincide.

In large-displacement analysis (an Abaqus/Explicit analysis or when the NLGEOM parameter is included on the **STEP* option in an Abaqus/Standard analysis), the local axes of inertia rotate with the rotation of the node to which the ROTARYI element is attached.

Data line to define the rotary inertia:

First (and only) line:

1. Rotary inertia about the local 1-axis, I_{11}.
2. Rotary inertia about the local 2-axis, I_{22}.
3. Rotary inertia about the local 3-axis, I_{33}.
4. Product of inertia, I_{12}.
5. Product of inertia, I_{13}.
6. Product of inertia, I_{23}.

The rotary inertia should be given in units of ML^2. Abaqus does not use any specific physical units, so the user's choice must be consistent.

S

This section describes each Abaqus input option that begins with S, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **SCALE MASS*
- **SCALE STIFFNESS*
- **SCALE STRESS DESIGN*
- **SCALE THERMAL CONDUCTIVITY*
- **SECTION CONTROLS*
- **SECTION CURVATURE*
- **SECTION FILE*
- **SECTION INERTIA*
- **SECTION ORIGIN*
- **SECTION POINTS*
- **SECTION PRINT*
- **SECTION STIFFNESS*
- **SECTION TANGENT*
- **SELECT CYCLIC SYMMETRY MODES*
- **SELECT EIGENMODES*
- **SFILM*
- **SFLOW*
- **SHEAR CENTER*
- **SHEAR FAILURE*
- **SHEAR RETENTION*
- **SHEAR TEST DATA*
- **SHELL GENERAL SECTION*
- **SHELL SECTION*
- **SHELL TO SOLID COUPLING*
- **SIMPEDANCE*
- **SIMPLE SHEAR TEST DATA*
- **SLIDE LINE*
- **SLOAD*
- **SOFT ROCK HARDENING*
- **SOFT ROCK PLASTICITY*
- **SOFTENING REGULARIZATION*
- **SOILS*
- **SOLID ELECTROLYTE THEORY*
- **SOLID SECTION*
- **SOLUBILITY*
- **SOLUTION TECHNIQUE*

- **SOLVER CONTROLS*
- **SORPTION*
- **SPECIFIC HEAT*
- **SPECTRUM*
- **SPH SURFACE BEHAVIOR*
- **SPRING*
- **SRADIATE*
- **STATIC*
- **STEADY STATE CRITERIA*
- **STEADY STATE DETECTION*
- **STEADY STATE DYNAMICS*
- **STEADY STATE TRANSPORT*
- **STEP*
- **STEP CONTROL*
- **STEP CYCLING*
- **STEP CYCLING CONTROL*
- **SUBCYCLING*
- **SUBMODEL*
- **SUBMODEL CONDITIONS*
- **SUBMODEL CUT*
- **SUBSTRUCTURE CHANGE*
- **SUBSTRUCTURE DAMPING*
- **SUBSTRUCTURE DAMPING CONTROLS*
- **SUBSTRUCTURE GENERATE*
- **SUBSTRUCTURE LOAD CASE*
- **SUBSTRUCTURE MATRIX OUTPUT*
- **SUBSTRUCTURE MODAL DAMPING*
- **SUBSTRUCTURE OUTPUT*
- **SUBSTRUCTURE PATH*
- **SUBSTRUCTURE PROPERTY*
- **SUPERELASTIC*
- **SUPERELASTIC HARDENING*
- **SUPERELASTIC HARDENING MODIFICATIONS*
- **SURFACE*
- **SURFACE BEHAVIOR*
- **SURFACE FLAW*
- **SURFACE INTERACTION*
- **SURFACE PROPERTY*
- **SURFACE PROPERTY ASSIGNMENT*
- **SURFACE SECTION*
- **SURFACE SMOOTHING*
- **SURFACE TENSION*
- **SURFACE TENSION CONTACT ANGLE*
- **SWELLING*
- **SYMMETRIC MODEL GENERATION*
- **SYMMETRIC RESULTS TRANSFER*

- *SYSTEM

*SCALE MASS: Specify scale factors and, for adjoint design sensitivity analysis, quantities that represent the derivatives of the scale factors divided by the scale factors for material mass.

This option is used to specify scale factors using a distribution for material mass. In addition, in an adjoint design sensitivity analysis, you can specify quantities that represent the derivatives of the scale factors divided by the scale factors. In the latter case, the scale factors and their derivatives are used in the computation of design responses and topology sensitivities.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

• *Adjoint Design Sensitivity Analysis*

There are no parameters associated with this option.

Data lines:

First (and only) line:

1. Distribution name.

*SCALE STIFFNESS: Specify scale factors and, for adjoint design sensitivity analysis, quantities that represent the derivatives of the scale factors divided by the scale factors for material stiffness.

This option is used to specify scale factors using a distribution for material stiffness. In addition, in an adjoint design sensitivity analysis, you can specify quantities that represent the derivatives of the scale factors divided by the scale factors. In the latter case, the scale factors and their derivatives are used in the computation of design responses and topology sensitivities.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

• *Adjoint Design Sensitivity Analysis*

There are no parameters associated with this option.

Data lines:

First (and only) line:

1. Distribution name.

*SCALE STRESS DESIGN: Specify scale factors and quantities that represent the derivatives of the scale factors divided by the scale factors for material stress design response.

This option is used to specify scale factors and quantities that represent the derivatives of the scale factors divided by the scale factors using a distribution for material stress design response. In an adjoint design sensitivity analysis, the scale factors and their derivatives are used in the computation of design responses and topology sensitivities.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

• *Adjoint Design Sensitivity Analysis*

There are no parameters associated with this option.

Data lines:

First (and only) line:

1. Distribution name.

*SCALE THERMAL CONDUCTIVITY: Specify scale factors for thermal conductivity.

This option is used to specify scale factors using a distribution for material thermal conductivity.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

There are no parameters associated with this option.

Data lines:

First (and only) line:

1. Distribution name.

*SECTION CONTROLS: Specify section controls.



Warning: Using values larger than the default values for hourglass control can produce excessively stiff response and sometimes can even lead to instability if the values are too large. Hourglassing that occurs with the default hourglass control parameters is usually an indication that the mesh is too coarse. Therefore, it is generally better to refine the mesh than to add stronger hourglass control.

This option is used to

- choose a nondefault hourglass control approach for reduced-integration elements in Abaqus/Standard and Abaqus/Explicit and modified tetrahedral or triangular elements in Abaqus/Standard,
- scale the default coefficients used in the hourglass control,
- activate distortion control for solid elements, and
- control the activation of bending stabilization for full-integration linear elements modeled with hyperelastic materials and specify the scale factor.

In Abaqus/Explicit this option is also used to

- select a nondefault kinematic formulation for 8-node brick elements,
- choose the second-order accurate formulation for solids and shells,
- scale the drill stiffness of shell elements,
- turn off the drill stiffness in small-strain shell elements S3RS and S4RS,
- introduce the stresses in membrane elements from the initial configuration gradually into the model,
- scale the linear and quadratic bulk viscosities,
- specify the size of the particle tracking box for discrete element method (DEM) analyses and smoothed particle hydrodynamic (SPH) analyses,
- specify additional control parameters for SPH analyses,
- specify whether elements must be deleted when they are completely damaged,
- specify a value of the scalar degradation parameter at or above which elements are assumed to be completely damaged,
- control the activation of the "improved" element time estimation method for three-dimensional continuum elements and elements with plane stress formulations,
- control shell element deletion based on integration point status, and
- control distortion-based element deletion.

This option is used in conjunction with the **MEMBRANE SECTION*, **SHELL GENERAL SECTION*, **SHELL SECTION*, and **SOLID SECTION* options. It can be used with **BEAM SECTION* and **BEAM GENERAL SECTION* to scale the linear and bulk viscosities. It can also be used with the **SOLID SECTION*, **SHELL SECTION*, **MEMBRANE SECTION*, **COHESIVE SECTION*, and **CONNECTOR SECTION* options to:

- specify whether elements must be deleted when they are completely damaged,
- specify a value of the scalar degradation parameter at or above which elements are assumed to be completely damaged, and,
- in Abaqus/Standard, specify the viscosity coefficient that controls the viscous regularization.

For an import analysis, you must select the necessary **SECTION CONTROLS* settings in the original analysis (even if some are not applicable for the original analysis); the settings specified in the original analysis are passed into the import analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Mesh module

References:

- *Section Controls*
- **COHESIVE SECTION*
- **CONNECTOR SECTION*
- **MEMBRANE SECTION*
- **SHELL GENERAL SECTION*
- **SHELL SECTION*
- **SOLID SECTION*

Required parameters:

NAME

Set this parameter equal to a label that is used to refer to the section control definition. All section control names in the same input file must be unique.

Optional parameters:

BENDING STABILIZATION

In Abaqus/Standard, this parameter applies to analyses involving full-integration linear elements (*Using Section Controls for Bending Stabilization*) modeled with built-in isotropic hyperelastic (without hysteresis), hyperfoam, low-density foam, and user-defined materials (defined through *UHYPER*, *UHYPER_STRETCH*, and *UMAT*) . In Abaqus/Explicit, this parameter applies to analyses involving C3D8 elements modeled with built-in hyperelastic, hyperfoam, and user-defined hyperelastic materials.

Set BENDING STABILIZATION=ON (default for built-in materials) to activate bending stabilization.

Set BENDING STABILIZATION=OFF (default for user-defined materials) to deactivate bending stabilization.

CONVERSION CRITERION

This parameter applies only to Abaqus/Explicit analyses involving the conversion of continuum elements to SPH particles and is valid only when ELEMENT CONVERSION=YES and SPH CONVERSION=PER ELEMENT.

Set CONVERSION CRITERION=TIME (default) to specify the time when continuum elements are to convert to SPH particles.

Set CONVERSION CRITERION=STRAIN to specify the maximum principal strain (absolute value) when continuum elements are to convert to SPH particles.

Set CONVERSION CRITERION=STRESS to specify the maximum principal stress (absolute value) when continuum elements are to convert to SPH particles.

Set CONVERSION CRITERION=USER to specify a used-defined criterion when continuum elements are to convert to SPH particles.

DELETE DISTORTED ELEMENT

This parameter applies only to Abaqus/Explicit for deformable elements (except cohesive elements) and is valid only when ELEMENT DELETION=YES.

Set DELETE DISTORTED ELEMENT=YES to allow element deletion when one of the specified distortion criterion is exceeded.

Set DELETE DISTORTED ELEMENT=NO (default) to deactivate distortion-based element deletion.

DISTORTION CONTROL

This parameter applies to Abaqus/Explicit analyses and to solid sections in Abaqus/Standard analyses containing C3D10HS elements.

Set DISTORTION CONTROL=YES to activate a constraint that acts to prevent negative element volumes or other excessive distortion for crushable materials. This is the default value for elements with hyperelastic or hyperfoam materials. The DISTORTION CONTROL parameter is not relevant for linear kinematics and cannot prevent elements from being distorted due to temporal instabilities, hourglass instabilities, or physically unrealistic deformation.

Set DISTORTION CONTROL=NO to deactivate a constraint that acts to prevent negative element volumes or other excessive distortion for crushable materials. This value is the default except for elements with hyperelastic or hyperfoam materials.

DRILL STIFFNESS

This parameter applies only to the small-strain shell elements S3RS and S4RS.

Set DRILL STIFFNESS=ON (default) to activate the constraint against the drill mode in S3RS and S4RS elements.

Set DRILL STIFFNESS=OFF to deactivate the constraint against the drill mode. Deactivating the drill constraint can result in large values for the rotation degrees of freedom at the nodes of these elements.

The drill constraint is always active for finite-strain conventional shell elements such as S4R.

ELEMENT CONVERSION

This parameter applies only to Abaqus/Explicit analyses involving the conversion of continuum elements to SPH particles.

Set ELEMENT CONVERSION=NO (default) to prevent continuum elements from converting to SPH particles.

Set ELEMENT CONVERSION=YES to allow continuum elements to convert to SPH particles. The default element conversion method is uniform background grid conversion. You can select the conversion method using the SPH CONVERSION parameter.

ELEMENT DELETION

This parameter applies to all elements with progressive damage behavior, except connector elements in Abaqus/Explicit.

This parameter is required when DELETE DISTORTED ELEMENT=YES.

Set ELEMENT DELETION=YES to allow element deletion when the element has completely damaged. This value is the default for all elements with a damage evolution model. However, this value is not applicable to pore pressure cohesive elements.

Set ELEMENT DELETION=NO to allow fully damaged elements to remain in the computations. The element retains a residual stiffness given by the specified value of MAX DEGRADATION. In

addition, elements with three-dimensional stress states (including generalized plane strain elements) can sustain hydrostatic compressive stresses, and elements with one-dimensional stress states can sustain compressive stresses. This value is the default for pore pressure cohesive elements and is not available for beam elements.

HOURGLASS

Set HOURGLASS=COMBINED to define the viscous-stiffness form of hourglass control for solid and membrane elements with reduced integration in Abaqus/Explicit.

Set HOURGLASS=ENHANCED (default for elements with hyperelastic and hyperfoam materials in Abaqus/Standard and Abaqus/Explicit; default in Abaqus/Standard and only option in Abaqus/Explicit for modified tetrahedral or triangular elements) to define hourglass control that is based on the assumed enhanced strain method for solid, membrane, finite-strain shell elements with reduced integration and modified tetrahedral or triangular elements in Abaqus/Standard and Abaqus/Explicit. Any data given on the data line is ignored for this case.

Set HOURGLASS=RELAX STIFFNESS (default for Abaqus/Explicit, except for elements with hyperelastic and hyperfoam materials) to use the integral viscoelastic form of hourglass control for all elements with reduced integration in Abaqus/Explicit. This value is not supported for Eulerian EC3D8R elements.

Set HOURGLASS=STIFFNESS (default for Abaqus/Standard, except for elements with hyperelastic and hyperfoam materials and modified tetrahedral or triangular elements) to define hourglass control that is strictly elastic for all elements with reduced integration in Abaqus/Standard and Abaqus/Explicit and modified tetrahedral or triangular elements in Abaqus/Standard.

Set HOURGLASS=VISCOUS (default for Eulerian EC3D8R elements) to define the hourglass damping used to control the hourglass modes for solid and membrane elements with reduced integration in Abaqus/Explicit.

HTINTEGRATION

This parameter applies only to Abaqus/Standard heat transfer analyses with temperature-dependent conductivity using linear brick or quadrilateral elements.

Set HTINTEGRATION=MIXED (default) to evaluate the conductivity term at Gauss integration points and the capacity term at nodes.

Set HTINTEGRATION=GAUSS to evaluate the conductivity and capacity terms at Gauss integration points.

IMPROVED DT METHOD

Include this parameter to activate the "improved" element time estimation method for three-dimensional continuum elements and elements with plane stress formulations in Abaqus/Explicit.

Set IMPROVED DT METHOD=GLOBAL (default) to match the setting of the "improved" element time estimation method defined globally for the whole model.

Set IMPROVED DT METHOD=YES to activate the "improved" element time estimation method.

Set IMPROVED DT METHOD=NO to deactivate the "improved" element time estimation method.

INITIAL GAP OPENING

This parameter applies only to Abaqus/Standard analyses using pore pressure cohesive elements or using enriched pore pressure continuum elements.

Set this parameter equal to the value of the initial gap opening used in the tangential flow continuity equation for pore pressure cohesive elements or enriched pore pressure continuum elements. The default value is 0.002.

KERNEL

This parameter applies only to Abaqus/Explicit analyses involving smoothed particle hydrodynamics (SPH).

Set KERNEL=CUBIC (default) to use a cubic spline interpolator for the SPH formalism.

Set KERNEL=QUADRATIC to use a quadratic interpolator for the SPH formalism.

Set KERNEL=QUINTIC to use a quintic interpolator for the SPH formalism.

KINEMATIC SPLIT

Include this parameter to change the kinematic formulation for 8-node brick elements only.

Set KINEMATIC SPLIT=AVERAGE STRAIN (default in Abaqus/Explicit) to use the uniform strain formulation and the hourglass shape vectors in the hourglass control. This is the only option available for Abaqus/Standard.

Set KINEMATIC SPLIT=CENTROID to use the centroid strain formulation and the hourglass base vectors in the hourglass control in Abaqus/Explicit.

Set KINEMATIC SPLIT=ORTHOGONAL to use the centroid strain formulation and the hourglass shape vectors in the hourglass control in Abaqus/Explicit.

If SECOND ORDER ACCURACY=YES, the KINEMATIC SPLIT parameter is reset to AVERAGE STRAIN in Abaqus/Explicit.

LENGTH RATIO

This parameter applies only to Abaqus/Explicit analyses and is valid only when the DISTORTION CONTROL parameter is used.

Set this parameter equal to r ($0.0 < r \leq 1.0$) to define the length ratio when using distortion control for crushable materials. The default value is $r = 0.1$.

LINEAR KINEMATIC CONVERSION

This parameter applies only to Abaqus/Explicit analyses to activate linear kinematic conversion. It applies to all solid elements and also to membrane elements if they share nodes with solid elements for which linear kinematic conversion is activated. This parameter is valid only when distortion control is activated; however, C3D10 elements do not require the use of distortion control.

Set LINEAR KINEMATIC CONVERSION=YES to activate the linear kinematic conversion when the element characteristic length ratio with respect to the original element size is below the specified LKC RATIO value.

Set LINEAR KINEMATIC CONVERSION=NO (default) to deactivate linear kinematic conversion.

LKC RATIO

This parameter applies only to Abaqus/Explicit analyses, and is valid only when LINEAR KINEMATIC CONVERSION = YES is defined.

Set this parameter equal to r ($0.0 < r \leq 1.0$) to define the ratio below which the linear kinematic conversion option is activated. The default value is $r = 0.2$.

MAX DEGRADATION

This parameter applies to all elements with progressive damage behavior, except connector elements in Abaqus/Explicit.

Set this parameter equal to the value of the damage variable at or above which a material point is assumed to be completely damaged. This parameter also determines the amount of residual stiffness that is retained by elements for which the ELEMENT DELETION parameter is set to NO.

For elements other than cohesive elements, connector elements, elements with the multiscale damage evolution model, and elements with plane stress formulations, the default value is 1.0 if the element is deleted from the mesh and 0.99 otherwise. For cohesive elements, connector elements, elements with the multiscale damage evolution model, and elements with plane stress formulations, the default value is always 1.0.

MIN GAP PARTICLE RATIO

This parameter applies only to Abaqus/Standard geotechnical analyses that model slurry transport and placement.

Set this parameter equal to the minimum ratio between the gap width and the particle diameter that is required for proppant particles to be transported into a gap/crack by the slurry flow. The default value is 1.0.

PARTICLE THICKNESS

This parameter applies only to Abaqus/Explicit analyses and is valid only when ELEMENT CONVERSION=BACKGROUND GRID.

Set PARTICLE THICKNESS=VARIABLE (default) to define the nonuniform thickness of the particles.

Set PARTICLE THICKNESS=UNIFORM to define the uniform thickness of the particles.

PERTURBATION

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to a small perturbation to be applied to the second orientation for the FLEXION-TORSION connectors.

PREACTIVATION SCALING

This parameter applies only to Abaqus/Standard analyses using progressive element activation.

Set this parameter equal to the coefficient to be used to scale material properties for elements that are inactive if **ELEMENT PROGRESSIVE ACTIVATION*, FOLLOW DEFORMATION=YES. The default value is 10^{-4}.

RAMP INITIAL STRESS

This parameter applies to solid elements, membrane elements, beam elements, and shell elements (S4R and S4RS) in Abaqus/Explicit analyses.

Set this parameter equal to the name of a total time-based amplitude defined to go from an initial value of zero to a final value of one. When this parameter is specified, the element stiffness is controlled until the amplitude value reaches its final value of one, so that the initial stresses are introduced gradually and not abruptly.

SECOND ORDER ACCURACY

This parameter applies only to Abaqus/Explicit analyses; the element formulation is always second-order accurate in Abaqus/Standard.

Set SECOND ORDER ACCURACY=YES to use a second-order accurate formulation for solid or shell elements suitable for problems undergoing a large number of revolutions (> 5).

Set SECOND ORDER ACCURACY=NO (default) to use the first-order accurate solid or shell elements.

The SECOND ORDER ACCURACY parameter is not relevant for linear kinematics.

SHELL DELETION NUMBER

This parameter applies only to shell elements in Abaqus/Explicit and is valid only when ELEMENT DELETION=YES.

This parameter allows you to delete shell elements based on the number of active or failed integration points through the shell section. The default value is 1.

Set SHELL DELETION NUMBER equal to a positive integer that represents the number of active integration points at which the shell element is deleted.

Set SHELL DELETION NUMBER equal to a negative integer that represents the number of failed integration points at which the shell element is deleted.

Set SHELL DELETION NUMBER equal to zero to delete the shell element when all the integration points through the shell section are failed.

SPH CONVERSION

This parameter applies only to an Abaqus/Explicit analysis involving the element conversion method and is valid only when ELEMENT CONVERSION=YES.

Set SPH CONVERSION=BACKGROUND GRID (default) to allow continuum elements to convert to SPH particles that are uniformly distributed in an equally spaced background grid. SPH CONVERSION=BACKGROUND GRID is the preferred option for element conversion because the uniform distribution of SPH particles can help achieve better simulation accuracy.

Set SPH CONVERSION=PER ELEMENT to allow continuum elements to convert to SPH particles based on each parent continuum element. If the parent continuum elements are not uniform in size, the converted SPH particles are not distributed uniformly, which can affect the simulation accuracy adversely.

SPH FORMULATION

This parameter applies only to Abaqus/Explicit analyses involving smoothed particle hydrodynamics (SPH).

Set SPH FORMULATION=CLASSICAL (default) to use the classical SPH method.

Set SPH FORMULATION=NSPH to use the normalized SPH method.

Set SPH FORMULATION=XSPH to use the XSPH method.

SPH SMOOTHING LENGTH

This parameter applies only to Abaqus/Explicit analyses involving smoothed particle hydrodynamics (SPH).

Set SPH SMOOTHING LENGTH=CONSTANT (default) to use the constant smoothing length.

Set SPH SMOOTHING LENGTH=VARIABLE to use the variable smoothing length.

SPH TENSILE INSTABILITY CONTROL

This parameter applies only to Abaqus/Explicit analyses and is valid only when ELEMENT CONVERSION=BACKGROUND GRID or when SPH particles are initially in a uniform distribution.

Set SPH TENSILE INSTABILITY CONTROL=NO (default) to not use the SPH tensile instability control.

Set SPH TENSILE INSTABILITY CONTROL=YES to use the SPH tensile instability control.

VISCOSITY

This parameter applies to cohesive elements, connector elements, and elements with plane stress formulations (plane stress, shell, continuum shell, and membrane elements) in Abaqus/Standard analyses.

Set this parameter equal to the value of the viscosity coefficient used in the viscous regularization scheme for cohesive elements or connector elements or equal to the value of the damping coefficient used in connector failure modeling. When this parameter is used to specify the viscosity coefficients

for the damage model for fiber-reinforced materials, the specified value is applied to all the damage modes. The default value is 0.0.

WEIGHT FACTOR

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to α ($0.0 \le \alpha \le 1.0$) to scale the contributions from the constant hourglass stiffness term and the hourglass damping term to the hourglass control formulation. Setting $\alpha = 0.0$ or $\alpha = 1.0$ corresponds to pure constant stiffness hourglass control and pure damping hourglass control, respectively. The default is $\alpha = 0.5$. This option is used only for solid and membrane elements when the HOURGLASS parameter is set equal to COMBINED.

Data lines to define the hourglass control, bulk viscosity, drill stiffness, bending stabilization, and smoothed particle hydrodynamics-related controls:

First line:

1. Scaling factor, s^s, for the hourglass stiffness, for use with displacement degrees of freedom. If this value is left blank, Abaqus uses the default value of 1.0. The suggested range for the value of s^s is between 0.2 and 3.0.
2. Scaling factor, s^r, for the hourglass stiffness, for use with rotational degrees of freedom in shell elements. If this value is left blank, Abaqus uses the default value of 1.0. The suggested range for the value of s^r is between 0.2 and 3.0.
3. Scaling factor, s^w, for the hourglass stiffness, for use with the out-of-plane displacement degree of freedom in small-strain shell elements and membrane elements with reduced integration in Abaqus/Explicit. If this value is left blank, Abaqus/Explicit uses the default value of 1.0. The suggested range for the value of s^w is between 0.2 and 3.0. This scaling factor is not relevant for Abaqus/Standard.
4. Scaling factor for the linear bulk viscosity in Abaqus/Explicit. If this value is left blank, Abaqus/Explicit uses the default value of 1.0. The suggested range for the value is between 0.0 and 1.0. This scaling factor is not relevant for Abaqus/Standard.
5. Scaling factor for the quadratic bulk viscosity in Abaqus/Explicit. If this value is left blank, Abaqus/Explicit uses the default value of 1.0. The suggested range for the value is between 0.0 and 1.0. This scaling factor is not relevant for Abaqus/Standard.
6. Scaling factor, s^p, for the hourglass stiffness for use with pressure Lagrange multiplier degrees of freedom for element type C3D4H in Abaqus/Standard. If this value is left blank, Abaqus/Standard uses the default value of 1.0. The suggested range for the value of s^p is between 0.2 and 3.0. This scaling factor is not relevant for Abaqus/Explicit.
7. Scaling factor, f_s for bending stabilization. If this value is left blank, Abaqus uses the default value of 1.0.
8. Scaling factor for the drill stiffness of shell elements in Abaqus/Explicit. This scaling factor is not relevant for Abaqus/Standard.

Second line (optional, to be used only in conjunction with smoothed particle hydrodynamics):

1. User-specified smoothing length.
2. User-specified smoothing length factor to scale the smoothing length.
3. Minimum number of particles within the sphere of influence for a given particle (default is 5).
4. Not used.
5. Mean velocity filtering coefficient for modified coordinate updates for particles used in the XSPH method. If this value is left blank, Abaqus/Explicit uses the default value of 0.5. The suggested range for the value is between 0.0 and 1.0.

Third line (optional, to be used only in conjunction with smoothed particle hydrodynamics and the discrete element method to control the size of the particle tracking box):

1. *X*-coordinate of the lower box corner.
2. *Y*-coordinate of the lower box corner.
3. *Z*-coordinate of the lower box corner.
4. *X*-coordinate of the opposite box corner.
5. *Y*-coordinate of the opposite box corner.
6. *Z*-coordinate of the opposite box corner.
7. For SPH analyses, enter a value of 1 (default) to track all the particles as the analysis progresses; enter a value of 0 to define a particle tracking box using the coordinates given on this data line. For DEM analyses, the default is 0 to track all the particles as the analysis progresses; enter a value of 1 to define a particle tracking box using the coordinates given on this data line.

Fourth line (optional, to be used only in conjunction with the conversion of continuum elements to smoothed particle hydrodynamic particles for ELEMENT CONVERSION=YES and SPH CONVERSION=PER ELEMENT):

1. Number of particles to be generated per isoparametric direction.
2. Threshold value for the conversion criterion.

Fourth line (optional, to be used only in conjunction with the conversion of continuum elements to smoothed particle hydrodynamic particles for ELEMENT CONVERSION=YES and SPH CONVERSION=BACKGROUND GRID):

1. Spacing of the background grid.
2. Name of an orientation definition (*Orientations*) to be used to define a local coordinate system for the background grid. Only the rectangular system can be used.

Fifth line (optional, to be used only in conjunction with the discrete element method to modify the scaling factor for one or more criterion for automatic time incrementation):

1. $f_{stability}$, scaling factor for stability criterion. The default is 0.9.
2. $f_{tracking}$, scaling factor for tracking accuracy criterion. The default is 0.1.
3. $f_{collision}$, scaling factor for duration of collision criterion. The default is 0.1.
4. $f_{Rayleigh}$, scaling factor for Rayleigh wave propagation criterion. The default is 0.9.
5. $f_{separation_dist}$, scaling factor for separation distance for JKR model criterion. The default is 0.3.
6. $f_{rotation}$, scaling factor for tangential tracking accuracy criterion. The default is 0.1.

Sixth line (optional, to be used only in conjunction with DELETE DISTORTED ELEMENT=YES to delete elements with excessive distortion):

1. Ratio of the element stable DT over the original element stable DT below which the element is deleted. The value must be greater than zero and less than one. If this entry is empty, this criterion is not used. The element stable DT used does not include damping (bulk viscosity), mass scaling, or penalty contact effects.
2. Ratio of the element volume/area over the original element volume/area below which the element is deleted. The value must be greater than zero and less than one. If this entry is empty, this criterion is not used.
3. Ratio of the element characteristic length over the original element characteristic length below which the element is deleted. The value must be greater than zero and less than one. If this entry is empty, this criterion is not used.
4. Element stable DT value below which the element is deleted. The value must be greater than zero. If this entry is empty, this criterion is not used. The element stable DT used does not include damping (bulk viscosity), mass scaling, or penalty contact effects.
5. Element volume or area value below which the element is deleted. The value must be greater than or equal to zero. The default is zero.

6. Element characteristic length value below which the element is deleted. The value must be greater than or equal to zero. The default is zero.

*SECTION CURVATURE: Define the precurvatures and the pretorsion of the beam centerline for 3-DOF warping elements.

This option is used in conjunction with the *BEAM SECTION GENERATE option to define the precurvatures and the pretorsion of the beam centerline. You use the *BEAM SECTION GENERATE option to compute the properties for the specified beam elements.

This option is available only for use with 3-DOF warping elements. If this option is not used, Abaqus assumes the precurvatures and the pretorsion of the beam centerline are zero.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *BEAM SECTION GENERATE
- *Meshed Beam Cross-Sections*

There are no parameters associated with this option.

Data line to define the beam centerline precurvatures and pretorsion:

First (and only) line:

1. Precurvature of the beam centerline in the plane perpendicular to the 1-axis of the cross-section (reciprocal of the osculating circle radius of the centerline projection onto that plane). (See *Figure 1*.) The default is 0.0.
2. Precurvature of the beam centerline in the plane perpendicular to the 2-axis of the cross-section (reciprocal of the osculating circle radius of the centerline projection onto that plane). The default is 0.0.
3. Pretorsion of the cross-section orientations along the beam centerline in radians per unit length. (See *Figure 2*.) The default is 0.0.

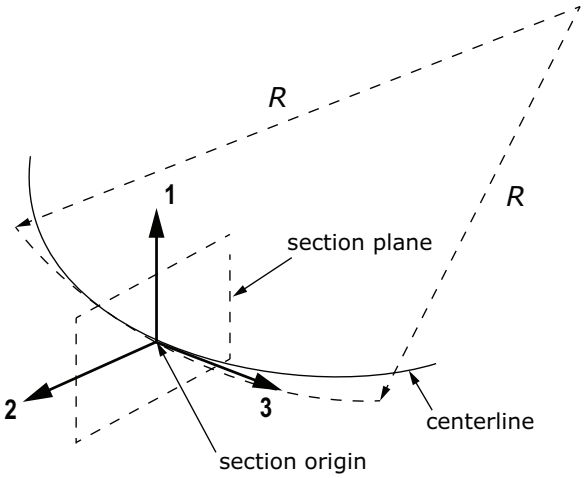


Figure 1: Precurvature.

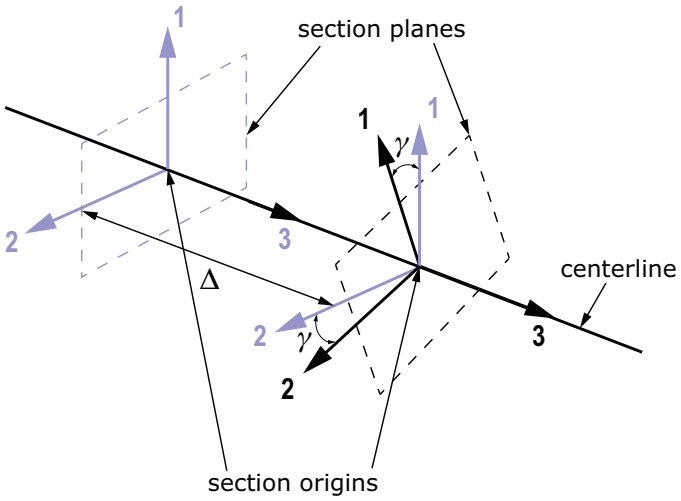


Figure 2: Pretorsion.

*SECTION FILE: Define results file requests of accumulated quantities on user-defined surface sections.

This option is used to control output to the results file of accumulated quantities associated with a user-defined section. Depending on the analysis type the output may include one or several of the following: the total force, the total moment, the total heat flux, the total current, the total mass flow, or the total pore fluid volume flux associated with the section. This option is not available for eigenfrequency extraction, eigenvalue buckling prediction, complex eigenfrequency extraction, or linear dynamics procedures.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Output to the Data and Results Files*
- *Abaqus/Standard Output Variable Identifiers*

Required parameters:

NAME

Set this parameter equal to a label that will be used to identify the output for the section. Section names in the same input file must be unique.

SURFACE

Set this parameter equal to the name used in the **SURFACE* option to define the surface.

Optional parameters:

AXES

Set AXES=LOCAL if output is desired in the local coordinate system. Set AXES=GLOBAL (default) to output quantities in the global coordinate system.

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

UPDATE

Set UPDATE=NO if output is desired in the original local system of coordinates. Set UPDATE=YES (default) to output quantities in a local system of coordinates that rotates with the average rigid body motion of the surface section. This parameter is relevant only if AXES=LOCAL and the NLGEOM parameter is active in the step.

Optional data lines:

First line:

1. Node number of the anchor point (blank if coordinates given).
2. First coordinate of the anchor point (ignored if node number given).
3. Second coordinate of the anchor point (ignored if node number given).

4. Third coordinate of the anchor point (for three-dimensional cases only; ignored if node number given).

Leave this line blank to allow Abaqus to define the anchor point.

Second line:

1. Node number used to specify point *a* in *Figure 1* (blank if coordinates given).
2. First coordinate of point *a* (ignored if node number given).
3. Second coordinate of point *a* (ignored if node number given).

The remaining data items are relevant only for three-dimensional cases.

9. Third coordinate of point *a* (ignored if node number given).
11. Node number used to specify point *b* (blank if coordinates given).
13. First coordinate of point *b* (ignored if node number given).
15. Second coordinate of point *b* (ignored if node number given).
17. Third coordinate of point *b* (ignored if node number given).

Leave this line blank to allow Abaqus to define the axes.

Third line:

1. Give the identifying keys for the variables to be output. The keys are defined in the "Section variables" section of *Abaqus/Standard Output Variable Identifiers*.

Omit both the first and second data lines for AXES=GLOBAL or to allow Abaqus to define the anchor point and the axes for AXES=LOCAL. Repeat the third data line as often as necessary to define the variables to be written to the results file. If this line is omitted, all appropriate variables (*Output to the Data and Results Files*) will be output.

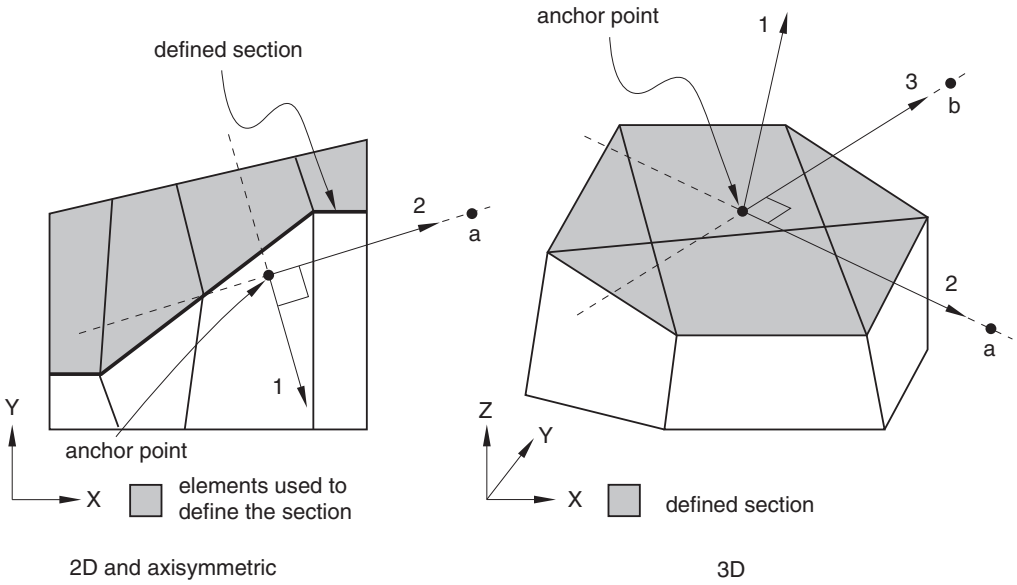


Figure 1: User-defined local coordinate system.

*SECTION INERTIA: Define inertia properties for meshed general beam sections.

This option must be used in conjunction with the **BEAM GENERAL SECTION*, SECTION=MESHED option to assign the beam section mass and inertia properties that result from the two-dimensional meshed cross-section generation procedure. The properties are written to the *jobname*`.bsp` file during the cross-section generation and are typically read into a subsequent beam analysis using the **INCLUDE* option.

The **SECTION STIFFNESS* and **SECTION INERTIA* options are mutually exclusive with the second and third data lines of the **BEAM GENERAL SECTION* option. See *Meshed Beam Cross-Sections* for details.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- **BEAM GENERAL SECTION*

There are no parameters associated with this option.

Data line:

First (and only) line:

1. Total mass of the section per unit length, (ρA).
2. Rotary inertia about the 1-axis of the section, $(\rho I)_{11}$.
3. Rotary product of inertia, $(\rho I)_{12}$.
4. Rotary inertia about the 2-axis of the section, $(\rho I)_{22}$.
5. Local 1-coordinate of the center of mass, x_{1cm}.
6. Local 2-coordinate of the center of mass, x_{2cm}.

*SECTION ORIGIN: Define a meshed cross-section origin.

This option is used in conjunction with the *BEAM SECTION GENERATE option to define the location of the beam node on a meshed beam cross-section.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Meshed Beam Cross-Sections*
- *BEAM SECTION GENERATE

Optional parameters:

ORIGIN

Set ORIGIN=CENTROID to place the beam node at the section centroid.

Set ORIGIN=SHEAR CENTER to place the beam node at the shear center.

Data line to define the location of the beam node if the ORIGIN parameter is omitted:

First (and only) data line:

1. Global *x*-coordinate of the beam node. The default is 0.0.
2. Global *y*-coordinate of the beam node. The default is 0.0.

If this option is not used, the section origin is placed at the origin of the coordinate system on the meshed cross-section model. If this option is used without the ORIGIN parameter and no data line is given, the section origin is placed at the centroid.

*SECTION POINTS: Locate points in the beam section for which stress and strain output are required.

This option is used as model data in Abaqus/Standard and Abaqus/Explicit in conjunction with the *BEAM GENERAL SECTION option and as history data in Abaqus/Standard in conjunction with the *BEAM SECTION GENERATE option.

When used in conjunction with the *BEAM GENERAL SECTION option and a predefined library section, it locates section points in the beam section for which axial stress and axial strain output are required.

When used in conjunction with the *BEAM GENERAL SECTION, SECTION=MESHED option, it locates elements and integration points in the beam cross-section model for which stress and strain output are required and provides material data and derivatives of the warping function required for determining stress and strain output.

When used in conjunction with the *BEAM SECTION GENERATE option, it associates a user-defined section point label with elements and integration points in the beam cross-section model for which stress and strain output are required during the subsequent beam analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Part, Part instance, Step

Abaqus/CAE: Property module

References:

- *Using a General Beam Section to Define the Section Behavior*
- *Meshed Beam Cross-Sections*
- *BEAM GENERAL SECTION

There are no parameters associated with this option.

Data lines to locate section points for output when used in conjunction with the *BEAM GENERAL SECTION option and a predefined library section:

First line:

1. Local x_1-position of first section point.
2. Local x_2-position of first section point.
3. Local x_1-position of second section point.
4. Local x_2-position of second section point.

Continue giving (x_1, x_2) coordinate pairs for as many points as needed. At most four pairs of points can be specified on any data line. If the point (0,0) is specified as the last entry on a line, it will be ignored unless it is the only point requested.

Data lines to locate elements and integration point numbers for meshed sections when used in conjunction with the *BEAM GENERAL SECTION, SECTION=MESHED option:

First line:

1. Section point label.
2. Two-dimensional model element number.
3. Integration point number.

Second line:

1. Young's modulus, *E*.
2. Shear modulus in user-defined 1-direction, G_1.
3. Shear modulus in user-defined 2-direction, G_2 (for isotropic materials G_2=G_1).
4. Material orientation angle, α, measured counterclockwise from the beam local 1-direction.
5. Local $-(x_1 - x_{1c})$-position of the integration point with respect to the centroid.
6. Local $(x_2 - x_{2c})$-position of the integration point with respect to the centroid.
7. Derivative of the warping function $\left(\frac{\partial \Psi}{\partial x_1} - (x_2 - x_{2s})\right)$ with respect to the local 1-direction and the shear center.
8. Derivative of the warping function $\left(\frac{\partial \Psi}{\partial x_2} + (x_1 - x_{1s})\right)$ with respect to the local 2-direction and the shear center.

Repeat this set of data lines for as many integration points as needed.

Data lines to locate elements and integration point numbers in the cross-section model when used in conjunction with the *BEAM SECTION GENERATE option:

First line:

1. Section point label.
2. Element number.
3. Integration point number.

Repeat this data line as often as necessary to specify the elements and integration points.

*SECTION PRINT: Define print requests of accumulated quantities on user-defined surface sections.

This option is used to provide tabular output of accumulated quantities associated with a user-defined section. Depending on the analysis type the output may include one or several of the following: the total force, the total moment, the total heat flux, the total current, the total mass flow, or the total pore fluid volume flux associated with the section. This option is not available for eigenfrequency extraction, eigenvalue buckling prediction, complex eigenfrequency extraction, or linear dynamics procedures.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Output to the Data and Results Files*
- *Abaqus/Standard Output Variable Identifiers*

Required parameters:

NAME

Set this parameter equal to a label that will be used to identify the output for the section. Section names in the same input file must be unique.

SURFACE

Set this parameter equal to the name used in the **SURFACE* option to define the surface.

Optional parameters:

AXES

Set AXES=LOCAL if output is desired in the local coordinate system. Set AXES=GLOBAL (default) to output quantities in the global coordinate system.

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

UPDATE

Set UPDATE=NO if output is desired in the original local system of coordinates. Set UPDATE=YES (default) to output quantities in a local system of coordinates that rotates with the average rigid body motion of the surface section. This parameter is relevant only if AXES=LOCAL and the NLGEOM parameter is active in the step.

Optional data lines:

First line:

1. Node number of the anchor point (blank if coordinates given).
2. First coordinate of the anchor point (ignored if node number given).
3. Second coordinate of the anchor point (ignored if node number given).

4. Third coordinate of the anchor point (for three-dimensional cases only; ignored if node number given).

Leave this line blank to allow Abaqus to define the anchor point.

Second line:

1. Node number used to specify point *a* in *Figure 1* (blank if coordinates given).
2. First coordinate of point *a* (ignored if node number given).
3. Second coordinate of point *a* (ignored if node number given).

The remaining data items are relevant only for three-dimensional cases.

9. Third coordinate of point *a* (ignored if node number given).
11. Node number used to specify point *b* (blank if coordinates given)
13. First coordinate of point *b* (ignored if node number given).
15. Second coordinate of point *b* (ignored if node number given).
17. Third coordinate of point *b* (ignored if node number given).

Leave this line blank to allow Abaqus to define the axes.

Third line:

1. Give the identifying keys for the variables to be output. The keys are defined in the "Section variables" section of *Abaqus/Standard Output Variable Identifiers*.

Omit both the first and second data lines for AXES=GLOBAL or to allow Abaqus to define the anchor point and the axes for AXES=LOCAL. Repeat the third data line as often as necessary: each line defines a table. If this line is omitted, all appropriate variables (*Output to the Data and Results Files*) will be output.

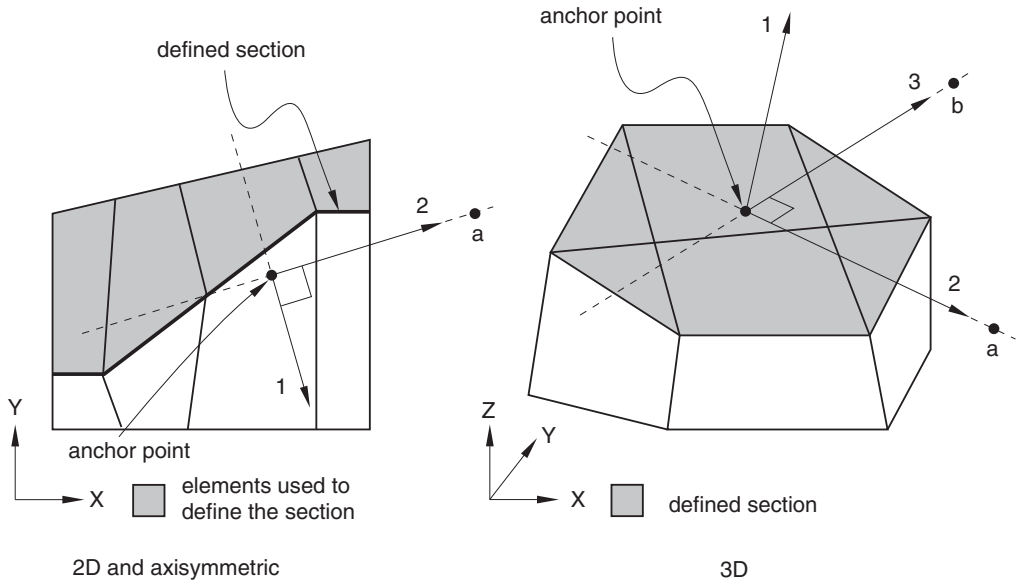


Figure 1: User-defined local coordinate system.

*SECTION STIFFNESS: Define stiffness properties for meshed general beam sections.

This option must be used in conjunction with the *BEAM GENERAL SECTION, SECTION=MESHED option to assign the beam section stiffness properties that result from the two-dimensional meshed cross-section generation procedure. The properties are written to the *jobname*`.bsp` file during the cross-section generation and are typically read into a subsequent beam analysis using the *INCLUDE option.

The *SECTION STIFFNESS and *SECTION INERTIA options are mutually exclusive with the second and third data lines of the *BEAM GENERAL SECTION option. The *SECTION STIFFNESS, FULL COUPLING option is also mutually exclusive with the *TRANSVERSE SHEAR STIFFNESS, CENTROID, and SHEAR CENTER options. See *Meshed Beam Cross-Sections* for details.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- *BEAM GENERAL SECTION

Optional parameter in Abaqus/Standard:

FULL COUPLING

Include this parameter to define a general beam section with full coupling between the six section strains (axial stretch, bending, torsion, and shear) in the constitutive relation. This includes all coupling effects that might otherwise be defined with the *TRANSVERSE SHEAR STIFFNESS, CENTROID, and SHEAR CENTER options.

If this parameter is omitted, it is assumed that the section stresses only relate directly to the respective section strains, except for cross-bending and for the additional selective coupling effects that might be defined with the *TRANSVERSE SHEAR STIFFNESS, CENTROID, and SHEAR CENTER options.

Data line if the FULL COUPLING parameter is omitted:

First (and only) line:

1. Axial stiffness of the section, (EA).
2. Bending stiffness about the 1-axis of the section, $(EI)_{11}$.
3. Stiffness for cross-bending, $(EI)_{12}$.
4. Bending stiffness about the 2-axis of the section, $(EI)_{22}$.
5. Torsional constant, (GJ).

Data lines if the FULL COUPLING parameter is included:

First line:

1. Axial stiffness of the section, $(K)_{11}$.
2. Shear stiffness of the section in the first direction, $(K)_{22}$.
3. Shear stiffness of the section in the second direction, $(K)_{33}$.

4. Torsion stiffness, $(K)_{44}$.
5. Bending stiffness about the 1-axis of the section, $(K)_{55}$.
6. Bending stiffness about the 2-axis of the section, $(K)_{66}$.
7. Coupling stiffness of the section for axial stretch–shear in the first direction, $(K)_{12}$.
8. Coupling stiffness of the section for axial stretch–shear in the second direction, $(K)_{13}$.

Second line:

1. Coupling stiffness of the section for axial stretch–torsion, $(K)_{14}$.
2. Coupling stiffness of the section for axial stretch–bending about the 1-axis, $(K)_{15}$.
3. Coupling stiffness of the section for axial stretch–bending about the 2-axis, $(K)_{16}$.
4. Stiffness of the section for cross-shear, $(K)_{23}$.
5. Coupling stiffness of the section for shear in the first direction–torsion, $(K)_{24}$.
6. Coupling stiffness of the section for shear in the first direction–bending about the 1-axis, $(K)_{25}$.
7. Coupling stiffness of the section for shear in the first direction–bending about the 2-axis, $(K)_{26}$.
8. Coupling stiffness of the section for shear in the second direction–torsion, $(K)_{34}$.

Third line:

1. Coupling stiffness of the section for shear in the second direction–bending about the 1-axis, $(K)_{35}$.
2. Coupling stiffness of the section for shear in the second direction–bending about the 2-axis, $(K)_{36}$.
3. Coupling stiffness of the section for torsion–bending about the 1-axis, $(K)_{45}$.
4. Coupling stiffness of the section for torsion–bending about the 2-axis, $(K)_{46}$.
5. Stiffness of the section for cross-bending, $(K)_{56}$.

*SECTION TANGENT: Define the inclination of the tangent of the beam centerline with respect to the cross-section normal direction for 3-DOF warping elements.

This option is used in conjunction with the **BEAM SECTION GENERATE* option to define the inclination of the tangent of the beam centerline with respect to the cross-section normal direction. You use the **BEAM SECTION GENERATE* option to compute the properties for the specified beam elements. The beam centerline is the curve that connects the corner nodes of these beam elements.

This option is available only for use with 3-DOF warping elements. If this option is not used, Abaqus assumes the beam centerline tangent direction is parallel to the cross-section normal direction.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- **BEAM SECTION GENERATE*
- *Meshed Beam Cross-Sections*

There are no parameters associated with this option.

Data line to define the beam centerline tangent direction:

First (and only) line:

1. First direction cosine of the beam centerline. The default is 0.
2. Second direction cosine of the beam centerline. The default is 0.
3. Third direction cosine of the beam centerline. The default is 0.

*SELECT CYCLIC SYMMETRY MODES: Specify the cyclic symmetry modes in an eigenvalue analysis of a cyclic symmetric structure.

This option is used to specify which cyclic symmetry modes should be used in an eigenvalue analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *Analysis of Models that Exhibit Cyclic Symmetry*
- **CYCLIC SYMMETRY MODEL*
- **TIE*

Optional parameters:

EVEN

Include this parameter to request the even cyclic symmetry modes only. If this parameter is omitted, all cyclic symmetry modes will be used within the NMIN, NMAX range.

NMIN

Set this parameter equal to the lowest cyclic symmetry mode number. The default value is 0.

NMAX

Set this parameter equal to the highest cyclic symmetry mode number. The default value is the highest number that is feasible for the number of sectors given in the **CYCLIC SYMMETRY MODEL* option.

There are no data lines associated with this option.

*SELECT EIGENMODES: Select the modes to be used in a modal dynamic, complex eigenvalue extraction, or substructure generation analysis.

This option selects the modes to be used in a dynamic analysis based on modes, in a complex eigenvalue extraction analysis, or in a substructure generation analysis. Only one option per step can be used. If this option is omitted in a dynamic analysis based on modes or in a complex eigenvalue extraction analysis, all modes extracted in the prior **FREQUENCY* step will be used, including residual modes if they were activated.

If this option is omitted in a substructure generation analysis, none of the modes extracted in the prior **FREQUENCY* step will be used, including residual modes if they were activated.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Supported in the *Step module* only for substructure generation.

References:

- *Implicit Dynamic Analysis Using Direct Integration*
- *Complex Eigenvalue Extraction*
- *Transient Modal Dynamic Analysis*
- *Mode-Based Steady-State Dynamic Analysis*
- *Subspace-Based Steady-State Dynamic Analysis*
- *Generating Substructures*
- *Response Spectrum Analysis*
- *Random Response Analysis*
- **MODAL DAMPING*

Optional parameters:

DEFINITION

Set DEFINITION=MODE NUMBERS (default) to indicate that the selected modes are given as a collection of mode numbers.

Set DEFINITION= FREQUENCY RANGE to indicate that the modes are selected from the specified frequency ranges including frequency boundaries. Frequency ranges can be discontinuous.

If both the **MODAL DAMPING* and **SELECT EIGENMODES* options are used in the same step, the DEFINITION parameter must be set equal to the same value in both options.

GENERATE

If this parameter is included, each data line should give a first mode, n_1; a last mode, n_2; and the increment in mode numbers between these modes, i. Then all modes going from n_1 to n_2 in steps of i will be added to the set. i must be an integer such that $(n_2 - n_1)/i$ is a whole number (not a fraction).

This parameter can be used only with DEFINITION=MODE NUMBERS.

Data lines if DEFINITION=MODE NUMBERS and the GENERATE parameter is omitted:

First line:

1. List of modes to be used.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

Data lines if DEFINITION=MODE NUMBERS and the GENERATE parameter is included:

First line:

1. First mode number.
2. Last mode number.
3. Increment in mode numbers between modes. The default is 1.

Repeat this data line as often as necessary.

Data lines if DEFINITION=FREQUENCY RANGE:

First line:

1. Lower boundary of the frequency range (in cycles/time).
2. Upper boundary of the frequency range (in cycles/time).

Repeat this data line as often as necessary.

*SFILM: Define film coefficients and associated sink temperatures over a surface.

This option is used to provide film coefficients and sink temperatures over a specified surface for all procedures that solve for nodal temperature degrees of freedom.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *Thermal Loads*
- *FILM*

Optional parameters:

FILM AMPLITUDE

Set this parameter equal to the name of the **AMPLITUDE* option that gives the variation of the film coefficient, h, with time.

If this parameter is omitted in an Abaqus/Standard analysis, the reference film coefficient is applied immediately at the beginning of the step and kept constant over the step, independent of the value assigned to the AMPLITUDE parameter on the **STEP* option. If this parameter is omitted in an Abaqus/Explicit analysis, the reference sink temperature given on the data lines is applied throughout the step.

The FILM AMPLITUDE parameter is ignored if a film coefficient is defined to be a function of temperature and field variables using the **FILM PROPERTY* option.

For nonuniform films of type FNU, the film coefficient amplitude is defined in user subroutine *FILM* in Abaqus/Standard or user subroutine *VFILM* in Abaqus/Explicit; and FILM AMPLITUDE references are used only to modify the film coefficient passed into the user subroutine.

OP

Set OP=MOD (default) to modify existing films or to define additional films.

Set OP=NEW if all existing **SFILM*s applied to the model should be removed.

SINK AMPLITUDE

Set this parameter equal to the name of the **AMPLITUDE* option that gives the variation of the sink temperature, θ^0, with time.

If this parameter is omitted in an Abaqus/Standard analysis, the reference sink temperature is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (*Defining an Analysis*). If this parameter is omitted in an Abaqus/Explicit analysis, the reference sink temperature is applied immediately at the beginning of the step.

For nonuniform films of type FNU, the sink temperature amplitude is defined in user subroutine *FILM* in Abaqus/Standard or user subroutine *VFILM* in Abaqus/Explicit; and SINK AMPLITUDE references are used only to modify the sink temperature passed into the user subroutine.

TRANSITION

Set TRANSITION=TOUCHING (default) to set the film coefficient to zero on portions of the specified surface when contact is active and the contact status is closed.

Set TRANSITION=NONE to apply the film coefficient to the specified surface independent of the contact status.

Data lines to define sink temperatures and film coefficients:

First line:

1. Surface name.
2. Film type label F or FNU.
3. Reference sink temperature value, θ^0. (Units of θ.) For nonuniform film coefficients the sink temperature must be defined in user subroutine *FILM* in Abaqus/Standard or user subroutine *VFILM* in Abaqus/Explicit. If given, this value will be modified by the SINK AMPLITUDE reference and passed into the user subroutine.
4. Reference film coefficient value, h (units of $JT^{-1}L^{-2}\theta^{-1}$), or name of the film property table defined with the **FILM PROPERTY* option. Nonuniform film coefficients must be defined in user subroutine *FILM* in Abaqus/Standard or user subroutine *VFILM* in Abaqus/Explicit. If given, this value will be modified by the FILM AMPLITUDE reference and passed into the user subroutine.
5. Time period over which the film coefficient is modified to its current magnitude or set to zero from its current value due to temporal changes in the contact status. The default time period is chosen to be 0.001 of the current step time. This parameter applies only to Abaqus/Standard and is neglected in a linear perturbation analysis or when TRANSITION=NONE.

Repeat this data line as often as necessary to define film conditions for different surfaces.

*SFLOW: Define seepage coefficients and associated sink pore pressures normal to a surface.

This option is used to provide seepage coefficients and sink pore pressures to control pore fluid flow normal to the surface in consolidation analysis.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Pore Fluid Flow*
- *FLOW*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that gives the variation of reference pore pressure with time. If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step or linearly over the step depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (see *Defining an Analysis*). The AMPLITUDE parameter is ignored for nonuniform seepage flow boundary conditions defined in user subroutine *FLOW* and for drainage-only seepage boundary conditions.

OP

Set OP=MOD (default) to modify existing flows or to define additional flows.

Set OP=NEW if all existing **SFLOW*s applied to the model should be removed. New flows can be defined.

Data lines to define uniform seepage:

First line:

1. Surface name.
2. Seepage flow type label Q.
3. Reference pore pressure value, u_w^{∞}. (Units of FL^{-2}.)
4. Reference seepage coefficient value, k_s. (Units of $F^{-1}L^3T^{-1}$.)

Repeat this data line as often as necessary to define uniform seepage for various surfaces.

Data lines to define drainage-only seepage:

First line:

1. Surface name.
2. Seepage flow type label QD.
3. Drainage-only seepage coefficient value, k_s. (Units of $F^{-1}L^3T^{-1}$.)

Repeat this data line as often as necessary to define drainage-only seepage for various surfaces.

Data lines to define nonuniform seepage:

First line:

1. Surface name.
2. Seepage flow type label QNU.
3. Optional reference pore pressure value. If given, this value is passed into user subroutine *FLOW* in the variable used to define the sink pore pressure.
4. Optional reference seepage coefficient. If given, this value is passed into user subroutine *FLOW* in the variable used to define the seepage coefficient.

The reference pore pressure value, u_w^∞, and reference seepage coefficient, k_s, are defined in user subroutine *FLOW* for nonuniform flow.

Repeat this data line as often as necessary to define nonuniform seepage for various surfaces.

*SHEAR CENTER: Define the position of the shear center of a beam section.

This option can be used only in conjunction with the **BEAM GENERAL SECTION*, SECTION=GENERAL or the **BEAM GENERAL SECTION*, SECTION=MESHED option. It cannot be used if you include the FULL COUPLING parameter for the **SECTION STIFFNESS* option.

It is used to define the position of the shear center of the section with respect to the local (1, 2) axis system.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- *Using a General Beam Section to Define the Section Behavior*
- **BEAM GENERAL SECTION*
- *Meshed Beam Cross-Sections*

There are no parameters associated with this option.

Data line to define the shear center:

First (and only) line:

1. Local x_1-coordinate of shear center, x_{1s}. The default is 0.
2. Local x_2-coordinate of shear center, x_{2s}. The default is 0.

*SHEAR FAILURE: Specify a shear failure model and criterion.

This option is used with the Mises or the Johnson-Cook plasticity models to specify shear failure of the material. It must be used in conjunction with the option *PLASTIC, HARDENING=ISOTROPIC or JOHNSON COOK.

Products: Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Classical Metal Plasticity*
- *Johnson-Cook Plasticity*
- *Dynamic Failure Models*
- *PLASTIC

Optional parameters:

DEPENDENCIES

This parameter is relevant only for TYPE=TABULAR.

Set this parameter equal to the number of field variable dependencies included in the definition of the equivalent plastic strain at failure, in addition to temperature. If this parameter is omitted, it is assumed that the strain at failure depends on the plastic strain, the plastic strain rate, the dimensionless pressure-deviatoric stress ratio and, possibly, on temperature. See *Material Data Definition* for more information.

ELEMENT DELETION

Set ELEMENT DELETION=YES (default) to allow element deletion when the failure criterion is met.

Set ELEMENT DELETION=NO to allow the element to take hydrostatic compressive stress only when the failure criterion is met.

TYPE

Set TYPE=JOHNSON COOK to define the Johnson-Cook shear failure criterion. It requires the use of the *PLASTIC, HARDENING=JOHNSON COOK option.

Set TYPE=TABULAR (default) to define the failure strain in tabular data form. It requires the use of the *PLASTIC, HARDENING=ISOTROPIC option.

Data lines to define the failure strain in tabular form (TYPE=TABULAR):

First line:

1. Equivalent plastic strain at failure, $\bar{\varepsilon}^{pl}_{f}$.
2. Rate of equivalent plastic strain, $\dot{\bar{\varepsilon}}^{pl}$.
3. Dimensionless pressure-deviatoric stress ratio, p/q.
4. Temperature.
5. First field variable.
6. Second field variable.

7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of strain at failure on plastic strain, plastic strain rate, stress ratio and, if needed, on temperature and other predefined field variables.

Data line to define the Johnson-Cook shear failure criterion (TYPE=JOHNSON COOK):

First (and only) line:

1. Johnson-Cook failure parameter, d_1.
2. Johnson-Cook failure parameter, d_2.
3. Johnson-Cook failure parameter, d_3.
4. Johnson-Cook failure parameter, d_4.
5. Johnson-Cook failure parameter, d_5.

*SHEAR RETENTION: Define the reduction of the shear modulus associated with crack surfaces in a CONCRETE model as a function of the tensile strain across the crack.

This option is used to give a multiplying factor, ϱ, that defines the modulus for shearing of cracks as a fraction of the elastic shear modulus of the uncracked concrete. If this option is used, it should follow the *CONCRETE option. The *SHEAR RETENTION option can also be used in conjunction with the *FAILURE RATIOS option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Concrete Smeared Cracking*
- *CONCRETE
- *FAILURE RATIOS

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of shear retention behavior, in addition to temperature. If this parameter is omitted, it is assumed that the shear retention behavior depends only on temperature. See *Material Data Definition* for more information.

Data lines to define the shear retention behavior:

First line:

1. ϱ^{close}. The default is 1.0.
2. ε^{max}. The default is a very large number (full shear retention).
3. Not used.
4. Not used.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the shear retention behavior on temperature and other predefined field variables.

*SHEAR TEST DATA: Used to provide shear test data.

This option can be used only in conjunction with the *VISCOELASTIC option. The *SHEAR TEST DATA option cannot be used for a viscoelastic material if the *COMBINED TEST DATA option is used.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Time Domain Viscoelasticity*
- *VISCOELASTIC

Using shear test data to define a viscoelastic material

Optional parameters:

SHRINF

To specify creep test data, set this parameter equal to the value of the long-term, normalized shear compliance $j_S(\infty)$.

To specify relaxation test data, set this parameter equal to the value of the long-term, normalized shear modulus $g_R(\infty)$. The shear compliance and shear modulus are related by $j_S(\infty) = 1/g_R(\infty)$.

The fitting procedure will use the specified value in the constraint $1 - \sum_{i=1}^{N} \overline{g}_i^P = g_R(\infty)$.

Data lines to specify creep test data:

First line:

1. Normalized shear compliance $j_S(t)$. $(j_S(t) \geq 1)$
2. Time t. $(t > 0)$

Repeat this data line as often as necessary to give the compliance-time data.

Data lines to specify relaxation test data:

First line:

1. Normalized shear relaxation modulus $g_R(t)$. $(0 \leq g_R(t) \leq 1)$
2. Time t. $(t > 0)$

Repeat this data line as often as necessary to give the modulus-time data.

*SHELL GENERAL SECTION: Define a general, arbitrary, elastic shell section.

This option is used to define a general, arbitrary, elastic shell section.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- *About Shell Elements*
- *Using a General Shell Section to Define the Section Behavior*
- *UGENS*
- *VUGENS*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the shell elements for which the section behavior is being defined.

Required parameter in Abaqus/Explicit, optional parameter in Abaqus/Standard:

DENSITY

Set this parameter equal to the mass per unit surface area of the shell.

If the MATERIAL and COMPOSITE parameters are omitted, this density accounts for the mass of the shell, since no material definition is given.

If the MATERIAL or COMPOSITE parameter is used, the mass of the shell includes a contribution from this parameter in addition to any contribution from the material definition.

Optional parameters:

BENDING ONLY

Include this parameter to ignore membrane stiffness effects in the shell. Bending stiffness and transverse shear stiffness coefficients are computed normally. Membrane-bending coupling coefficients are set to zero. Diagonal membrane stiffness coefficients are set to 1×10^{-6} times the largest diagonal bending stiffness term. Off-diagonal membrane stiffness coefficients are set to zero.

CONTROLS

In an Abaqus/Explicit analysis, set this parameter equal to the name of a section controls definition (see *Section Controls*) to be used to specify the second-order accurate element formulation option, a nondefault hourglass control formulation option, or scale factors.

In an Abaqus/Standard analysis, set this parameter equal to the name of a section controls definition to be used to specify the enhanced hourglass control formulation (see *Section Controls*) or to be used in a subsequent Abaqus/Explicit import analysis.

LAYUP

This parameter is relevant only when the COMPOSITE parameter is used.

Set this parameter equal to the name of a composite layup (see *Composite layups*). Abaqus/CAE uses this name to identify the composite layup that contains the shell section.

MEMBRANE ONLY

Include this parameter to ignore bending stiffness effects in the shell. Membrane stiffness and transverse shear stiffness coefficients are computed normally. Membrane-bending coupling coefficients are set to zero. Diagonal bending stiffness coefficients are set to 1×10^{-6} times the largest diagonal membrane stiffness term. Off-diagonal bending stiffness coefficients are set to zero.

OFFSET

Include this parameter to define the distance (as a fraction of the shell thickness) from the shell midsurface to the reference surface (containing the nodes of the element). This parameter accepts positive or negative values, the labels SPOS or SNEG, or in an Abaqus/Standard analysis the name of a distribution (see *Distribution Definition*).

Positive values of the offset are in the positive normal direction (see *About Shell Elements*). When OFFSET=0.5 (or SPOS), the top surface of the shell is the reference surface. When OFFSET=−0.5 (or SNEG), the bottom surface of the shell is the reference surface. The default is OFFSET=0, which indicates that the middle surface of the shell is the reference surface.

In an Abaqus/Standard analysis a spatially varying offset can be specified by setting OFFSET equal to the name of a distribution. The distribution used to define the shell offset must have a default value. The default offset is used by any shell element assigned to the shell section that is not specifically assigned a value in the distribution.

ORIENTATION

Set this parameter equal to the name of an orientation definition (see *Orientations*) to be used with the section forces and section strains.

POISSON

Include this parameter to define the shell thickness direction behavior.

Set this parameter equal to a nonzero value to cause the thickness direction strain under plane stress conditions to be a linear function of the membrane strains. The value of the POISSON parameter must be between −1.0 and 0.5.

Set POISSON=ELASTIC to automatically select this parameter value based on the initial isotropic elastic part of the material definition.

The default is POISSON=0.5.

SMEAR ALL LAYERS

This parameter is relevant only when the COMPOSITE parameter is used.

Include this parameter to ignore the material layer stacking sequence. Transverse shear stiffness coefficients are computed normally. Membrane-bending coupling terms are set to zero, and bending stiffness terms are computed as $T^2/12$ times the corresponding membrane stiffness terms, where T is the total thickness of the shell.

STACK DIRECTION

This parameter is relevant only for continuum shells.

Set this parameter equal to 1, 2, 3, or ORIENTATION to define the continuum shell stack or thickness direction. Specify one of the numerical values to select the corresponding isoparametric direction of the element as the stack or thickness direction. The default is STACK DIRECTION=3.

If STACK DIRECTION=ORIENTATION, the ORIENTATION parameter is also required.

To obtain a desired thickness direction, the appropriate numerical value for the STACK DIRECTION parameter depends on the element connectivity. For a mesh-independent specification, use STACK DIRECTION=ORIENTATION. If the orientation assigned to the ORIENTATION parameter is defined with a distribution (*Distribution Definition*), STACK DIRECTION=ORIENTATION is not supported.

SYMMETRIC

This parameter is relevant only when the COMPOSITE parameter is used.

Include this parameter if the layers in the composite shell are symmetric about a central core. This parameter cannot be used if a spatially varying thickness or orientation angle is defined on any composite layer using a distribution (*Distribution Definition*).

THICKNESS MODULUS

This parameter is relevant only for continuum shells.

Set this parameter equal to the effective thickness modulus.

Set THICKNESS MODULUS=ELASTIC to compute the effective thickness modulus based on the in-plane elastic properties. In this case, the effective thickness modulus is set to the initial tensile modulus in the out-of-plane direction. If you use this option, you must define the shell thickness direction behavior using POISSON=ELASTIC.

ZERO

If the section is defined by its general stiffness, set this parameter equal to θ^0, the reference temperature for thermal expansion (for example, ZERO=50 means $\theta^0 = 50°$).

This parameter is ignored if the COMPOSITE, the MATERIAL, or the USER parameter is specified.

The following parameters are optional, mutually exclusive, and used only if the section is not defined by its general stiffness on the data lines:

COMPOSITE

Include this parameter to indicate that the shell is composed of layers with different linear elastic material behavior.

MATERIAL

Set this parameter equal to the name of the single linear elastic material of which the shell is made.

USER

This parameter cannot be used with continuum shell elements. See *Using a General Shell Section to Define the Section Behavior* for use of this option in linear perturbation analyses.

Include this parameter to indicate that the shell section stiffness is defined in user subroutine *UGENS* (Abaqus/Standard) or user subroutine *VUGENS* (Abaqus/Explicit).

The following parameters are optional, mutually exclusive, and can be used only in combination with the MATERIAL, the COMPOSITE, or the USER parameter:

NODAL THICKNESS

Include this parameter to indicate that the shell thickness should not be read from the data lines but should be interpolated from the thickness specified at the nodes with the **NODAL THICKNESS* option. For composite sections the total thickness is interpolated from the nodes, and the thicknesses of the layers specified on the data lines are scaled proportionally. This parameter is ignored for continuum shells.

SHELL THICKNESS

Set this parameter equal to the name of a distribution (*Distribution Definition*) to define spatially varying thickness. If this parameter is used for a noncomposite section, the thickness on the data line is ignored. For composite sections the total thickness is defined by the distribution, and the thicknesses of the layers specified on the data lines are scaled proportionally. This parameter is ignored for continuum shells.

The distribution used to define shell thickness must have a default value. The default thickness is used by any shell element assigned to the shell section that is not specifically assigned a value in the distribution.

The following optional parameters can be used only in combination with the USER parameter:

DELETE

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the number of the solution-dependent variable (of those defined with the VARIABLES parameter) that controls the deletion of the element.

I PROPERTIES

Set this parameter equal to the number of integer property values needed as data in user subroutine *UGENS* or *VUGENS*. The default is I PROPERTIES=0.

PROPERTIES

Set this parameter equal to the number of real (floating point) property values needed as data in user subroutine *UGENS* or *VUGENS*. The default is PROPERTIES=0.

TVS DAMAGE

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the number of the solution-dependent variable (of those defined with the VARIABLES parameter) that controls the transverse shear damage variable of the element.

UNSYMM

Include this parameter if the section stiffness matrices are not symmetric. This parameter will invoke the unsymmetric equation solution capability.

VARIABLES

Set this parameter equal to the number of solution-dependent variables that must be stored for the section. The default is VARIABLES=1.

Optional parameter for use when the MATERIAL, the COMPOSITE, and the USER parameters are omitted:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the scaling moduli, in addition to temperature. If this parameter is omitted, it is assumed that the moduli are constant or depend only on temperature.

Data line if the MATERIAL parameter is included:

First (and only) line:

1. Shell thickness. This value is ignored if the NODAL THICKNESS or SHELL THICKNESS parameter is included.

Data lines if the COMPOSITE parameter is included:

First line:

1. Positive scalar value defining layer thickness or the name of a distribution (*Distribution Definition*) that defines spatially varying layer thicknesses. A distribution for composite layer thickness can be used only for conventional shell elements (not continuum shell elements). The layer thickness is modified if the NODAL THICKNESS or SHELL THICKNESS parameter is included.
2. This field is not used (any value given will be ignored). It is included for compatibility with the **SHELL SECTION* option.
3. Name of the material forming this layer.
4. Orientation angle, ϕ, or the name of a distribution (*Distribution Definition*) that defines spatially varying orientation angles. Orientation angles (in degrees) are measured positive counterclockwise relative to the orientation definition given with the ORIENTATION parameter. If the ORIENTATION parameter is not included, ϕ is measured relative to the default shell local directions (see *Conventions*).
5. Name of the ply. Required only for composite layups defined in Abaqus/CAE.

Repeat this data line as often as necessary to define the layers of the shell. The order of the laminated shell layers with respect to the positive direction of the shell normal is defined by the order of the data lines. If the SYMMETRIC parameter is included, specify only half the layers, from the bottom layer to the midplane.

Data lines to define the shell section directly if the MATERIAL, the COMPOSITE, and the USER parameters are omitted:

First line:

1. Symmetric half of the $[\mathbf{D}]$ matrix in the order D_{11}, D_{12}, D_{22}, D_{13}, D_{23}, D_{33}, D_{14},..., D_{44}, D_{15},..., D_{55},..., D_{16},..., D_{66}

Repeat this data line three times. Enter 21 entries total, 8 per line on the first two lines and 5 on the third line.

Second line (optional):

1. $\{\mathbf{F}\}$ vector (F_1, F_2, ..., F_6).

Third line (optional):

1. Y, scaling modulus for $[\mathbf{D}]$. The default is 1.0.
2. α, scaling modulus for $[\mathbf{F}]$. The default is 0.0.
3. θ, temperature for these values of Y and α.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define Y and α as functions of temperature and other predefined field variables.

Data lines to define spatially varying shell section stiffness with a distribution if the MATERIAL, the COMPOSITE, and the USER parameters are omitted:

First line:

1. Distribution name. The data in the distribution define the symmetric half of the $[\mathbf{D}]$ matrix.

Second line (optional):

1. $\{\mathbf{F}\}$ vector (F_1, F_2, ..., F_6).

Third line (optional):

1. Y, scaling modulus for $[\mathbf{D}]$. The default is 1.0.
2. α, scaling modulus for $[\mathbf{F}]$. The default is 0.0.
3. θ, temperature for these values of Y and α.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define Y and α as functions of temperature and other predefined field variables.

Data lines if the USER parameter is included:

First line:

1. Shell thickness. This value is ignored if the NODAL THICKNESS or SHELL THICKNESS parameters are included.

Second line:

1. Enter the data to be used as properties in user subroutine *UGENS* or *VUGENS*. Enter all floating point values first, followed immediately by the integer values.

Repeat this data line as often as necessary to define the properties required in user subroutine *UGENS* or *VUGENS*. Enter eight values per line for both real and integer values.

*SHELL SECTION: Specify a shell cross-section.

This option is used to specify a shell cross-section.NODAL THICKNESS

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- *About Shell Elements*
- *Using a Shell Section Integrated during the Analysis to Define the Section Behavior*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the shell elements for which the section behavior is being defined.

Required, mutually exclusive parameters:

COMPOSITE

Include this parameter if the shell is made up of several layers of material.

MATERIAL

Set this parameter equal to the name of the material of which the shell is made.

Optional parameters:

CONTROLS

In an Abaqus/Explicit analysis, set this parameter equal to the name of a section controls definition (see *Section Controls*) to be used to specify the second-order accurate element formulation option, a nondefault hourglass control formulation option, or scale factors.

In an Abaqus/Standard analysis, set this parameter equal to the name of a section controls definition to be used to specify the enhanced hourglass control formulation (see *Section Controls*) or to be used in a subsequent Abaqus/Explicit import analysis.

DENSITY

Set this parameter equal to a mass per unit surface area of the shell.

If this parameter is used, the mass of the shell includes a contribution from this parameter in addition to any contribution from the material definition.

LAYUP

This parameter is relevant only when the COMPOSITE parameter is used.

Set this parameter equal to the name of a composite layup. This name identifies the composite layup that contains the shell section.

NODAL THICKNESS

Include this parameter to indicate that the shell thickness should not be read from the data lines but should be interpolated from the thickness specified at the nodes with the **NODAL THICKNESS* option. For composite sections the total thickness is interpolated from the nodes and the thicknesses of the layers specified on the data lines are scaled proportionally. This parameter is ignored for continuum shells.

The NODAL THICKNESS and SHELL THICKNESS parameters are mutually exclusive.

OFFSET

Include this parameter to define the distance (as a fraction of the shell thickness) from the shell midsurface to the reference surface (containing the nodes of the element). This parameter accepts positive or negative values, the labels SPOS or SNEG, or the name of a distribution. See *Distribution Definition*.

Positive values of the offset are in the positive normal direction (see *About Shell Elements*). When OFFSET=0.5 (or SPOS), the top surface of the shell is the reference surface. When OFFSET=−0.5 (or SNEG), the bottom surface of the shell is the reference surface. The default is OFFSET=0, which indicates that the middle surface of the shell is the reference surface.

In both Abaqus/Standard and Abaqus/Explicit analyses a spatially varying offset can be specified by setting OFFSET equal to the name of a distribution. The distribution used to define the shell offset must have a default value. The default offset is used by any shell element assigned to the shell section that is not specifically assigned a value in the distribution.

ORIENTATION

Set this parameter equal to the name of an orientation definition (*Orientations*) to be used with material calculations in this definition of shell section behavior. This orientation will be used for material calculations and stress output in the individual layers, for the section forces output, and for the transverse shear stiffness.

It is possible to use a different orientation definition for material calculations in the individual layers of a composite shell by referencing an orientation definition or giving an orientation angle (in degrees, positive counterclockwise relative to the shell local directions) on each layer definition data line. Any layer definition line that does not have an orientation reference or an angle specified will use the orientation defined here. While an orientation defined with a distribution can be used to specify the overall orientation for the shell section, an orientation defined with a distribution cannot be specified on the layers of a composite shell.

POISSON

Include this parameter to define the shell thickness direction behavior.

Set this parameter equal to a nonzero value to cause the thickness direction strain under plane stress conditions to be a linear function of the membrane strains. The value of the POISSON parameter must be between −1.0 and 0.5.

Set POISSON=ELASTIC to automatically select this parameter value based on the initial elastic part of the material definition.

Set POISSON=MATERIAL in an Abaqus/Explicit analysis to cause the thickness direction strain under plane stress conditions to be a function of the membrane strains and the in-plane material properties.

In Abaqus/Standard the default is POISSON=0.5; in Abaqus/Explicit the default is POISSON=MATERIAL.

SECTION INTEGRATION

Set SECTION INTEGRATION=SIMPSON (default) to use Simpson's rule for the shell section integration.

Set SECTION INTEGRATION=GAUSS to use Gauss quadrature for the shell section integration. Gauss quadrature cannot be used for heat transfer or thermally coupled shell elements.

SHELL THICKNESS

Set this parameter equal to the name of a distribution (*Distribution Definition*) to define spatially varying thickness. If this parameter is used for a noncomposite section the thickness on the data line is ignored. For composite sections the total thickness is defined by the distribution and the thicknesses of the layers specified on the data lines are scaled proportionally. This parameter is ignored for continuum shells.

The distribution used to define shell thickness must have a default value. The default thickness is used by any shell element assigned to the shell section that is not specifically assigned a value in the distribution.

The NODAL THICKNESS and SHELL THICKNESS parameters are mutually exclusive.

STACK DIRECTION

This parameter is relevant only for continuum shells.

Set this parameter equal to 1, 2, 3, or ORIENTATION to define the continuum shell stack or thickness direction. Specify one of the numerical values to select the corresponding isoparametric direction of the element as the stack or thickness direction. The default is STACK DIRECTION=3.

If STACK DIRECTION=ORIENTATION, the ORIENTATION parameter is also required.

To obtain a desired thickness direction, the appropriate numerical value for the STACK DIRECTION parameter depends on the element connectivity. For a mesh-independent specification, use STACK DIRECTION=ORIENTATION. If the orientation assigned to the ORIENTATION parameter is defined with a distribution (*Distribution Definition*), STACK DIRECTION=ORIENTATION is not supported.

SYMMETRIC

This parameter is relevant only when the COMPOSITE parameter is used.

Include this parameter if the layers in the composite shell are symmetric about a central core. This parameter cannot be used if a spatially varying thickness or orientation angle is defined on any composite layer using a distribution (*Distribution Definition*).

TEMPERATURE

Use this parameter to select the mode of temperature and field variable input used on the **FIELD*, the **INITIAL CONDITIONS*, or the **TEMPERATURE* options.

Omit the TEMPERATURE parameter to define the predefined field by its magnitude on the reference surface of the shell and its gradient through the thickness.

Set TEMPERATURE=*n*, where *n* (1) is the number of predefined field variable points in the shell or in each layer if the COMPOSITE parameter is used, to define the predefined field at *n* equally spaced points through each layer of the shell section.

In a heat transfer analysis step or a coupled temperature-displacement analysis step when the **FIELD* option is used to specify values of predefined field variables, the TEMPERATURE parameter is required to specify the number of field variable points in the shell. The number of temperature points in the shell with temperature degrees of freedom is defined by the number of integration points specified on the data lines.

THICKNESS MODULUS

This parameter is relevant only for continuum shells.

Set this parameter equal to the effective thickness modulus.

Set THICKNESS MODULUS=ELASTIC to compute the effective thickness modulus based on the in-plane elastic properties. In this case, the effective thickness modulus is set to the initial tensile modulus in the out-of-plane direction. If you use this option, you must define the shell thickness direction behavior using POISSON=ELASTIC.

Data line to define a homogeneous shell (the MATERIAL parameter is included):

First (and only) line:

1. Shell thickness. This value is ignored if the NODAL THICKNESS or SHELL THICKNESS parameters are included.
2. Number of integration points to be used through the shell section. The default is five points if Simpson's rule is used and three points if Gauss quadrature is used. The number of integration points must be an odd number for Simpson's rule and is equal to the number of temperature degrees of freedom at a node of the element if this section is associated with heat transfer or coupled temperature-displacement elements. The maximum number of points for Simpson's rule is 99, and in the case of heat transfer or coupled temperature-displacement elements it is 19. This number must be at least 2 and less than or equal to 15 for Gauss quadrature. For Simpson's rule it must be at least 3, except in a pure heat transfer analysis, where the number of integration points can be 1 for a constant temperature through the shell thickness.

Data lines to define a composite shell (the COMPOSITE parameter is included):

First line:

1. For conventional shell elements enter a positive scalar value defining the actual layer thickness or the name of a distribution (*Distribution Definition*) that defines spatially varying layer thicknesses. The layer thickness is modified if the NODAL THICKNESS or SHELL THICKNESS parameter is included.

   For continuum shell elements enter a positive scalar value defining the relative thickness of this layer with respect to the other layers. The thickness ratios for the layers need not be given in physical units, nor is it required that the layer relative thicknesses add to one.
2. Number of integration points to be used through the layer. The default is three points if Simpson's rule is used and two points if Gauss quadrature is used. The number of integration points must be an odd number for Simpson's rule, and it determines the number of temperature degrees of freedom at a node of the element if this section is associated with heat transfer or coupled temperature-displacement elements. The maximum number of points for Simpson's rule is 99, and in the case of heat transfer or coupled temperature-displacement elements it is 19. This number must be less than or equal to 15 for Gauss quadrature.
3. Name of the material forming this layer.
4. Name of the orientation to be used with this layer, an orientation angle, ϕ, or the name of a distribution (*Distribution Definition*) that defines spatially varying orientation angles. If the name of an orientation is used, the orientation cannot be defined with distributions. Orientation angles (in degrees) are measured positive counterclockwise relative to the orientation definition given with the ORIENTATION parameter. If the ORIENTATION parameter is not included, ϕ is measured relative to the default shell local directions (see *Orientations*).
5. Name of the ply. Required only for composite layups defined in Abaqus/CAE. You must specify ply names that are unique throughout the entire model to ensure the correct display of ply-based results.

Repeat this data line as often as necessary. Use one data line for each layer of the shell. The order of the laminated shell layers with respect to the positive direction of the shell normal is defined by the order of the data lines. If the SYMMETRIC parameter is included, specify only half the layers, from the bottom layer to the midplane.

*SHELL TO SOLID COUPLING: Define a surface-based coupling between a shell edge and a solid face.

This surface-based option allows for a transition from shell element modeling to solid element modeling in a three-dimensional analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Interaction module

References:

- *Element-Based Surface Definition*
- *Node-Based Surface Definition*
- *Shell-to-Solid Coupling*

Required parameters:

CONSTRAINT NAME

Set this parameter equal to a label that will be used to refer to this constraint.

Optional parameters:

INFLUENCE DISTANCE

Set this parameter equal to the perpendicular distance from the edge-based surface within which all nodes or element facets on the solid surface (depending on the solid surface type) must lie to be included in the coupling constraint. The default value is half the thickness of the shell that was used to define the edge-based surface.

POSITION TOLERANCE

Set this parameter equal to the distance within which nodes on the edge-based surface must lie from the solid surface to be included in the coupling definition. The default tolerance is 5% of the length of a typical facet on the shell edge.

Data lines to define the surfaces forming the coupling definitions:

First line:

1. The edge-based surface name.
2. The solid surface name.

Repeat this data line as often as necessary to define all the surfaces forming the coupling definition. Each data line defines a pair of surfaces that will be coupled.

*SIMPEDANCE: Define impedances of acoustic surfaces.

This option is used to provide surface impedance information or nonreflecting boundaries for acoustic and coupled acoustic-structural analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *Coupled Acoustic-Structural Analysis*
- *Acoustic and Shock Loads*
- **IMPEDANCE*
- **IMPEDANCE PROPERTY*

Required, mutually exclusive parameters:

PROPERTY

Set this parameter equal to the name of the **IMPEDANCE PROPERTY* option defining the table of impedance values to be used.

NONREFLECTING

Set NONREFLECTING=PLANAR (default) to specify the impedance corresponding to that of a normal incidence plane wave.

Set NONREFLECTING=IMPROVED to specify the impedance corresponding to that of a plane wave at an arbitrary angle of incidence. This parameter can be used only for transient dynamics.

Set NONREFLECTING=CIRCULAR to specify a radiation condition appropriate for a circular boundary in two dimensions or a right circular cylinder in three dimensions.

Set NONREFLECTING=SPHERICAL to specify a radiation condition appropriate for a spherical boundary.

Set NONREFLECTING=ELLIPTICAL to specify a radiation condition appropriate for an elliptical boundary in two dimensions or a right elliptical cylinder in three dimensions.

Set NONREFLECTING=PROLATE SPHEROIDAL to specify a radiation condition appropriate for a prolate spheroidal boundary.

Optional parameters:

OP

Set OP=MOD (default) to modify existing impedances or to define additional impedances.

Set OP=NEW if all existing impedances applied to the model should be removed. To remove only selected impedances, use OP=NEW and respecify all impedances that are to be retained.

Data line to define an impedance for PROPERTY, NONREFLECTING=PLANAR, or NONREFLECTING=IMPROVED:

First (and only) line:

1. Surface name.

Data line to define an absorbing boundary impedance for NONREFLECTING=CIRCULAR or NONREFLECTING=SPHERICAL:

First (and only) line:

1. Surface name.
2. r_1, the radius of the circle or sphere defining the absorbing boundary surface.

Data line to define an absorbing boundary impedance for NONREFLECTING=ELLIPTICAL or NONREFLECTING=PROLATE SPHEROIDAL:

First (and only) line:

1. Surface name.
2. The semimajor axis, *a*, of the ellipse or prolate spheroid defining the surface. *a* is 1/2 of the maximum distance between two points on the ellipse or spheroid, analogous to the radius of a circle or sphere.
3. The eccentricity, ϵ, of the ellipse or prolate spheroid. The eccentricity is the square root of one minus the square of the ratio of the minor axis, *b*, to the major axis, *a*: $\epsilon = \sqrt{1 - (b/a)^2}$.
4. Global *X*-coordinate of the center of the radiating surface.
5. Global *Y*-coordinate of the center of the radiating surface.
6. Global *Z*-coordinate of the center of the radiating surface.
7. *X*-component of the direction cosine of the major axis of the radiating surface. The components of this vector need not be normalized to unit magnitude.
8. *Y*-component of the direction cosine of the major axis of the radiating surface.
9. *Z*-component of the direction cosine of the major axis of the radiating surface.

*SIMPLE SHEAR TEST DATA: Used to provide simple shear test data.

This option is used to provide simple shear test data. It can be used only in conjunction with the **HYPERFOAM* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Hyperelastic Behavior in Elastomeric Foams*
- **HYPERFOAM*

There are no parameters associated with this option.

Data lines to specify simple shear test data for a hyperfoam:

First line:

1. Nominal shear stress, T_S.
2. Nominal shear strain, γ.
3. Nominal transverse stress, T_T (normal to edge with shear stress). This stress value is optional but strongly recommended. If given, a more accurate material response will be obtained.

Repeat this data line as often as necessary to give the stress-strain data.

*SLIDE LINE: Specify slide line surfaces on which deformable structures may interact.

This option is relevant only for slide line and tube-to-tube contact elements. It is used to define the slide line and to specify which set of contact elements interacts with it.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Assembly

Abaqus/CAE: Unsupported; the *Interaction module* uses surface-based contact.

References:

- *Tube-to-Tube Contact Elements*
- *Slide Line Contact Elements*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the slide line or tube-to-tube contact elements that interact with the slide line being defined.

TYPE

Set TYPE=LINEAR to define a slide line made up of linear segments. This parameter should be used when first-order elements make up the model.

Set TYPE=PARABOLIC to define a slide line made up of parabolic segments. This parameter should be used when second-order elements make up the model. In this case the slide line should consist of an odd number of nodes, where the odd nodes on the slide line are associated with the ends of parabolic segments.

Optional parameters:

EXTENSION ZONE

Set this parameter equal to a fraction of the end segment length by which either end of an open slide line is to be extended to avoid numerical round off errors associated with contact modeling. The value given must lie between 0.0 and 0.2. The default value is 0.1.

GENERATE

Include this parameter to allow incremental generation of node numbers along a slide line.

SMOOTH

Set this parameter equal to the smoothing fraction, f, to round discontinuities between line segments of a slide line. The default is 0. The limit is $f < 0.5$.

Data lines if the GENERATE parameter is omitted:

First line:

1. First node number on this slide line.
2. Second node number on this slide line.
3. Third node number on this slide line.

4. Etc.

Repeat this data line as often as necessary to specify the nodes forming the slide line. Enter up to 16 integer values per line.

Data lines if the GENERATE parameter is included:

First line:

1. First node number of the series.
2. Last node number of the series.
3. Increment between node numbers.

Repeat this data line as often as necessary to specify the nodes forming the slide line.

*SLOAD: Apply loads to a substructure.

This option is used to activate a substructure load case defined by the **SUBSTRUCTURE LOAD CASE* option.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

References:

- *Using Substructures*
- **SUBSTRUCTURE LOAD CASE*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name given to an amplitude defined by the **AMPLITUDE* option (*Amplitude Curves*). This amplitude defines the time variation of the load case (**SUBSTRUCTURE LOAD CASE*) magnitude throughout the step. If this parameter is omitted, the default amplitude is that defined in *Defining an Analysis* for the particular procedure used in the step.

OP

Set OP=MOD (default) to modify or add to the currently active **SLOAD*s.

Set OP=NEW if all existing **SLOAD*s applied to the model should be removed and new ones possibly defined.

Optional, mutually exclusive parameters for steady-state dynamic analysis:

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the loading.

REAL

Include this parameter (default) to define the real (in-phase) part of the loading.

Data lines to define the loading:

First line:

1. Element number or element set name. The elements must be substructures.
2. Load case name (as defined on the associated **SUBSTRUCTURE LOAD CASE* option).
3. Load magnitude. This magnitude is a multiplier for the load case magnitude defined under the **SUBSTRUCTURE LOAD CASE* option. It can be adjusted throughout the step by the amplitude specification in the usual way.

Repeat this data line as often as necessary to define the loadings on the substructures.

*SOFT ROCK HARDENING: Specify hardening for the soft rock plasticity model.

This option is used to define piecewise linear hardening/softening of the soft rock plasticity yield surface. It can be used only in conjunction with the **SOFT ROCK PLASTICITY* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- **SOFT ROCK PLASTICITY*
- *Soft Rock Plasticity Model*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies, in addition to temperature, included in the definition of the hydrostatic pressure stress. If this parameter is omitted, the hydrostatic pressure stress may depend only on the volumetric plastic strain and, possibly, on the temperature. See *Material Data Definition* for more information.

SR

This parameter is relevant only when it is used in conjunction with the **SOFTENING REGULARIZATION* option.

Set SR=ON (default) to use softening regularization for the soft rock hardening data.

Set SR=OFF if softening regularization is not required.

TYPE

Set TYPE=COMPRESSION (default) to define the hardening behavior by providing the hydrostatic compression yield stress, p_c, as a function of volumetric plastic strain.

Set TYPE=TENSION to define the hardening behavior by providing the hydrostatic tension yield stress, p_t, as a function of volumetric plastic strain.

Compression hardening must always be defined. Tensile hardening is optional and can be specified, if needed, by repeating the **SOFT ROCK HARDENING* option with TYPE=TENSION.

Data lines to define hardening for soft rock plasticity:

First line:

1. Value of the hydrostatic pressure stress at yield, p_c or p_t. p_c is given as a positive value and must increase with increasing plastic strain. p_t can be zero or negative and must decrease with increasing plastic strain.
2. Absolute value of the corresponding compressive volumetric plastic strain. The first tabular value must always be zero.
3. Temperature.
4. First field variable.

5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the yield surface size on volumetric plastic strain and, if needed, on temperature and other predefined field variables.

*SOFT ROCK PLASTICITY: Specify the soft rock plasticity model.

This option is used to specify the plastic part of the material behavior for elastic-plastic materials that use the soft rock plasticity model.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- **SOFT ROCK HARDENING*
- *Soft Rock Plasticity Model*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies, in addition to temperature, included in the definition of the soft rock parameters. If this parameter is omitted, the soft rock parameters may depend only on temperature. See *Material Data Definition* for more information.

ECCENTRICITY

This parameter is used to define the eccentricity, ϵ. The default value of ϵ is 0.001.

Data lines to define soft rock plasticity:

First line:

1. Friction angle, β, in the p–t plane. Give the value in degrees. The friction angle can range from $0° \le \beta \le 89.9°$.
2. Dilation angle, ψ, in the p–t plane. Give the value in degrees. The dilation angle can range from $0° \le \psi \le 89.9°$.
3. Material parameter, n_y. If this value is left blank or set to zero, a value of 1.6 is assumed.
4. Deviatoric plane correction parameter, f_0. This value must be greater than or equal to zero.
5. Deviatoric plane correction parameter, f_1. This value must be greater than or equal to zero.
6. Deviatoric plane correction parameter, α. This value must be greater than or equal to zero.
7. Absolute value of the initial compressive volumetric plastic strain, $\varepsilon^{pl}_{vol}\big|_0$.
8. Temperature.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than zero):

1. First field variable.
2. Second field variable.
3. Third field variable.
4. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the soft rock parameters on temperature and other predefined field variables.

*SOFTENING REGULARIZATION: Specify softening regularization for the clay plasticity model and the soft rock plasticity model.

This option is used to specify a regularization scheme to mitigate potential mesh dependence of results in models that exhibit strain localization with increasing plastic deformation. It can be used only in conjunction with the *CLAY PLASTICITY or the *SOFT ROCK PLASTICITY option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Critical State (Clay) Plasticity Model*
- *CLAY PLASTICITY
- *SOFT ROCK PLASTICITY

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies, in addition to temperature, included in the definition of softening regularization. If this parameter is omitted, the softening regularization behavior may depend only on temperature.

TYPE

Set TYPE=POWER LAW (default) to define the regularization behavior in terms of the crack band length, $l_c^{(m)}$; exponent, n_r; and bound on the magnitude of regularization, f_{max}.

Data lines to define softening regularization:

First line:

1. Crack band length, $l_c^{(m)}$. This value must be greater than zero.
2. Exponent, n_r. This value must be greater than zero.
3. Bound on the magnitude of regularization, f_{max}. This value must be greater than zero.
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the softening regularization data on temperature and other predefined field variables.

*SOILS: Effective stress analysis for fluid-filled porous media.

This option is used to specify transient (consolidation) or steady-state response analysis of partially or fully saturated fluid-filled porous media.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Coupled Pore Fluid Diffusion and Stress Analysis*
- *Rate-Dependent Plasticity: Creep and Swelling*

Optional parameters:

ALLSDTOL

Include this parameter to indicate that an adaptive automatic damping algorithm will be activated in this step. Set this parameter equal to the maximum allowable ratio of the stabilization energy to the total strain energy. The initial damping factor is specified via the STABILIZE parameter or the FACTOR parameter. This damping factor will then be adjusted through the step based on the convergence history and the value of ALLSDTOL. If this parameter is set equal to zero, the adaptive automatic damping algorithm is not activated; a constant damping factor will be used throughout the step. If this parameter is included without a specified value, the default value is 0.05. If this parameter is omitted but the STABILIZE parameter is included with the default value of dissipated energy fraction, the adaptive automatic damping algorithm will be activated automatically with ALLSDTOL=0.05.

This parameter must be used in conjunction with the STABILIZE parameter (see *Solving Nonlinear Problems*).

CETOL

This parameter will invoke automatic time incrementation. If the UTOL, DELTMX, and CETOL parameters are omitted, fixed time increments will be used.

This parameter is meaningful only when the material response includes time-dependent creep behavior; CETOL controls the accuracy of the creep integration. Set this parameter equal to the maximum allowable difference in the creep strain increment calculated from the creep strain rates at the beginning and at the end of the increment (see *Rate-Dependent Plasticity: Creep and Swelling*).

The tolerance can be calculated by choosing an acceptable stress error tolerance and dividing by a typical elastic modulus.

CONSOLIDATION

Include this parameter to choose transient (consolidation) analysis. Omit this parameter to choose steady-state analysis.

CONTINUE

Set CONTINUE=NO (default) to specify that this step will not carry over the damping factors from the results of the preceding general step. In this case the initial damping factors will be recalculated based on the declared damping intensity and on the solution of the first increment of the step or can be specified directly.

Set CONTINUE=YES to specify that this step will carry over the damping factors from the end of the immediately preceding general step.

This parameter must be used in conjunction with the ALLSDTOL and the STABILIZE parameters.

DELTMX

This parameter will invoke automatic time incrementation. Set this parameter equal to the maximum temperature change allowed within an increment. Abaqus/Standard will restrict the time step to ensure that this value is not exceeded at any node during any increment of the step. If this parameter, the CETOL parameter, and the UTOL parameter are all omitted in a transient analysis, fixed time increments will be used, with a constant time increment equal to the initial time increment.

END

This parameter is meaningful only for transient analysis. Set END=PERIOD (default) to analyze a specified time period. Set END=SS to end the step when steady state is reached.

CREEP

Set CREEP=NONE to specify that there is no creep or viscoelastic response occurring during this step even if creep or viscoelastic material properties have been defined.

FACTOR

Set this parameter equal to the damping factor to be used in the automatic damping algorithm (see *Solving Nonlinear Problems*) if the problem is expected to be unstable due to local instabilities and the damping factor calculated by Abaqus/Standard is not suitable. This parameter must be used in conjunction with the STABILIZE and CONSOLIDATION parameters and overrides the automatic calculation of the damping factor based on a value of the dissipated energy fraction.

HEAT

This parameter is relevant if there are regions in the model that use coupled temperature–pore pressure elements; it specifies whether heat transfer effects are to be modeled in these regions.

This parameter is not relevant if only coupled pore pressure–displacement elements are used in a model.

Set HEAT=YES (default) to specify that heat transfer effects are to be modeled in these regions. In this case Abaqus/Standard solves the heat transfer equation in conjunction with the mechanical equilibrium and the fluid flow continuity equations.

Set HEAT=NO to specify that heat transfer will not be modeled in these regions.

SLURRY FLOW

Set SLURRY FLOW=YES (default) to activate the solution of the slurry continuity equation in a step. In this case, the slurry concentration field will evolve with time within the step..

Set SLURRY FLOW=NO to deactivate the solution of the slurry continuity equation in a step. In this case, the slurry concentration field will not evolve in time within the step.

This parameter is relevant only when the slurry concentration degree of freedom is active.

SLURRY MAX

Set this parameter equal to the maximum allowed value of the slurry concentration at a material point. The default value is 0.67.

This parameter is relevant only when the slurry concentration degree of freedom is active.

SLURRY MIN

Set this parameter equal to the minimum allowed value of the slurry concentration at a material point. The default value is zero.

This parameter is relevant only when the slurry concentration degree of freedom is active.

STABILIZE

Include this parameter to use automatic stabilization if the problem is expected to be unstable due to local instabilities. Set this parameter equal to the dissipated energy fraction of the automatic damping algorithm (see *Solving Nonlinear Problems*). If this parameter is omitted, the stabilization algorithm is not activated. If this parameter is included without a specified value, the default value of the dissipated energy fraction is 2×10^{-4} and the adaptive automatic damping algorithm will be activated by default with ALLSDTOL =0.05 in this step; set ALLSDTOL=0 to deactivate the adaptive automatic damping algorithm. If the FACTOR parameter is used, any value of the dissipated energy fraction will be overridden by the damping factor.

This parameter can be used only in conjunction with the CONSOLIDATION parameter.

UTOL

This parameter will invoke automatic time incrementation. If the UTOL, DELTMX, and the CETOL parameters are omitted, fixed time increments will be used.

Set this parameter equal to the maximum pore pressure change permitted in any increment (in pressure units) in a transient consolidation analysis. Abaqus/Standard will restrict the time step to ensure that this value will not be exceeded at any node (except nodes with boundary conditions) during any increment of the analysis.

Set this value equal to any nonzero value in a steady-state analysis (to activate automatic time incrementation).

Data line to define incrementation for a soils analysis:

First (and only) line:

1. Initial time increment. This value will be modified as required if the automatic time stepping scheme is used; it will be used as a constant time increment otherwise.
2. Time period. If END=SS is chosen, the step ends after this time period or when steady-state conditions are reached, whichever comes first.
3. Minimum time increment allowed. If Abaqus/Standard finds it needs a smaller time increment than this value, the analysis is terminated. If this entry is zero, a default value of the smaller of the suggested initial time increment or 10^{-5} times the total time period is assumed. Only used for automatic time incrementation.
4. Maximum time increment allowed. If this value is not specified, the upper limit is the total time period. Only used for automatic time incrementation.
5. The rate of change of pore pressure with time, used to define steady state: only needed if END=SS is chosen. When all nodal wetting liquid pressures are changing at rates below this value, the solution terminates.
6. The rate of change of temperature with time, used to define steady state: only needed if END=SS is chosen. When all nodal temperatures are changing at rates below this value, the solution terminates.

*SOLID ELECTROLYTE THEORY: Activate use of the electrochemistry framework to model a solid electrolyte.

This option is used in a material data block to activate the usage of the electrochemistry framework to model a solid electrolyte.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

• *Modeling Solid Electrolytes and Solid-State Batteries*

There are no parameters or data lines associated with this option.

*SOLID SECTION: Specify element properties for solid, infinite, acoustic, particle, and truss elements.

This option is used to define properties of solid (continuum) elements, infinite elements, acoustic finite and infinite elements, particle elements, and truss elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- *Solid (Continuum) Elements*
- *Infinite Elements*
- *Continuum Particle Elements*
- *Truss Elements*

Required parameters:

COMPOSITE

This parameter applies only to Abaqus/Standard analyses.

This parameter can be used only with three-dimensional brick solid elements, wedge elements, and continuum solid shell elements that have only displacement degrees of freedom. Include this parameter if the solid is made up of several layers of material.

The COMPOSITE and MATERIAL parameters are mutually exclusive.

ELSET

Set this parameter equal to the name of the element set containing the elements for which the material behavior is being defined.

MATERIAL

Set this parameter equal to the name of the material to be used with these elements.

The COMPOSITE and MATERIAL parameters are mutually exclusive.

REF NODE

This parameter is required only for generalized plane strain elements and acoustic infinite elements; it is ignored for all other element types.

Set this parameter equal to either the node number of the reference node or the name of a node set containing the reference node. If the name of a node set is chosen, the node set must contain exactly one node.

Required parameter for anisotropic materials; optional parameter for isotropic materials:

ORIENTATION

Set this parameter equal to the name of an orientation definition (*Orientations*) to be used to define a local coordinate system for material calculations in the elements in this set. This parameter is required when the material is anisotropic.

For a composite solid this orientation, together with the orientation angle specified on the layer data lines, can also be used to define the material orientations in the individual layers. Alternatively, a material orientation can be specified by referencing an orientation definition on each layer data line. In this case the reference given on the ORIENTATION parameter is ignored. Any layer definition line that does not have an orientation reference or an angle specified will use the section orientation defined on the keyword line.

Optional parameters:

CONTROLS

In an Abaqus/Explicit analysis, set this parameter equal to the name of a section controls definition (see *Section Controls*) to be used to specify a nondefault hourglass control formulation option or scale factor. The **SECTION CONTROLS* option can be used to select the hourglass control and order of accuracy of the formulation for two- and three-dimensional solid elements and to select the kinematic formulation for 8-node brick elements.

In an Abaqus/Standard analysis, set this parameter equal to the name of a section controls definition (see *Section Controls*) to be used to specify the enhanced hourglass control formulation or to be used in a subsequent Abaqus/Explicit import analysis.

LAYUP

This parameter is relevant only when the COMPOSITE parameter is used.

Set this parameter equal to the name of a composite layup (see *Composite layups*). Abaqus/CAE uses this name to identify the composite layup that contains the solid section.

ORDER

This parameter can be used only with acoustic infinite elements in Abaqus/Explicit. It defines the number of ninth-order polynomials that will be used to resolve the variation of the acoustic field in the infinite direction. Set this parameter equal to *N* to indicate that the first *N* members of the set of ninth-order polynomials are to be used. The default is ORDER=10, which is the value always used in Abaqus/Standard.

STACK DIRECTION

This parameter applies only to Abaqus/Standard analyses.

This parameter can be used only with composite elements. It defines the stacking direction with respect to a pair of element faces. For brick elements, set this parameter equal to 1, 2, or 3. For wedge elements and continuum solid shell elements, set this parameter equal to 3. The default is STACK DIRECTION=3.

SYMMETRIC

This parameter is relevant only when the COMPOSITE parameter is used.

Include this parameter if the layers in the composite solid are symmetric about a central core. This parameter cannot be used if spatially varying orientation angles are defined on any composite layer using distributions (see *Distribution Definition*).

Data line to define homogeneous solid elements, infinite elements, acoustic elements, particle, or truss elements:

First (and only) line:

1. Enter any attribute values required. The default for the first attribute is 1.0. See the description in *About the Element Library* of the element type being used for a definition of the data required.

Data lines to define a composite solid:

First line:

1. Layer thickness or the name of a distribution that defines spatially varying layer thickness (see *Distribution Definition*). The layer thickness will be adjusted such that the sum of the layer thicknesses corresponds to the element length in the stack direction.
2. Number of integration points to be used through the layer. This number must be an odd number. The default is one integration point.
3. Name of the material forming this layer.
4. Name of the orientation to be used with this layer, an orientation angle, ϕ, or in Abaqus/Standard the name of a distribution (see *Distribution Definition*) that defines spatially varying orientation angles. If the name of an orientation is used, the orientation cannot be defined with distributions. Orientation angles (in degrees) are measured positive counterclockwise relative to the local direction, which must be defined on the **ORIENTATION* definition. If the local directions for a composite solid section are defined with user subroutine *ORIENT* (see *ORIENT*), orientation angles defined on the data lines of the section definition are ignored.
5. Name of the ply. Required only for composite layups defined in Abaqus/CAE.

Repeat this data line as often as necessary to define the properties for each layer of the composite solid. If the SYMMETRIC parameter is included, specify only half the layers, from the bottom layer to the midplane.

*SOLUBILITY: Specify solubility.

This option is used to define the solubility for a material diffusing through a base material. It must be used in conjunction with the **DIFFUSIVITY* option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Solubility*
- **DIFFUSIVITY*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the solubility. If this parameter is omitted, it is assumed that the solubility is constant or that it depends only on temperature. See *Material Data Definition* for more information.

Data lines to define solubility:

First line:

1. Solubility. (Units of P.)
2. Temperature, θ.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the solubility as a function of temperature and other predefined field variables.

*SOLUTION TECHNIQUE: Specify alternative solution methods.

This option is used to specify the quasi-Newton method instead of the standard Newton method for solving nonlinear equations, to specify a separated solution scheme for coupled temperature-displacement and coupled thermal-electrical procedures, or to activate the linearized contact capability.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Fully Coupled Thermal-Stress Analysis*
- *Coupled Thermal-Electrical Analysis*
- *Solving Nonlinear Problems*
- *Linear Complementarity Problem (LCP) Solution Technique for Solving Contact Problems*
- *Convergence Criteria for Nonlinear Problems*

Required parameters:

TYPE

Set TYPE=QUASI-NEWTON to specify a quasi-Newton solution method.

Set TYPE=SEPARATED to specify that linearized equations for the individual fields in a fully coupled procedure are to be decoupled and solved separately for each field. This option can be specified only with the **COUPLED THERMAL-ELECTRICAL* procedure and the **COUPLED TEMPERATURE-DISPLACEMENT* procedure without the ELECTRICAL parameter.

Set TYPE=LCP CONTACT to activate the linearized contact capability to solve for contact status and contact stresses. This option can be specified only within a static perturbation step with small sliding, frictionless contact.

Optional parameters:

REFORM KERNEL

This parameter can be used only with TYPE=QUASI-NEWTON. Set this parameter equal to the number of quasi-Newton iterations allowed before the kernel matrix is reformed. The default is REFORM KERNEL=8.

Data line for TYPE=LCP CONTACT:

First (and only) line:

1. Initial gap distance to specify a zone within which potential contact constraints are exposed to the LCP solver. In the case of multiple contact pairs, this global setting applies to all of them. By default, a gap distance is calculated internally for each contact pair based on a characteristic facet size associated with the contact pair.
2. Scale factor for the initial gap distance specified in the previous entry or the internally computed (contact) pair-wise gap distances. The default is 1.0.

*SOLVER CONTROLS: Specify controls for the iterative and direct linear solvers.

This option is used to set the control parameters for both the direct and iterative linear equation solvers.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Direct Linear Equation Solver*
- *Iterative Linear Equation Solver*

Optional parameters:

CONSTRAINT OPTIMIZATION

Include this parameter to optimize the handling of hard contact and compressibility constraints associated with hybrid elements.

RESET

Include this parameter to reset all of the iterative solver controls to their default values. The option should have no data lines when this parameter is used.

If this parameter is omitted, only the iterative solver specified controls will be changed in the current step; the other controls will remain at their settings from previous steps.

Data line to define control parameters for the iterative solver:

First (and only) line:

1. Relative tolerance for convergence. The default is 10^{-6} for all linear perturbation and general nonlinear procedures.
2. Maximum number of iterations allowed. The default is 500.
3. ILU factorization fill-in level. The default is 0; the maximum is 3. This parameter applies only to porous media (soils) analyses.

*SORPTION: Define absorption and exsorption behavior.

This option is used to define absorption and exsorption behaviors of a partially saturated porous medium in the analysis of coupled wetting liquid flow and porous medium stress.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Sorption*

Optional parameters:

LAW

Set LAW=LOG to define the absorption or exsorption behavior by the analytical logarithmic form.

Set LAW=TABULAR (default) to define the absorption or exsorption behavior in tabulated form.

TYPE

Set TYPE=ABSORPTION (default) to define the absorption behavior.

Set TYPE=EXSORPTION to define the exsorption behavior (this must be a repeated use of the option for the same material).

Set TYPE=SCANNING to define the scanning line (this must be a repeated use of the option for the same material).

Data lines for TYPE=ABSORPTION or TYPE=EXSORPTION and LAW=TABULAR:

First line:

1. Pore pressure, u_w, with the condition $u_w \leq 0.0$. (Units of FL^{-2}.)
2. Saturation, s. This value must lie in the range $0.01 \leq s \leq 1.0$.

Repeat this data line as often as necessary to define the relationship between u_w and s from $s = 0^+$ to $s = 1.0$ in increasing values of s. At least two data lines must be specified.

Data line for TYPE=ABSORPTION or TYPE=EXSORPTION and LAW=LOG:

First (and only) line:

1. A. This value must be positive. (Dimensionless.)
2. B. This value must be positive. (Units of L^2F^{-1}.)
3. s_0. This value must lie in the range $0.01 \leq s_0 < s_1 < 0.9$. The default is 0.01.
4. s_1. This value must lie in the range $0.01 \leq s_0 < s_1 < 0.9$. The default is 0.01 plus a very small positive number (since s_1 cannot be equal to s_0).

Data line for TYPE=SCANNING:

First (and only) line:

1. Slope of the scanning line, $(du_w/ds)|_s$. This slope must be positive and larger than the slope of any segment of the absorption or exsorption behavior definitions.

*SPECIFIC HEAT: Define specific heat.

This option is used to specify a material's specific heat.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

• *Specific Heat*

Optional parameters:

DEPENDENCIES

This parameter applies only to Abaqus/Standard and Abaqus/Explicit analyses.

Set this parameter equal to the number of field variables included in the definition of specific heat. If this parameter is omitted, it is assumed that the specific heat is constant or depends only on temperature. See *Material Data Definition* for more information.

PORE FLUID

This parameter applies only to Abaqus/Standard analyses.

Include this parameter if the specific heat of the pore fluid in a porous medium is being defined.

SLURRY

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to define the specific heat of the carrier fluid and the solid particles. You can use this parameter only if you included the PORE FLUID parameter.

Data lines to specify a material's specific heat:

First line:

1. Specific heat per unit mass. (Units of $JM^{-1}\theta^{-1}$.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the specific heat as a function of temperature and other predefined field variables.

Data lines to specify a material's specific heat with the SLURRY parameter:

First line:

1. Specific heat per unit mass of the carrier fluid. (Units of $JM^{-1}\theta^{-1}$.)

2. Specific heat per unit mass of the proppant particles. (Units of $JM^{-1}\theta^{-1}$.)
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the specific heat of the carrier fluid and the proppant particles as a function of temperature and other predefined field variables.

*SPECTRUM: Define or create a response spectrum.

This option is used to convert a dynamic event into a spectrum to be used in a **RESPONSE SPECTRUM* analysis. You can define a spectrum using values of S as a function of frequency and damping or create a spectrum from a user-specified amplitude that describes a dynamic event.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: *Amplitude toolset*; supported only for defining a spectrum.

References:

- *Response Spectrum Analysis*
- **RESPONSE SPECTRUM*

Defining a spectrum using values of S as a function of frequency and damping

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the spectrum. This label is used to cross-reference a spectrum on the **RESPONSE SPECTRUM* option.

Optional parameters:

G

Set this parameter equal to the value of the acceleration of gravity. This parameter is used only when TYPE=G is specified.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

TYPE

Set TYPE=DISPLACEMENT, TYPE=VELOCITY, or TYPE=ACCELERATION (default) to define the units in which the spectrum is defined.

Set TYPE=G to enter an acceleration spectrum in g-units. The parameter G then defines the gravitational acceleration.

Data lines to define a spectrum using values of S as a function of frequency and damping:

First line:

1. Magnitude of the spectrum.
2. Frequency, in cycles per time, at which this magnitude is used.

3. Associated damping, given as ratio of critical damping.

Repeat this data line as often as necessary to define the spectrum at all frequencies at each damping value.

Creating a spectrum from a user-specified amplitude

Required parameters:

CREATE

Use this parameter to invoke spectrum building from a dynamic event provided by the user specified amplitude.

AMPLITUDE

Set this parameter equal to an amplitude name that describes the dynamic event used to create the spectrum.

NAME

Set this parameter equal to a label that will be used to refer to this created spectrum. This label is used to cross-reference a spectrum on the **RESPONSE SPECTRUM* option.

TIME INCREMENT

Set this parameter equal to the time increment for implicit time integration scheme used to build this spectrum.

Optional parameters:

G

Set this parameter equal to the value of the acceleration of gravity. This parameter is used when TYPE=G or EVENT TYPE=G is specified.

TYPE

Set TYPE=DISPLACEMENT, TYPE=VELOCITY, or TYPE=ACCELERATION (default) to define the units in which the spectrum is created.

Set TYPE=G to create an acceleration spectrum in *g*-units. The parameter G then defines the gravitational acceleration.

ABSOLUTE

This parameter is only relevant for acceleration spectrum. Use this parameter to request creation of absolute acceleration spectrum (default, if both ABSOLUTE and RELATIVE parameters are omitted).

RELATIVE

This parameter is only relevant for acceleration spectrum. Use this parameter to request creation of relative acceleration spectrum.

DAMPING GENERATE

Use this parameter to request generation of the list containing the fraction of critical damping for which spectra will be created.

EVENT TYPE

Set EVENT TYPE=DISPLACEMENT, EVENT TYPE=VELOCITY, or EVENT TYPE=ACCELERATION (default) to define the units in which the amplitude is defined.

Set EVENT TYPE=G to enter an acceleration amplitude in *g*-units. The parameter G then defines the gravitational acceleration.

FILE

Set this parameter equal to the name of the alternate output file containing the created spectrum data. Each line of this data file will contain magnitude of the spectrum, frequency, in cycles per time, at which this magnitude is used, and associated damping, given as ratio of critical damping. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the created spectrum can only be used in a subsequent response spectrum procedures of the same or restarted job.

Data lines to create a spectrum from a user-specified amplitude:

First line:

1. Lower limit of the frequency range, in cycles per time, at which the spectrum will start.
2. Upper limit of the frequency range, in cycles per time, at which the spectrum will end.
3. Total number of points in the frequency range at which the spectrum value is requested, including the end points. The minimum value is 2. If the value given is less than 2 (or omitted), the default value of 2 points is assumed.

Second line if the DAMPING GENERATE parameter is omitted:

1. List of damping values (fraction of critical damping).

Second line if the DAMPING GENERATE parameter is included:

1. Starting value of the fraction of critical damping.
2. End value of the fraction of critical damping.
3. Increment of the fraction of critical damping.

*SPH SURFACE BEHAVIOR: Define boundary surface interaction properties between SPH particles and Lagrangian surfaces.

This option is used to define boundary surface interaction properties between SPH particles and Lagrangian surfaces. The **SPH SURFACE BEHAVIOR* option must be used in conjunction with the **SURFACE INTERACTION* option.

Products: Abaqus/Explicit

Type: Model or history data

Level: Model, Step

References:

• *Smoothed Particle Hydrodynamics*

Optional parameters:

TYPE

Set this parameter equal to FREE SLIP (default) for nonsticky-type flow problems.

Set this parameter equal to NO SLIP for sticky-type flow problems.

There are no data lines associated with this option.

*SPRING: Define spring behavior.

This option is used to define the spring behavior for spring elements. It is also used to assign a structural damping factor to form the imaginary part of the spring stiffness matrix. The imaginary stiffness represents an element-level damping contribution to the frequency domain dynamic equations and to the time domain mode-based dynamic analyses that support nondiagonal damping (see *Modal dynamic analysis*).

In Abaqus/Standard analyses it is also used to define the spring behavior for ITS and JOINTC elements. If the **SPRING* option is being used to define part of the behavior of ITS or JOINTC elements, it must be used in conjunction with the **ITS* or **JOINT* options and the ELSET and ORIENTATION parameters should not be used.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: *Property module* and *Interaction module*; supported only for linear behavior independent of field variables. For nonlinear behavior or to include field variables, model connectors in the *Interaction module.*

References:

- *Springs*
- *Flexible Joint Element*
- *Tube Support Elements*

Required parameter if the behavior of spring elements is being defined:

ELSET

Set this parameter equal to the name of the element set containing the spring elements for which this behavior is being defined.

Optional parameters:

COMPLEX STIFFNESS

This parameter is relevant in direct-solution and subspace-based steady-state analyses and in mode-based analyses with **FREQUENCY*, DAMPING PROJECTION=ON in Abaqus/Standard that support nondiagonal damping.

Include this parameter to define both the real and imaginary parts of the stiffness. The imaginary part represents structural damping.

If this parameter is omitted, the default is real stiffness only.

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the real spring stiffness data, in addition to temperature. If this parameter is omitted, it is assumed that the spring stiffness is independent of field variables. See “Using the DEPENDENCIES parameters to define field variable dependence” in *Material Data Definition* for more information.

NONLINEAR

Include this parameter to define nonlinear spring behavior. Omit this parameter to define linear spring behavior.

ORIENTATION

This parameter applies only to Abaqus/Standard analyses.

If the option is being used to define the behavior of SPRING1 or SPRING2 elements, this parameter can be used to refer to an orientation definition so that the spring is acting in a local system. Set this parameter equal to the name of the **ORIENTATION* definition (*Orientations*).

RTOL

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the tolerance to be used for regularizing the material data. The default is RTOL=0.03. See *Material Data Definition* for a discussion of data regularization.

Data lines to define linear spring behavior for SPRINGA or ITS elements:

First line:

1. Enter a blank line.

Second line:

1. Spring stiffness (force per relative displacement).
2. Frequency (in cycles per time). Applicable only for **STEADY STATE DYNAMICS*, DIRECT; **STEADY STATE DYNAMICS*, SUBSPACE PROJECTION; and **STEADY STATE DYNAMICS* and **MODAL DYNAMIC* analyses that support nondiagonal damping.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the spring stiffness as a function of frequency, temperature, and other predefined field variables.

Data lines to define nonlinear spring behavior for SPRINGA or ITS elements:

First line:

1. Enter a blank line.

Second line:

1. Force.
2. Relative displacement.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the spring stiffness as a function of temperature and other predefined field variables.

Data lines to define linear spring behavior for SPRING1, SPRING2, or JOINTC elements:

First line:

1. Degree of freedom with which the springs are associated at their first nodes or, for JOINTC elements, the degree of freedom for which the spring behavior is being defined.
2. For SPRING2 elements give the degree of freedom with which the springs are associated at their second nodes.

If the ORIENTATION parameter is included on the **SPRING* option when defining spring elements or on the **JOINT* option when defining joint elements, the degrees of freedom specified here are in the local system defined by the **ORIENTATION* option referenced.

Second line:

1. Real spring stiffness (force per relative displacement).
2. Frequency (in cycles per time). Applicable only for **STEADY STATE DYNAMICS*, DIRECT; **STEADY STATE DYNAMICS*, SUBSPACE PROJECTION; and **STEADY STATE DYNAMICS* and **MODAL DYNAMIC* analyses that support nondiagonal damping.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the spring stiffness as a function of frequency, temperature, and other predefined field variables.

Data lines to define nonlinear spring behavior for SPRING1, SPRING2, or JOINTC elements:

First line:

1. Degree of freedom with which the springs are associated at their first nodes or, for JOINTC elements, the degree of freedom for which the spring behavior is being defined.
2. For SPRING2 elements give the degree of freedom with which the springs are associated at their second nodes.

If the ORIENTATION parameter is included on the **SPRING* option when defining spring elements or on the **JOINT* option when defining joint elements, the degrees of freedom specified here are in the local system defined by the **ORIENTATION* option referenced.

Second line:

1. Force.
2. Relative displacement.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the spring stiffness as a function of temperature and other predefined field variables.

Data lines to define linear spring behavior for SPRINGA elements with COMPLEX STIFFNESS:

First line:

1. Enter a blank line.

Second line:

1. Real spring stiffness (force per relative displacement).
2. Structural damping factor.
3. Frequency (in cycles per time). Applicable only for **STEADY STATE DYNAMICS*, DIRECT; **STEADY STATE DYNAMICS*, SUBSPACE PROJECTION; and **STEADY STATE DYNAMICS* and **MODAL DYNAMIC* analyses that support nondiagonal damping.

Repeat this set of data lines as often as necessary to define the spring stiffness and structural damping factor as a function of frequency.

Data lines to define linear spring behavior for SPRING1 and SPRING2 elements with COMPLEX STIFFNESS:

First line:

1. Degree of freedom with which the springs are associated at their first nodes.
2. For SPRING2 elements give the degree of freedom with which the springs are associated at their second nodes.

If the ORIENTATION parameter is included on the **SPRING* option when defining spring elements, the degrees of freedom specified here are in the local system defined by the **ORIENTATION* option referenced.

Second line:

1. Real spring stiffness (force per relative displacement).
2. Structural damping factor.
3. Frequency (in cycles per time). Applicable only for **STEADY STATE DYNAMICS*, DIRECT; **STEADY STATE DYNAMICS*, SUBSPACE PROJECTION; and **STEADY STATE DYNAMICS* and **MODAL DYNAMIC* analyses that support nondiagonal damping.

Repeat this set of data lines as often as necessary to define the spring stiffness and the structural damping factor as a function of frequency.

*SRADIATE: Specify surface radiation conditions in heat transfer analysis.

This option is used to apply surface radiation boundary conditions between a nonconcave surface and a nonreflecting environment in fully coupled thermal-stress analysis or to define an approximate cavity radiation interaction. In Abaqus/Standard it is also used for all procedures that solve for nodal temperature degrees of freedom.

It must be used in conjunction with the **PHYSICAL CONSTANTS* option, which is used to define the Stefan-Boltzmann constant.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- *Thermal Loads*
- *Cavity Radiation in Abaqus/Standard*

Optional parameters:

AMBIENT AMPLITUDE

Set this parameter equal to the name of the **AMPLITUDE* option that gives the variation of the ambient temperature with time.

If this parameter is omitted in an Abaqus/Standard analysis, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (see *Defining an Analysis*). If this parameter is omitted in an Abaqus/Explicit analysis, the reference magnitude is applied immediately at the beginning of the step.

OP

Set OP=MOD (default) for existing **SRADIATE* definitions to remain, with this option modifying existing radiation conditions or defining additional radiation conditions.

Set OP=NEW if all existing **SRADIATE* definitions applied to the model should be removed.

RADIATION AMPLITUDE

Set this parameter equal to the name of the **AMPLITUDE* option that gives the variation of the total radiation flux with time.

If this parameter is omitted in an Abaqus/Standard analysis, the total radiation flux, accounting for any AMBIENT AMPLITUDE, is applied immediately, independent of the value assigned to the AMPLITUDE parameter on the **STEP* option. If this parameter is omitted in an Abaqus/Explicit analysis, the total radiation flux, accounting for any AMBIENT AMPLITUDE, is applied immediately at the beginning of the step.

TRANSITION

Set TRANSITION=TOUCHING (default in Abaqus/Standard; not supported in Abaqus/Explicit) to set emissivity to zero on portions of the specified surface when contact is active and the contact status is closed.

Set TRANSITION=NONE (only option in Abaqus/Explicit) to apply emissivity to the specified surface independent of the contact status.

Data lines to define surface radiation conditions:

First line:

1. Surface name.
2. Radiation type label R.
3. Reference ambient temperature value, θ^0. (Units of θ.)
4. Emissivity, ϵ.
5. Time period over which emissivity is modified to its current magnitude or set to zero from its current value due to temporal changes in contact status. The default time period is chosen to be 0.001 of the current step time. This value is neglected in a linear perturbation analysis or when TRANSITION=NONE.

Repeat this data line as often as necessary to define radiation conditions for different surfaces.

Data lines to define approximate cavity radiation (available only in Abaqus/Standard):

First line:

1. Surface name.
2. Radiation type label AVG.
3. Leave this field blank.
4. Emissivity, ϵ.

Repeat this data line as often as necessary to define radiation conditions for different surfaces.

*STATIC: Static stress/displacement analysis.

This option is used to indicate that the step should be analyzed as a static load step.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Static Stress Analysis*
- *Unstable Collapse and Postbuckling Analysis*
- *Adiabatic Analysis*
- *Solving Nonlinear Problems*
- *Deformation Plasticity*

Optional parameters for a linear perturbation static analysis:

RESIDUAL MODES

Include this parameter to define residual modes for the subsequent substructure generation analyses. This parameter outputs the static responses corresponding to all the load cases to a file named *jobname*_RM_*stepnumber*.sim.

This parameter is ignored if it is specified for a general analysis.

Optional parameters for a general static analysis:

ADIABATIC

Include this parameter to perform an adiabatic stress analysis. This parameter is relevant only for isotropic metal plasticity materials with a Mises yield surface and when the **INELASTIC HEAT FRACTION* option has been specified.

ALLSDTOL

Include this parameter to indicate that an adaptive automatic damping algorithm will be activated in this step. Set this parameter equal to the maximum allowable ratio of the stabilization energy to the total strain energy. The initial damping factor is specified via the STABILIZE parameter or the FACTOR parameter. This damping factor will then be adjusted through the step based on the convergence history and the value of ALLSDTOL. If this parameter is set equal to zero, the adaptive automatic damping algorithm is not activated; a constant damping factor will be used throughout the step. If this parameter is included without a specified value, the default value is 0.05. If this parameter is omitted but the STABILIZE parameter is included with the default value of dissipated energy fraction, the adaptive automatic damping algorithm will be activated automatically with ALLSDTOL=0.05.

This parameter must be used in conjunction with the STABILIZE parameter (see *Solving Nonlinear Problems*).

CONTINUE

Set CONTINUE=NO (default) to specify that this step will not carry over the damping factors from the results of the preceding general step. In this case the initial damping factors will be recalculated

based on the declared damping intensity and on the solution of the first increment of the step or can be specified directly.

Set CONTINUE=YES to specify that this step will carry over the damping factors from the end of the immediately preceding general step.

This parameter must be used in conjunction with the ALLSDTOL and the STABILIZE parameters.

DIRECT

This parameter selects direct user control of the incrementation through the step. If this parameter is used, constant increments of the size defined by the first item on the data line are used. If this parameter is omitted, Abaqus/Standard will choose the increments (after trying the user's initial time increment for the first attempt at the first increment).

The parameter can have the value NO STOP. If this value is included, the solution to an increment is accepted after the maximum number of iterations allowed has been completed (as defined by the **CONTROLS* option), even if the equilibrium tolerances are not satisfied. Very small increments and a minimum of two iterations are usually necessary if this value is used. *This approach is not recommended; it should be used only in special cases when the analyst has a thorough understanding of how to interpret results obtained in this way.*

FACTOR

Set this parameter equal to the damping factor to be used in the automatic damping algorithm (see *Solving Nonlinear Problems*) if the problem is expected to be unstable due to local instabilities and the damping factor calculated by Abaqus is not suitable. This parameter must be used in conjunction with the STABILIZE parameter and overrides the automatic calculation of the damping factor based on a value of the dissipated energy fraction.

This parameter cannot be used if the RIKS parameter is included.

FULLY PLASTIC

This parameter is relevant only for cases where “fully plastic” analysis is required with deformation theory plasticity. For that purpose set this parameter equal to the name of the element set being monitored for fully plastic behavior.

The step will end when the solutions at all constitutive calculation points in the element set are fully plastic (defined by the equivalent strain being 10 times the offset yield strain). The step will end before this occurs if either the maximum number of increments given on the **STEP* option or the time period given on the data line of **STATIC* is exceeded.

LONG TERM

Include this parameter to obtain the fully relaxed long-term elastic solution with time-domain viscoelasticity or the long-term elastic-plastic solution for two-layer viscoplasticity. If the LONG TERM parameter is omitted, the instantaneous elastic solution is obtained for time-domain viscoelasticity and the combined response of the elastic-plastic and elastic-viscous networks is obtained for two-layer viscoplasticity. The parameter is relevant only for time-domain viscoelastic and two-layer viscoplastic materials.

RATE DEPENDENCE

Set RATE DEPENDENCE=ON (default) to consider the strain rate–dependence when evaluating the yield stress of a material and to consider the slip rate–dependence when evaluating the friction coefficient.

Set RATE DEPENDENCE=OFF to ignore the strain rate–dependence when evaluating the yield stress of a material and to ignore the slip rate–dependence when evaluating the friction coefficient.

RIKS

Include this parameter to use the modified Riks method for proportional loading cases (*Unstable Collapse and Postbuckling Analysis*).

STABILIZE

Include this parameter to use automatic stabilization if the problem is expected to be unstable due to local instabilities. Set this parameter equal to the dissipated energy fraction of the automatic damping algorithm (see *Solving Nonlinear Problems*). If this parameter is omitted, the stabilization algorithm is not activated. If this parameter is included without a specified value, the default value of the dissipated energy fraction is $\mathbf{2 \times 10^{-4}}$ and the adaptive automatic damping algorithm will be activated by default with ALLSDTOL =0.05 in this step; set ALLSDTOL=0 to deactivate the adaptive automatic damping algorithm. If the FACTOR parameter is used, any value of the dissipated energy fraction will be overridden by the damping factor.

This parameter cannot be used if the RIKS parameter is included.

Data line for a general static analysis:

First (and only) line:

1. Initial time increment. This value will be modified as required if the automatic time stepping scheme is used or will be used as the constant time increment if the DIRECT parameter is used. If this entry is zero or is not specified, a default value that is equal to the total time period of the step is assumed.
2. Time period of the step. If this entry is zero or is not specified, a default value of $\mathbf{1.0}$ is assumed.
3. Minimum time increment allowed. Only used for automatic time incrementation. If Abaqus/Standard finds it needs a smaller time increment than this value, the analysis is terminated. If this entry is zero, a default value of the smaller of the suggested initial time increment or 10^{-5} times the total time period is assumed.
4. Maximum time increment allowed. Only used for automatic time incrementation. If this value is not specified, no upper limit is imposed.

Data line for the Riks method:

First (and only) line:

1. Initial increment in arc length along the static equilibrium path in scaled load-displacement space, Δl_{in}. If the DIRECT parameter is used, this value will be used as a constant arc length increment. If this entry is zero or is not specified, a default value that is equal to the total arc length of the step is assumed.
2. Total arc length scale factor associated with this step, l_{period}. If this entry is zero or is not specified, a default value of $\mathbf{1.0}$ is assumed.
3. Minimum arc length increment, Δl_{min}. If this entry is zero, a default value of the smaller of the suggested initial arc length or 10^{-5} times the total arc length is assumed.
4. Maximum arc length increment, Δl_{max}. If this value is not specified, no upper limit is imposed.
5. Maximum value of the load proportionality factor, λ_{end}. This value is used to terminate the step when the load exceeds a certain magnitude.
6. Node number at which the finishing displacement value is being monitored.
7. Degree of freedom being monitored.
8. Value of the total displacement (or rotation) at the node and degree of freedom that, if crossed during an increment, ends the step at the current increment.

*STEADY STATE CRITERIA: Specify steady-state criteria for terminating a quasi-static unidirectional simulation.

This option is used to specify the norms that must be satisfied to halt a quasi-static unidirectional simulation based on achieving a steady-state condition. It must be used in conjunction with the **STEADY STATE DETECTION* option.

Products: Abaqus/Explicit

Type: History data

Level: Step

References:

- *Output to the Output Database*
- *Steady-State Detection*
- **STEADY STATE DETECTION*

There are no parameters associated with this option.

Data lines to define steady-state detection norms SSPEEQ and SSSPRD:

First line:

1. Norm type.
2. Norm tolerance. The default tolerance is .001.
3. Global *X*-coordinate of a point on the exit plane.
4. Global *Y*-coordinate of a point on the exit plane.
5. Global *Z*-coordinate of a point on the exit plane.

Repeat this data line as often as necessary. Each line defines a criterion that must be satisfied to achieve steady state.

Data lines to define steady-state detection norms SSFORC and SSTORQ:

First line:

1. Norm type.
2. Norm tolerance. The default tolerance is .005.
3. Global *X*-coordinate of a point on the exit plane.
4. Global *Y*-coordinate of a point on the exit plane.
5. Global *Z*-coordinate of a point on the exit plane.
6. Rigid body reference node number.
7. First direction cosine of force or torque norm output at the reference node.
8. Second direction cosine of force or torque norm output at the reference node.
9. Third direction cosine of force or torque norm output at the reference node.

Repeat this data line as often as necessary. Each line defines a criterion that must be satisfied to achieve steady state.

*STEADY STATE DETECTION: Specify steady-state requirements for terminating a quasi-static unidirectional simulation.

This option is used to define the conditions that must be satisfied to determine that steady state has been reached. It must be used in conjunction with the **STEADY STATE CRITERIA* option.

Products: Abaqus/Explicit

Type: History data

Level: Step

References:

- *Steady-State Detection*
- *Output to the Output Database*
- **STEADY STATE CRITERIA*

Required parameters:

ELSET

Set this parameter equal to the name of the element set for which this steady-state detection definition is being applied.

SAMPLING

This parameter is used to specify the method used to sample all steady-state norms associated with this option. See *Steady-State Detection* for more information regarding steady-state norm definitions.

Set SAMPLING=PLANE BY PLANE to calculate the steady-state norms as each plane of elements crosses the exit plane. This method should be used only for non-Eulerian analyses with uniform planes of elements sequentially passing the exit plane.

Set SAMPLING=UNIFORM to calculate the steady-state norms at an interval defined by the time required for material to flow through an element of average length. This interval is determined at the beginning of the step and remains constant throughout the step. This method should be used only for analyses with material flowing in and out of Eulerian boundary regions in the primary direction.

Data line to define primary direction and cutting plane position:

First (and only) line:

1. First direction cosine of primary direction.
2. Second direction cosine of primary direction.
3. Third direction cosine of primary direction.
4. Global *X*-coordinate of a point on the cutting plane.
5. Global *Y*-coordinate of a point on the cutting plane.
6. Global *Z*-coordinate of a point on the cutting plane.

*STEADY STATE DYNAMICS: Steady-state dynamic response based on harmonic excitation.

This option is used to calculate the system's linearized steady-state response to harmonic excitation.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Direct-Solution Steady-State Dynamic Analysis*
- *Mode-Based Steady-State Dynamic Analysis*
- *Subspace-Based Steady-State Dynamic Analysis*

Optional and mutually exclusive parameters:

If none of these optional mutually exclusive parameters is specified, Abaqus/Standard assumes MODE BASED=CONVENTIONAL.

DIRECT

Include this parameter to compute the steady-state harmonic response directly in terms of the physical degrees of freedom of the model. This usually makes the procedure significantly more expensive, but it can be used if model parameters depend on frequency, if the stiffness of the system is unsymmetric and the unsymmetric terms are important, or if the system contains discrete damping (such as dashpot elements).

MODE BASED

Include this parameter to compute the steady-state harmonic response using the mode superposition method.

Set MODE BASED=CONVENTIONAL (default) to select the conventional modal solver that solves the system of linear equations of motion for every frequency directly by factorizing the complex matrix.

Set MODE BASED=SPECTRAL to select the spectral modal solver that uses the dense complex eigenvalue problem solution to represent the steady-state harmonic response of the damped model. This solver can perform better than the conventional solver in analyses with many modes, frequency points, and load cases for finite element models with special, but commonly used, damping properties (for example, no viscous damping in the model, viscous damping specified for relatively small portions of the model, or viscous modal damping specified for several selected modes).

SUBSPACE PROJECTION

Include this parameter to compute the steady-state harmonic response on the basis of the subspace projection method. In this case a direct solution is obtained for the model projected onto the eigenvectors obtained in the preceding **FREQUENCY* step. This is a cost-effective approach to including consideration of unsymmetric stiffness and frequency-dependent model parameters. It is more expensive than the modal superposition method but less expensive than the direct-solution method.

Set SUBSPACE PROJECTION=ALL FREQUENCIES (default) if the projection of the dynamic equations onto the modal subspace is to be performed at each frequency requested on the data lines.

Set SUBSPACE PROJECTION=CONSTANT if a single projection of the dynamic equations onto the modal subspace is to be used for all frequencies requested on the data lines. The projection is performed using model properties evaluated at the center frequency determined on a logarithmic or linear scale depending on the value of the FREQUENCY SCALE parameter.

Set SUBSPACE PROJECTION=EIGENFREQUENCY if the projections onto the modal subspace of the dynamic equations are to be performed at each eigenfrequency within the requested ranges and at the eigenfrequencies immediately outside these ranges. The projections are then interpolated at each frequency requested on the data lines. The interpolation is done on a logarithmic or linear scale depending on the value of the FREQUENCY SCALE parameter.

Set SUBSPACE PROJECTION=PROPERTY CHANGE to select how often subspace projections onto the modal subspace are performed based on material property changes as a function of frequency. The interpolation is done on a logarithmic or linear scale depending on the value of the FREQUENCY SCALE parameter.

Set SUBSPACE PROJECTION=RANGE if the projections onto the modal subspace of the dynamic equations are to be performed at the lower limit of each frequency range and at the upper limit of the last frequency range. The interpolation is done on a linear scale. This value can be used only with the SIM architecture.

Optional parameters:

DAMPING CHANGE

This parameter is relevant only for SUBSPACE PROJECTION=PROPERTY CHANGE.

Set this parameter equal to the maximum relative change in damping material properties before a new projection is to be performed. The default value is 0.1.

FREQUENCY SCALE

This parameter is relevant only if INTERVAL=EIGENFREQUENCY or INTERVAL=RANGE. If INTERVAL=SPREAD linear scale is used.

Set this parameter equal to LOGARITHMIC (default) or LINEAR to determine whether a logarithmic or linear scale is used for output. If the SUBSPACE PROJECTION parameter is included and is set equal to either EIGENFREQUENCY or PROPERTY CHANGE, the same scale will be used for the interpolation of the subspace projections.

FRICTION DAMPING

This parameter is relevant only if the DIRECT or the SUBSPACE PROJECTION parameter is included.

Set FRICTION DAMPING=NO (default) or YES to ignore or to include friction-induced damping effects at the slipping contact interface for which a velocity differential is imposed.

INTERVAL

Set INTERVAL=EIGENFREQUENCY if the frequency ranges specified on each data line are to be subdivided using the system's eigenfrequencies. This option requires a preceding **FREQUENCY* step and is the default if the DIRECT parameter is omitted.

Set INTERVAL=RANGE if the frequency range specified on each data line is to be used directly. This option is the default if the DIRECT parameter is included.

Set INTERVAL=SPREAD to define frequency points around eigenfrequencies found in the frequency ranges specified on each data line. This option requires a preceding **FREQUENCY* step.

REAL ONLY

This parameter is relevant only if the DIRECT or the SUBSPACE PROJECTION parameter is included.

Include this parameter if damping terms are to be ignored so that a real, rather than a complex, system matrix is factored. This option can reduce computational time significantly for the DIRECT procedure and, to a lesser extent, for the SUBSPACE PROJECTION procedure.

STIFFNESS CHANGE

This parameter is relevant only for SUBSPACE PROJECTION=PROPERTY CHANGE.

Set this parameter equal to the maximum relative change in stiffness material properties before a new projection is to be performed. The default value is 0.1.

Data lines for a steady-state dynamics analysis if INTERVAL=EIGENFREQUENCY:

First line:

1. Lower limit of frequency range or a single frequency, in cycles/time.
2. Upper limit of frequency range, in cycles/time. If this value is given as zero, it is assumed that results are required at only one frequency and the remaining data items on the line are ignored.
3. Number of points in the frequency range at which results should be given, including the end points. It is the number of points from the lower limit of the frequency range to the first eigenfrequency in the range; in each interval from eigenfrequency to eigenfrequency; and from the highest eigenfrequency in the range to the high limit of the frequency range. The minimum value is 2. If the value given is less than 2 (or omitted), the default value of 20 points is assumed.
4. Bias parameter. This parameter is useful only if results are requested at four or more frequency points. It is used to bias the results points toward the ends of the intervals so that better resolution is obtained there. This is generally useful since the ends of each interval are the eigenfrequencies where the response amplitudes vary most rapidly. The default bias parameter is 3.0.
5. Frequency scale factor. All the frequency points, except the lower and upper limit of the frequency range, are scaled by this factor. The default frequency scale factor is 1.0.

Repeat this data line as often as necessary to define frequency ranges in which results are required.

Data lines for a steady-state dynamics analysis if INTERVAL=RANGE:

First line:

1. Lower limit of frequency range or a single frequency, in cycles/time.
2. Upper limit of frequency range, in cycles/time. If this value is given as zero, it is assumed that results are required at only one frequency and the remaining data items on the line are ignored.
3. Total number of points in the frequency range at which results should be given, including the end points. The minimum value is 2. If the value given is less than 2 (or omitted), the default value of 20 points is assumed.
4. Bias parameter. This parameter is useful only if results are requested at four or more frequency points. It is used to bias the results points toward the ends of the intervals so that better resolution is obtained there. The default bias parameter is 1.0 .

Repeat this data line as often as necessary to define frequency ranges in which results are required.

Data lines for a steady-state dynamics analysis if INTERVAL=SPREAD:

First line:

1. Lower limit of frequency range or a single frequency, in cycles/time.
2. Upper limit of frequency range, in cycles/time. If this value is given as zero, it is assumed that results are required at only one frequency and the remaining data items on the line are ignored.

3. Number of equally spaced points around the eigenfrequency at which results should be given, including the eigenfrequency and end points. The minimum value is 3. If the value given is less than 3 (or omitted), the default value of 3 points is assumed.
4. Bias parameter. If the value given is different from 1.0, the bias parameter is reset to 1.0.
5. Frequency scale factor. All the frequency points are scaled by this factor. The default frequency scale factor is 1.0.
6. Frequency spread. Determines the spread as a fractional value of each eigenfrequency in the specified range. The value must be greater than 0.0 and less than 1.0. The default frequency spread is 0.1.

Repeat this data line as often as necessary to define frequency ranges in which results are required.

*STEADY STATE TRANSPORT: Steady-state transport analysis.

This option is used to indicate that the step should be analyzed as a steady-state transport analysis.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Steady-State Transport Analysis*
- *Symmetric Model Generation*

Optional parameters:

ALLSDTOL

Include this parameter to indicate that an adaptive automatic damping algorithm will be activated in this step. Set this parameter equal to the maximum allowable ratio of the stabilization energy to the total strain energy. The initial damping factor is specified via the STABILIZE parameter or the FACTOR parameter. This damping factor will then be adjusted through the step based on the convergence history and the value of ALLSDTOL. If this parameter is set equal to zero, the adaptive automatic damping algorithm is not activated; a constant damping factor will be used throughout the step. If this parameter is included without a specified value, the default value is 0.05. If this parameter is omitted but the STABILIZE parameter is included with the default value of dissipated energy fraction, the adaptive automatic damping algorithm will be activated automatically with ALLSDTOL=0.05.

This parameter must be used in conjunction with the STABILIZE parameter (see *Solving Nonlinear Problems*).

CONTINUE

Set CONTINUE=NO (default) to specify that this step is not to carry over the damping factors from the results of the preceding general step. In this case the initial damping factors will be recalculated based on the declared damping intensity and on the solution of the first increment of the step or can be specified directly.

Set CONTINUE=YES to specify that this step is to carry over the damping factors from the end of the immediately preceding general step.

This parameter must be used in conjunction with the ALLSDTOL and the STABILIZE parameters.

DIRECT

This parameter selects direct user control of the incrementation through the step. If this parameter is used, constant increments of the size defined by the first item on the data line are used. If this parameter is omitted, Abaqus/Standard will choose the increments (after trying the user's initial time increment for the first attempt at the first increment).

The parameter can have the value NO STOP. If this value is included, the solution to an increment is accepted after the maximum number of iterations allowed has been completed (as defined by the **CONTROLS* option), even if the equilibrium tolerances are not satisfied. Very small increments and a minimum of two iterations are usually necessary if this value is used. *This approach is not recommended; it should be used only in special cases when the analyst has a thorough understanding of how to interpret results obtained in this way.*

ELSET

Set this parameter equal to the name of an element set for which the rigid body motion will be described in a spatial or Eulerian manner. The rest of the elements in the model will be treated in a classical Lagrangian manner. Only one Eulerian element set can be specified in the whole model.

If this parameter is omitted, the rigid body motion in the whole model will be described in a spatial or Eulerian manner.

FACTOR

Set this parameter equal to the damping factor to be used in the automatic damping algorithm (see *Solving Nonlinear Problems*) if the problem is expected to be unstable due to local instabilities and the damping factor calculated by Abaqus is not suitable. This parameter must be used in conjunction with the STABILIZE parameter and overrides the automatic calculation of the damping factor based on a value of the dissipated energy fraction.

INERTIA

Include this parameter to indicate that inertia effects must be accounted for.

Set INERTIA=NO (default) to ignore inertia effects.

Set INERTIA=YES to include inertia effects. This setting includes the inertia load stiffness terms. At high rolling speeds these terms can improve the convergence behavior of the simulation significantly. Therefore, it is recommended that you include these terms.

Set INERTIA=LOW SPEED to include inertia effects but ignore unsymmetric inertia load stiffness matrix terms. The unsymmetric inertia load stiffness terms increase the solver time for each iteration. At low rolling speeds the inertia load stiffness terms are often not critical for convergence and using INERTIA=LOW SPEED may shorten the simulation time.

INERTIA STABILIZATION FACTOR

Set this parameter equal to the stabilization factor to use in the inertia calculations for first-order elements in three-dimensional meshes created by revolving an axisymmetric mesh cross-section during symmetric model generation (see *Inertia Stabilization*). This factor must be set to a value greater than or equal to zero. By default, it is set to one.

Set this parameter equal to zero to turn off inertia stabilization.

The inertia stabilization factor is not related to the STABILIZE parameter; however, when inertia stabilization is used, it is often not necessary to include automatic stabilization.

LONG TERM

Include this parameter to indicate that there is no viscoelastic or viscoplastic material response during this step. The solution must be based on the long-term elastic moduli if the material description includes viscoelastic material properties or be based on the long-term response of the elastic-plastic network alone if the two-layer viscoplastic material model is used.

MULLINS

Include this parameter when the Mullins effect material model is used to indicate how the Mullins effect should be applied over the step.

Set MULLINS=RAMP to indicate that the Mullins effect should be ramped up over the time period of the current step.

Set MULLINS=STEP (default) to indicate that the Mullins effect should be applied instantaneously at the beginning of the current step.

PASS BY PASS

Include this parameter to indicate that a quasi-steady-state (pass-by-pass) procedure will be used to obtain the steady-state solution.

If this parameter is omitted, the steady-state solution will be obtained directly.

STABILIZE

Include this parameter to use automatic stabilization if the problem is expected to be unstable due to local instabilities. Set this parameter equal to the dissipated energy fraction of the automatic damping algorithm (see *Solving Nonlinear Problems*). If this parameter is omitted, the stabilization algorithm is not activated. If this parameter is included without a specified value, the default value of the dissipated energy fraction is $\mathbf{2 \times 10^{-4}}$ and the adaptive automatic damping algorithm will be activated by default with ALLSDTOL =0.05 in this step; set ALLSDTOL=0 to deactivate the adaptive automatic damping algorithm. If the FACTOR parameter is used, any value of the dissipated energy fraction will be overridden by the damping factor.

Data line to define a steady-state transport analysis:

First (and only) line:

1. Initial time increment. This value will be modified as required if the automatic time stepping scheme is used or will be used as the constant time increment if the DIRECT parameter is used.
2. Time period of the step.
3. Minimum time increment allowed. Only used for automatic time incrementation. If Abaqus/Standard finds it needs a smaller time increment than this value, the analysis is terminated. If this entry is zero, a default value of the smaller of the suggested initial time increment or 10^{-5} times the total time period is assumed.
4. Maximum time increment allowed. Only used for automatic time incrementation. If this value is not specified, no upper limit is imposed.

*STEP: Begin a step.

This option is used to begin each step definition. It must be followed by a procedure definition option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Model

Abaqus/CAE: Step module

References:

- *Defining an Analysis*
- *Convergence Criteria for Nonlinear Problems*
- *Direct Design Sensitivity Analysis*
- **END STEP*

Beginning a step in an Abaqus/Standard analysis

Optional parameters:

AMPLITUDE

This parameter defines the default amplitude variation for loading magnitudes during the step.

Set AMPLITUDE=STEP if the load is to be applied instantaneously at the start of the step and remain constant throughout the step.

Set AMPLITUDE=RAMP if the load magnitude is to vary linearly over the step, from the value at the end of the previous step (or zero, at the start of the analysis) to the value given on the loading option.

If this parameter is omitted, the default amplitude choice depends on the procedure chosen, as shown in *Defining an Analysis*. The default amplitude variation can be overwritten for individual loadings by using the AMPLITUDE parameter on the loading options (*Amplitude Curves*).

This parameter is rarely needed, and changing the defaults might cause problems. For example, the automatic load incrementation scheme in procedures without a real time scale (such as the **STATIC* option) applies the loads gradually by incrementing the normalized time scale. The use of AMPLITUDE=STEP specifies that the entire load will be applied immediately, so Abaqus/Standard might not be able to choose suitable small increments if the loading causes strongly nonlinear response.

CONVERT SDI

This parameter determines how severe discontinuities (such as contact changes) are accounted for during nonlinear analysis.

Set CONVERT SDI=YES (default) to use local convergence criteria to determine whether a new iteration is required. Abaqus/Standard will determine the maximum penetration and estimated force errors associated with severe discontinuities and check whether these errors are within the tolerances. Hence, a solution might converge if the severe discontinuities are small.

Set CONVERT SDI=NO to force a new iteration if severe discontinuities occur during an iteration, regardless of the magnitude of the penetration and force errors. This option also changes some time incrementation parameters and uses different criteria to determine whether to do another iteration or to make a new attempt with a smaller increment size.

If the CONVERT SDI parameter is omitted, Abaqus/Standard will use the value specified in the previous general analysis step. An exception is the first new step of a restart analysis, which will use CONVERT SDI=YES by default regardless of the setting in the previous step.

This parameter has no relevance and will be ignored for heat transfer analysis and linear perturbation steps.

DSA

This parameter applies only to Abaqus/Design.

Set DSA=YES to activate design sensitivity analysis for the step. Once DSA is activated in a general step, it remains active in all subsequent general steps until it is deactivated in a subsequent general step by setting DSA=NO. Once DSA is activated in a perturbation step, it remains active in all subsequent consecutive perturbation steps until it is deactivated in a subsequent consecutive perturbation step. However, if DSA is activated in a step whose procedure is not supported for DSA, DSA will be deactivated until it is activated again by setting DSA=YES.

EXTRAPOLATION

This parameter is useful only for nonlinear analyses.

Set EXTRAPOLATION=LINEAR (default for procedures other than **DYNAMIC*, APPLICATION=TRANSIENT FIDELITY) to indicate that the process is essentially monotonic, so that Abaqus/Standard should use a 100% linear extrapolation, in time, of the previous incremental solution to begin the nonlinear equation solution for the current increment (a 1% extrapolation is used with the Riks method).

Set EXTRAPOLATION=PARABOLIC to indicate that the process should use a quadratic displacement-based extrapolation, in time, of the previous two incremental solutions to begin the nonlinear equation solution for the current increment.

Set EXTRAPOLATION=VELOCITY PARABOLIC (available for **DYNAMIC* procedure only and default for the **DYNAMIC*APPLICATION=TRANSIENT FIDELITY procedure) to indicate that the process should use a quadratic velocity-based extrapolation, in time, of the previous incremental solutions to begin the nonlinear equation solution for the current increment.

Set EXTRAPOLATION=NO to suppress any extrapolation.

INC

Set this parameter equal to the maximum number of increments in a step (or in a single loading cycle for direct cyclic analysis). This value is only an upper bound. The default value is 100.

The INC parameter has no effect in procedures where automatic incrementation cannot be used (for example, **BUCKLE*, **STEADY STATE DYNAMICS*, and **MODAL DYNAMIC*).

NAME

Set this parameter equal to a label that will be used to refer to the step on the output database. Step names in the same input file must be unique. Step names from the original input file can be reused in a restart input file.

NLGEOM

Omit this parameter or set NLGEOM=NO to perform a geometrically linear analysis during the current step. Include this parameter or set NLGEOM=YES to indicate that geometric nonlinearity should be accounted for during the step (stress analysis, fully coupled thermal-stress analysis, and

coupled thermal-electrical-stress analysis only). Once the NLGEOM option has been switched on, it will be active during all subsequent steps in the analysis.

PERTURBATION

Include this parameter to indicate that this is a linear perturbation step. For this type of analysis Abaqus/Standard expects that load, boundary, and temperature changes should be given and that the results will be changes relative to the previous step. *Please read the discussions in General and Perturbation Procedures, Mesh-to-Mesh Solution Mapping, and About Loads before using this option.*

SENSITIVITY

Set SENSITIVITY=ADJOINT to output adjoint design sensitivities for topology, sizing, lattice, or shape optimization.

If this parameter is omitted, adjoint design sensitivities are not output.

SOLVER

Set SOLVER=ITERATIVE to use the iterative linear equation solver. *Please read the discussion in Iterative Linear Equation Solver before using this option.*

You cannot use the iterative solver and the direct sparse solver together in the same input file.

If this parameter is omitted for the first step, the default direct sparse solver is used. If this parameter is omitted for a step after the first step, the solver for the previous step is used.

UNSYMM

Set UNSYMM=YES to indicate that unsymmetric matrix storage and solution should be used.

Set UNSYMM=NO to indicate that symmetric storage and solution should be used.

The default value for this parameter depends on the model and procedure options used. The user is allowed to change the default value only in certain cases. If the UNSYMM parameter is not used in such cases, Abaqus/Standard will use the value specified in the previous general analysis step. See *Defining an Analysis* for a more detailed discussion of the use of this parameter.

Optional data lines:

First line:

1. Subheading for this step.

The subheading can be several lines long, but only the first 80 characters of the first line will be saved and printed as a subheading.

Beginning a step in an Abaqus/Explicit analysis

Optional parameters:

NAME

Set this parameter equal to the name used to identify the step on the output database. Step names in the same input file must be unique.

NLGEOM

Set NLGEOM=YES (default) to indicate that geometric nonlinearity should be accounted for during the step (stress analysis and fully coupled thermal-stress analysis only). Once the NLGEOM option

has been switched on, it will be active during all subsequent steps in the analysis. Set NLGEOM=NO to perform a geometrically linear analysis during the current step.

The default value for the NLGEOM parameter in an Abaqus/Explicit analysis is YES unless the Abaqus/Explicit analysis is an import analysis, in which case the default value of the NLGEOM parameter is the same as the value of the parameter at the time of import.

Optional data lines:

First line:

1. Subheading for this step.

The subheading can be several lines long, but only the first 80 characters of the first line will be saved and printed as a subheading.

*STEP CONTROL: Control progress of the simulation based on the solution state.

This option is used to control the progress of a simulation based on the solution state. If you specify a threshold value for a particular measure of the solution state that has been associated with a sensor definition, the action taken on reaching the threshold can include terminating the current step or terminating the analysis.

You can define an action for a step control to end the current step or analysis when the algebraic or the absolute value of the sensor has reached either a maximum or a minimum threshold. You can also specify that Abaqus/Standard utilizes an alternative time incrementation that is smaller than that computed by the normal time-incrementation algorithms in Abaqus/Standard when the sensor value falls within a user-specified range.

The **STEP CONTROL* option applies only in the step in which it is defined. You can define multiple **STEP CONTROL* definitions within a single step.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Defining an Analysis*
- **STEP*

Required parameters:

NAME

Set this parameter equal to the name associated with this sensor-based step control definition. The sensor's name itself is given on the data line.

Optional parameters:

ACTION

By default, the step control definition is utilized for time increment refinement when the value of the sensor is within a specified interval. In this case, no threshold value of the sensor is necessary. The nondefault actions of a step control definition require the specification of a threshold value of the sensor.

Set ACTION=CONTINUE (default) to trigger time increment refinement when the value of the sensor is within a specified interval. It must be used in conjunction with DT REFINEMENT=YES.

Set ACTION=END ANALYSIS to end the analysis completely when the sensor reaches a specified threshold value.

Set ACTION=END STEP to conclude the current step and to continue to the next step when the sensor reaches a specified threshold value.

DT REFINEMENT

You can specify two limits and control the time incrementation when the sensor value is within the range defined by the two limits to get finer resolution on the solution within a specified range of values for the sensor while using normal time increments outside of this range. Alternatively, you can specify a limit and a threshold value and control the time incrementation when the sensor value is in that range.

Set DT REFINEMENT=AUTO to indicate that Abaqus/Standard should refine the time increment automatically to reach the threshold as closely as possible.

Set DT REFINEMENT=NO to leave the time increment unaffected.

Set DT REFINEMENT=YES (default) to indicate that a user-specified fixed time increment should be used.

TOLERANCE

Set this parameter equal to the tolerance to be used to determine when the sensor has reached the threshold value. The TOLERANCE parameter must be used in conjunction with DT REFINEMENT=AUTO.

Abaqus/Standard enforces the relative error in the sensor value with respect to its threshold value to be less than the specified tolerance. The default value of the tolerance is 0.001.

Data lines to define a sensor-based step control to end the current step or the analysis without refining the time incrementation (DT REFINEMENT=NO, ACTION=END STEP or ACTION=END ANALYSIS):

First data line:

1. Sensor name.
2. Threshold criteria (Max, Min, AbsMax, or AbsMin).
3. Sensor threshold value.

Repeat this data line as often as necessary to define different stopping criteria, potentially using different sensors.

Data lines to define a sensor-based step control to end the current step or the analysis with automatic refinement of the time incrementation (DT REFINEMENT=AUTO, ACTION=END STEP or ACTION=END ANALYSIS):

First data line:

1. Sensor name.
2. Threshold criteria (Max, Min, AbsMax, or AbsMin).
3. Sensor threshold value.

Repeat this data line as often as necessary to define different stopping criteria, potentially using different sensors.

Data lines to define a sensor-based step control to end the current step or the analysis with a user-specified refinement of the time incrementation (DT REFINEMENT=YES, ACTION=END STEP or ACTION=END ANALYSIS):

First data line:

1. Sensor name.
2. Threshold criteria (Max, Min, AbsMax, or AbsMin).
3. Sensor threshold value.
4. Time increment value to be used, provided that it is lower than the time increment computed by the automatic time-incrementation algorithms in Abaqus/Standard and the condition for time increment refinement is met.
5. Sensor limit value when the time increment refinement starts.
6. Blank.

Repeat this data line as often as necessary to define different stopping criteria, potentially using different sensors.

Data lines to define a sensor-based step control, with a user-specified refinement of the time incrementation, to get a finer resolution of the solution within a specified interval (DT REFINEMENT=YES, ACTION=CONTINUE):

First data line:

1. Sensor name.
2. Blank.
3. Blank.
4. Time increment value to be used, provided that it is lower than the time increment computed by the automatic time-incrementation algorithms in Abaqus/Standard and the condition for time increment refinement is met.
5. Sensor limit value when the time increment refinement starts.
6. Sensor limit value when the time increment refinement ends.

Repeat this data line as often as necessary to refine the time increment in regions of interest, potentially using different sensors.

*STEP CYCLING: Define a step or series of steps to be repeated.

This option is used to specify a step or series of steps to be repeated over a set number of cycles. Early termination of step cycling might be triggered from within a step using step cycling controls with sensor-based or wear-based criteria.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- **STEP CYCLING CONTROL*
- *Step Cycling*

One of the following mutually exclusive parameters is required:

START

Include this parameter to indicate the start of a sequence of steps that will be cycled.

END

Include this parameter to indicate the end of a sequence of steps that will be cycled.

Optional parameters:

NAME

Set this parameter equal to the name of the cycling sequence. This parameter can be used only in conjunction with the START parameter.

Step cycling names in the same input file must be unique. Step cycling names from the original input file cannot be reused in a restart input file.

Data line to define the start of a step cycling sequence:

First line:

1. Number of cycles.
2. Cycle frequency at which to write output. The default is to write output every cycle.
3. Cycle frequency at which to create a new output database file . By default, Abaqus/Standard writes all output to the same output database file.

Second line (optional):

1. Initial cycle scale factor for use with contact wear.
2. Maximum cycle scale factor for use with contact wear.

*STEP CYCLING CONTROL: Specify sensor-based or contact wear-based criteria to control early termination of step cycling.

This option is used to specify a sensor-based or a contact wear-based criterion to trigger early termination of step cycling before a set number of step cycles are complete.

For a sensor-based criterion, the trigger involves satisfying a termination condition that depends on the sensor state and a sensor threshold value. Alternatively, you can specify accumulated contact wear distances and a threshold wear distance to trigger early termination of step cycling. For the wear-based criterion, you also specify a target maximum accumulated wear distance per step cycle to determine a step cycle scale factor used for wear calculations in the next step cycle.

This option can be activated in a step only if the step is part of a step cycle definition. You can include this option multiple times within a step, with different control definitions; early termination might be triggered by any of the definitions.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- **STEP*
- **STEP CYCLING*
- *Defining an Analysis*
- *Contact Wear*

Required parameters:

NAME

Set this parameter equal to the name associated with this step cycling control definition.

Optional parameters:

ACTION

Set ACTION=END CYCLING (default) to break out of the step cycling loop altogether and consequently skip the remaining steps in the current step cycle.

CRITERION

Set CRITERION=SENSOR-BASED (default) to specify a threshold sensor value to trigger the intended action.

Set CRITERION=WEAR-BASED to specify a threshold wear distance to trigger the intended action.

Data lines to define sensor-based step cycling control:

First line:

1. Sensor name.
2. Label for threshold criterion (Max, Min, AbsMax, or AbsMin).
3. Sensor threshold value.

Repeat this data line as often as necessary to define different termination criteria, potentially using different sensors.

Data lines to define wear-based step cycling control:

First line:

1. Node set name or blank to denote the entire model.
2. Label for threshold criterion (AbsMax to end cycling if any nodal wear value in the node set is over the threshold, or AbsMin to end cycling if all nodal wear values in the node set are over the threshold).
3. Threshold wear distance.
4. Target maximum wear distance accumulated per simulated wear cycle to determine step cycle scaling factor for contact wear in the next step cycle.

Repeat this data line as often as necessary to define local termination criteria.

*SUBCYCLING: Define a subcycling zone.

This option is used to define a subcycling zone.

Products: Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Explicit Dynamic Analysis*
- *Fully Coupled Thermal-Stress Analysis*
- *Selective Subcycling*

Required parameters:

ELSET

Set this parameter equal to the name of the element set to define a subcycling zone.

There are no data lines associated with this option.

*SUBMODEL: Specify driven boundary nodes in submodeling analysis.

This option is used to specify the total list of “driven regions” for a submodel.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: *Load module* and *model attribute*

References:

• *About Submodeling*

Optional, mutually exclusive parameters:

ACOUSTIC TO STRUCTURE

Include this parameter if the submodel will be driven at the specified surface by the acoustic pressure from a global coupled acoustic-structural model.

SHELL TO SOLID

Include this parameter if a solid element submodel will be driven by a global shell model. If this parameter is included, all driven nodes must be on solid elements and must be located in regions modeled with shell elements in the global model. If this parameter is included on any **SUBMODEL* option in an input file, it must be included on all **SUBMODEL* options in the input file.

Required parameter for SHELL TO SOLID submodeling:

SHELL THICKNESS

If the OFFSET parameter is not used on the **SHELL SECTION* or **SHELL GENERAL SECTION* option in the global model, set this parameter equal to the maximum value of the shell thickness in the global model (given in the units used in the model). If the OFFSET parameter is used in the global model, set this parameter equal to twice the maximum distance from the reference surface to the top or bottom shell surface.

Optional parameters:

ABSOLUTE EXTERIOR TOLERANCE

Set this parameter equal to the absolute value (given in the units used in the model) by which a driven node of the submodel may lie outside the region of the elements of the global model. If this parameter is not used or has a value of 0.0, the EXTERIOR TOLERANCE will apply. For shell-to-solid submodeling the driven node may lie within a region defined by half the value of the SHELL THICKNESS parameter plus the exterior tolerance.

EXTERIOR TOLERANCE

Set this parameter equal to the fraction of the average element size in the global model by which a driven node of the submodel may lie outside the region of the elements of the global model. The default is 0.05. For shell-to-solid submodeling the driven node may lie within a region defined by half the value of the SHELL THICKNESS parameter plus the exterior tolerance.

If both tolerance parameters are specified by the user, Abaqus uses the tighter tolerance.

GLOBAL ELSET

Set this parameter equal to the name of the element set in the global model that will be searched for elements whose responses will be used to drive the submodel. If this parameter is omitted, Abaqus will search all elements in the global model that lie in the vicinity of the submodel.

This parameter must be used with the ACOUSTIC TO STRUCTURE parameter when the acoustic pressures act on both sides of a shell.

INTERSECTION ONLY

Include this parameter to specify that Abaqus ignore driven nodes found to lie outside the region of elements of the global model, after accounting for the exterior search tolerance.

This parameter can be used only with TYPE=NODE. This parameter cannot be used with the ACOUSTIC TO STRUCTURE or SHELL TO SOLID parameters.

TYPE

This parameter applies only to Abaqus/Standard analyses. It determines whether global model communication to the submodel occurs through nodes or through surfaces.

Set TYPE=NODE (default) for node-based submodeling. Node-based submodel definitions will be accompanied by **BOUNDARY*, SUBMODEL definitions.

Set TYPE=SURFACE for surface-based submodeling. This parameter setting cannot be used with the ACOUSTIC TO STRUCTURE, SHELL TO SOLID, or SHELL THICKNESS parameters. Surface-based submodel definitions will be accompanied by **DSLOAD*, SUBMODEL definitions.

Data lines to define the driven boundary for general and shell-to-solid submodeling:

First line:

1. List of nodes or node set labels. Up to 16 entries are allowed per line.

Repeat this data line as often as necessary.

Data lines to define the driven boundary for acoustic-to-structure submodeling:

First line:

1. The name of the element-based structural surface.

Repeat this data line as often as necessary.

Data lines to define the driven surfaces for surface-based submodeling:

First line:

1. The name of the element-based surface.

Repeat this data line as often as necessary.

*SUBMODEL CONDITIONS: Specify that submodel conditions are to be imposed on cut surfaces during this step.

This option is used together with instances of the **SUBMODEL CUT* suboption to impose submodel conditions on the cut surface of a submodel. Submodel conditions can be specified only once per step.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *About Submodeling*
- *Beam-Based Submodeling*

Required parameters:

GLOBAL STEP

Set this parameter equal to the step number in the global analysis for which the values of the driven variables will be read during this step of the submodel analysis.

INC

This parameter can be used only in a static linear perturbation step.

Set this parameter equal to the increment in the selected step of the global analysis at which the solution will be used to specify the values of the driven variables. By default, Abaqus/Standard uses the solution at the last increment of the selected step.

Optional parameters:

AMPLITUDE SCALE

Set this parameter equal to the value by which the driven variables read from the global analysis are to be scaled.

The default is AMPLITUDE SCALE=1.0.

TIME SCALE

If the submodel analysis step time is different from the global analysis step time, use the TIME SCALE parameter to adjust the time variable for the driven nodes' amplitude functions. The time variable of each driven node's amplitude function is scaled to match the submodel analysis step time.

If this parameter is omitted, the time variable is not scaled.

There are no data lines associated with this option.

*SUBMODEL CUT: Define a submodel cut surface.

This option is used to identify a submodel cut where the submodel conditions will be imposed and to specify the submodel control settings specific to a submodel cut. It must be used as a suboption of the **SUBMODEL CONDITIONS* option.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *About Submodeling*
- *Beam-Based Submodeling*

Required parameters:

LOCAL SURFACE

Set this parameter equal to the name of the element-based surface where the submodel conditions are being imposed.

Optional parameters:

ABSOLUTE EXTERIOR TOLERANCE

Set this parameter equal to the absolute value (given in the units used in the model) by which a driven node of the submodel can lie outside the region of the elements of the global model.

If this parameter is omitted or has a value of 0.0, the EXTERIOR TOLERANCE applies.

EXTERIOR TOLERANCE

Set this parameter equal to the fraction of the average element size in the global model by which a driven node can lie outside the region of the elements of the global model.

The default is 0.05.

GLOBAL ELSET

Set this parameter equal to the name of the element set in the global model that will be searched for elements whose responses are being used to impose the submodel conditions.

If this parameter is omitted, Abaqus/Standard searches all the elements in the global model.

MAIN IMPOSED CONDITIONS

Set MAIN IMPOSED CONDITIONS=PRIMARY (default) to impose conditions on the primary degrees of freedom.

Set MAIN IMPOSED CONDITIONS=NONE to impose no conditions on the primary degrees of freedom.

OP

Set OP = MOD (default) to modify the imposed conditions on this submodel cut surface.

Set OP = NEW to remove the previously imposed conditions on this submodel cut surface and to apply the newly specified conditions.

Data line to define the degrees of freedom for imposed boundary conditions:

First line:

1. The degree of freedom to be imposed or exempted.
2. Specify NONE if this degree of freedom should not be imposed; specify PRIMARY if this degree of freedom should be imposed.
3. Etc., up to four degrees of freedom per line.

Repeat this data line as often as necessary.

*SUBSTRUCTURE CHANGE: Change substructure properties for the current analysis step.

This option is used to change substructure properties for the current analysis step.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

• *Generating Substructures*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the substructure elements for which to modify the operators. Abaqus/Standard ignores any elements in the element set that are not substructures.

Optional parameters:

STIFFNESS UNSYMMETRY FACTOR

Set this parameter equal to the unsymmetry factor to use for the substructure element set.

Set STIFFNESS UNSYMMETRY FACTOR=1.0 (default) to use the unsymmetric instance of the reduced stiffness matrix (if available) for the substructure elements.

Set STIFFNESS UNSYMMETRY FACTOR=0.0 to use the symmetric instance of the reduced stiffness matrix for the substructure elements.

For any value other than 1.0, Abaqus/Standard requires both the symmetric and unsymmetric instances of the reduced stiffness matrix for the substructure elements specified in the element set (see *Generating a Reduced Stiffness Matrix for a Substructure*).

There are no data lines associated with this option.

*SUBSTRUCTURE DAMPING: Specify mass proportional and stiffness proportional damping for a substructure.

This option is used to specify substructure damping factors proportional to the substructure mass and stiffness matrices at the substructure usage stage. The mass and stiffness proportional damping factors can be used in combination with the generated substructure viscous and structural damping matrices and substructure modal damping to form the substructure element damping matrix at the substructure usage stage.

The *SUBSTRUCTURE DAMPING option can be used only in conjunction with the *SUBSTRUCTURE PROPERTY option. This option is supported in procedures where nondiagonal damping is allowed in the model domain, such as general analysis procedures, linear dynamic procedures, and the direct steady-state dynamics procedure.

Products: Abaqus/Standard Abaqus/AMS

Type: Model data

Level: Part, Part instance, Model

References:

- *Using Substructures*
- *Generating Substructures*

Optional parameters:

ALPHA

Set this parameter equal to the α_R coefficient to create Rayleigh mass proportional damping. The default is ALPHA=0. (Units of T^{-1}.)

BETA

Set this parameter equal to the β_R coefficient to create Rayleigh stiffness proportional damping. The default is BETA=0. (Units of T.)

STRUCTURAL

Set this parameter equal to the γ coefficient to create imaginary stiffness proportional damping. The default is STRUCTURAL=0.

There are no data lines associated with this option.

*SUBSTRUCTURE DAMPING CONTROLS: Specify damping controls for a substructure property.

This option is used in conjunction with the **SUBSTRUCTURE PROPERTY* option to control the type and the source of substructure damping in all analysis procedures using substructures that take damping into account. The rules for using the STRUCTURAL and VISCOUS parameters are the same as the rules for using them for the **DAMPING CONTROLS* option within a step definition (see *Defining Substructure Damping* for details).

This option identifies which of the supplied damping options will participate within a substructure. It does not affect the use of the **SUBSTRUCTURE MODAL DAMPING* option.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- *Defining Substructure Damping*
- *Generating Substructures*

Optional parameters:

STRUCTURAL

Set this parameter equal to ELEMENT to request only the condensed structural damping matrix.

Set this parameter equal to FACTOR to request only the structural damping matrix proportional to the condensed substructure stiffness matrix.

Set this parameter equal to COMBINED to request the structural damping matrix that includes the combination of both ELEMENT and FACTOR.

Set this parameter equal to NONE to exclude the structural damping matrix for the elements associated with this substructure property.

If this parameter is omitted or the option is not used as a suboption of the **SUBSTRUCTURE PROPERTY* option, the substructure property uses COMBINED as the default with the structural factor specified in the **SUBSTRUCTURE DAMPING* option.

VISCOUS

Set this parameter equal to ELEMENT to request only the substructure viscous damping matrix defined as a linear combination of the condensed substructure mass and stiffness matrices.

Set this parameter equal to FACTOR to request only the viscous damping matrix.

Set this parameter equal to COMBINED to request a viscous damping matrix that includes a combination of ELEMENT and FACTOR.

Set this parameter equal to NONE to exclude the viscous damping matrix for the elements associated with this substructure property.

If this parameter is omitted or the option is not used as a suboption of the **SUBSTRUCTURE PROPERTY* option, the substructure property uses COMBINED as the default with the mass and stiffness proportional Rayleigh damping factors specified under the substructure damping as ALPHA and/or BETA.

There are no data lines associated with this option.

*SUBSTRUCTURE GENERATE: Substructure generation analysis.

This option is used to indicate that the step is analyzed as a substructure generation step.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

Abaqus/CAE: Step module

References:

• *Generating Substructures*

Optional parameters:

DISPLAY ELSET

Set this parameter equal to the name of the element set that contains all the elements you want to use to define the coarse substructure display representation for visualization in the substructure usage analysis.

This element set can include any 2-node elements (truss, beam, connector, etc.) and 3- or 4-node elements with two-dimensional geometry (surface, membrane, shell).

EIGENPROBLEM

Set this parameter equal to YES (default) if the substructure eigenproblem needs to be solved during substructure generation in Abaqus. Substructure eigenvectors can be used to define substructure modal damping for a given substructure.

Set this parameter equal to NO if the eigensolution is not required.

ELSET

If element output recovery is needed, including all element nodes in the selective recovery node set generally is insufficient since an element can have Abaqus internal nodes.

Set this parameter equal to the name of the element set that contains all the elements in the regions of the substructure where you want to recover results.

FRICTION DAMPING

Set FRICTION DAMPING=NO (default) to ignore friction-induced viscous damping effects.

Set FRICTION DAMPING=YES to include friction-induced viscous damping effects.

GRAVITY LOAD

Set GRAVITY LOAD=YES to calculate the substructure's gravity load vectors. The default is GRAVITY LOAD=NO.

LIBRARY

Set this parameter equal to the prefix in the generated substructure name. The prefix and the identifier (specified using the TYPE parameter) constitute a substructure name that must be unique. The default prefix is *jobname*.

The LIBRARY and NAME parameters are mutually exclusive. Using the NAME parameter is the preferred method.

MASS MATRIX

Set MASS MATRIX=YES to calculate the substructure's reduced mass matrix. The default is MASS MATRIX=NO.

MODEL DATA

Set MODEL DATA=ODB (default) to generate the substructure model data file, which contains the finite element model data required for visualization of results recovered within the substructure.

Set MODEL DATA=NONE to suppress generation of the substructure model data file.

MONITOR NSET

Set this parameter equal to the name of the node set that contains the nodes whose solution you want to monitor in the substructure usage analysis.

Monitor nodes are created automatically in the usage-level model, and history output requests are applied to these nodes in the usage-level analysis procedures (where applicable).

NAME

Set this parameter equal to the substructure name for which the substructure data are written. See *Input Syntax Rules* for the syntax of such names. The default substructure name is *jobname_Zn*, where *n* is the current step number of the job.

The NAME parameter and the LIBRARY and TYPE parameters are mutually exclusive. Using the NAME parameter is the preferred method.

NSET

Set this parameter equal to the name of the node set that contains the nodes of the substructure where you want to recover results. This node set must contain all nodes for which node output can be requested in a substructure usage analysis.

If the NSET parameter is omitted but the ELSET parameter is used, the recovery matrix corresponding to all the element nodes in the specified element set is generated. If both the NSET and ELSET parameters are used, the recovery matrix for the union of the node set and the set of all the element nodes for all the elements in the element set is generated. If both the NSET and ELSET parameters are omitted, the recovery matrix for all eliminated nodes is generated (default case).

OVERWRITE

Include this parameter to overwrite existing files in the substructure database with the same name. The default is no overwrite.

PROPERTY EVALUATION

Set this parameter equal to the frequency at which to evaluate frequency-dependent properties for viscoelasticity, springs, and dashpots during the substructure generation. If this parameter is omitted, Abaqus/Standard evaluates the stiffness associated with frequency-dependent springs and dashpots at zero frequency and does not consider the stiffness contributions from frequency-domain viscoelasticity in the **SUBSTRUCTURE GENERATE* step.

RECOVERY MATRIX

Set RECOVERY MATRIX=NO to specify that output of element or nodal information is not available within this substructure. The default is RECOVERY MATRIX=YES, indicating that recovery of eliminated variables is possible for most analysis procedures.

If RECOVERY MATRIX=NO, the NSET and ELSET parameters are ignored.

RESIDUAL MODES

Include this parameter to add the following to the substructure basis:

- The load modes corresponding to the substructure load cases specified in the substructure generation analysis

- The residual modes computed in the static perturbation analyses between the most recent eigenfrequency extraction analysis and the current substructure generation analysis

If no eigenfrequency extraction analysis is specified before the substructure generation analysis, Abaqus/Standard adds the residual modes computed from all the static perturbation analyses specified before the substructure generation analysis, as well as the load modes corresponding to the substructure load cases, to the substructure basis.

STIFFNESS MATRIX

For acoustic-structural substructures, this parameter is valid only for substructures generated using coupled modes.

If this parameter is omitted, a symmetric instance of the substructure's reduced stiffness matrix is generated when Abaqus/Standard uses the symmetric solver for the current analysis step. An unsymmetric instance of the substructure's reduced stiffness matrix is generated when Abaqus/Standard uses the unsymmetric solver for the current analysis step. For more information, see *Matrix Storage and Solution Scheme in Abaqus/Standard*.

Set STIFFNESS MATRIX=SYMMETRIC to generate a symmetric instance of the reduced stiffness matrix regardless of whether Abaqus/Standard uses the symmetric or unsymmetric solver.

Set STIFFNESS MATRIX=UNSYMMETRIC to generate an unsymmetric instance of the reduced stiffness matrix regardless of whether Abaqus/Standard uses the symmetric or unsymmetric solver.

Set STIFFNESS MATRIX=BOTH to generate both a symmetric and an unsymmetric instance of the reduced stiffness matrix regardless of whether Abaqus/Standard uses the symmetric or unsymmetric solver.

STRUCTURAL DAMPING MATRIX

Set STRUCTURAL DAMPING MATRIX=YES to calculate the substructure's reduced structural damping matrix. A symmetric instance of the reduced structural damping matrix is generated when Abaqus/Standard uses the symmetric solver. An unsymmetric instance of the reduced structural damping matrix is generated when Abaqus/Standard uses the unsymmetric solver. For more information, see *Matrix Storage and Solution Scheme in Abaqus/Standard*.

The default is STRUCTURAL DAMPING MATRIX=NO.

Set STRUCTURAL DAMPING MATRIX=SYMMETRIC to generate a symmetric instance of the reduced structural damping matrix regardless of whether Abaqus/Standard uses the symmetric or unsymmetric solver. For acoustic-structural substructures, this parameter value is valid only for substructures generated using coupled modes.

Set STRUCTURAL DAMPING MATRIX=UNSYMMETRIC to generate an unsymmetric instance of the reduced structural damping matrix regardless of whether Abaqus/Standard uses the symmetric or unsymmetric solver. For acoustic-structural substructures, this parameter value is valid only for substructures generated using coupled modes.

Set STRUCTURAL DAMPING MATRIX=BOTH to generate both a symmetric and an unsymmetric instance of the reduced structural damping matrix regardless of whether Abaqus/Standard uses the symmetric or unsymmetric solver. For acoustic-structural substructures, this parameter value is valid only for substructures generated using coupled modes.

TYPE

Set this parameter equal to the identifier in the generated substructure name. The identifier and the prefix (specified using the LIBRARY parameter) constitute a substructure name that must be unique. The identifier must be Z followed by a number that cannot exceed 9999. The default identifier is Z*n*, where *n* is the current step number of the job.

The TYPE and NAME parameters are mutually exclusive. Using the NAME parameter is the preferred method.

VISCOUS DAMPING MATRIX

Set VISCOUS DAMPING MATRIX=YES to calculate the substructure's reduced viscous damping matrix. A symmetric instance of the reduced viscous damping matrix is generated when Abaqus/Standard uses the symmetric solver. An unsymmetric instance of the reduced viscous damping matrix is generated when Abaqus/Standard uses the unsymmetric solver. For more information, see *Matrix Storage and Solution Scheme in Abaqus/Standard*.

The default is VISCOUS DAMPING MATRIX=NO.

Set VISCOUS DAMPING MATRIX=SYMMETRIC to generate a symmetric instance of the reduced viscous damping matrix regardless of whether Abaqus/Standard uses the symmetric or unsymmetric solver. For acoustic-structural substructures, this parameter value is valid only for substructures generated using coupled modes.

Set VISCOUS DAMPING MATRIX=UNSYMMETRIC to generate an unsymmetric instance of the reduced viscous damping matrix regardless of whether Abaqus/Standard uses the symmetric or unsymmetric solver. For acoustic-structural substructures, this parameter value is valid only for substructures generated using coupled modes.

Set VISCOUS DAMPING MATRIX=BOTH to generate both a symmetric and an unsymmetric instance of the reduced viscous damping matrix regardless of whether Abaqus/Standard uses the symmetric or unsymmetric solver. For acoustic-structural substructures, this parameter value is valid only for substructures generated using coupled modes.

There are no data lines associated with this option.

*SUBSTRUCTURE LOAD CASE: Begin the definition of a substructure load case.

This option is used to begin the definition of a substructure load case for the substructure currently being generated. It can be used only in a **SUBSTRUCTURE GENERATE* analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

References:

- *Generating Substructures*
- **SLOAD*

Required parameters:

NAME

Set this parameter equal to a label that is used to refer to the load case in **SLOAD* option specifications when applying loads to the substructure during an analysis.

Required parameter for substructure load cases that include nodal pressure loading:

GENERATE

Include this parameter to generate multiple substructure load cases.

This parameter applies only to a load case definition that refers to the nodal pressure distributed load type label (NP) in the **DSLOAD* option.

Nodal pressure substructure load cases are generated for each node of the referenced surface. The label that is used to refer to the individual load case in **SLOAD* option specifications is formatted using the specified load case name, the referenced surface name, and the node number as follows:

```
name-surface_name-node_number
```

To define the loads:

Enter any mechanical loading options (*Concentrated Loads* and *Distributed Loads*) or thermal loading options (*Thermal Loads*) to define the loads forming the load case. Specify a magnitude for each load. This magnitude is scaled by a magnitude and amplitude reference specified in the **SLOAD* option. The load case definition continues until an option is encountered that is not one of the loading options. If boundary conditions are included in a **SUBSTRUCTURE LOAD CASE*, they are always active, even if the **SLOAD* option is not used.

*SUBSTRUCTURE MATRIX OUTPUT: Write a substructure's recovery matrix, reduced stiffness matrix, mass matrix, load case vectors, and gravity load vectors to a file.

This option is used to write a substructure's recovery matrix, reduced stiffness matrix, mass matrix, load case vectors, and gravity load vectors to a file. It can be used only in a **SUBSTRUCTURE GENERATE* analysis.

Products: Abaqus/Standard

Type: History data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

References:

• *Generating Substructures*

Optional parameters:

FILE NAME

This parameter is intended for use with OUTPUT FILE=USER DEFINED.

Set this parameter equal to the name of the file (without an extension) to which the data will be written. The extension `.mtx` will be added to the file name provided by the user; see *Input Syntax Rules* for the syntax of such file names.

If the OUTPUT FILE parameter is omitted or set equal to RESULTS FILE, use of the FILE NAME parameter overrides the OUTPUT FILE setting; the data will be written to the named file, not to the results (`.fil`) file.

GRAVITY LOAD

Set GRAVITY LOAD=YES to write the substructure gravity load vectors (only available if the gravity load vectors are requested with the **SUBSTRUCTURE GENERATE* option). The default is GRAVITY LOAD=NO.

MASS

Set MASS=YES to write the substructure mass matrix (only available if the mass matrix is requested with the **SUBSTRUCTURE GENERATE* option). The default is MASS=NO.

OUTPUT FILE

Set OUTPUT FILE=RESULTS FILE (default) to write the data to the results (`.fil`) file in the format specified in *Results File*.

Set OUTPUT FILE=USER DEFINED to write the results to a user-specified file in the format of the **USER ELEMENT*, LINEAR option (*User-Defined Elements*). The name of the file is specified using the FILE NAME parameter.

RECOVERY MATRIX

Set RECOVERY MATRIX=YES to write the substructure recovery matrix (only available if the recovery matrix is requested with the **SUBSTRUCTURE GENERATE* option). The default is RECOVERY MATRIX=NO.

SLOAD

Set SLOAD=YES to write the substructure load case vectors. The default is SLOAD=NO.

STIFFNESS

Set STIFFNESS=YES to write the substructure stiffness matrix. The default is STIFFNESS=NO.

There are no data lines associated with this option.

*SUBSTRUCTURE MODAL DAMPING: Specify damping for a substructure property.

This option is used to specify diagonal modal damping coefficients associated with the substructure eigenmodes. Substructure modal damping contributes to the substructure damping matrices at the substructure usage stage. This option is used in conjunction with the **SUBSTRUCTURE PROPERTY* option. If this option is omitted, the element set containing the substructures for which the properties are being defined will exclude modal damping.

Products: Abaqus/Standard

Type: Model data

Level: Model

References:

- *Using Substructures*
- *Generating Substructures*

Optional, mutually exclusive parameters (if no parameter is specified, Abaqus assumes that the modal damping coefficients are provided on the data lines):

STRUCTURAL

Include this parameter to select structural damping, which means that the damping is proportional to the internal forces but opposite in direction to the velocity.

This parameter can be used only with the **STEADY STATE DYNAMICS*, **RANDOM RESPONSE*, SIM-based **MODAL DYNAMIC*, or **COMPLEX FREQUENCY* procedures (see *Mode-Based Steady-State Dynamic Analysis*, *Random Response Analysis*, *Transient Modal Dynamic Analysis*, and *Complex Eigenvalue Extraction*).

The value of the damping coefficient, γ, that multiplies the internal forces is entered on the data line.

VISCOUS

Set VISCOUS=FRACTION OF CRITICAL DAMPING to select modal damping using the damping coefficients given in this option. The data lines specify the modal damping values to be used in the analysis.

Set VISCOUS=RAYLEIGH to indicate that the damping for a particular mode is defined as $\alpha_M m_M + \beta_M k_M$, where α_M and β_M are coefficients defined on the first data line of the option and m_M is the modal mass and k_M is the modal stiffness for mode M.

Optional parameters:

DEFINITION

Set DEFINITION=MODE NUMBERS (default) to indicate that the damping values are given for the specified mode numbers.

Set DEFINITION=FREQUENCY RANGE to indicate that the damping values are given for the specified frequency ranges. Frequency ranges can be discontinuous.

Data lines to define a fraction of critical damping by specifying mode numbers (if no parameters are specified or if VISCOUS=FRACTION OF CRITICAL DAMPING and DEFINITION=MODE NUMBERS):

First line:

1. Mode number of the lowest mode of a range.
2. Mode number of the highest mode of a range. (If this entry is left blank, it is assumed to be the same as the previous entry so that values are being given for one mode only.)
3. Fraction of critical damping, ξ.

Repeat this data line as often as necessary to define modal damping for different modes.

Data lines to define Rayleigh damping by specifying mode numbers (VISCOUS=RAYLEIGH and DEFINITION=MODE NUMBERS):

First line:

1. Mode number of the lowest mode of a range.
2. Mode number of the highest mode of a range. (If this entry is left blank, it is assumed to be the same as the previous entry so that values are being given for one mode only.)
3. Mass proportional damping, α_M.
4. Stiffness proportional damping, β_M.

Repeat this data line as often as necessary to define modal damping for different modes.

Data lines to define structural damping by specifying mode numbers (STRUCTURAL and DEFINITION=MODE NUMBERS):

First line:

1. Mode number of the lowest mode of a range.
2. Mode number of the highest mode of a range. (If this entry is left blank, it is assumed to be the same as the previous entry so that values are being given for one mode only.)
3. Damping coefficient, γ.

Repeat this data line as often as necessary to define modal damping for different modes.

Data lines to define a fraction of critical damping by specifying frequency ranges (VISCOUS=FRACTION OF CRITICAL DAMPING and DEFINITION=FREQUENCY RANGE):

First line:

1. Frequency value (in cycles/time).
2. Fraction of critical damping, ξ.

Repeat this data line as often as necessary to define modal damping for different frequencies. Abaqus interpolates linearly between frequencies and keeps the damping value constant and equal to the closest specified value outside the frequency range.

Data lines to define Rayleigh damping by specifying frequency ranges (VISCOUS=RAYLEIGH and DEFINITION=FREQUENCY RANGE):

First line:

1. Frequency value (in cycles/time).
2. Mass proportional damping, α_M.
3. Stiffness proportional damping, β_M.

Repeat this data line as often as necessary to define modal damping for different frequencies. Abaqus interpolates linearly between frequencies and keeps the damping value constant and equal to the closest specified value outside the frequency range.

Data lines to define structural damping by specifying frequency ranges (STRUCTURAL and DEFINITION=FREQUENCY RANGE):

First line:

1. Frequency value (in cycles/time).
2. Damping coefficient, γ.

Repeat this data line as often as necessary to define modal damping for different frequencies. Abaqus interpolates linearly between frequencies and keeps the damping value constant and equal to the closest specified value outside the frequency range.

*SUBSTRUCTURE OUTPUT: Output the solution in the substructure usage analysis.

This option is used to output the solution at all of the substructure degrees of freedom in the substructure usage analysis. The full set of the substructure's degrees of freedom is formed by the retained nodal degrees of freedom and the generalized degrees of freedom associated with the substructure dynamic modes. Output of the solution at the generalized substructure degrees of freedom is not supported by other Abaqus output options.

This option can be used with a transient dynamic Abaqus/Standard analysis (**DYNAMIC* option).

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- **DYNAMIC*
- *Using Substructures*

Optional parameters:

ELSET

Set this parameter equal to the name of the element set that contains all substructure-type elements where you want to output the solution. If this parameter is omitted, the output is performed for all substructures in the model.

FORMAT

Set FORMAT=OP4 to output the substructure solution in the Output4 text format that can be used, for example, by the MSC Nastran finite element solver from MSC.Software Corporation or by the AVL EXCITE™ flexible body dynamics solver from AVL LIST GmbH.

Set FORMAT=SIM (default) to output the substructure solution on SIM, which is a high-performance database available in Abaqus.

There are no data lines associated with this option.

*SUBSTRUCTURE PATH: Enter into a substructure to obtain output or return back from a previously entered substructure.

This option is used to navigate through “levels” of substructures to obtain output of results.

Products: Abaqus/Standard

Type: History data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

References:

- *Using Substructures*

At least one of the following parameters is required:

ENTER ELEMENT

Include this parameter if output requests within a substructure are to follow. Set this parameter equal to the element number used for the substructure at the usage level at which Abaqus/Standard is operating before the appearance of this option.

LEAVE

Include this parameter to leave a substructure and go back up to its usage level.

There are no data lines associated with this option.

*SUBSTRUCTURE PROPERTY: Translate, rotate, and/or reflect substructures.

This option is used to define properties for a substructure. It is required for all substructures in a model.

Products: Abaqus/Standard

Type: Model data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

References:

- *Using Substructures*
- *Generating Frequency-Based Substructures*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the substructures for which properties are being defined.

Optional parameters:

DISPLAY

Set DISPLAY=YES to apply the coarse substructure display representation defined at substructure generation for the substructures for which properties are being defined.

Set DISPLAY=NO (default) to disable the coarse substructure display representation in this model for the substructures for which properties are being defined.

FREQUENCY BASED

This parameter is relevant to frequency-based substructures. Frequency-based substructures are used only in a direct steady-state dynamic analysis (see *Direct-Solution Steady-State Dynamic Analysis*). All other analyses use conventional substructures regardless of the availability of frequency-based substructures.

Set FREQUENCY BASED=ALL FREQUENCIES to use the frequency-based substructure at all frequency points of interest. If the frequency-based substructure is not available at a certain frequency, an operator-averaging-based approximate solution is given at that frequency. If a frequency of interest is beyond the range of frequencies at which the frequency-based substructure is constructed at generation, Abaqus issues an error.

Set FREQUENCY BASED=NO FREQUENCIES to disable the use of the frequency-based substructure at any frequency. The conventional substructure is used instead at all frequencies.

Set FREQUENCY BASED=MATCHED FREQUENCIES (default) to use the frequency-based substructure only at frequencies at which the frequency-based substructure is constructed at generation. At all nonmatching frequencies, the conventional substructure is used.

LARGE ROTATIONS

Set LARGE ROTATIONS=YES (default) to allow large rotations of the substructure in a geometrically nonlinear analysis.

Set LARGE ROTATIONS=NO to suppress large rotations of the substructure in a geometrically nonlinear analysis.

MONITOR

Set MONITOR=YES to monitor the substructure solution at the nodes selected during substructure generation for the substructures for which properties are being defined.

Set MONITOR=NO (default) to disable the solution monitoring.

POSITION TOL

Set this parameter equal to the tolerance on the distance between usage level nodes and the corresponding substructure nodes. If this parameter is omitted, the default is a tolerance of 10^{-4} times the largest overall dimension within the substructure. If the parameter is given with a value of 0.0, the position of the retained nodes is not checked.

Data line to translate a substructure:

First (and only) line:

1. Value of the translation to be applied in the global *X*-direction.
2. Value of the translation to be applied in the global *Y*-direction.
3. Value of the translation to be applied in the global *Z*-direction.

Data lines to translate and/or rotate a substructure:

First line:

1. Value of the translation to be applied in the global *X*-direction.
2. Value of the translation to be applied in the global *Y*-direction.
3. Value of the translation to be applied in the global *Z*-direction.

Enter values of zero to apply a pure rotation.

Second line:

1. Global *X*-coordinate of point *a* on the axis of rotation (see *Figure 1*).
2. Global *Y*-coordinate of point *a* on the axis of rotation.
3. Global *Z*-coordinate of point *a* on the axis of rotation.
4. Global *X*-coordinate of point *b* on the axis of rotation.
5. Global *Y*-coordinate of point *b* on the axis of rotation.
6. Global *Z*-coordinate of point *b* on the axis of rotation.
7. Angle of rotation about the axis *a*–*b*, in degrees.

Data lines to translate and/or reflect a substructure:

First line:

1. Value of the translation to be applied in the global *X*-direction.
2. Value of the translation to be applied in the global *Y*-direction.
3. Value of the translation to be applied in the global *Z*-direction.

Enter values of zero to apply a pure reflection.

Second line:

1. Enter a blank line.

Third line:

1. Global *X*-coordinate of point *a* in the plane of reflection (see *Figure 2*).
2. Global *Y*-coordinate of point *a* in the plane of reflection.
3. Global *Z*-coordinate of point *a* in the plane of reflection.
4. Global *X*-coordinate of point *b* in the plane of reflection.
5. Global *Y*-coordinate of point *b* in the plane of reflection.
6. Global *Z*-coordinate of point *b* in the plane of reflection.

Fourth line:

1. Global *X*-coordinate of point *c* in the plane of reflection.
2. Global *Y*-coordinate of point *c* in the plane of reflection.
3. Global *Z*-coordinate of point *c* in the plane of reflection.

Data lines to translate, rotate, and reflect a substructure:

First line:

1. Value of the translation to be applied in the global *X*-direction.
2. Value of the translation to be applied in the global *Y*-direction.
3. Value of the translation to be applied in the global *Z*-direction.

Second line:

1. Global *X*-coordinate of point *a* on the axis of rotation (see *Figure 1*).
2. Global *Y*-coordinate of point *a* on the axis of rotation.
3. Global *Z*-coordinate of point *a* on the axis of rotation.
4. Global *X*-coordinate of point *b* on the axis of rotation.
5. Global *Y*-coordinate of point *b* on the axis of rotation.
6. Global *Z*-coordinate of point *b* on the axis of rotation.
7. Angle of rotation about the axis *a*–*b*, in degrees.

Third line:

1. Global *X*-coordinate of point *a* in the plane of reflection (see *Figure 2*).
2. Global *Y*-coordinate of point *a* in the plane of reflection.
3. Global *Z*-coordinate of point *a* in the plane of reflection.
4. Global *X*-coordinate of point *b* in the plane of reflection.
5. Global *Y*-coordinate of point *b* in the plane of reflection.
6. Global *Z*-coordinate of point *b* in the plane of reflection.

Fourth line:

1. Global *X*-coordinate of point *c* in the plane of reflection.
2. Global *Y*-coordinate of point *c* in the plane of reflection.
3. Global *Z*-coordinate of point *c* in the plane of reflection.

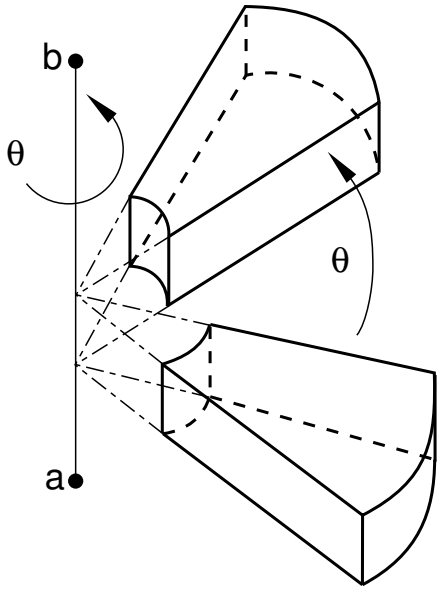


Figure 1: Substructure rotation.

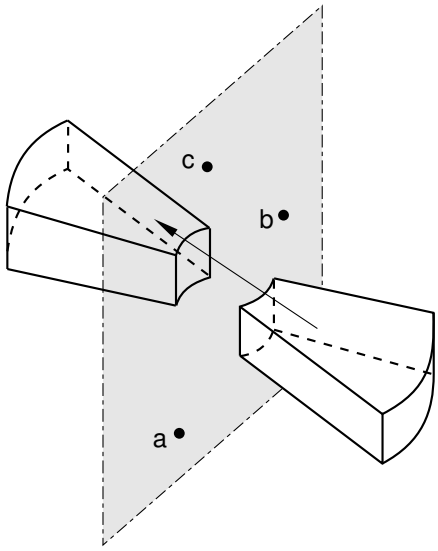


Figure 2: Substructure reflection. Points a, b, and c cannot be colinear.

*SUPERELASTIC: Define the data for the superelastic model.

This option is used to define the data for the superelastic material model. It can be used only in conjunction with the *ELASTIC option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *SUPERELASTIC HARDENING MODIFICATIONS
- *SUPERELASTIC HARDENING
- *Superelasticity*

Optional parameters:

NONASSOCIATED

Set this parameter equal to the volumetric transformation strain, ε_V^L. If this parameter is omitted, the volumetric transformation strain is assumed to be equal to the uniaxial transformation strain, ε^L.

Data lines to define the superelastic model:

First line:

1. Young's modulus of martensite, E_M.
2. Poisson's ratio of martensite, ν_M.
3. Uniaxial transformation strain, ε^L.
4. Stress at which the transformation begins during loading in tension, σ_{tL}^S.
5. Stress at which the transformation ends during loading in tension, σ_{tL}^E.
6. Stress at which the reverse transformation begins during unloading in tension, σ_{tU}^S.
7. Stress at which the reverse transformation ends during unloading in tension, σ_{tU}^E.
8. Stress at which the transformation begins during loading in compression, as a positive value, σ_{cL}^S.

Second line:

1. Reference temperature, T_0.
2. Slope of the stress versus temperature curve for loading, $\left(\frac{\delta\sigma}{\delta T}\right)_L$.
3. Slope of the stress versus temperature curve for unloading, $\left(\frac{\delta\sigma}{\delta T}\right)_U$.

*SUPERELASTIC HARDENING: Specify hardening data for martensite.

This option is used to define piecewise linear hardening of martensite.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- **SUPERELASTIC*
- **SUPERELASTIC HARDENING MODIFICATIONS*
- *Plasticity Model for Superelastic Materials*

There are no parameters associated with this option.

Data lines to specify hardening of martensite:

First line:

1. Yield stress.
2. Total strain.

Repeat this data line as often as necessary to define the dependence of the yield stress on the total strain.

*SUPERELASTIC HARDENING MODIFICATIONS: Specify the stress-strain response of superelastic materials as a function of plastic strain.

This option is used to specify the variation of the transformation stress levels of a superelastic material as a function of the plastic strain.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- **SUPERELASTIC*
- **SUPERELASTIC HARDENING*
- *Plasticity Model for Superelastic Materials*
- *USUPERELASHARDMOD*
- *VUSUPERELASHARDMOD*

Optional parameters:

USER

Include this parameter to specify the dependence of the transformation plateau on the plastic strain in a user subroutine.

Optional parameter when the USER parameter is included:

PROPERTIES

Set this parameter equal to the number of properties being entered. The properties are available for use in the user subroutine.

Data lines when the USER parameter is omitted:

First :

1. Stress at which the transformation begins during loading in tension, σ_{tL}^{S}.
2. Stress at which the transformation ends during loading in tension, σ_{tL}^{E}.
3. Stress at which the reverse transformation begins during unloading in tension, σ_{tU}^{S}.
4. Stress at which the reverse transformation ends during unloading in tension, σ_{tU}^{E}.
5. Plastic strain.

Repeat this data line as often as necessary to define the dependence of the transformation stress levels on the plastic strain.

Data lines when the USER and PROPERTIES parameters are included:

First:

1. Enter the material properties, eight per line.

Repeat this data line as often as necessary to define all material properties.

*SURFACE: Define a surface or region in a model.

This option is used to define surfaces for contact simulations, tie constraints, fasteners, and coupling, as well as regions for distributed surface loads, acoustic radiation, acoustic impedance, and output of integrated quantities on a surface. In Abaqus/Standard it is also used to define surfaces for cavity radiation analysis and assembly loads. In Abaqus/Explicit this option can also be used to define boundary regions on adaptive mesh domains.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Element-based surfaces are supported by the *Surface toolset*. Node-based surfaces are not supported; if node-based surfaces are imported into Abaqus/CAE, they are treated as sets.

References:

- *About Surfaces*
- *Element-Based Surface Definition*
- *Node-Based Surface Definition*
- *Analytical Rigid Surface Definition*
- *Eulerian Surface Definition*
- *Operating on Surfaces*
- *Eulerian Analysis*
- *About Contact Interactions*
- *RSURFU*
- *Modeling Discontinuities as an Enriched Feature Using the Extended Finite Element Method*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the surface.

Required parameter for cavity radiation simulations:

PROPERTY

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the name of the **SURFACE PROPERTY* definition associated with this surface. See *Cavity Radiation in Abaqus/Standard*.

Optional parameter for surface tension loads:

PROPERTY

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the name of the **SURFACE PROPERTY* definition associated with this surface. See *Surface Tension Loads*.

Optional parameters:

COMBINE

Set COMBINE=UNION to create a surface based on the union of two or more surfaces of the same type.

Set COMBINE=INTERSECTION to create a surface based on the intersection of two surfaces of the same type.

Set COMBINE=DIFFERENCE to create a surface based on the difference of two surfaces of the same type (the second surface is subtracted from the first).

Only the NAME parameter and, in cavity radiation simulations, the PROPERTY parameter can be used in conjunction with this parameter.

CROP

Include this parameter to create a new surface that will contain only those faces from an existing surface that have nodes in a specified rectangular box.

Only the NAME parameter and, in cavity radiation simulations, the PROPERTY parameter can be used in conjunction with this parameter.

DEFINITION

This parameter is relevant only for surfaces defined using TYPE=CUTTING SURFACE.

Set DEFINITION=COORDINATES (default) to define the cutting plane by giving the coordinates of a point on the cutting plane and the normal to the cutting plane.

Set DEFINITION=NODES to define the cutting plane by giving global node numbers for point *a* on the cutting plane and point *b* that lies off the cutting plane with the cutting plane normal determined by the vector from *a* to *b*.

FILLET RADIUS

This parameter can be used with TYPE=SEGMENTS, TYPE=CYLINDER, or TYPE= REVOLUTION to define a radius of curvature to smooth discontinuities between adjoining straight-line segments, adjoining circular-arc segments, and adjoining straight-line and circular-arc segments.

INTERNAL

Abaqus/CAE uses the INTERNAL parameter to identify surfaces that are created internally. The INTERNAL parameter is used only in models defined in terms of an assembly of part instances. The default is to omit the INTERNAL parameter.

REGION TYPE

This parameter is relevant only for surfaces defined on the boundary of an adaptive mesh domain. A surface defined in the interior of an adaptive mesh domain will move independently of the material unless the surface is constrained by mesh constraints. See *Defining ALE Adaptive Mesh Domains in Abaqus/Explicit*.

Set REGION TYPE=LAGRANGIAN to create a Lagrangian boundary region. The edge of a Lagrangian boundary region will follow the material while allowing adaptive meshing along the edge and within the interior of the region.

Set REGION TYPE=SLIDING (default) to create a sliding boundary region. The edge of a sliding boundary region will slide over the material. Adaptive meshing will occur on the edge and within the interior of the region. Mesh constraints are typically applied on the edge of a sliding boundary region to fix it spatially.

Set REGION TYPE=EULERIAN to create an Eulerian boundary region in an adaptive mesh domain. This option is used to create a boundary region across which material can flow. Mesh constraints must be used normal to an Eulerian boundary region to allow material to flow through the region.

If no mesh constraints are applied, an Eulerian boundary region will behave in the same way as a sliding boundary region.

TRIM

Set TRIM=YES to invoke trimming of open free surfaces. Set TRIM=NO to suppress surface trimming. The default value is TRIM=YES unless the surface is used as a main surface in a finite-sliding contact formulation in Abaqus/Standard or the surface is used with the contact pair algorithm in Abaqus/Explicit. TRIM=YES has no effect on surfaces used with the contact pair algorithm in Abaqus/Explicit.

TYPE

Set TYPE=ELEMENT (default) to define a free surface automatically for the elements specified or to define a surface on the elements by using element face identifiers.

Set TYPE=NODE to define a surface by specifying a list of nodes or node set labels.

Set TYPE=SEGMENTS to create a two-dimensional analytical surface in the (X, Y) plane for planar models or in the (r, z) plane for axisymmetric models by defining connected line segments.

Set TYPE=CYLINDER to define a three-dimensional analytical surface by sweeping connected line segments defined in a local (x, y) plane along a specified generator vector.

Set TYPE=REVOLUTION to define a three-dimensional analytical surface by providing connected line segments, which are given in an (r, z) plane and are rotated about an axis.

Set TYPE=CUTTING SURFACE to generate an interior element-based surface using a cutting plane passing through an element set. The generated surface is an approximation to the cutting plane.

Set TYPE=EULERIAN MATERIAL to define a surface on the exterior boundary of an Eulerian material instance. This option applies only to Abaqus/Explicit.

Set TYPE=USER to define an analytical surface via user subroutine *RSURFU* in Abaqus/Standard.

Set TYPE=XFEM to generate a crack surface for enriched cracked elements. This option applies only to Abaqus/Standard. Crack surfaces for XFEM are not supported with the **CONTACT PAIR* option.

Additional optional parameters used for contact pair analyses in Abaqus/Explicit:

MAX RATIO

Set this parameter equal to the upper bound of the ratio of a facet's contact thickness to its minimum edge (or diagonal) length. This ratio is computed after the SCALE THICK parameter is applied to the contact thickness. The contact thickness for individual facets will be adjusted, if necessary, to conform to this maximum ratio.

If this parameter is omitted, no upper bound will be enforced. If this parameter is included without a value specified, the default value is 0.6.

NO OFFSET

Include this parameter to indicate that this surface will ignore the midplane offset of any shell, membrane, or rigid elements that form the surface.

NO THICK

Include this parameter to indicate that this surface will ignore the thickness of any shell, membrane, or rigid elements that form the surface. A surface defined with this parameter cannot be used to define a double-sided surface or for self-contact.

SCALE THICK

This parameter applies only when the surface is used with the **CONTACT PAIR* option.

Set this parameter equal to the amount by which to scale the thicknesses of the underlying elements to compute the contact thicknesses. The default is 1.0 (i.e., no scaling).

Data lines for COMBINE=UNION:

First line:

1. List of surfaces.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

Data line for COMBINE=INTERSECTION or COMBINE=DIFFERENCE:

First (and only) line:

1. First surface name.
2. Second surface name.

For COMBINE=DIFFERENCE the second surface is subtracted from the first.

Data lines to define a surface when the CROP parameter is included:

First line:

1. Surface name.

Second line:

1. *X*-coordinate of the lower box corner.
2. *Y*-coordinate of the lower box corner.
3. *Z*-coordinate of the lower box corner.
4. *X*-coordinate of the opposite box corner.
5. *Y*-coordinate of the opposite box corner.
6. *Z*-coordinate of the opposite box corner.

Third line (optional):

1. *X*-coordinate of the first point defining the orientation.
2. *Y*-coordinate of the first point defining the orientation.
3. *Z*-coordinate of the first point defining the orientation.
4. *X*-coordinate of the second point defining the orientation.
5. *Y*-coordinate of the second point defining the orientation.
6. *Z*-coordinate of the second point defining the orientation.

Data lines to define a surface using elements or element sets when the TYPE=ELEMENT parameter is used:

First line:

1. Element set name or element number. In Abaqus/Explicit a blank data line can be specified to automatically generate the exterior (free) faces of every element in the model.
2. Face or edge identifier label (see *Element-Based Surface Definition* for the face and edge identifiers for various elements) or the “word” EDGE (optional).

Repeat this data line as often as necessary to define the surface.

Data lines to define a surface using nodes or node sets when the TYPE=NODE parameter is used:

First line:

1. Node set name or node number.
2. Cross-sectional area or distributing weight factor. In Abaqus/Standard contact calculations, the default is the area specified in the associated **SURFACE INTERACTION* option if the surface is defined in a contact pair; otherwise, a unit area is used. In Abaqus/Explicit the cross-sectional area used for contact pair calculations for node-based surface nodes is always set to 1.0 regardless of the value specified here. If the surface is used in a **COUPLING* or **SHELL TO SOLID COUPLING* definition, the default distributing weight factor is zero in Abaqus/Standard and 1.0 in Abaqus/Explicit.

Repeat this data line as often as necessary to define the surface.

Data lines to define a surface using a plane cutting through the given element sets when TYPE=CUTTING SURFACE, DEFINITION=COORDINATES:

First line:

1. *X*-coordinate of a point on the cutting plane in the initial configuration.
2. *Y*-coordinate of a point on the cutting plane in the initial configuration.
3. *Z*-coordinate of a point on the cutting plane in the initial configuration.
4. *X*-component of a normal to the cutting plane in the initial configuration.
5. *Y*-component of a normal to the cutting plane in the initial configuration.
6. *Z*-component of a normal to the cutting plane in the initial configuration.

Second line:

1. List of elements or element set labels to be cut by the cutting plane to generate an element-based surface that is an approximation to the cutting plane. A blank data line can be specified to generate a surface by cutting the whole model.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

Data lines to define a surface using a plane cutting through the given element sets when TYPE=CUTTING SURFACE, DEFINITION=NODES:

First line:

1. Node number of the node at point *a*.
2. Node number of the node at point *b*.

Second line:

1. List of elements or element set labels to be cut by the cutting plane to generate an element-based surface that is an approximation to the cutting plane. A blank data line can be specified to generate a surface by cutting the whole model.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

Data line to define surfaces created with TYPE=EULERIAN MATERIAL:

First line:

1. Name of the material instance as defined in the **EULERIAN SECTION*.

Abaqus/Explicit will automatically create a surface on the exterior of the given material.

Data lines to define a crack surface created with TYPE=XFEM:

First line:

1. List of names of the enriched features as defined with the **ENRICHMENT* option.

   Abaqus/Standard will automatically generate a crack surface as the enriched elements crack.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

No data lines are needed for TYPE=USER

Data lines to define surfaces created with TYPE=SEGMENTS:

First line:

1. The "word" START.
2. Global *X*-coordinate or *r*-coordinate of the starting point of the line segments.
3. Global *Y*-coordinate or *z*-coordinate of the starting point of the line segments.

Second and subsequent data lines define the various line, circular, and parabolic segments (see below for their format) that form the profile of the analytical surface.

Data lines to define surfaces created with TYPE=CYLINDER:

First line (leave blank if this surface is being defined within a part or a part instance):

1. Global *X*-coordinate of point *a*, the origin of the local (x, y) system (see *Figure 1*).
2. Global *Y*-coordinate of point *a*, the origin of the local (x, y) system.
3. Global *Z*-coordinate of point *a*, the origin of the local (x, y) system.
4. Global *X*-coordinate of point *b* on the local *x*-axis.
5. Global *Y*-coordinate of point *b* on the local *x*-axis.
6. Global *Z*-coordinate of point *b* on the local *x*-axis.

Second line (leave blank if this surface is being defined within a part or a part instance):

1. Global *X*-coordinate of point *c* on the local cylinder generator vector.
2. Global *Y*-coordinate of point *c* on the local cylinder generator vector.
3. Global *Z*-coordinate of point *c* on the local cylinder generator vector.

Third line:

1. The "word" START.
2. Local *x*-coordinate of the starting point of the line segments.
3. Local *y*-coordinate of the starting point of the line segments.

Fourth and subsequent data lines define the various line, circular, and parabolic segments (see below for their format) that form the profile of the analytical surface.

Data lines to define surfaces created with TYPE=REVOLUTION:

First line (leave blank if this surface is being defined within a part or a part instance):

1. Global *X*-coordinate of point *a*, the origin of the local (r, z) system (see *Figure 2*).
2. Global *Y*-coordinate of point *a*, the origin of the local (r, z) system.

3. Global Z-coordinate of point a, the origin of the local (r, z) system.
4. Global X-coordinate of point b on the symmetry axis (the local z-axis).
5. Global Y-coordinate of point b on the symmetry axis (the local z-axis).
6. Global Z-coordinate of point b on the symmetry axis (the local z-axis).

Second line:

1. The "word" START.
2. Local r-coordinate of the starting point of the line segments.
3. Local z-coordinate of the starting point of the line segments.

Third and subsequent data lines define the various line, circular, and parabolic segments (see below for their format) that form the profile of the analytical surface.

Data lines that define the line segments that form the analytical surface for TYPE=SEGMENTS, TYPE=CYLINDER, and TYPE=REVOLUTION:

Data line to define a straight line segment:

1. The "word" LINE.
2. x-coordinate of the end point of the line.
3. y-coordinate of the end point of the line.

Data line to define a circular arc segment (the arc must be less than 179.74°):

1. The "word" CIRCL.
2. x-coordinate of the end point of the circular arc.
3. y-coordinate of the end point of the circular arc.
4. x-coordinate of the center (origin) of the circular arc.
5. y-coordinate of the center (origin) of the circular arc.

Data line to define a parabolic arc segment:

1. The "word" PARAB.
2. Local x-coordinate of the middle point along the parabolic arc.
3. Local y-coordinate of the middle point along the parabolic arc.
4. Local x-coordinate of the end point of the parabolic arc.
5. Local y-coordinate of the end point of the parabolic arc.

For surfaces created with TYPE=SEGMENTS, the x- and y-coordinates are the global X- and Y-coordinates or r- and z-coordinates. For surfaces created with TYPE=CYLINDER, the x- and

y-coordinates are the local x- and y-coordinates. For surfaces created with TYPE=REVOLUTION, the x- and y-coordinates are the local r- and z-coordinates.

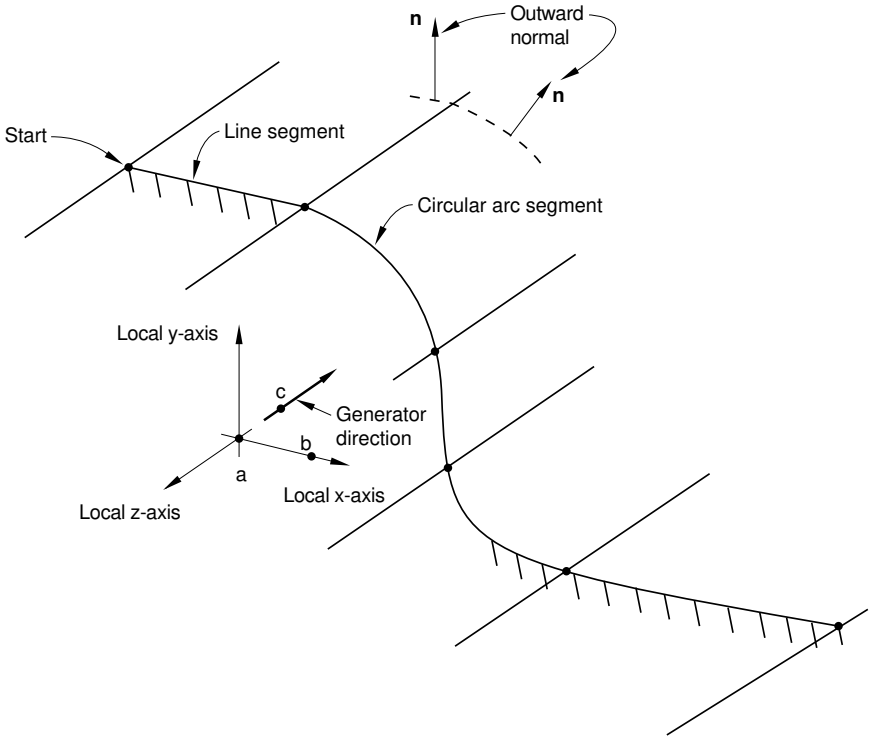


Figure 1: *SURFACE, TYPE=CYLINDER.

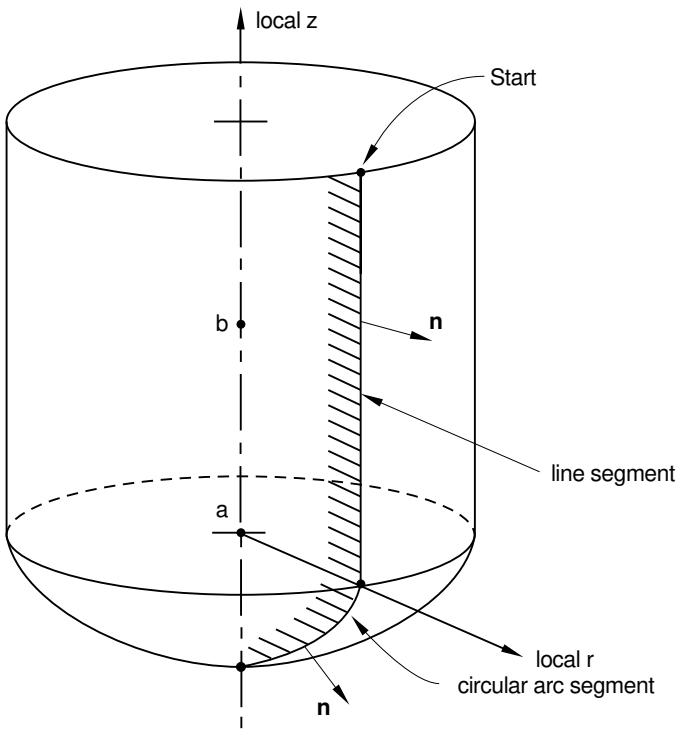


Figure 2: *SURFACE, TYPE=REVOLUTION.

*SURFACE BEHAVIOR: Define alternative pressure-overclosure relationships for contact.

This option is used to modify the default hard contact pressure-overclosure relationship in a mechanical contact analysis. Mechanical interactions normal to the surfaces are influenced by this option. It must be used in conjunction with the **SURFACE INTERACTION* option or in an Abaqus/Standard analysis with the **GAP* option or the **INTERFACE* option. By default, Abaqus/Standard will determine whether the contact constraint will be enforced with or without a Lagrange multiplier. You can use the **CONTACT CONTROLS* option to override the default.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data in Abaqus/Standard; Model or history data in Abaqus/Explicit

Level: Part, Part instance, Assembly, Model in Abaqus/Standard; Step in Abaqus/Explicit

Abaqus/CAE: Interaction module

References:

- *About Mechanical Contact Properties*
- *Contact Pressure-Overclosure Relationships*
- *Discrete Element Method*
- **CONTACT CONTROLS*
- **GAP*
- **INTERFACE*
- **SURFACE INTERACTION*

Optional, mutually exclusive parameters:

AUGMENTED LAGRANGE

This parameter applies only to Abaqus/Standard analyses with a default “hard” pressure-overclosure relationship.

Include this parameter to choose the augmented Lagrange method for enforcement of the contact constraint. See *Contact Constraint Enforcement Methods in Abaqus/Standard* for a discussion of the default penalty stiffness and penetration tolerance associated with this method (the default penalty stiffness used by this method is often stiffer than that with the straight penalty method). You can specify or modify the penalty stiffness on the data line.

DIRECT

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to choose direct enforcement of contact constraints without approximation or use of augmentation iterations.

PENALTY

This parameter applies only to Abaqus/Standard analyses with a default “hard” pressure-overclosure relationship.

Set PENALTY=LINEAR (default) to choose the linear penalty method for enforcement of the contact constraint. See *Contact Constraint Enforcement Methods in Abaqus/Standard* for a discussion of the default linear penalty stiffness. You can specify or modify the penalty stiffness on the data line.

Set PENALTY=NONLINEAR to choose the nonlinear penalty method for enforcement of the contact constraint. See *Contact Constraint Enforcement Methods in Abaqus/Standard* for a discussion of the default nonlinear penalty stiffness. You can specify or modify the final nonlinear penalty stiffness and other nonlinear penalty control parameters on the data line.

Optional parameters:

NO SEPARATION

Include this parameter to prevent any separation of the two surfaces once contact has been established.

PRESSURE-OVERCLOSURE

Use this parameter to choose a contact pressure-overclosure relationship other than the default hard contact.

Set PRESSURE-OVERCLOSURE=HARD (default) to choose a pressure-overclosure relationship without physical softening. Some numerical softening will occur if a penalty or augmented Lagrange constraint enforcement method is used.

Set PRESSURE-OVERCLOSURE=EXPONENTIAL to define an exponential pressure-overclosure relationship.

Set PRESSURE-OVERCLOSURE=LINEAR to define a linear pressure-overclosure relationship.

Set PRESSURE-OVERCLOSURE=SCALE FACTOR to define a piecewise linear pressure-overclosure relationship based on scaling the default contact stiffness. This option is available only for the general contact algorithm in Abaqus/Explicit.

Set PRESSURE-OVERCLOSURE=TABULAR to define a piecewise linear pressure-overclosure relationship in tabular form.

Set PRESSURE-OVERCLOSURE=HERTZ to define a Hertz-type pressure-overclosure relationship for discrete element method analysis.

Set PRESSURE-OVERCLOSURE=JKR to define a JKR-type pressure-overclosure relationship for discrete element method analysis.

Set PRESSURE-OVERCLOSURE=LKM to define contact for lumped kinetic molecular method analysis.

If a contact area is not defined, such as might occur for node-based surfaces or for GAP- or ITT-type contact elements, “pressure” should be interpreted as force. For contact with three-dimensional beams or trusses, “pressure” should be interpreted as force per unit length.

When used to modify the default surface behavior, the PRESSURE-OVERCLOSURE parameter cannot be used with the NO SEPARATION parameter in an Abaqus/Standard analysis; the same restriction applies to Abaqus/Explicit when general contact is used except for PRESSURE-OVERCLOSURE=LINEAR.

Optional data line for AUGMENTED LAGRANGE and PENALTY=LINEAR:

First (and only) line:

1. User-defined linear penalty stiffness. If this field is left blank or is zero, the linear penalty stiffness has the default value or is multiplied by the scale factor given in the third field of the data line.
2. Clearance at which the contact pressure is zero, c_0. The default is zero.
3. Scale factor for the default linear penalty stiffness or for the linear penalty stiffness if specified in the first field of the data line. The default is one.

Optional data line for PENALTY=NONLINEAR:

First (and only) line:

1. User-defined final nonlinear penalty stiffness. If this field is left blank or is zero, the final nonlinear penalty stiffness has the default value or is multiplied by the scale factor given in the third field of the data line.
2. Clearance at which the contact pressure is zero, c_0 (see *Figure 1*). The default is zero.
3. Scale factor for the default final nonlinear penalty stiffness or for the final nonlinear penalty stiffness if specified in the first field of the data line. The default is one.
4. Scale factor for the upper quadratic limit d (see *Figure 1*), which is equal to the scale factor times the characteristic contact facet length. The default is 0.03.
5. The ratio of initial penalty stiffness over the final penalty stiffness. The default is 0.01.
6. The ratio $(e-c_0)/(d-c_0)$ that defines the lower quadratic limit e (see *Figure 1*). The default is 1/3.

Data line for PRESSURE-OVERCLOSURE=EXPONENTIAL:

First (and only) line:

1. Clearance at which the contact pressure is zero, c_0 (see *Figure 2*).
2. Pressure at zero clearance, p_0.

The following data item is available only in Abaqus/Explicit analyses:

7. Value of the maximum stiffness, k_{max}. When using penalty contact, large stiffness values obtained from the exponential law might significantly lower the stable time increment size. Choose a suitable maximum stiffness to mitigate this effect (see *Figure 2*). By default, k_{max} is set equal to infinity for kinematic contact and the default penalty stiffness for penalty contact.

Data line for PRESSURE-OVERCLOSURE=LINEAR:

First (and only) line:

1. Slope of the pressure-overclosure curve. This value must be positive.

Data line for PRESSURE-OVERCLOSURE=SCALE FACTOR:

First (and only) line:

1. Overclosure factor, r (see *Figure 3*), which defines the overclosure measure (used to delineate the segments of the pressure-overclosure curve) as a percentage of the minimum element size in the contact region. This value must be greater than zero. Leave blank if an overclosure measure (second data item) is specified.
2. Overclosure measure, d (see *Figure 3*), which defines the overclosure measure (used to delineate the segments of the pressure-overclosure curve) directly. This value must be greater than zero. Leave blank if an overclosure factor (first data item) is specified.
3. Stiffness scale factor, s (see *Figure 3*), which defines the geometric scaling of the “base” stiffness and must be greater than one.
4. Initial stiffness scale factor, s_0 (see *Figure 3*), which defines an additional scale factor for the “base” default contact stiffness and must be greater than zero. The default value is one.

Data lines for PRESSURE-OVERCLOSURE=TABULAR:

First line:

1. Pressure.
2. Overclosure.

Repeat this data line in ascending order of overclosure value as often as necessary to define the overclosure as a function of pressure. A minimum of two data lines are required. The pressure-overclosure relationship is extrapolated beyond the last overclosure point by continuing the same slope (see *Figure 4*).

Optional data line for PRESSURE-OVERCLOSURE=HERTZ:

First (and only) line:

1. User-defined limiting value of Hertz stiffness.

Optional data line for PRESSURE-OVERCLOSURE=JKR:

First (and only) line:

1. Surface energy per unit area.
2. User-defined limiting value of stiffness.
3. Specify the label SHIFTED for the shifted JKR force-displacement curve; otherwise, leave this term blank.

There are no data lines if PRESSURE-OVERCLOSURE=LKM

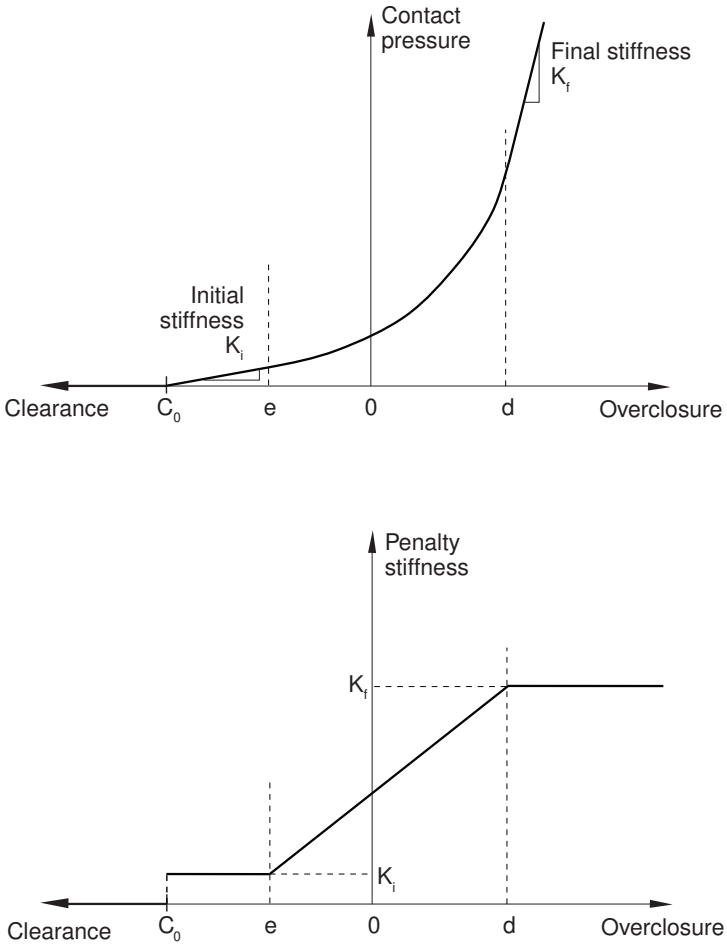


Figure 1: Nonlinear penalty pressure-overclosure relationship.

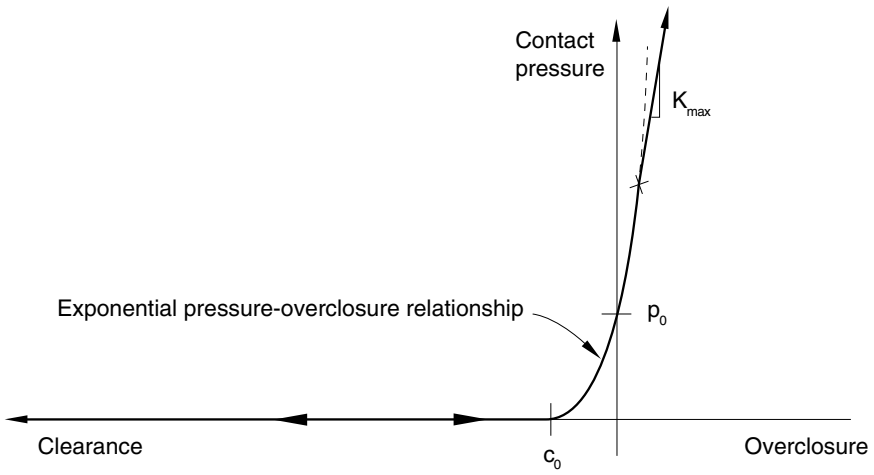


Figure 2: Exponential pressure-overclosure relationship.

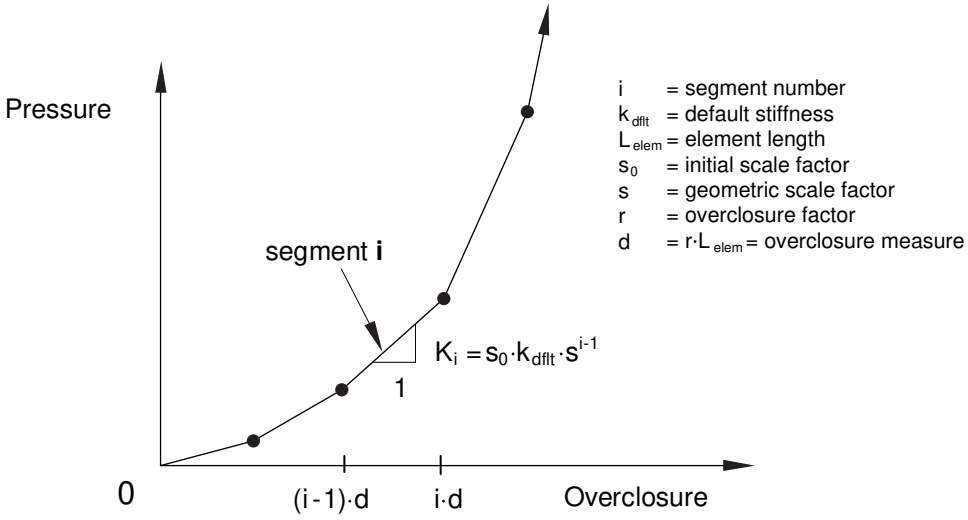


Figure 3: Scale factor pressure-overclosure relationship.

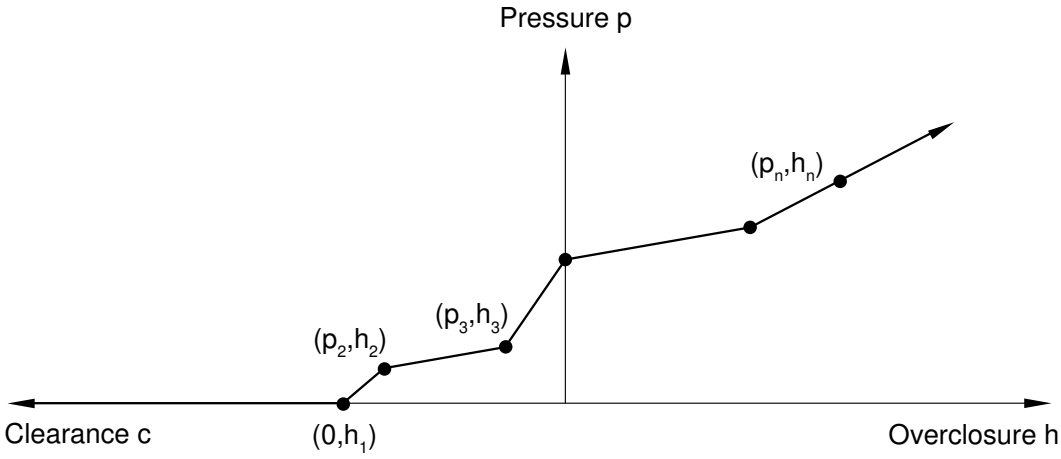


Figure 4: Pressure-overclosure relationship defined in tabular form.

*SURFACE FLAW: Define geometry of surface flaws.

This option is used with line spring elements to define the geometry of the part-through crack of the shell.

Products: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

• *Line Spring Elements for Modeling Part-through Cracks in Shells*

Required parameters:

SIDE

Set SIDE=POSITIVE or SIDE=NEGATIVE to indicate which surface is cracked.

Optional parameters:

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

Data lines to define the flaw:

First line:

1. Node number or node set label. This node or node set can be at either side of the crack for element LS6.
2. Crack depth at this location.

Repeat this data line as often as necessary to define the crack depth at all nodal locations along the crack.

*SURFACE INTERACTION: Define surface interaction properties.

This option is used to create a surface interaction property definition. The surface interaction properties will govern any contact interactions that reference this surface interaction.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data in Abaqus/Standard; Model or history data in Abaqus/Explicit

Level: Model in Abaqus/Standard; Model or Step in Abaqus/Explicit

Abaqus/CAE: Interaction module

References:

- *Assigning Surface Properties for Contact Pairs in Abaqus/Explicit*
- *About Mechanical Contact Properties*
- *Contact Pressure-Overclosure Relationships*
- *Contact Damping*
- *Frictional Behavior*
- *User-Defined Interfacial Constitutive Behavior*
- *Thermal Contact Properties*
- *Electrical Contact Properties*
- *Pore Fluid Contact Properties*
- *UINTER*
- *VUINTER*
- *VUINTERACTION*
- *Defining Contact of Cracked Element Surfaces Using a Small-Sliding Formulation*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to this surface interaction property.

Use this label on the INTERACTION parameter of the **CONTACT PAIR* option (for an Abaqus/Standard contact analysis or an Abaqus/Explicit analysis using the contact pair algorithm) or on the data line of the **CONTACT PROPERTY ASSIGNMENT* option (for an Abaqus/Explicit analysis using the general contact algorithm) to assign this surface interaction property to a contact interaction.

Optional parameters:

DEPVAR

This parameter is relevant only when the USER parameter is included.

Set DEPVAR equal to the number of state-dependent variables required for user subroutine *UINTER* in an Abaqus/Standard analysis or for user subroutine *VUINTER* or *VUINTERACTION* in an Abaqus/Explicit analysis. The default is DEPVAR=0.

PAD THICKNESS

This parameter applies only to Abaqus/Explicit analyses using the contact pair algorithm.

Set this parameter equal to the thickness of an interfacial layer between the contacting surfaces. The value can be positive or negative.

PROPERTIES

This parameter is relevant only when the USER parameter is included.

Set this parameter equal to the number of property values needed as data to define the surface interaction model in user subroutine *UINTER* in an Abaqus/Standard analysis or in user subroutine *VUINTER* or *VUINTERACTION* in an Abaqus/Explicit analysis. The default is PROPERTIES=0.

This parameter is ignored when the option is used in association with a connector element.

RATE FACTOR

This parameter applies only to Abaqus/Explicit analyses using general contact in conjunction with rate-dependent surface-based cohesive damage or rate-dependent friction behavior.

Set this parameter equal to the factor ω in the equation that is used to filter the rate of separation or rate of slip (see *Filtering Rates of Relative Motion*). The default value is 0.9.

RATE INTERPOLATION

This parameter applies only to Abaqus/Explicit analyses using general contact and influences the interpolation of friction coefficients and cohesive damage parameters with respect to the slip rate and rate of separation.

Set RATE INTERPOLATION=LOGARITHMIC (default) to use logarithmic interpolation with respect to slip rate and rate of separation dependencies.

Set RATE INTERPOLATION=LINEAR to use linear interpolation with respect to slip rate and rate of separation dependencies.

TRACKING THICKNESS

Set this parameter equal to the thickness that determines the contacting surfaces to be tracked.

In Abaqus/Standard and Abaqus/Explicit the tracking range will be at least as large as the setting of this parameter; an internally computed default will remain in effect if this parameter setting is less than the default. This parameter extends the range over which contact opening distance (COPEN) output is provided for separated surfaces.

In Abaqus/Explicit this parameter also applies if user subroutine *VUINTER* or *VUINTERACTION* is in effect. Only contacting surfaces whose proximity is within this thickness are available for user-defined interactions. This parameter affects only node-to-surface contact, and the input value for this parameter cannot be negative. An internal default value is used if a zero value is input or if the parameter is omitted.

UNSYMM

This parameter applies only to Abaqus/Standard analyses in which the USER parameter is included.

Include this parameter when the interface stiffness matrix is not symmetric. This parameter causes Abaqus/Standard to use its unsymmetric equation solution procedures.

USER

In an Abaqus/Standard analysis, include this parameter if the surface interaction model is to be defined in user subroutine *UINTER*

In an Abaqus/Explicit analysis, include this parameter without any assignment if the surface interaction model is to be defined in user subroutine *VUINTER*. Set USER=INTERACTION if the surface interaction model is to be defined in user subroutine *VUINTERACTION*. *VUINTER* is applicable to contact pairs, whereas *VUINTERACTION* is applicable to general contact.

When this parameter is included, the **SURFACE BEHAVIOR* option and its various suboptions cannot be used under the same interaction definition.

UXFEMCRACK

This parameter applies only to Abaqus/Standard analyses.

Include this parameter if the surface interaction between the cracked element surfaces for enriched elements using a small-sliding formulation is to be defined in user subroutine *UXFEMCRACK*.

Optional data line if both the USER and the UXFEMCRACK parameters are omitted for two-dimensional models in Abaqus/Standard or for contact pairs involving node-based surfaces in Abaqus/Standard:

First (and only) line:

1. Out-of-plane thickness of the surface or cross-sectional area for every node in the node-based surface. The default value is 1.0. Abaqus/Standard uses this value to compute the contact nodal areas and contact stresses.

Data lines to define the surface interaction in an Abaqus/Standard analysis if the USER or the UXFEMCRACK parameter is used:

First line:

1. Out-of-plane thickness of the surface for two-dimensional models or cross-sectional area for every node in the node-based surface. The default value for both is 1.0. In other situations, enter a blank line.

Second line (needed only if the PROPERTIES parameter is used):

1. Enter the values of the surface interaction properties, eight per line.

Repeat this data line as often as necessary to define all material constants.

Data lines to define the surface interaction in an Abaqus/Explicit analysis if the PROPERTIES parameter is used:

First line:

1. Enter a blank line.

Second line:

1. Enter the values of the surface interaction properties, eight per line.

Repeat this data line as often as necessary to define all material constants.

*SURFACE PROPERTY: Define named surface properties for cavity radiation or contact-based wear.

This option is used to define surface properties for cavity radiation analysis or contact-based wear modeling.

In the case of cavity radiation, this option must immediately precede the **EMISSIVITY* option. For contact based wear, this option must precede the **WEAR SURFACE PROPERTIES* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- *Cavity Radiation in Abaqus/Standard*
- *Contact Wear*
- **EMISSIVITY*
- **WEAR SURFACE PROPERTIES*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the property name for cross-referencing to the surface definitions for which properties are being defined. For cavity radiation, only element-based surfaces can be used.

There are no data lines associated with this option.

*SURFACE PROPERTY ASSIGNMENT: Assign surface properties to a surface for the general contact algorithm.

This option is used to modify surface properties for surfaces that are involved in general contact interactions. It must be used in conjunction with the *CONTACT option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data in Abaqus/Standard; Model or history data in Abaqus/Explicit

Level: Model in Abaqus/Standard; Model or Step in Abaqus/Explicit

Abaqus/CAE: Interaction module

References:

- *Surface Properties for General Contact in Abaqus/Standard*
- *Assigning Surface Properties for General Contact in Abaqus/Explicit*
- *Contact Wear*
- *CONTACT
- *SURFACE PROPERTY
- *WEAR SURFACE PROPERTIES

Required parameters:

PROPERTY

Use this parameter to specify the property type being assigned. To modify more than one type of surface property, include the *SURFACE PROPERTY ASSIGNMENT option more than once with different values for this parameter.

Set PROPERTY=BEAM SMOOTHING to control smoothing of beam segments in beam-to-beam contact. This parameter value applies only to Abaqus/Standard analyses.

Set PROPERTY=CRUSH TRIGGER to control crush initiation of a crushable surface. This parameter value applies only to Abaqus/Explicit analyses.

Set PROPERTY=DISTRIBUTION FACTOR to specify the preferred fraction of frictional work directed to this surface. This parameter value applies only to Abaqus/Explicit analyses.

Set PROPERTY=FEATURE EDGE CRITERIA to control which primary feature edges and secondary feature edges should be activated in the general contact domain.

Set PROPERTY=FRICTION to assign a surface-based friction coefficient. This parameter value applies only to Abaqus/Explicit analyses.

Set PROPERTY=GEOMETRIC CORRECTION to assign geometric corrections.

Set PROPERTY=OFFSET FRACTION to assign the surface offset as a fraction of the surface thickness.

Set PROPERTY=ORIENTATION to assign a coordinate system for local tangent directions to the surface and/or specify preferential frictional directions to the surface in the context of surface-based anisotropic friction. This parameter value applies only to Abaqus/Explicit analyses.

Set PROPERTY=THICKNESS to assign the surface thickness.

Set PROPERTY=VERTEX CRITERIA to control which nodes of feature edges should be considered vertex nodes in the general contact domain. This parameter value applies only to Abaqus/Standard analyses.

Set PROPERTY=WEAR to assign contact wear properties.

Optional parameters:

DEFINITION

This parameter is relevant only for surface properties defined using PROPERTY=GEOMETRIC CORRECTION.

Set DEFINITION=COORDINATES (default) to define the parameters for the geometry correction by giving the coordinates of one or more points, depending on the surface shape.

Set DEFINITION=NODES to define the parameters for the geometry correction by giving the global number of one or more nodes, depending on the surface shape.

FRICTION ANISOTROPY

This parameter applies only to Abaqus/Explicit analyses with PROPERTY=ORIENTATION.

Set FRICTION ANISOTROPY=EPSILON (default) to define the frictional directional preference factor ϵ on the data lines.

Set FRICTION ANISOTROPY=RATIO to define the ratio r on the data lines.

Data lines for PROPERTY=BEAM SMOOTHING:

First line:

1. Surface name.
2. A scalar value between 0.0 and 0.5 (0.2 is the default).

Repeat this data line as often as necessary. If the beam smoothing assignments overlap, the last assignment applies in the overlap region.

Data lines for PROPERTY=CRUSH TRIGGER:

First line:

1. Surface name or material name.
2. Specify TRIGGER (default), NO TRIGGER, or NO CRUSH. If NO TRIGGER or NO CRUSH is specified, no further entries are needed.
3. A scalar value representing the stress required to initiate crushable behavior as a factor of the crush stress (**CRUSH STRESS*) defined on the material associated with the surface.
4. Crush initiation angle.
5. Crush continuation angle.
6. Specify SURFACE (default) or omit if the first entry is a surface name. Specify MATERIAL if the first entry is a material name.

Repeat this data line as often as necessary. If the crush trigger assignments overlap, the last assignment applies in the overlap region.

Data lines for PROPERTY=DISTRIBUTION FACTOR:

First line:

1. Surface name or material name.

2. Preferred fraction of frictional work of an interaction directed to this surface. The default value is 0.5.
3. Specify SURFACE (default) or omit if the first entry is a surface name. Specify MATERIAL if the first entry is a material name.

Repeat this data line as often as necessary. If the distribution factor assignments overlap, the last assignment applies in the overlap region.

Data lines for PROPERTY=FEATURE EDGE CRITERIA in Abaqus/Standard:

First line:

1. Surface name or material name. If this entry is omitted, a default surface that encompasses the entire general contact domain is assumed.
2. A scalar value between 0° and 180° representing the cutoff feature angle (in degrees), PERIMETER EDGES, or NO FEATURE EDGES. The default is 45°. This field activates edges to participate in edge-to-surface contact.
3. This field is intentionally left blank and is unused.
4. A scalar value between 0° and 180° representing the cutoff feature angle (in degrees), PERIMETER EDGES, or NO FEATURE EDGES. The entry on this field activates edges to participate in edge-to-edge contact. The default is NO FEATURE EDGES.
5. Specify SURFACE (default) or omit if the first entry is a surface name. Specify MATERIAL if the first entry is a material name.

Repeat this data line as often as necessary. If the feature edge criteria assignments overlap, the last assignment applies in the overlap region.

Data lines for PROPERTY=FEATURE EDGE CRITERIA in Abaqus/Explicit:

First line:

1. Surface name or material name. If this entry is omitted, a default surface that encompasses the entire general contact domain is assumed. A non-blank entry is required if the ALL EDGES or PICKED EDGES options are specified for the second entry and ORIGINAL is in effect for the fifth entry.
2. The criterion for primary feature edges. A scalar value representing the cutoff feature angle in degrees (the default value is 40°), PERIMETER EDGES, ALL EDGES, PICKED EDGES, or NO FEATURE EDGES.
3. The criterion for secondary feature edges. A scalar value representing the cutoff feature angle in degrees (the default is 30°), ALL REMAINING EDGES, PERIMETER EDGES, or PICKED EDGES.
4. This field is intentionally left blank and is unused.
5. The word ORIGINAL to apply feature edge criteria statically or the word CURRENT (default) to apply feature edge criteria for this surface dynamically.
6. This field is intentionally left blank and is unused.
7. Specify SURFACE (default) or omit if the first entry is a surface name. Specify MATERIAL if the first entry is a material name.

Repeat this data line as often as necessary. If the feature edge criteria assignments overlap, the last assignment applies in the overlap region.

Data lines for PROPERTY=FRICTION:

First line:

1. Surface name or material name.
2. Surface-based friction coefficient.
3. Specify SURFACE (default) or omit if the first entry is a surface name. Specify MATERIAL if the first entry is a material name.

Repeat this data line as often as necessary. If the feature edge criteria assignments overlap, the last assignment applies in the overlap region.

Data lines for PROPERTY=GEOMETRIC CORRECTION, DEFINITION=COORDINATES to define smoothing on regions of surfaces:

First line:

1. Surface name.
2. Specify CIRCUMFERENTIAL, SPHERICAL, TOROIDAL, NONE, or leave blank. If NONE or blank is specified, no further entries are required.
3. Global *X*-coordinate of point *a* (see *Figure 1* through *Figure 4*, depending on surface shape).
4. Global *Y*-coordinate of point *a*.
5. Global *Z*-coordinate of point *a* (only for three-dimensional).
6. Global *X*-coordinate of point *b* (only for three-dimensional circumferential or toroidal; see *Figure 2* or *Figure 4*, depending on surface shape).
7. Global *Y*-coordinate of point *b* (only for three-dimensional circumferential or toroidal).
8. Global *Z*-coordinate of point *b* (only for three-dimensional circumferential or toroidal).
9. Distance *R* (only for three-dimensional toroidal; see *Figure 4*).

Repeat this data line as often as necessary. If the geometry correction assignments overlap, the last assignment applies in the overlap region. For toroidal shapes the line joining point *a* and the center of the circular arc should be perpendicular to the axis of revolution.

Data lines for PROPERTY=GEOMETRIC CORRECTION, DEFINITION=NODES to define smoothing on regions of surfaces:

First line:

1. Surface name.
2. Specify CIRCUMFERENTIAL, SPHERICAL, TOROIDAL, NONE, or leave blank. If NONE or blank is specified, no further entries are required.
3. Node number of the node at point *a* (see *Figure 1* through *Figure 4*, depending on surface shape).
4. Node number of the node at point *b* (only for three-dimensional circumferential or toroidal; see *Figure 2* or *Figure 4*, depending on surface shape).
5. Distance *R* (only for three-dimensional toroidal; see *Figure 4*).

Repeat this data line as often as necessary. If the geometry correction assignments overlap, the last assignment applies in the overlap region. For toroidal shapes the line joining point a and the center of the circular arc should be perpendicular to the axis of revolution.

Data lines for PROPERTY=OFFSET FRACTION:

First line:

1. Surface name or material name. If this entry is omitted, a default surface that encompasses the entire general contact domain is assumed. Faces specified on elements other than shell elements, membrane elements, rigid elements, and surface elements will be ignored.
2. ORIGINAL (default), SPOS, SNEG, or a value between –0.5 and 0.5. The offset defines the distance (as a fraction of the thickness) from the midsurface to the reference surface (containing the nodes of the element). Positive values of the offset are in the positive element normal direction. The default is ORIGINAL, which indicates that the offset specified in element section definitions (via the OFFSET parameter on **SHELL SECTION* or **RIGID BODY*) will be used.
3. Specify SURFACE (default) or omit if the first entry is a surface name. Specify MATERIAL if the first entry is a material name.

Repeat this data line as often as necessary. If the offset fraction assignments overlap, the last assignment applies in the overlap region.

Data lines for PROPERTY=ORIENTATION:

First line:

1. Surface name or material name.
2. A valid orientation system name. If the orientation system name is omitted or the surface is an analytic rigid one, a default orientation is used.
3. Scalar value of the extra rotation (in degrees) applied to the orientation system once it has been projected to the surface. The default value is 0.
4. Scalar value of the frictional directional preference factor ϵ (within the range –1.0 to 1.0, default 0.0) or the friction coefficients ratio r (greater than 0.0, default 1.0) depending on the value of the FRICTION ANISOTROPY parameter.
5. Specify SURFACE (default) or omit if the first entry is a surface name. Specify MATERIAL if the first entry is a material name.

Repeat this data line as often as necessary. If the directional preference assignments overlap, the last assignment applies in the overlap region.

Data lines for PROPERTY=THICKNESS:

First line:

1. Surface name or material name. If this entry is omitted, a default surface that encompasses the entire general contact domain is assumed.
2. ORIGINAL (default), CURRENT (available only in Abaqus/Explicit), THINNING (available only in Abaqus/Explicit), or a scalar value representing a nominal thickness.
3. A constant scaling factor (default is 1.0).
4. Specify SURFACE (default), or omit if the first entry is a surface name. Specify MATERIAL if the first entry is a material name.

Repeat this data line as often as necessary. If the thickness assignments overlap, the last assignment applies in the overlap region.

Data lines for PROPERTY=VERTEX CRITERIA:

First line:

1. Surface name or material name. If this entry is omitted, a default surface that encompasses the entire general contact domain is assumed.
2. ALL VERTICES, NO VERTICES, or a scalar value between 10° and 90° representing the vertex angle threshold (in degrees). The default is 20°. This field activates vertex nodes to participate in vertex-to-surface contact.
3. Specify SURFACE (default) or omit if the first entry is a surface name. Specify MATERIAL if the first entry is a material name.

Repeat this data line as often as necessary. If the vertex criteria assignments overlap, the last assignment applies in the overlap region.

Data lines for PROPERTY=WEAR:

First line:

1. Surface name or material name. If this entry is omitted, a default surface that encompasses the entire general contact domain is assumed.
2. Named contact wear property specified with the **SURFACE PROPERTY* option.

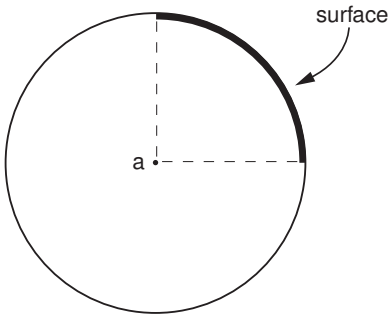


Figure 1: Two-dimensional circumferential smoothing.

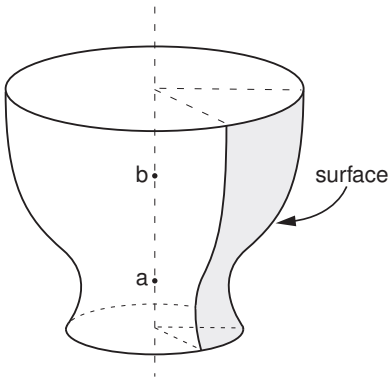


Figure 2: Three-dimensional circumferential smoothing.

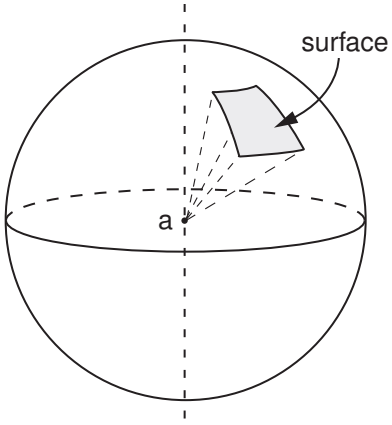


Figure 3: Spherical smoothing.

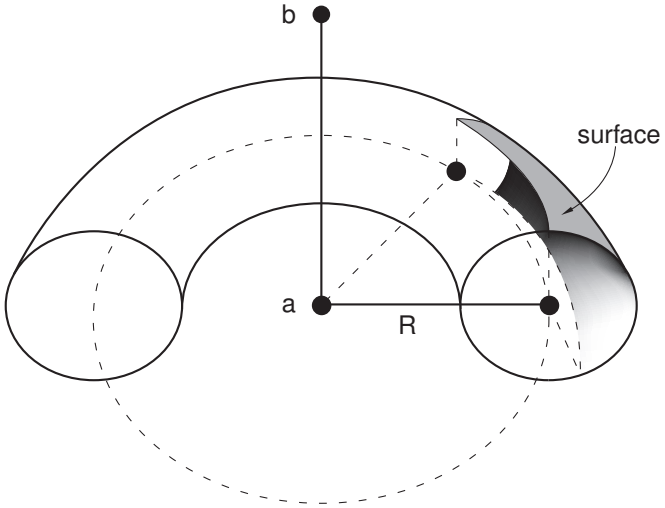


Figure 4: Three-dimensional toroidal smoothing.

*SURFACE SECTION: Specify section properties for surface elements.

This option is used to specify a surface element cross-section. In an Abaqus/Aqua analysis the surface section can be used to visualize the behavior of the water surface under gravity waves.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE Abaqus/Aqua

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- *Surface Elements*
- *Defining Reinforcement*
- **REBAR LAYER*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the surface elements for which the section properties are being defined.

Optional, mutually exclusive parameters:

AQUAVISUALIZATION

This parameter applies only to Abaqus/Aqua analyses.

Set AQUAVISUALIZATION=YES to specify that the referenced surface elements will be used to visualize gravity waves.

DENSITY

Set this parameter equal to the mass density per unit area of the surface element section.

Data line for a constant thickness surface element:

First (and only) line:

1. Section thickness. The default is 0.

*SURFACE SMOOTHING: Define surface smoothing methods.

This option is used to create a surface smoothing definition for contact interactions. It must be used in conjunction with the **CONTACT PAIR* option. The defined smoothing methods apply to the specified regions of the surfaces in the referenced contact pair.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

• *Smoothing Contact Surfaces in Abaqus/Standard*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to this surface smoothing definition.

This label is referred to by the GEOMETRIC CORRECTION parameter on the **CONTACT PAIR* option.

Data lines to define the surface regions on which smoothing is applied:

Data line to define smoothing on regions of two-dimensional surfaces that correspond (or nearly correspond) to a circular arc (see *Figure 1*):

1. Name of secondary surface or surface corresponding to a subregion of the secondary surface. The name can be omitted when smoothing is applied only to a subregion of the main surface.
2. Name of main surface or surface corresponding to a subregion of the main surface. The name can be omitted when smoothing is applied only to a subregion of the secondary surface.
3. The "word"CIRCUMFERENTIAL.
4. Global *X*-coordinate of point *a*, the approximate center (origin) of the circular arc.
5. Global *Y*-coordinate of point *a*, the approximate center (origin) of the circular arc.

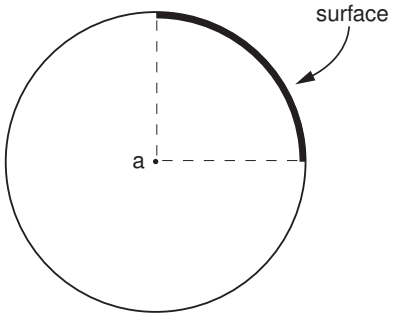


Figure 1: Two-dimensional circumferential smoothing.

Data line to define smoothing on regions of surfaces that correspond (or nearly correspond) to a surface of revolution (see *Figure 2*):

1. Name of secondary surface or surface corresponding to a subregion of the secondary surface. The name can be omitted when smoothing is applied only to a subregion of the main surface.

2. Name of main surface or surface corresponding to a subregion of the main surface. The name can be omitted when smoothing is applied only to a subregion of the secondary surface.
3. The “word”CIRCUMFERENTIAL.
4. Global *X*-coordinate of point *a* on the approximate axis of revolution for the surface.
5. Global *Y*-coordinate of point *a* on the approximate axis of revolution for the surface.
6. Global *Z*-coordinate of point *a* on the approximate axis of revolution for the surface.
7. Global *X*-coordinate of point *b* on the approximate axis of revolution for the surface (see *Figure 2*).
8. Global *Y*-coordinate of point *b* on the approximate axis of revolution for the surface.
9. Global *Z*-coordinate of point *b* on the approximate axis of revolution for the surface.

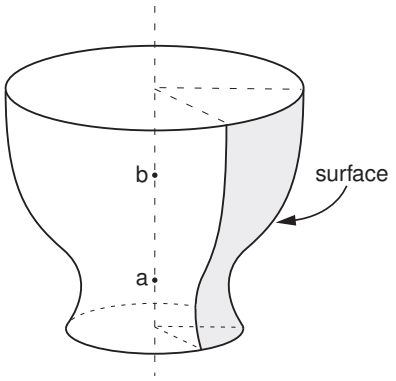


Figure 2: Three-dimensional circumferential smoothing.

Data line to define smoothing on regions of surfaces that correspond (or nearly correspond) to a sphere section (see *Figure 3*):

1. Name of secondary surface or surface corresponding to a subregion of the secondary surface. The name can be omitted when smoothing is applied only to a subregion of the main surface.
2. Name of main surface or surface corresponding to a subregion of the main surface. The name can be omitted when smoothing is applied only to a subregion of the secondary surface.
3. The “word”SPHERICAL.
4. Global *X*-coordinate of point *a*, the approximate center (origin) of the sphere.
5. Global *Y*-coordinate of point *a*, the approximate center (origin) of the sphere.
6. Global *Z*-coordinate of point *a*, the approximate center (origin) of the sphere.

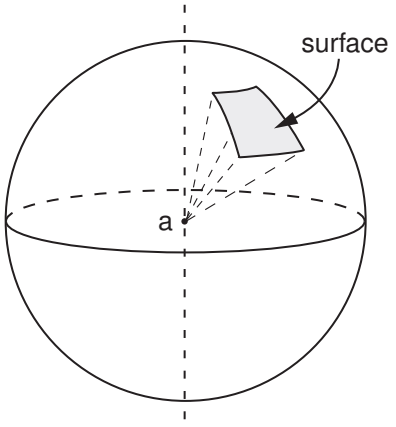


Figure 3: Spherical smoothing.

Data line to define smoothing on regions of surfaces that correspond (or nearly correspond) to a toroidal surface (see *Figure 4*):

1. Name of secondary surface or surface corresponding to a subregion of the secondary surface. The name can be omitted when smoothing is applied only to a subregion of the main surface.
2. Name of main surface or surface corresponding to a subregion of the main surface. The name can be omitted when smoothing is applied only to a subregion of the secondary surface.
3. The "word"TOROIDAL.
4. Global *X*-coordinate of point *a* on the approximate axis of revolution for the surface (see *Figure 4*).
5. Global *Y*-coordinate of point *a* on the approximate axis of revolution for the surface.
6. Global *Z*-coordinate of point *a* on the approximate axis of revolution for the surface.
7. Global *X*-coordinate of point *b* on the approximate axis of revolution for the surface (see *Figure 4*).
8. Global *Y*-coordinate of point *b* on the approximate axis of revolution for the surface.
9. Global *Z*-coordinate of point *b* on the approximate axis of revolution for the surface.
10. Distance *R* of the center of the circular arc from the axis of revolution (see *Figure 4*).

Repeat the above data lines as often as necessary to define all surface regions that require smoothing.

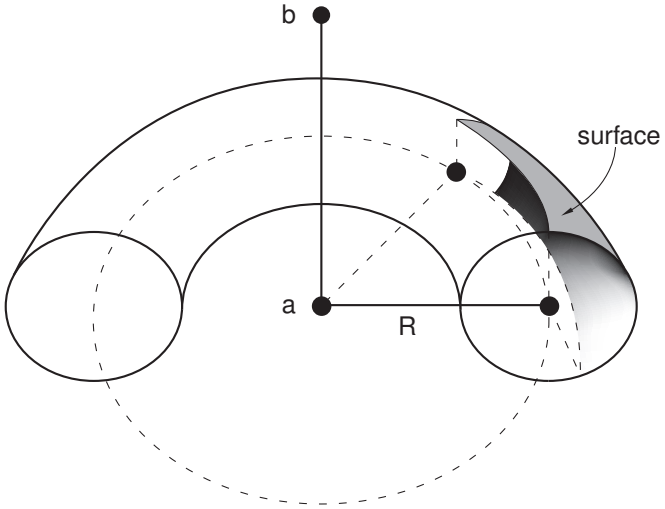


Figure 4: Three-dimensional toroidal smoothing.

*SURFACE TENSION: Define the surface tension of an element-based surface.

This option is used to define surface tension properties on an element-based surface. It must appear immediately after the *SURFACE PROPERTY option.

Products: Abaqus/Explicit

Type: Model data

Level: Assembly

References:

- *Surface Tension Loads*
- *SURFACE PROPERTY

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the surface tension, in addition to temperature. If this parameter is omitted, it is assumed that the surface tension depends only on temperature. See *Material Data Definition*.

Data lines to define the surface tension:

First line:

1. Surface tension, γ.
2. Temperature, T . If this value is omitted, the surface tension is assumed to be independent of the temperature.
3. First field variable, f^1.
4. Second field variable, f^2.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable, f^7.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the surface tension as a function of the temperature and predefined field variables.

*SURFACE TENSION CONTACT ANGLE: Specify the surface tension contact angle property for a given contact interaction.

This option is used to specify the surface tension contact angle associated with a surface interaction definition where one of the surfaces has surface tension. It must appear immediately after the *SURFACE INTERACTION option.

Products: Abaqus/Explicit

Type: History data

Level: Model, Step

References:

- *Surface Tension Loads*
- *SURFACE INTERACTION

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the surface tension contact angle, in addition to temperature. If this parameter is omitted, it is assumed that the surface tension depends only on temperature. See *Material Data Definition*.

Data lines to define the surface tension contact angle:

First line:

1. Surface tension contact angle, θ (unit degree).
2. Temperature, T . If this value is omitted, the surface tension is assumed to be independent of the temperature.
3. First field variable, f^1.
4. Second field variable, f^2.
5. Etc., up to six field variables

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable, f^7.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the surface tension contact angle as a function of the temperature and predefined field variables.

*SWELLING: Specify time-dependent volumetric swelling.

This option is used to specify time-dependent metal swelling for a material. Swelling behavior defined by this option is active only during *SOILS, CONSOLIDATION; *COUPLED TEMPERATURE-DISPLACEMENT; and *VISCO procedures.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Rate-Dependent Plasticity: Creep and Swelling*
- *CREEP*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the volumetric swelling strain rate in addition to temperature. If this parameter is omitted, it is assumed that the volumetric swelling strain rate is constant or depends only on temperature. See *Material Data Definition* for more information.

LAW

Set LAW=INPUT (default) to define the swelling behavior on the data lines.

Set LAW=USER to define the swelling behavior in user subroutine *CREEP*.

Data lines for LAW=INPUT:

First line:

1. Volumetric swelling strain rate.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of volumetric swelling strain rate on temperature and other predefined field variables.

*SYMMETRIC MODEL GENERATION: Create a three-dimensional model from an axisymmetric or partial three-dimensional model.

This option is used to create a three-dimensional model by revolving the cross-section of an axisymmetric model about a symmetry axis, by revolving a single three-dimensional sector about the symmetry axis, or by combining two parts of a symmetric three-dimensional model where one part is the original model and the other part is obtained by reflecting the original part through a line or a plane.

Products: Abaqus/Standard

Type: Model data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

References:

• *Symmetric Model Generation*

Required, mutually exclusive parameters:

PERIODIC

Use this parameter to indicate that a periodic three-dimensional model must be generated by revolving a single three-dimensional sector of the model about the symmetry axis (see *Figure 1*).

Set PERIODIC=CONSTANT (default) to indicate that each generated sector in the periodic model has a constant angle.

Set PERIODIC=VARIABLE to indicate that each generated sector in the periodic model can have a variable angle in the circumferential direction.

In both cases each sector always has the same geometry and mesh. The surfaces on both sides of the original sector must be completely planar when PERIODIC=VARIABLE. If the surface meshes on either side of the original sector are not matched completely, constraints between the neighboring pairs of corresponding surfaces specified on the data lines will be applied with an automatically generated **TIE* option when the periodic three-dimensional model is generated.

REFLECT

Set REFLECT=LINE to indicate that a three-dimensional model must be generated by reflecting a partial three-dimensional model through a symmetry line (*Figure 3*).

Set REFLECT=PLANE to indicate that a three-dimensional model must be generated by reflecting a partial three-dimensional model through a symmetry plane (*Figure 4*).

REVOLVE

Include this parameter to indicate that a three-dimensional model must be generated by revolving the cross-section of an axisymmetric mesh about the symmetry axis. See *Figure 2*.

Optional parameters:

ELEMENT OFFSET

Set this parameter equal to an integer to define the offset for element numbering. When the REVOLVE parameter is used, the offset is added to each element number on the previous cross-section to obtain the numbering of the elements on the next cross-section, starting at the reference cross-section, $\theta = 0.0$. The reference cross-section uses the same numbering as the original axisymmetric model. When the REFLECT parameter is used, the offset is added to the original

element numbers to define the numbering on the reflected part. The default and minimum value is the largest element number used in the original model.

FILE NAME

Set this parameter equal to the name of an external file (without an extension) to which keyword and data lines for the model definition will be written. The extension `.axi` will be added to the file name provided by the user. See *Input Syntax Rules* for the syntax of such file names.

NODE OFFSET

Set this parameter equal to an integer to define the offset used for node numbering. When the REVOLVE parameter is used, the offset is added to each node number on the previous cross-section to obtain the numbering of the nodes on the next cross-section, starting at the reference cross-section, $\theta = 0.0$. The reference cross-section uses the same numbering as the original axisymmetric model. When the REFLECT parameter is used, the offset is added to the original node numbers to define the numbering on the reflected part. The default and minimum value is the largest node number used in the original model.

TOLERANCE

Set this parameter equal to the distance to be used in the search for duplicate nodes. Duplicate nodes on the axis of revolution of a revolved model, on the connection planes between sectors of a periodic model, and on the connection plane between the two parts of a reflected model will be eliminated. The default distance is 1.0% of the average element dimension.

Data lines if each generated sector in the periodic model has a constant angle (PERIODIC=CONSTANT):

First line:

1. *X*-coordinate of point *a*.
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.
4. *X*-coordinate of point *b*.
5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.

Second line:

1. Segment angle, θ (in degrees), of the original three-dimensional sector. $0.0° < \theta \leq 360.0°$.
2. Number of three-dimensional repetitive sectors, including the original sector in the generated periodic model. The default is 1.

Third line (needed if the surface meshes on either side of the original sector are not matched completely):

1. The surface name on one side of the original sector.
2. The corresponding surface name on the other side of the original sector, measured at a positive angle, θ, around the revolving axis.
3. Tolerance distance within which nodes on one surface of a sector must lie from the corresponding surface of the neighboring sector to be constrained. Nodes on the surface of the sector that are further away from the corresponding surface of the neighboring sector than this distance are not constrained. The default value for this tolerance distance is 5% or 10% of the typical element size in the surfaces of the original sector, depending on whether node-to-surface or surface-to-surface type of constraints are used, respectively.
4. Include the “word” SURFACE (default) to generate a surface-to-surface type of constraint or the “word” NODE to generate a node-to-surface type of constraint.

Repeat the third data line as often as necessary to define pairs of corresponding surfaces on each side of the original repetitive sector. Constraints between the neighboring pairs of corresponding surfaces will be applied with an automatically generated *TIE option when the periodic three-dimensional model is generated.

Data lines if each generated sector in the periodic model has a variable angle (PERIODIC=VARIABLE):

First line:

1. X-coordinate of point a.
2. Y-coordinate of point a.
3. Z-coordinate of point a.
4. X-coordinate of point b.
5. Y-coordinate of point b.
6. Z-coordinate of point b.

Second line:

1. Segment angle, θ (in degrees), of the original three-dimensional sector. $0.0° < \theta \leq 360.0°$.
2. Number of three-dimensional sectors that have the same angle as the original sector to be generated, including the original sector. The default is 1.

Third line:

1. Number of additional three-dimensional sectors to be generated.
2. Angular scaling factor in the circumferential direction with respect to the original sector. The default is 1.0.

Repeat the third data line as often as necessary to define all the sectors of the model in the circumferential direction.

Subsequent lines (needed if the surface meshes on either side of the original sector are not matched completely):

1. The surface name on one side of the original sector.
2. The corresponding surface name on the other side of the original sector, measured at a positive angle, θ, around the revolving axis.
3. Tolerance distance within which nodes on one surface of a sector must lie from the corresponding surface of the neighboring sector to be constrained. Nodes on the surface of the sector that are further away from the corresponding surface of the neighboring sector than this distance are not constrained. The default value for this tolerance distance is 5% or 10% of the typical element size in the surfaces of the original sector, depending on whether node-to-surface or surface-to-surface type of constraints are used, respectively.
4. Include the “word” SURFACE (default) to generate a surface-to-surface type of constraint or the “word” NODE to generate a node-to-surface type of constraint.

Repeat the subsequent data line as often as necessary to define more pairs of corresponding surfaces on each side of the original sector. Constraints between the neighboring pairs of corresponding surfaces will be applied with the automatically generated *TIE option when the periodic three-dimensional model is generated.

Data line if REFLECT=LINE:

First (and only) line:

1. X-coordinate of point a.
2. Y-coordinate of point a.
3. Z-coordinate of point a.
4. X-coordinate of point b.

5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.

Data lines if REFLECT=PLANE:

First line:

1. *X*-coordinate of point *a*.
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.
4. *X*-coordinate of point *b*.
5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.

Second line:

1. *X*-coordinate of point *c*.
2. *Y*-coordinate of point *c*.
3. *Z*-coordinate of point *c*.

Data lines if the REVOLVE parameter is included:

First line:

1. *X*-coordinate of point *a*.
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.
4. *X*-coordinate of point *b*.
5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.

Second line:

1. *X*-coordinate of point *c*.
2. *Y*-coordinate of point *c*.
3. *Z*-coordinate of point *c*.

Third line:

1. Segment angle, θ, through which the cross-section must be revolved. $0.0° < \theta \leq 360.0°$. The segments are connected, so, except for the first segment, each segment starts at the end point of the segment given on the previous data line.
2. Number of subdivisions or elements to be used in the segment. The default is 1. A single element subdivision must not exceed 45° when general elements are used or 180° when cylindrical elements are used.
3. Bias ratio to be used in the spacing of nodes generated over the segment. The value is set to the ratio of adjacent angles between nodes along each arc of nodes generated. Thus, if the value is less than one, the nodes are concentrated toward the beginning of the segment; and if the value is greater than one, the nodes are concentrated toward the end of the segment. The default is 1.0.
4. Include the “word” GENERAL (default) to generate general three-dimensional elements or the “word” CYLINDRICAL to generate cylindrical elements.

Repeat the third data line as often as necessary to define the discretization of the model in the circumferential direction.

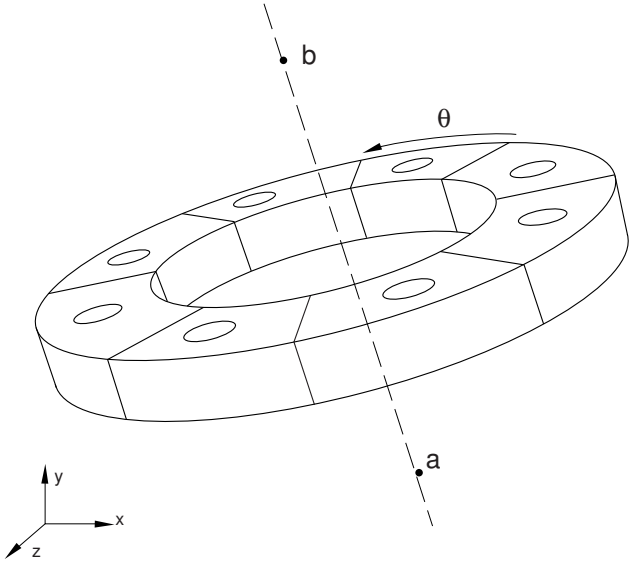


Figure 1: Revolving a single three-dimensional repetitive sector to create a periodic structure.

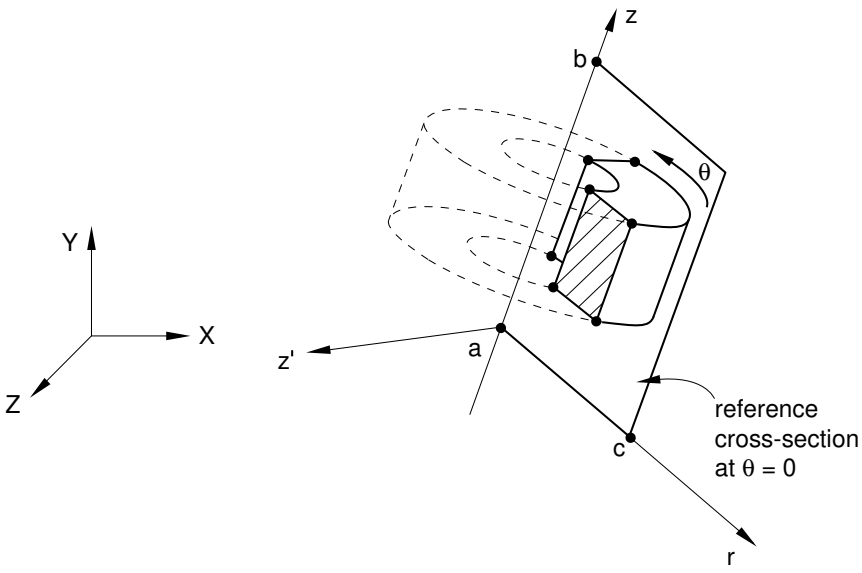


Figure 2: Revolving an axisymmetric cross-section.

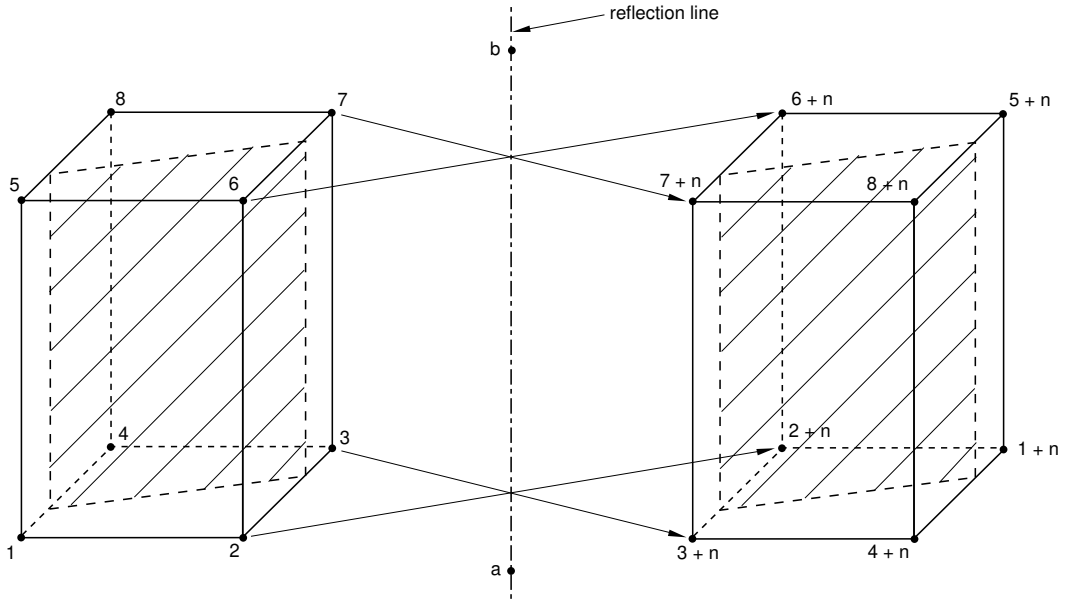


Figure 3: Reflecting a three-dimensional model through line *ab* with node offset *n*.

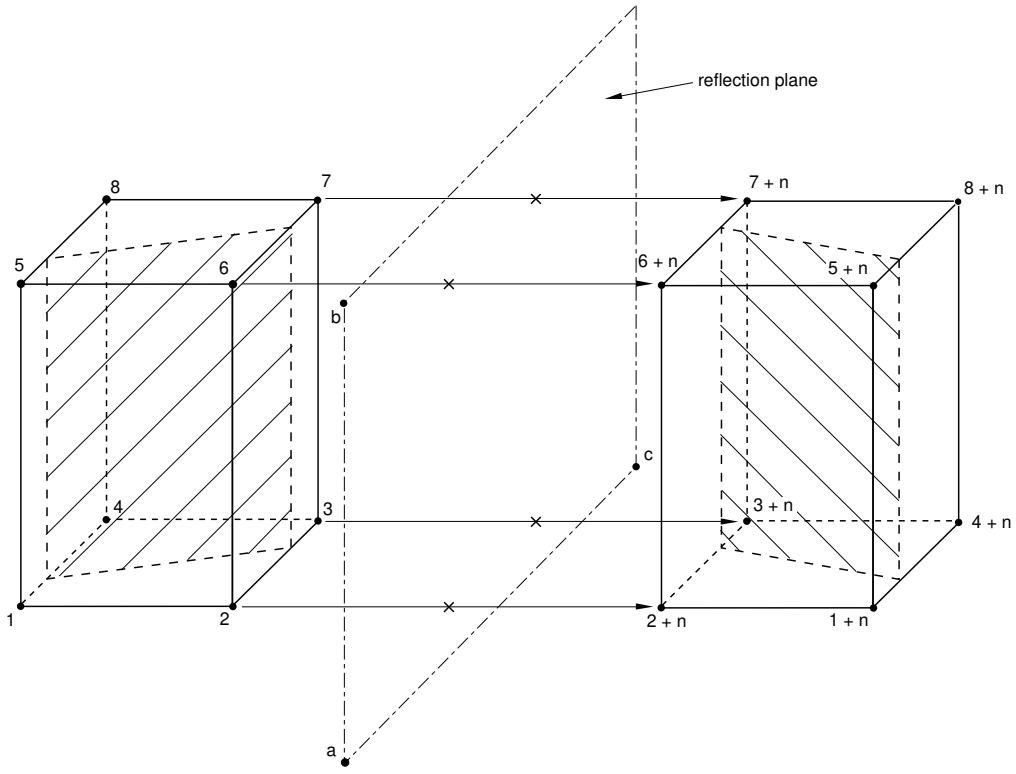


Figure 4: Reflecting a three-dimensional model through a plane *abc* with node offset *n*.

*SYMMETRIC RESULTS TRANSFER: Import results from an axisymmetric or partial three-dimensional analysis.

This option is used to transfer a solution from an axisymmetric analysis to a three-dimensional model or to transfer the solution of a partial three-dimensional model to a full three-dimensional model. It can be used only in conjunction with the **SYMMETRIC MODEL GENERATION* option.

Products: Abaqus/Standard

Type: Model data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

References:

- *Transferring Results from a Symmetric Mesh or a Partial Three-Dimensional Mesh to a Full Three-Dimensional Mesh*
- *Symmetric Model Generation*
- **SYMMETRIC MODEL GENERATION*

Optional parameters:

STEP

Set this parameter equal to the step number on the restart file at which the results must be obtained. If this parameter is omitted, the results will be obtained at the last available step found on the restart file.

INC

Set this parameter equal to the increment number on the restart file at which the results must be obtained. If this parameter is omitted, the results will be obtained at the end of the step specified by the STEP parameter.

ITERATION

This parameter is relevant only if the solution is transferred from a previous direct cyclic analysis.

Set this parameter equal to the iteration number on the restart file at which the results must be obtained. Since restart information can be written only at the end of an iteration in a direct cyclic analysis, the INC parameter is irrelevant and is ignored if the ITERATION parameter is specified.

If this parameter is omitted, the results will be obtained at the end of the step specified by the STEP parameter.

LIBRARY

Set this parameter equal to a value that specifies the previous analysis from which to transfer results to a three-dimensional model. You can specify the name of the previous analysis or the name including the full path.

If you do not specify the path, all input files and results files from the previous analyses must reside in the current (working) directory.

UNBALANCED STRESS

Set UNBALANCED STRESS=STEP (default) if the stress unbalance is to be resolved in the first increment.

Set UNBALANCED STRESS=RAMP if the stress unbalance is to be resolved linearly over the step.

There are no data lines associated with this option.

*SYSTEM: Specify a local coordinate system in which to define nodes.

This option is used to define nodes by accepting coordinates relative to a specified local rectangular coordinate system and generating the nodal coordinates in the global coordinate system.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Not applicable; instancing a part in the *Assembly module*creates a local coordinate system.

References:

• *Node Definition*

There are no parameters associated with this option.

Data lines to define a local coordinate system:

First line:

1. X_a, global *X*-coordinate of the origin of the local coordinate system (point *a* in *Figure 1*).
2. Y_a, global *Y*-coordinate of the origin of the local coordinate system.
3. Z_a, global *Z*-coordinate of the origin of the local coordinate system.

The following entries are not needed for a pure translation:

9. X_b, global *X*-coordinate of a point on the X^1-axis of the local coordinate system (point *b* in *Figure 1*).
11. Y_b, global *Y*-coordinate of a point on the X^1-axis of the local coordinate system.
13. Z_b, global *Z*-coordinate of a point on the X^1-axis of the local coordinate system.

Second line (optional; if not provided, the *Z*-axis direction remains unchanged, and the X^1-axis is projected onto the (X, Y) plane):

1. X_c, global *X*-coordinate of a point in the (X^1, Y^1) plane of the local coordinate system, on the side of the positive Y^1-axis (for example, point *c* in *Figure 1*).
2. Y_c, global *Y*-coordinate of a point in the (X^1, Y^1) plane of the local coordinate system, on the side of the positive Y^1-axis.
3. Z_c, global *Z*-coordinate of a point in the (X^1, Y^1) plane of the local coordinate system, on the side of the positive Y^1-axis.

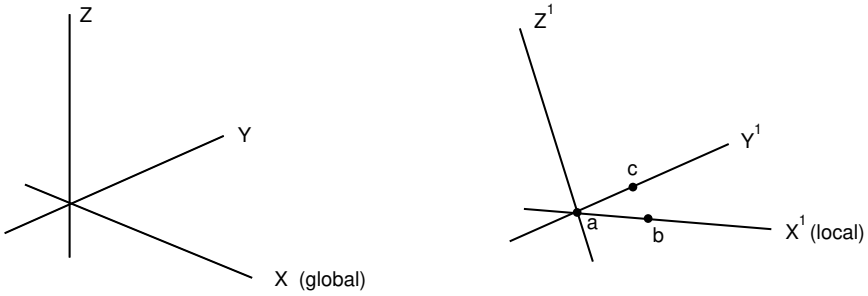


Figure 1: Local coordinate system.

T

This section describes each Abaqus input option that begins with T, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **TABLE COLLECTION*
- **TEMPERATURE*
- **TENSILE FAILURE*
- **TENSION CUTOFF*
- **TENSION STIFFENING*
- **THERMAL EXPANSION*
- **TIE*
- **TIME POINTS*
- **TORQUE*
- **TORQUE PRINT*
- **TRACER PARTICLE*
- **TRANSFORM*
- **TRANSPORT VELOCITY*
- **TRANSVERSE SHEAR*
- **TRANSVERSE SHEAR STIFFNESS*
- **TRIAXIAL TEST DATA*
- **TRS*

*TABLE COLLECTION: Define a table collection.

This option is used to define a table collection. A table collection is a named container that encapsulates property tables and parameter tables. It is used to transfer data defined as property tables and parameter tables from an input file to a user subroutine. This option must be used with **PARAMETER TABLE* and **PROPERTY TABLE*.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- *About Table Collections, Parameter Tables, and Property Tables*
- *Accessing Abaqus Table Collections*
- **PARAMETER TABLE TYPE*
- **PROPERTY TABLE TYPE*
- **PARAMETER TABLE*
- **PROPERTY TABLE*

Required parameters:

NAME

Name of the table collection.

There are no data lines associated with this option.

*TEMPERATURE: Specify temperature as a predefined field.

This option is used to specify temperature as a predefined field during an analysis.

To use this option in a restart analysis of Abaqus/Standard, either **TEMPERATURE* or **INITIAL CONDITIONS*, TYPE=TEMPERATURE must have been specified in the original analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- *Predefined Fields*
- *UTEMP*

Optional parameters for using the data line format:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that gives the time variation of the temperature throughout the step (see *Amplitude Curves*).

If this parameter is omitted in an Abaqus/Standard analysis, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (see *Defining an Analysis*). If this parameter is omitted in an Abaqus/Explicit analysis, a linear interpolation is done over the step.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

OP

Set OP=MOD (default) for existing **TEMPERATURE* values to remain, with this option modifying existing temperatures or defining additional temperatures.

Set OP=NEW if all existing **TEMPERATURE* values should be removed. New temperatures can be defined.

For a general analysis step, a temperature that is removed via OP=NEW is reset to the value given on the **INITIAL CONDITIONS* option or to zero if no initial temperature was defined. For a linear perturbation step, a temperature that is removed via OP=NEW is always reset to zero. If temperatures are being returned to their initial condition values, the AMPLITUDE parameter described above does not apply. Rather, the AMPLITUDE parameter given on the **STEP* option governs the behavior in an Abaqus/Standard analysis, and the temperatures are always ramped back to their initial conditions in Abaqus/Explicit analyses. If temperatures are being reset to new values (not to initial conditions) via OP=NEW, the AMPLITUDE parameter described above applies.

SECTION SPECIFICATION

Set SECTION SPECIFICATION=UNIFORM to define uniform temperature values.

Required parameter for reading temperatures from the results or output database file:

FILE

Set this parameter equal to the name of the results or output database file from which the data are read. The file extension is optional; however, if both `.fil` and `.odb` files exist, the results file will be used if the INTERPOLATE parameter is omitted. If the INTERPOLATE parameter is used, an output database file must exist. See *Input Syntax Rules* for the syntax of such file names. This parameter cannot be used in a **STATIC*, RIKS analysis step.

Optional parameters for reading temperatures from the results or output database file:

ABSOLUTE EXTERIOR TOLERANCE

This parameter is relevant only for use with the INTERPOLATE parameter. Set this parameter equal to the absolute value (given in the units used in the model) by which nodes of the current model may lie outside the region of the model in the output database specified by the FILE parameter. If this parameter is not used or has a value of 0.0, the EXTERIOR TOLERANCE parameter will apply.

EXTERIOR TOLERANCE

This parameter is relevant only for use with the INTERPOLATE parameter. Set this parameter equal to the fraction of the average element size by which nodes of the current model may lie outside the region of the elements of the model in the output database specified by the FILE parameter. The default value is 0.05.

If both tolerance parameters are specified, Abaqus uses the tighter tolerance.

BSTEP

Set this parameter equal to the step number (of the analysis whose results or output database file is being used as input to this option) that begins the history data to be read. If no value is supplied, Abaqus will begin reading temperature data from the first step available on the file read.

BINC

Set this parameter equal to the increment number (of the analysis whose results or output database file is being used as input to this option) that begins the history data to be read. If no value is supplied, Abaqus will begin reading temperature data from the first increment available (excluding any zero increments if the results file was written in Abaqus/Standard using **FILE FORMAT*, ZERO INCREMENT) for step BSTEP on the results or output database file.

ESTEP

Set this parameter equal to the step number (of the analysis whose results or output database file is being used as input to this option) that ends the history data to be read. If no value is supplied, ESTEP is taken as equal to BSTEP.

EINC

Set this parameter equal to the increment number (of the analysis whose results or output database file is being used as input to this option) that ends the history data to be read. If no value is supplied, EINC is taken as the last available increment of step ESTEP on the file read.

BTRAMP

Set this parameter equal to the starting time (measured relative to the total step time period) after which the temperatures read from the results file will be ramped to their initial condition values.

The default value is **1.0**, in which case no temperature ramping takes place. This feature is used to create a cyclic temperature history from a prior heat transfer analysis that is not cyclic.

DRIVING ELSETS

This parameter is relevant only for use with the INTERPOLATE parameter. Include this parameter to indicate that the temperature field is interpolated from a user-specified element set from the

previous analysis to a user specified node set in the current job. This parameter is used to eliminate mapping ambiguity in cases where element regions in the previous analysis are close or touching. To accomplish part instance to part instance mapping, define your element and node sets to correspond to the respective instances in the previous and current analysis.

INTERPOLATE

Include this parameter to indicate that the temperature field needs to be interpolated between dissimilar meshes. This feature is used to read temperatures from an output database file generated during a heat transfer analysis or generated during a global model analysis used with the submodeling capability. This parameter and the MIDSIDE parameter are mutually exclusive. If the heat transfer analysis uses first-order elements and the current mesh is the same but uses second-order elements, use the MIDSIDE parameter instead.

MIDSIDE

Include this parameter to indicate that midside node temperatures in second-order elements are to be interpolated from corner node temperatures. This feature is used to read temperatures from a results or an output database file generated during a heat transfer analysis using first-order elements. This parameter and the INTERPOLATE parameter are mutually exclusive.

Required parameter for defining data in user subroutine `UTEMP` or `VUTEMP`:

USER

Include this parameter to indicate that user subroutine *UTEMP* or *VUTEMP* will be used to define temperature values. In an Abaqus/Standard analysis *UTEMP* is called for each node given on the data lines. In an Abaqus/Explicit analysis *VUTEMP* is called for a block of nodes specified on the data lines.

If you also specify values on the data lines, these values are ignored. If you specify a results or an output database file in addition to user subroutine *UTEMP* in Abaqus/Standard, values read from this file will be passed into *UTEMP* for possible modification. In Abaqus/Explicit user subroutine *VUTEMP* cannot be used with a results or output database file.

Data lines to define gradients of temperature in beams and shells:

First line:

1. Node set or node number.
2. Reference temperature value. If the AMPLITUDE parameter is present, this value and subsequent gradient values will be modified by the AMPLITUDE specification.
3. Temperature gradient in the n_2-direction for beams or temperature gradient through the thickness for shells.
4. Temperature gradient in the n_1-direction for beams.

Repeat this data line as often as necessary to define temperatures at different nodes or node sets.

Data lines to define temperatures at temperature points in beams and shells:

First line:

1. Node set or node number.
2. Temperature at the first temperature point. If the AMPLITUDE parameter is present, this value and subsequent temperature values will be modified by the AMPLITUDE specification.
3. Temperature of the second temperature point.
4. Temperature of the third temperature point.
5. Etc., up to seven temperatures.

Subsequent lines (only needed if there are more than seven temperature points in the element):

1. Temperature of the eighth temperature point.
2. Etc., up to eight temperatures per line.

If more than seven temperature values are needed at any node, continue on the next line. It may be necessary to leave blank data lines for some nodes if any other node in the model has more than seven temperature points because the total number of temperatures that Abaqus expects to read for any node is based on the maximum number of temperature values of all the nodes in the model. These trailing initial values will be zero and will not be used in the analysis.

Repeat this set of data lines as often as necessary to define temperatures at different nodes or node sets.

Data lines to define temperatures for solid or frame elements using the data line format:

First line:

1. Node set or node number.
2. Temperature. If the AMPLITUDE parameter is present, this value will be modified by the AMPLITUDE specification.

Repeat this data line as often as necessary to prescribe temperature at different nodes or node sets.

Data lines to define uniform temperature (SECTION SPECIFICATION=UNIFORM):

First line:

1. Node set, node number, or blank.
2. Uniform temperature value.

Repeat this data line as often as necessary to define uniform temperatures at various nodes and node sets. If you omit both node numbers and node sets (that is, enter a blank space), the specified uniform temperature is applied to all nodes in the model.

No data lines are needed if temperatures are read from an Abaqus results or output database file (except when the DRIVING ELSETS parameter is present) and for Abaqus/Standard a user subroutine is not used (FILE parameter included, USER parameter omitted)

Data lines to define temperatures using user subroutine `UTEMP` or `VUTEMP` using the data line format (FILE parameter omitted in Abaqus/Standard, USER parameter included):

First line:

1. Node set or node number.

Repeat this data line as often as necessary. *UTEMP* will be called for each node listed, and *VUTEMP* will be called for a block of nodes.

Data lines to define temperatures using user subroutine `UTEMP` when temperatures are read from an Abaqus results or output database file (FILE and USER parameters included):

First line:

1. Node set or node number.

Repeat this data line as often as necessary. The nodes identified on the data lines will be assigned values from the results or output database file; optionally, these values can be modified in user subroutine *UTEMP*.

Data lines to define temperatures when the FILE, INTERPOLATE, and DRIVING ELSETS parameters are included:

First line:

1. Element set, node set.

Repeat this data line as often as necessary. The node set identified on the data lines will be assigned values from the element set in the results (`.fil`) or output database (`.odb`) file. If a duplicate node is defined on a subsequent data line, it will be removed from the subsequent temperature mapping and printed out to the data (`.dat`) file.

*TENSILE FAILURE: Specify a tensile failure model and criterion.

This option is used with the Mises or the Johnson-Cook plasticity models or the equation of state model to specify a tensile failure model and criterion. It must be used in conjunction with the *PLASTIC, HARDENING=ISOTROPIC option; the *PLASTIC, HARDENING=JOHNSON COOK option; or the *EOS option.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Equation of State*
- *Classical Metal Plasticity*
- *Johnson-Cook Plasticity*
- *Dynamic Failure Models*
- *EOS
- *PLASTIC

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the hydrostatic cutoff stress, in addition to temperature. If this parameter is omitted, it is assumed that the hydrostatic cutoff stress is constant or depends only on temperature. See *Material Data Definition* for more information.

ELEMENT DELETION

Set ELEMENT DELETION=YES (default) to allow element deletion when the failure criterion is met.

Set ELEMENT DELETION=NO to allow BRITTLE/DUCTILE-type failure for the deviatoric and hydrostatic parts of stresses.

Required parameters for use with ELEMENT DELETION=NO:

PRESSURE

Set PRESSURE=BRITTLE to model the case where the pressure stress is required to be compressive when the failure criterion is met.

Set PRESSURE=DUCTILE to model the case where the pressure stress will be limited by the hydrostatic cutoff stress when the failure criterion is met.

SHEAR

Set SHEAR=BRITTLE to model the case where the deviatoric stresses will be set to zero when the failure criterion is met.

Set SHEAR=DUCTILE to model the case where the deviatoric stresses will be unaffected when the failure criterion is met.

Data lines to specify a tensile failure model:

First line:

1. Hydrostatic cutoff stress (positive in tension). (Units of FL^{-2}.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of hydrostatic cutoff stress as a function of temperature and other predefined field variables.

*TENSION CUTOFF: Specify tension cutoff data for the Mohr-Coulomb plasticity model.

This option is used to specify tension cutoff data to limit the load-carrying capacity of the Mohr-Coulomb plasticity model near the tensile region. This option must be used in conjunction with *MOHR COULOMB and *MOHR COULOMB HARDENING options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Mohr-Coulomb Plasticity*
- *MOHR COULOMB
- *MOHR COULOMB HARDENING

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the tension cutoff stress, in addition to temperature. If this parameter is omitted, it is assumed that the tensile yield stress depends only on the plastic strain and, possibly, on temperature.

Data lines to define tension cutoff:

First data line:

1. Yield stress in uniaxial tension, σ_t.
2. Corresponding plastic strain. (The first tabular value entered must always be zero.)
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent data lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of tension cutoff on predefined field variables.

*TENSION STIFFENING: Define the retained tensile stress normal to a crack in a CONCRETE model.

This option is used to define the retained tensile stress normal to a crack as a function of the deformation in the direction of the normal to the crack. It must be used with and appear after the *CONCRETE option. The *TENSION STIFFENING option can also be used in conjunction with the *SHEAR RETENTION and *FAILURE RATIOS options.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Concrete Smeared Cracking*
- *CONCRETE
- *FAILURE RATIOS
- *SHEAR RETENTION

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the postcracking behavior, in addition to temperature. If this parameter is omitted, it is assumed that the postcracking behavior depends only on temperature. See *Material Data Definition* for more information.

TYPE

Set TYPE=DISPLACEMENT to define the postcracking behavior by entering the displacement, u_0, at which a linear loss of strength after cracking gives zero stress.

Set TYPE=STRAIN (default) to specify the postcracking behavior by entering the postfailure stress-strain relationship directly.

Data lines if the parameter TYPE=STRAIN is included (default):

First line:

1. Fraction of remaining stress to stress at cracking.
2. Absolute value of the direct strain minus the direct strain at cracking.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

The first point at each value of temperature must be a stress fraction of 1.0 at a strain of 0.0.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the postcracking behavior on temperature and other predefined field variables.

Data lines if the parameter TYPE=DISPLACEMENT is included:

First line:

1. Displacement, u_0, at which a linear loss of strength after cracking gives zero stress. (Units of L.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the postcracking behavior on temperature and other predefined field variables.

*THERMAL EXPANSION: Define the thermal expansion behavior of beams.

This option can be used only in conjunction with the *BEAM GENERAL SECTION, SECTION=NONLINEAR GENERAL option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- *Structural Relaxation in Glass*
- *Using a General Beam Section to Define the Section Behavior*
- *BEAM GENERAL SECTION

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies for the thermal-expansion coefficient, in addition to temperature. If this parameter is omitted, it is assumed that the thermal-expansion coefficient is constant or depends only on temperature. See *Material Data Definition* for more information.

Data lines to define the thermal expansion behavior:

First line:

1. Coefficient of thermal expansion.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the thermal expansion coefficient as a function of temperature and other predefined field variables.

*TIE: Define surface-based tie and cyclic symmetry constraints or coupled acoustic-structural interactions.

This option is used to impose tie constraints, cyclic symmetry constraints, or coupled acoustic-structural interactions between pairs of surfaces.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Interaction module

References:

- *Mesh Tie Constraints*
- *Element-Based Surface Definition*
- *Node-Based Surface Definition*
- *Analysis of Models that Exhibit Cyclic Symmetry*
- *Coupled Acoustic-Structural Analysis*

Required parameters:

NAME

Set this parameter equal to a label to use to refer to the constraint.

Optional, mutually exclusive parameters:

POSITION TOLERANCE

Set this parameter equal to a cutoff distance that is used to determine which nodes on the secondary surface are tied to the main surface. The calculation of the distance between the secondary and main surface for a particular secondary node depends on factors such as shell element thickness, the setting of the TYPE parameter, and the types of surfaces involved. Secondary nodes that do not satisfy the position tolerance are not tied to the main surface. The default value for this tolerance distance depends on the type of formulation and surfaces used in the constraints.

TIED NSET

Set this parameter equal to the node set label of the node set that includes the nodes on the secondary surface that will be tied to the main surface. Nodes not included in this node set will not be tied.

Optional parameters:

ADJUST

Set ADJUST=YES (default) to move all tied nodes on the secondary surface onto the main surface in the initial configuration, without any strain.

Set ADJUST=NO if the secondary nodes will not be moved. This is the default if the secondary surface belongs to a substructure or if one or more of the surfaces is beam element-based.

CONSTRAINT RATIO

This parameter applies only when two surfaces with rotational degrees of freedom are tied with an offset but the NO ROTATION parameter is used.

Set this parameter equal to the fractional distance between the main reference surface and the secondary node at which the translational constraint should act. By default, Abaqus attempts to choose this distance such that the translational constraint acts precisely at the interface.

CYCLIC SYMMETRY

This parameter can be used only in conjunction with the **CYCLIC SYMMETRY MODEL* option.

Include this parameter to invoke a constraint between the faces bounding a repetitive sector of a cyclic symmetric structure.

NO ELECTRIC POTENTIAL

Include this parameter if electric potential in solid degrees of freedom should not be tied.

If this parameter is omitted, any electric potential in solid degrees of freedom are tied if applicable.

NO FLUID ELECTRIC POTENTIAL

Include this parameter if electric potential in fluid degrees of freedom should not be tied.

If this parameter is omitted, any electric potential in fluid degrees of freedom are tied if applicable.

NO ION CONCENTRATION

Include this parameter if ion concentration in fluid degrees of freedom should not be tied.

If this parameter is omitted, any ion concentration in fluid degrees of freedom are tied if applicable.

NO PORE

Include this parameter if pore pressure degrees of freedom should not be tied.

If this parameter is omitted, any pore pressure degrees of freedom are tied if applicable.

NO ROTATION

Include this parameter if rotation degrees of freedom should not be tied.

If this parameter is omitted, any existing rotation degrees of freedom are tied if applicable (in addition to the translation degrees of freedom).

NO SPECIES CONCENTRATION

Include this parameter if species concentration degrees of freedom should not be tied.

If this parameter is omitted, any species concentration degrees of freedom are tied if applicable.

NO TEMPERATURE

Include this parameter if temperature degrees of freedom should not be tied.

If this parameter is omitted, any existing temperature degrees of freedom are tied if applicable.

NO THICKNESS

Include this parameter to ignore shell thickness effects in calculations involving position tolerances and adjustments for initial gaps.

TYPE

Set TYPE=SURFACE TO SURFACE (default for most cases in Abaqus/Standard) to have the tie coefficients generated such that stress accuracy is optimized for the specified surface type pairings.

Set TYPE=NODE TO SURFACE (default for all cases in Abaqus/Explicit) to have the tie coefficients generated according to the interpolation functions at the point where the secondary node projects onto the main surface.

Data lines to define the surfaces forming the constraint pairs:

First line:

1. The secondary surface name.
2. The main surface name.

Repeat this data line as often as necessary to define all the surfaces forming the constraint pairs. Each data line defines a pair of surfaces to tie together.

*TIME POINTS: Specify time points at which data are written to the output database file, or specify time points in the loading history at which the response of a structure will be evaluated in a direct cyclic analysis.

This option is used to specify time points at which data are written to the output database file or, if it is used in conjunction with the **DIRECT CYCLIC* option, to specify time points in the loading history at which the response of a structure will be evaluated.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Step module

References:

- **DIRECT CYCLIC*
- **OUTPUT*

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the time points.

Optional parameters:

GENERATE

If this parameter is included, each data line should give a starting time point, t_1; an ending time point, t_2; and the time increment between these two specified time points, Δt.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

Data lines if the GENERATE parameter is omitted:

First line:

1. List of time points; the points must be arranged in ascending order.

Repeat this data line as often as necessary. Up to eight entries are allowed per line. If you use the **TIME POINTS* option in conjunction with the **DIRECT CYCLIC* option, the listed time points must include the starting time and ending time in a single loading cycle. The time points must be specified in the step time.

Data lines if the GENERATE parameter is included:

First line:

1. Starting time point.
2. Ending time point, which must be later than the starting time point.
3. Increment in time between the two specified time points.

Repeat this data line as often as necessary. If you use the *TIME POINTS option in conjunction with the *DIRECT CYCLIC option, the listed time points must include the starting time and ending time in a single loading cycle. The time points must be specified in the step time.

*TORQUE: Define the torsional behavior of beams.

This option can be used only in conjunction with the *BEAM GENERAL SECTION, SECTION=NONLINEAR GENERAL option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- *Using a General Beam Section to Define the Section Behavior*
- *BEAM GENERAL SECTION

Optional parameters (if neither ELASTIC nor LINEAR is included, elastic-plastic response is assumed):

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the torque–twist relationship, in addition to temperature. If this parameter is omitted, it is assumed that the torque–twist relationship is constant or depends only on temperature. See *Material Data Definition* for more information.

ELASTIC

Include this parameter if the torque–twist relationship is nonlinear but elastic.

LINEAR

Include this parameter if the torque varies linearly with twist.

Data lines if the LINEAR parameter is included:

First line:

1. Torsional stiffness of the section.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the torsional stiffness as a function of temperature and other predefined field variables.

Data lines if the LINEAR parameter is omitted:

First line:

1. Torque.
2. Twist.
3. Temperature.
4. First field variable.

5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the torsional behavior as a function of temperature and other predefined field variables.

*TORQUE PRINT: Print a summary of the total torque that can be transmitted across axisymmetric slide lines.

This option is used to obtain a summary of the total torque that can be transmitted across all axisymmetric slide lines in a model.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

• *Slide Line Contact Elements*

Optional parameters:

FREQUENCY

Set this parameter equal to the output frequency, in increments. When this option is used, the torque output will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1.

There are no data lines associated with this option.

*TRACER PARTICLE: Define tracer particles for tracking the location of and results at material points during a step.

This option is used to define tracer particles and assign them to tracer sets for tracking the location of and results at material points during a step. The tracer set name is used in conjunction with the *ELEMENT OUTPUT and/or the *NODE OUTPUT options to request output for the tracer particles associated with the tracer set name.

Products: Abaqus/Explicit

Type: History data

Level: Step

References:

- *Output to the Output Database*
- *ELEMENT OUTPUT
- *NODE OUTPUT

Required parameters:

TRACER SET

Set this parameter equal to the name of the tracer set to which these tracer particles will be assigned.

Optional parameters:

PARTICLE BIRTH STAGES

Set this parameter equal to the number of tracer particle births within the step. If this parameter is omitted, a single particle birth will occur at the beginning of the step. If this parameter has a value *n* greater than one, tracer particles will leave their parent nodes *n* times during the step at equally spaced intervals in time.

Data lines to define the tracer particles associated with the tracer set:

First line:

1. List of nodes or node set labels to be assigned as tracer particles corresponding to this tracer set.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

*TRANSFORM: Specify a local coordinate system at nodes.

This option is used to specify a local coordinate system for displacement and rotation degrees of freedom at a node.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Define nodal coordinate systems for prescribed conditions in the *Load module*.

References:

• *Transformed Coordinate Systems*

Required parameters:

NSET

Set this parameter equal to the name of the node set for which the local transformed system is being given.

Optional parameters:

TYPE

Set TYPE=R (default) to indicate a rectangular Cartesian system (*Figure 1*). Set TYPE=C to indicate a cylindrical system (*Figure 2*). Set TYPE=S to indicate a spherical system (*Figure 3*).

Data line to define a transformed coordinate system:

First (and only) line:

1. Global *X*-coordinate of point *a* specifying transformation.
2. Global *Y*-coordinate of point *a* specifying transformation.
3. Global *Z*-coordinate of point *a* specifying transformation.
4. Global *X*-coordinate of point *b* specifying transformation.
5. Global *Y*-coordinate of point *b* specifying transformation.
6. Global *Z*-coordinate of point *b* specifying transformation.

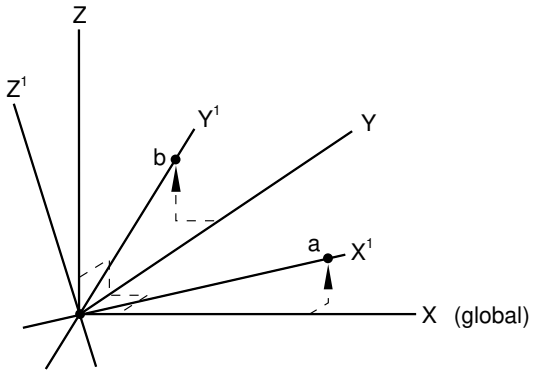


Figure 1: Cartesian transformation option.

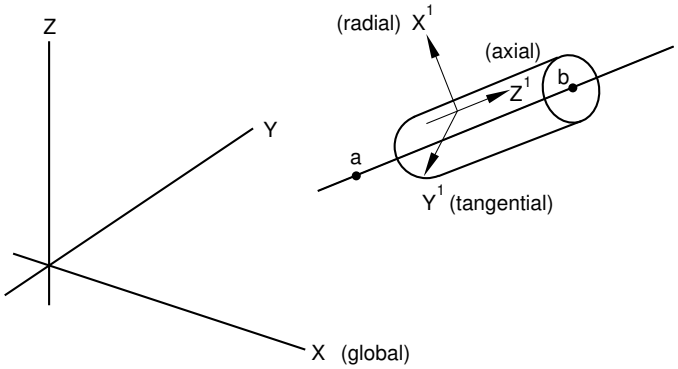


Figure 2: Cylindrical transformation option.

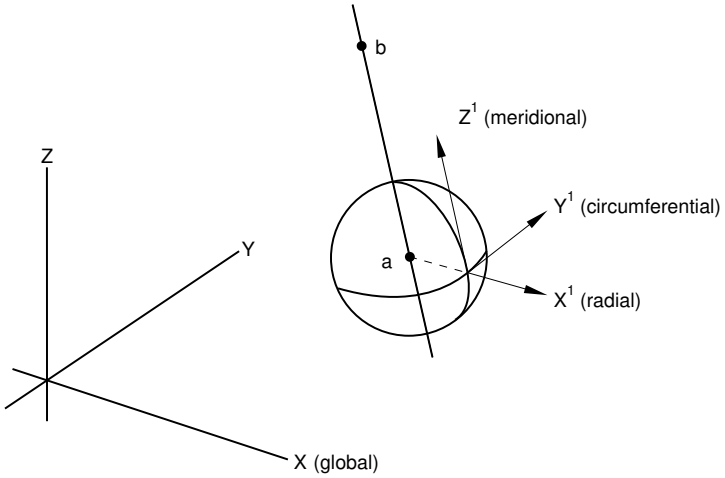


Figure 3: Spherical transformation option.

*TRANSPORT VELOCITY: Specify angular transport velocity.

This option is used to define the angular velocity of material transported through the mesh of a deformable body or the transport of material relative to the reference node of a rigid body during a steady-state transport analysis.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Steady-State Transport Analysis*
- *Symmetric Model Generation*
- *UMOTION*

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve (defined in the **AMPLITUDE* option) that gives the time variation of the velocity throughout the step (*Amplitude Curves*).

If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the **STEP* option (see *Defining an Analysis*).

USER

Include this parameter to indicate that the magnitude of the rotational velocity will be defined in user subroutine *UMOTION*. If this parameter is used, any magnitudes defined by the data lines can be redefined in the user subroutine.

Data lines to define rotational motion:

First line:

1. Node set label or node number.
2. Magnitude of the rotational velocity (in radians/time). This magnitude will be modified by the **AMPLITUDE* specification if the AMPLITUDE parameter is used. For a deformable body the rotation is assumed to be about the axis defined on the **SYMMETRIC MODEL GENERATION* option. For a rigid body of type REVOLUTION the rotation is assumed to be about the axis of revolution of the body.

Repeat this data line as often as necessary to define rotational motion on nodes of different parts of the model.

*TRANSVERSE SHEAR: Specify the transverse shear moduli for computing the transverse shear stiffness of shells and beams.

This option is used to define the transverse shear moduli to be used to compute the transverse shear stiffness for shells and beams. The **TRANSVERSE SHEAR* option can be used in materials that are associated with the **BEAM SECTION*, **SHELL SECTION*, or **SHELL GENERAL SECTION* options.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Linear Elastic Behavior*
- *Choosing a Beam Element*
- *Shell Section Behavior*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the moduli. If this parameter is omitted, it is assumed that the moduli are constant or depend only on temperature. See *Material Data Definition* for more information.

TYPE

Set TYPE=ISOTROPIC (default) to define the isotropic transverse shear modulus.

Set TYPE=ORTHOTROPIC to define fully anisotropic transverse shear moduli.

Set TYPE=ANISOTROPIC to define anisotropic transverse shear moduli.

Data lines to define the isotropic transverse shear modulus (TYPE=ISOTROPIC):

First line:

1. Shear modulus, G. (Units of FL^{-2}.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Second line (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the transverse shear modulus as a function of temperature and other predefined field variables.

Data lines to define orthotropic transverse shear moduli (TYPE=ORTHOTROPIC):

First line:

1. G_{12}. (Units of FL^{-2}.)

2. G_{13}.
3. G_{23}.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Second line (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the transverse shear moduli as a function of temperature and other predefined field variables.

Data lines to define anisotropic transverse shear moduli (TYPE=ANISOTROPIC):

First line:

1. D_{1212}. (Units of FL^{-2}.)
2. D_{1313}.
3. D_{2323}.
4. D_{1323}.
5. This entry is currently not used.
6. Temperature.
7. First field variable.
8. Second field variable

Second line (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the transverse shear moduli as a function of temperature and other predefined field variables.

*TRANSVERSE SHEAR STIFFNESS: Define transverse shear stiffness for beams and shells.

This option must be used in conjunction with the *BEAM GENERAL SECTION option, the *BEAM SECTION option, the *COHESIVE SECTION option, the *SHELL GENERAL SECTION option, or the *SHELL SECTION option.

It cannot be used if you include the FULL COUPLING parameter for the *SECTION STIFFNESS option.

The transverse shear stiffness defined with this option affects only the transverse shear flexible elements whose section properties are defined by the immediately preceding section option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- *Shell Section Behavior*
- *Choosing a Beam Element*
- *Defining the Constitutive Response of Cohesive Elements Using a Continuum Approach*
- *Defining the Constitutive Response of Cohesive Elements Using a Traction-Separation Description*
- *SHELL GENERAL SECTION
- *SHELL SECTION
- *BEAM GENERAL SECTION
- *BEAM SECTION
- *SECTION STIFFNESS
- *COHESIVE SECTION

There are no parameters associated with this option.

Data line when used with cohesive sections, shell sections, and *BEAM GENERAL SECTION, SECTION=MESHED:

First (and only) line:

1. Value of the shear stiffness of the section in the first direction, K_{11}^{ts}.
2. Value of the shear stiffness of the section in the second direction, K_{22}^{ts}.
3. Value of the coupling term in the shear stiffness of the section, K_{12}^{ts}.
4. Value of the slenderness compensation factor. If this field is left blank, a default value of 0.25 is assumed.

If either value K_{11}^{ts} or K_{22}^{ts} is omitted or given as zero, the nonzero value is used for both.

Data line when used with all other beam sections:

First (and only) line:

1. Value of the K_{23} shear stiffness of the section.
2. Value of the K_{13} shear stiffness of the section.
3. Value of the slenderness compensation factor or the label SCF. If this field is left blank, a default value of 0.25 is assumed. If the label SCF is specified, the values of the shear stiffness specified by the user will be ignored. They and the slenderness compensation factor will be calculated from the elastic material definition with the beam section.

If either value $K_{\alpha 3}$ is omitted or given as zero, the nonzero value will be used for both when the label SCF is not used.

*TRIAXIAL TEST DATA: Provide triaxial test data.

This option is required if some or all of the material parameters that define the exponent form of the *DRUCKER PRAGER option are to be calibrated from triaxial test data. This option can be used only as a suboption of the *DRUCKER PRAGER option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Extended Drucker-Prager Models*

Optional parameters:

A

Set this parameter equal to the value of the material constant a if it is known and is held fixed at the input value. Omit the parameter and its value if a is to be obtained from the calibration.

B

Set this parameter equal to the value of the material constant b if it is known and is held fixed at the input value. Omit the parameter and its value if b is to be obtained from the calibration.

PT

Set this parameter equal to the value of the material constant p_t if it is known and is held fixed at the input value. Omit the parameter and its value if p_t is to be obtained from the calibration.

Data lines to specify triaxial test data:

First line:

1. Sign and magnitude of confining stress, $\sigma_1 = \sigma_2$.
2. Sign and magnitude of the stress in loading direction, σ_3.

Repeat this data line as often as necessary to give the yield stress at different levels of confining stress.

*TRS: Used to define temperature-time shift for time history viscoelastic analysis.

This option can be used in conjunction with the **VISCOELASTIC* option. In addition, in Abaqus/Explicit it can be used with the **VISCOSITY* option, and in Abaqus/Standard it can be used in conjunction with the *EXPANSION* option to define the Tool-Narayanaswamy-Moynihan (TNM) model.

In an Abaqus/Standard analysis, the nonlinear viscoelasticity must be defined by using the **VISCOELASTIC*, NONLINEAR option; viscoelasticity must be defined in the time domain by using the **VISCOELASTIC*, TIME option; or viscoelasticity must be defined in the frequency domain by using the **VISCOELASTIC*, FREQUENCY option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Frequency Domain Viscoelasticity*
- *Time Domain Viscoelasticity*
- *Parallel Rheological Framework*
- *Structural Relaxation in Glass*
- *UTRS*
- *UTRSNETWORK*
- *VUTRS*
- **EXPANSION*
- **VISCOELASTIC*
- **VISCOSITY*

Optional parameters:

DEFINITION

Set DEFINITION=WLF (default) to define the shift function by the Williams-Landel-Ferry approximation.

Set DEFINITION=ARRHENIUS to define the shift function by the Arrhenius approximation.

Set DEFINITION=USER to define the shift function in user subroutines. In Abaqus/Standard analyses define the shift function in user subroutine *UTRS* for the linear viscoelastic model (see *Time Domain Viscoelasticity*) or in user subroutine *UTRSNETWORK* for the nonlinear viscoelastic model defined using the parallel rheological framework (see *Parallel Rheological Framework*). In Abaqus/Explicit analyses define the shift function in user subroutine *VUTRS* for the linear viscoelastic model.

Set DEFINITION=TNM to define the shift and response functions for the Tool-Narayanaswamy-Moynihan model.

Set DEFINITION=TABULAR to define the shift function in tabular form.

DEPENDENCIES

This parameter is relevant only for DEFINITION=TABULAR.

Set this parameter equal to the number of field variables included in the specification of the shift function.

If this parameter is omitted, it is assumed that the shift function is constant or depends only on temperature. For more information, see *Material Data Definition*.

PROPERTIES

This parameter applies only to Abaqus/Explicit analyses and to Abaqus/Standard analyses if user subroutine *UTRSNETWORK* is used to define the shift function.

Set this parameter equal to the number of properties being entered. The properties are available for use in user subroutines *VUTRS* or *UTRSNETWORK*.

Data line to define the shift function by the Williams-Landel-Ferry approximation:

First (and only) line:

1. Reference temperature, θ_0.
2. Calibration constant, C_1.
3. Calibration constant, C_2.

Data line to define the shift function by the Arrhenius approximation by specifying the reference temperature and the activation energy:

First (and only) line:

1. Reference temperature, θ_0.
2. Activation energy, E_0.

In addition, you need to specify the universal gas constant and absolute zero using the **PHYSICAL CONSTANTS* option.

Data line to define the shift function by the Arrhenius approximation by specifying the reference temperature and the ratio of the activation energy to the universal gas constant:

First (and only) line:

1. Reference temperature, θ_0.
2. Blank space.
3. Ratio of the activation energy to the universal gas constant, E_0/R.

In addition, you need to specify absolute zero using the **PHYSICAL CONSTANTS* option.

Data lines to define material properties for a user-defined shift function (DEFINITION=USER):

No data lines are needed if the PROPERTIES parameter is omitted or set to 0. Otherwise, first line:

1. Enter the material properties, eight per line.

Repeat this data line as often as necessary to define all material properties.

Data lines to define the shift and response functions for the TNM model:

First line:

1. Reference temperature, θ_R.
2. Activation energy, E_0.

3. Material parameter, χ.
4. Initial fictive temperature, θ^0_{fict}.
5. Number of terms in the response function.

Second line:

1. Coefficient, C_1, in the first term of the response function.
2. Relaxation time, τ_1, in the first term of the response function.

In addition, you need to specify the universal gas constant and absolute zero using the **PHYSICAL CONSTANTS* option.

Repeat this data line as often as necessary to define the second, third, etc. terms of the response function. There is no restriction on the number of terms in the response function.

Data lines to define the shift function in tabular form:

First line:

1. Logarithm to base 10 of the shift function, $\log_{10}(A)$.
2. Temperature, θ.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the logarithm of the shift function as a function of temperature and other predefined field variables.

U

This section describes each Abaqus input option that begins with U, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **UEL PROPERTY*
- **UNDEX CHARGE PROPERTY*
- **UNIAXIAL*
- **UNIAXIAL TEST DATA*
- **UNIFORM*
- **UNIT SYSTEM*
- **UNLOADING DATA*
- **USER DEFINED FIELD*
- **USER ELEMENT*
- **USER MATERIAL*
- **USER OUTPUT VARIABLES*

*UEL PROPERTY: Define property values to be used with a user element type.

This option is required to define the properties of a user element, where the element set must be provided. User element properties provided on the data line are relevant only for nonlinear user elements. The damping parameters provided via parameters on this option are relevant for all procedure types except for nonlinear user elements given by the user subroutine in direct-integration transient dynamic analyses.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Model

Abaqus/CAE: Actuator/sensor interaction properties can be defined in the *Interaction module.*

References:

- *User-Defined Elements*
- **USER ELEMENT*

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the user elements for which these property values are being defined.

Optional parameters:

MATERIAL

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the name of the material to be used with these elements.

ORIENTATION

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the name of an orientation definition (*Orientations*) to be used to define a local coordinate system for material calculations in the elements in this set.

Optional parameters (relevant for mode based procedures, direct steady-state dynamic analyses, and direct-integration transient dynamic analysis with linear user elements):

ALPHA

Set this parameter equal to the Rayleigh mass damping factor, α.

BETA

Set this parameter equal to the Rayleigh stiffness damping factor, β.

STRUCTURAL

Set this parameter equal to the structural stiffness proportional damping factor, s.

There are no data lines required to define the properties of linear user elements

Data lines to define the properties of nonlinear user elements if the PROPERTIES and/or I PROPERTIES parameters are used on the *USER ELEMENT option with a value of one or more:

First line:

1. Enter the values of the element properties. Enter all floating point values first, followed immediately by the integer values.

Repeat this data line as often as necessary. Eight values per line are used for both real and integer values.

*UNDEX CHARGE PROPERTY: Define an UNDEX charge for incident waves.

This option defines parameters that create the time histories of load, displacement, and other variables used to simulate an underwater explosion. This option must be used in conjunction with the **INCIDENT WAVE INTERACTION PROPERTY* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Model, Step

Abaqus/CAE: Interaction module

References:

- *Acoustic and Shock Loads*
- **INCIDENT WAVE INTERACTION PROPERTY*

There are no parameters associated with this option.

Data lines to define the UNDEX charge:

First line:

1. Charge material constant, K.
2. Charge material constant, k.
3. Charge material constant, A.
4. Charge material constant, B.
5. Adiabatic charge constant, K_c.
6. Ratio of specific heats for gas, γ.

Second line:

1. Density of charge material, ρ_c.
2. Mass of charge material, m_c.
3. Acceleration due to gravity, g.
4. Atmospheric pressure, p_{atm}.
5. Wave effect parameter, η. Set to 1.0 for wave effects in the fluid and gas; set to 0.0 to neglect these effects. The default is 1.0.
6. Flow drag coefficient, C_D. The default is 0.0.
7. Flow drag exponent, E_D ($E_D \geq 0$). The default is 2.0.

Third line:

1. Time duration, T_{final}.
2. Maximum number of time steps for the bubble simulation, N_{steps}. The bubble amplitude simulation ceases when the number of steps reaches N_{steps} or the time duration, T_{final}, is reached. The default is 1500.
3. Relative step size control parameter, Ω_{rel}. The default is 1×10^{-11}.
4. Absolute step size control parameter, X_{abs}. The default is 1×10^{-11}.

5. Step size control exponent, β. The step size, Δt, is decreased or increased according to the error estimate: $(\Omega_{\text{rel}} |x| + X_{\text{abs}})^{\beta} \leq \Delta t \left| \frac{dx}{dt} \right|^{\beta}$. The default is 0.2.

Fourth line:

1. Depth magnitude of charge material, d_I.
2. *X*-direction cosine of fluid surface normal.
3. *Y*-direction cosine of fluid surface normal.
4. *Z*-direction cosine of fluid surface normal.

*UNIAXIAL: Characterize fabric materials through loading and unloading test data.

This option is used to indicate the start of shear or uniaxial test data along a particular direction to define the behavior of a fabric material. It must be used in conjunction with the *FABRIC option.

Products: Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Fabric Material Behavior*
- *FABRIC
- *LOADING DATA
- *UNLOADING DATA

Required parameters:

COMPONENT

Set COMPONENT=1 to define uniaxial behavior of fabric fibers in the “fill” direction.

Set COMPONENT=2 to define uniaxial behavior of fabric fibers in the “warp” direction.

Set COMPONENT=SHEAR to define shear response of the fabric.

There are no data lines associated with this option.

*UNIAXIAL TEST DATA: Used to provide uniaxial test data (compression and/or tension).

This option is used to provide uniaxial test data. It can be used only in conjunction with the *HYPERELASTIC option, the *HYPERFOAM option, the *LOW DENSITY FOAM option, and the *MULLINS EFFECT option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Hyperelastic Behavior of Rubberlike Materials*
- *Hyperelastic Behavior in Elastomeric Foams*
- *Low-Density Foams*
- *Stress Softening in Rubberlike Materials*
- *Stress Softening in Elastomeric Foams*
- *HYPERELASTIC
- *HYPERFOAM
- *LOW DENSITY FOAM
- *MULLINS EFFECT

Using uniaxial test data to define a hyperelastic material

Optional parameters:

SMOOTH

Include this parameter to apply a smoothing filter to the stress-strain data. If the parameter is omitted, no smoothing is performed.

Set this parameter equal to the number n such that $2n + 1$ is equal to the total number of data points in the moving window through which a cubic polynomial is fit using the least-squares method. n should be larger than 1. The default is SMOOTH=3.

Optional parameter when the *UNIAXIAL TEST DATA option is used in conjunction with the *HYPERELASTIC, MARLOW option or the *HYPERELASTIC, VALANIS-LANDEL option:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the test data. If this parameter is omitted, it is assumed that the test data depend only on temperature.

Data lines to specify uniaxial test data for hyperelasticity other than the Marlow model and Valanis-Landel model (the nominal strains must be arranged in either ascending or descending order if the SMOOTH parameter is used):

First line:

1. Nominal stress, T_U.
2. Nominal strain, ϵ_U.

Repeat this data line as often as necessary to give the stress-strain data.

Data lines to specify uniaxial test data for the Marlow model or the Valanis-Landel model (the nominal strains must be arranged in ascending order if the SMOOTH parameter is used):

First line:

1. Nominal stress, T_U.
2. Nominal strain, ϵ_U.
3. Nominal lateral strain, $\epsilon_2 = \epsilon_3$. Not needed if the POISSON parameter is specified on the **HYPERELASTIC* option or if the **VOLUMETRIC TEST DATA* option is used.
4. Temperature, θ.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the test data as a function of temperature and other predefined field variables. Nominal strains and nominal stresses must be given in ascending order.

Using uniaxial test data to define an elastomeric foam

There are no parameters associated with this option.

Data lines to specify uniaxial test data for a hyperfoam:

First line:

1. Nominal stress, T_L.
2. Nominal strain, ϵ_U.
3. Nominal lateral strain, $\epsilon_2 = \epsilon_3$. Default is zero. Not needed if the POISSON parameter is specified on the **HYPERFOAM* option.

Repeat this data line as often as necessary to give the stress-strain data.

Using uniaxial test data to define a low-density foam material

Required parameters:

DIRECTION

Set DIRECTION=TENSION to define tensile behavior.

Set DIRECTION=COMPRESSION to define compressive behavior.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the test data. If this parameter is omitted, it is assumed that the test data depend only on temperature.

Data lines to specify uniaxial test data for *LOW DENSITY FOAM, LATERAL STRAIN DATA=NO:

First line:

1. Nominal stress, T_U. (Provide the absolute value if DIRECTION=COMPRESSION.)
2. Nominal strain, ϵ_U. (Provide the absolute value if DIRECTION=COMPRESSION.)
3. Nominal strain rate, $\dot{\epsilon}_U$. (Provide positive values to specify the loading response and negative values to specify unloading.)
4. Temperature, θ.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the test data as a function of temperature and other predefined field variables. Nominal strains, nominal strain rates and nominal stresses must be given in ascending order.

Data lines to specify uniaxial test data for *LOW DENSITY FOAM, LATERAL STRAIN DATA=YES:

First line:

1. Nominal stress, T_U. (Provide the absolute value if DIRECTION=COMPRESSION.)
2. Nominal strain, ϵ_U. (Provide the absolute value if DIRECTION=COMPRESSION.)
3. Nominal lateral strain, $\epsilon_2 = \epsilon_3$. Default is zero. The value must be negative if DIRECTION=TENSION and positive if DIRECTION=COMPRESSION. (A negative Poisson's ratio is not supported for this model.)
4. Nominal strain rate, $\dot{\epsilon}_U$. (Provide positive values to specify the loading response and negative values to specify unloading.)
5. Temperature, θ.

6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the test data as a function of temperature and other predefined field variables. Nominal strains, nominal strain rates and nominal stresses must be given in ascending order.

Using uniaxial test data to define the Mullins effect material model

There are no parameters associated with this option.

Data lines to specify uniaxial test data for defining the unloading-reloading response of the Mullins effect material model:

First line:

1. Nominal stress, T_L.
2. Nominal strain, ϵ_U.

Repeat this data line as often as necessary to give the stress-strain data.

*UNIFORM: Define a uniform coupling constraint.

This option is used to constrain the nonmechanical degrees of freedom of surface nodes to the same degree of freedom of a reference node. It must be used in conjunction with the **COUPLING* option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance, Assembly

References:

- **COUPLING*
- *Coupling Constraints*

There are no parameters associated with this option.

Data lines to specify the degrees of freedom to be constrained:

First (and only) line:

1. Degree of freedom constrained.

   Only one degree of freedom can be constrained per **UNIFORM*. Only nonmechanical degrees of freedom can be constrained.

*UNIT SYSTEM: Define a unit system for an Abaqus model for translation to other solvers.

This option is used to define the units system used in the Abaqus model for the purpose of translating to other formats. It is used for importing external fields, for co-simulation, and to create a Simpack Flexible Body Interface (FBI) file.

This information has no effect on the Abaqus simulation; you must ensure that a consistent set of units is used.

The units of a mechanical system can be defined by specifying one of the basic triples: *length-mass-time* or *length-force-time*. These two methods are mutually exclusive. If you specify *length-mass-time*, the force unit is defined implicitly. If you specify *length-force-time*, the mass unit is defined implicitly.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

• *Defining a Unit System for Translating Data with Other Solvers*

Optional parameters:

SYSTEM

Set SYSTEM=SI if the model uses the International System of units.

Set SYSTEM=MKS if the model uses meter-kilogram-second units.

Set SYSTEM=CGS if the model using centimeter-gram-second units.

Set SYSTEM=FPS if the model uses foot-pound-second units.

The MKS, CGS, and FPS unit systems apply to mechanical simulations only and do not define other quantities such as temperature.

Optional data lines to define units for the physical dimensions if the SYSTEM parameter is omitted:

First line:

1. Physical dimension name LENGTH, MASS, FORCE, TIME, TEMPERATURE, ELECTRIC CURRENT, AMOUNT SUBSTANCE, or LUMINOSITY.
2. Unit symbol to derive the conversion factor. The unit symbol defines the conversion factor between the Abaqus model and the SI system. If this field is left blank, the following SI units are assumed: length (m), mass (kg) or force (N), time (s), temperature (K), electric current (A), amount substance (mole), and luminosity (candela).

   The following unit symbols are available: length (m, cm, mm, km, in, ft, yd, and mi); mass (kg, g, t, lb, klb, oz, slug, slinch, and uton); force (N, lbf, kgf, ozf, dyne, kN, klbf, tf, and pdl); time (s, ms, h, min, and d); temperature (K, C, and F); electric current (A, mA, microA, nA, and pA); amount of substance (mole); and luminosity (candela).

   For a description of unit names, unit symbols, and the associated conversion factor, see *Defining a Unit System for Translating Data with Other Solvers*.

3. The "To SI" conversion factor. For units not defined in the list of unit symbols, specify the conversion factor from the Abaqus model to the SI system. In this case the second field should be left blank. If this field is left blank, a conversion factor of 1.0 is used.

Repeat this data line as often as necessary to define units for the physical dimensions.

If the data line for a physical dimension is omitted, the default SI unit is assumed. Any rotations are in radians. Data lines for the mass and force physical dimensions are mutually exclusive. If neither mass nor force is specified, the SI unit of mass is assumed, and the force unit is derived. When converting temperatures, you must define the absolute zero value by defining physical constants. If both the unit symbol and the “To SI” factors are specified on the data line, then resulting conversion factor will be the product from both entries.

*UNLOADING DATA: Provide unloading data for uniaxial behavior models in connectors or unloading data from uniaxial and shear tests for fabrics.

This option is used to define unloading response for the uniaxial behavior of connector elements when used in conjunction with the **CONNECTOR BEHAVIOR*, **CONNECTOR UNIAXIAL BEHAVIOR*, and **LOADING DATA* options.

This option is used to define the unloading response from a uniaxial or a shear test for fabric materials when used in conjunction with the **FABRIC*, **UNIAXIAL*, and **LOADING DATA* options. A fabric uniaxial test is specified with increasing strains along the specified yarn direction. A fabric shear test is specified with increasing shear strains as the fill and the warp yarns rotate with respect to each other.

Products: Abaqus/Explicit

Type: Model data

Level: Model

References:

- *Connector Uniaxial Behavior*
- *Fabric Material Behavior*
- **CONNECTOR BEHAVIOR*
- **CONNECTOR UNIAXIAL BEHAVIOR*
- **FABRIC*
- **LOADING DATA*
- **UNIAXIAL*

Defining the unloading response for uniaxial behavior in connectors

Required parameters:

DEFINITION

Set DEFINITION=COMBINED to define an unloading path based on the specified unloading curve and a transition slope to transition from the loading to the unloading curve.

Set DEFINITION=EXPONENTIAL to define an exponential unloading path.

Set DEFINITION=INTERPOLATED CURVE to define an unloading path based on an interpolation between the specified unloading curves.

Set DEFINITION=QUADRATIC to define a quadratic unloading path.

Set DEFINITION=SHIFTED CURVE to define an unloading path based on shifting the specified unloading curve to the point of unloading.

Optional parameters:

RATE DEPENDENT

Include this parameter to define rate-dependent unloading data. If this parameter is omitted, the data are assumed to be rate independent. This parameter can be used only if the loading data are elastic and rate dependent.

Data lines for DEFINITION=COMBINED to define rate-independent unloading behavior that does not depend on independent components:

First line:

1. Transition slope.

Second line:

1. Force or moment. (Provide the absolute value.)
2. Constitutive relative displacement or rotation. (Provide the absolute value.)
3. Temperature.
4. First field variable.
5. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Do not repeat the first data line. Repeat the subsequent data lines as often as necessary to define the unloading curve data.

Data lines for DEFINITION=COMBINED to define rate-independent unloading behavior that depends on independent components:

First line:

1. Transition slope.

Second line:

1. Force or moment. (Provide the absolute value.)
2. Constitutive relative displacement or rotation. (Provide the absolute value.)
3. Constitutive relative motion in the first independent component.
4. Constitutive relative motion in the second independent component.
5. Etc., up to N_i entries as identified on the loading data definition.
6. Temperature.
7. First field variable.
8. Second field variable.

If the number of data entries exceeds the limit of eight entries per line, continue the input on the next data line.

Do not repeat the first data line. Repeat the subsequent data lines as often as necessary to define the unloading curve data.

Data line for DEFINITION=EXPONENTIAL and DEFINITION=QUADRATIC:

First (and only) line:

1. Energy dissipation factor.
2. Permanent deformation factor. (Should be defined only in conjunction with the **LOADING DATA*, TYPE=PERMANENT DEFORMATION option.)

Data lines for DEFINITION=INTERPOLATED CURVE and DEFINITION=SHIFTED CURVE to define rate-independent unloading behavior (the RATE DEPENDENT parameter is omitted) that does not depend on independent components:

First line:

1. Force or moment. (Provide the absolute value.)
2. Constitutive relative displacement or rotation. (Provide the absolute value.)
3. Temperature.
4. First field variable.
5. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the unloading curve data.

Data lines for DEFINITION=INTERPOLATED CURVE and DEFINITION=SHIFTED CURVE to define rate-independent unloading behavior (the RATE DEPENDENT parameter is omitted) that depends on independent components:

First line:

1. Force or moment. (Provide the absolute value.)
2. Constitutive relative displacement or rotation. (Provide the absolute value.)
3. Constitutive relative motion in the first independent component.
4. Constitutive relative motion in the second independent component.
5. Etc., up to N_i entries as identified on the loading data definition.
6. Temperature.
7. First field variable.
8. Second field variable.

   If the number of data entries exceeds the limit of eight entries per line, continue the input on the next data line.

Repeat this set of data lines as often as necessary to define the unloading curve data.

Data lines for DEFINITION=INTERPOLATED CURVE to define rate-dependent unloading behavior (the RATE DEPENDENT parameter is included) that does not depend on independent components:

First line:

1. Force or moment. (Provide the absolute value.)
2. Constitutive relative displacement or rotation. (Provide the absolute value.)
3. Relative velocity. (Provide the absolute value.)
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the unloading curve data.

Data lines for DEFINITION=INTERPOLATED CURVE to define rate-dependent unloading behavior (the RATE DEPENDENT parameter is included) that depends on independent components:

First line:

1. Force or moment. (Provide the absolute value.)
2. Constitutive relative displacement or rotation. (Provide the absolute value.)
3. Relative velocity. (Provide the absolute value.)
4. Connector relative position or constitutive relative motion in the first independent component.
5. Connector relative position or constitutive relative motion in the second independent component.
6. Etc., up to N_i entries as identified on the loading data definition.
7. Temperature.
8. First field variable.

If the number of data entries exceeds the limit of eight entries per line, continue the input on the next data line.

Repeat this set of data lines as often as necessary to define the unloading curve data.

Defining the unloading response data from uniaxial tests of fabric materials

Required parameters:

DEFINITION

Set DEFINITION=COMBINED to define an unloading path based on the specified unloading curve and a transition slope to transition from the loading to the unloading curve.

Set DEFINITION=EXPONENTIAL to define an exponential unloading path.

Set DEFINITION=INTERPOLATED CURVE to define an unloading path based on an interpolation between the specified unloading curves.

Set DEFINITION=QUADRATIC to define a quadratic unloading path.

Set DEFINITION=SHIFTED CURVE to define an unloading path based on shifting the specified unloading curve to the point of unloading.

The available unloading path types depend on the behavior type assigned to the test data (as defined on the **LOADING DATA* option).

Optional parameters:

RATE DEPENDENT

Include this parameter equal to define rate-dependent unloading data. If this parameter is omitted, the data are assumed to be rate independent. This parameter can be used only if the loading data are elastic and rate dependent.

Data lines for DEFINITION=COMBINED:

First line:

1. Transition slope.

Second line:

1. Nominal stress. (Provide the absolute value.)
2. Nominal strain. (Provide the absolute value.)
3. Temperature.
4. First field variable.
5. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Do not repeat the first data line. Repeat the subsequent data lines as often as necessary to define the unloading curve data.

Data line for DEFINITION=EXPONENTIAL and DEFINITION=QUADRATIC:

First (and only) line:

1. Energy dissipation factor.
2. Permanent deformation factor. (Should be defined only in conjunction with the **LOADING DATA*, TYPE=PERMANENT DEFORMATION option.)

Data lines for DEFINITION=INTERPOLATED CURVE and DEFINITION=SHIFTED CURVE to define uniaxial rate-independent unloading behavior (the RATE DEPENDENT parameter is omitted):

First line:

1. Nominal stress. (Provide the absolute value.)
2. Nominal strain. (Provide the absolute value.)
3. Temperature.
4. First field variable.
5. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the unloading curve data.

Data lines for DEFINITION=INTERPOLATED CURVE to define uniaxial rate-dependent unloading behavior (the RATE DEPENDENT parameter is included):

First line:

1. Nominal stress. (Provide the absolute value.)
2. Nominal strain. (Provide the absolute value.)
3. Strain rate. (Provide the absolute value.)
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the unloading curve data.

*USER DEFINED FIELD: Redefine field variables at a material point.

This material option is used to allow the values of field variables at a material point to be redefined within an increment.

There are two approaches for redefining field variables within an increment:

- User subroutine–based approach: Redefine the field variables via user subroutine *USDFLD* in Abaqus/Standard analyses or user subroutine *VUSDFLD* in Abaqus/Explicit analyses.
- Direct approach: Redefine the field variable directly by associating it with a scalar output variable.

If the **USER DEFINED FIELD* option is used, it must appear within a **MATERIAL* definition (*Material Data Definition*).

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *USDFLD*
- *VUSDFLD*
- *Specifying Material Data as Functions of Solution-Dependent Variables*

Optional parameters:

PROPERTIES

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the number of properties being entered. The properties are available for use in user subroutine *VUSDFLD*.

TYPE

Set TYPE=USER (default) to redefine the field variables in a user subroutine.

Set TYPE=SPECIFIED to redefine the field variables by direct specification.

Data lines to define material properties when PROPERTIES is used:

First line:

1. Enter the material properties, eight per line.

Repeat this data line as often as necessary to define all material properties.

Data lines to redefine the field variables when TYPE=SPECIFIED:

First line:

1. Predefined field variable number.
2. Abaqus output variable identifier (scalar values only). The output variable identifers are defined in *Abaqus/Standard Output Variable Identifiers* and *Available Output Variable Keys*.

Repeat this data line as often as necessary to define the field variables.

*USER ELEMENT: Introduce a user-defined element type.

This option is used to introduce a linear or a general user-defined element. It must precede any reference to this user element on an *ELEMENT option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Model

Abaqus/CAE: Actuator/sensor interactions can be defined in the *Interaction module.*

References:

- *User-Defined Elements*
- *UEL*
- *VUEL*
- *ELEMENT
- *MATRIX
- *UEL PROPERTY

Introducing a linear user-defined element (Abaqus/Standard only)

Required parameters:

TYPE

Set this parameter equal to the element type key used to identify this element on the *ELEMENT option. The format of this type key must be U*n* in Abaqus/Standard, where *n* is a positive integer less than 10000. To use this element type, set TYPE=U*n* on the *ELEMENT option.

Optional parameters:

FILE

Set this parameter equal to the name of the results file (with no extension) from which the data are to be read. See *Input Syntax Rules* for the syntax of such file names.

This parameter can be used only if the user-defined element type is linear and its stiffness and/or mass matrices are to be read from the Abaqus/Standard results file of a previous analysis (in which they were written by using the *ELEMENT MATRIX OUTPUT or *SUBSTRUCTURE MATRIX OUTPUT options). When this parameter is used, all values are taken from the results file. For example, if the stiffness or mass being read from the results file is not symmetric, the UNSYMM parameter will be invoked automatically.

If this parameter is omitted, the data will be read from a standard input file.

INTEGRATION

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the number of integration points to be used in Gauss integration. This parameter must be used in conjunction with the TENSOR parameter.

TENSOR

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to specify the element type. This parameter must be used in conjunction with the INTEGRATION parameter.

Set TENSOR=THREED to specify that it is a three-dimensional element in a stress/displacement or heat transfer analysis.

Set TENSOR=TWOD to specify that it is a two-dimensional element in a heat transfer analysis.

Set TENSOR=PSTRAIN to specify that it is a plane strain element in a stress/displacement analysis.

Set TENSOR=PSTRESS to specify that it is a plane stress element in a stress/displacement analysis.

Required parameters if the FILE parameter is included:

OLD ELEMENT

Set this parameter equal to the element number that was assigned to the element whose matrices are being read. This parameter can also be set to a substructure identifier to read a substructure matrix from an Abaqus/Standard results file.

STEP

Set this parameter equal to the step number in which the element matrix was written. This parameter is not required if using a substructure whose matrix was output during its generation.

INCREMENT

Set this parameter equal to the increment number in which the element matrix was written. This parameter is not required if using a substructure whose matrix was output during its generation.

Required parameters if the FILE parameter is omitted:

LINEAR

Include this parameter to indicate that the behavior of the element type is linear and is defined by a stiffness matrix and/or a mass matrix. The **MATRIX* option is required to define the element's behavior.

NODES

Set this parameter equal to the number of nodes associated with an element of this type.

Optional parameters if the FILE parameter is omitted:

COORDINATES

Abaqus/Standard assigns space to store the coordinate values at each node in user subroutine *UEL*. The default number of coordinate values is equal to the largest active degree of freedom of the user element with a maximum of 3. Use the COORDINATES parameter to increase the number of coordinate values.

UNSYMM

Include this parameter if the element matrices are not symmetric. This parameter will cause Abaqus/Standard to use its unsymmetric equation solution capability.

The presence or absence of this parameter determines the form in which the matrices must be provided for reading.

Data lines if the FILE parameter is omitted:

First line:

1. Enter the list of active degrees of freedom at the first node of the element (as determined by the connectivity list). The rule in *Conventions* regarding which degrees of freedom can be used for displacement, rotation, temperature, etc. must be conformed to.

Second line if the active degrees of freedom are different at subsequent nodes:

1. Enter the position in the connectivity list (node position on the element) where the new list of active degrees of freedom first applies.
2. Enter the new list of active degrees of freedom.

Repeat the second data line as often as necessary.

Introducing a general user-defined element

Required parameters:

TYPE

Set this parameter equal to the element type key used to identify this element on the **ELEMENT* option. The format of this type key must be U*n* in Abaqus/Standard and VU*n* in Abaqus/Explicit, where *n* is a positive integer less than 10000. To use this element type, set TYPE=U*n* (or VU*n*) on the **ELEMENT* option.

NODES

Set this parameter equal to the number of nodes associated with an element of this type.

Optional parameters:

COORDINATES

Set this parameter equal to the maximum number of coordinates needed in user subroutine *UEL* in Abaqus/Standard and user subroutine *VUEL* in Abaqus/Explicit at any node point of the element. Abaqus assigns space to store the coordinate values at all the nodes associated with elements of this type. The default is COORDINATES=1.

Abaqus will change the value of COORDINATES to be the maximum of the user-specified value of the COORDINATES parameter or the value of the largest active degree of freedom of the user element that is less than or equal to 3. For example, if COORDINATES=1 and the active degrees of freedom of the user element are 2, 3, and 6, the value of COORDINATES will be changed to 3. If COORDINATES=2 and the active degrees of freedom of the user element are 11 and 12, the value of COORDINATES will remain as 2.

I PROPERTIES

Set this parameter equal to the number of integer property values needed as data in user subroutine *UEL* (or *VUEL*) to define such an element. The default is I PROPERTIES=0.

PROPERTIES

Set this parameter equal to the number of real (floating point) property values needed as data in user subroutine *UEL* (or *VUEL*) to define such an element. The default is PROPERTIES=0.

UNSYMM

This parameter applies only to Abaqus/Standard analyses.

Include this parameter if the element matrices are not symmetric. This parameter will cause Abaqus/Standard to use its unsymmetric equation solution capability.

VARIABLES

Set this parameter equal to the number of solution-dependent state variables that must be stored within the element. Its value must be greater than 0. The default is VARIABLES=1.

Data lines to define a general user-defined element:

First line:

1. Enter the list of active degrees of freedom at the first node of the element (as determined by the connectivity list). The rule in *Conventions* regarding which degrees of freedom can be used for displacement, rotation, temperature, etc. must be conformed to.

Second line if the active degrees of freedom are different at subsequent nodes:

1. Enter the position in the connectivity list (node position on the element) where the new list of active degrees of freedom first applies.
2. Enter the new list of active degrees of freedom.

Repeat the second data line as often as necessary.

*USER MATERIAL: Define material constants for use in subroutine UMAT, UMATHT, VUMAT, or VUMATHT.

This option is used to input material constants for use in a user-defined mechanical model (user subroutine *UMAT* in Abaqus/Standard or user subroutine *VUMAT* in Abaqus/Explicit). In addition, it can be used to input material constants for use in a user-defined thermal material model (user subroutine *UMATHT* in Abaqus/Standard or user subroutine *VUMATHT* in Abaqus/Explicit).

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *User-Defined Mechanical Material Behavior*
- *User-Defined Thermal Material Behavior*
- *UMAT*
- *UMATHT*
- *VUMAT*
- *VUMATHT*

Required parameters:

CONSTANTS

Set this parameter equal to the number of constants being entered.

Optional parameters:

EFFMOD

This parameter applies only to Abaqus/Explicit analyses.

Include this parameter if the effective bulk modulus and shear modulus are returned by user subroutine *VUMAT*. Abaqus/Explicit computes the stable time increment using the returned values.

HYBRID FORMULATION

This parameter applies only to Abaqus/Standard analyses and is relevant only for hybrid elements.

Set HYBRID FORMULATION=INCREMENTAL (default) to use an incremental Lagrange multiplier–based formulation.

Set HYBRID FORMULATION=TOTAL to use a total Lagrange multiplier–based formulation.

Set HYBRID FORMULATION=INCOMPRESSIBLE to define an incompressible material response.

TYPE

Set TYPE=MECHANICAL (default) if the constants are used in defining the mechanical behavior of the material.

Set TYPE=THERMAL if the constants are used in defining the thermal constitutive behavior of the material.

If both the user-defined mechanical and the user-defined thermal behavior of the material are being modeled, the **USER MATERIAL* option should be repeated in the material data block such that each value of the TYPE parameter is used.

UNSYMM

This parameter applies only to Abaqus/Standard analyses.

Include this parameter if the material stiffness matrix, $\partial \Delta \boldsymbol{\sigma} / \partial \Delta \boldsymbol{\varepsilon}$, is not symmetric or when a thermal constitutive model is used and $\partial \mathbf{f} / \partial (\partial \theta / \partial \mathbf{x})$ is not symmetric. This parameter causes Abaqus/Standard to use its unsymmetric equation solution procedures.

Data lines to define material constants:

First line:

1. Give the material constants, eight per line.

Repeat this data line as often as necessary to define all material constants.

*USER OUTPUT VARIABLES: Specify number of user variables.

This option is used to allow Abaqus to allocate space at each material calculation point for user-defined output variables defined in user subroutine *UVARM*. If the **USER OUTPUT VARIABLES* option is used, it must appear within each of the relevant material or gasket behavior definitions.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *UVARM*

There are no parameters associated with this option.

Data line to specify the number of user variables:

First (and only) line:

1. Number of user-defined variables required at each material point.

V

This section describes each Abaqus input option that begins with V, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **VARIABLE MASS SCALING*
- **VIEW FACTOR OUTPUT*
- **VISCO*
- **VISCOELASTIC*
- **VISCOSITY*
- **VISCOUS*
- **VOID NUCLEATION*
- **VOLUMETRIC TEST DATA*

*VARIABLE MASS SCALING: Specify mass scaling during the step.

This option is used to specify mass scaling during the step for part or all of the model.

Products: Abaqus/Explicit

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Mass Scaling*
- *About Output*

Optional parameters:

DT

Set this parameter equal to the desired element stable time increment for the element set provided. The mass scaling is applied according to the method specified with the TYPE parameter. If the DT parameter is omitted, all variable mass scaling definitions from previous steps are removed, and the scaled mass matrix from the end of the previous step is carried over to the current step.

ELSET

Set this parameter equal to the name of the element set for which this mass scaling definition is being applied. If this parameter is omitted, the mass scaling definition will apply to all elements in the model.

The **VARIABLE MASS SCALING* option can be repeated with different ELSET definitions to define different mass scaling for the specified element sets.

TYPE

Set TYPE=UNIFORM to scale the masses of the elements equally so that the smallest element stable time increment of the scaled elements equals the value assigned to DT.

Set TYPE=BELOW MIN (default) to scale the masses of only the elements whose element stable time increments are less than the value assigned to DT. The masses of these elements will be scaled so that the element stable time increments equal the value assigned to DT.

Set TYPE=SET EQUAL DT to scale the masses of all elements so that they have the same element stable time increment equal to the value assigned to DT.

Set TYPE=ROLLING to scale the element masses automatically for the simulation of a rolling process. The appropriate value for the target stable time increment is determined by Abaqus from several parameters of the rolling process. The DT parameter will be ignored in this case.

Required, mutually exclusive parameters if the DT parameter or the TYPE=ROLLING parameter is used:

FREQUENCY

Set this parameter equal to the frequency, in increments, at which mass scaling calculations are to be performed during the step. For example, FREQUENCY=5 will scale the mass at the beginning of the step and at increments 5, 10, 15, etc. The value of this parameter must be a positive integer.

NUMBER INTERVAL

Set this parameter equal to the number of intervals during the step at which mass scaling calculations will be performed. For example, if NUMBER INTERVAL=2, mass scaling calculations will be performed at the beginning of the step, the increment immediately following the half-way point in the step, and the final increment in the step.

Required parameters for TYPE=ROLLING:

CROSS SECTION NODES

Set this parameter equal to the number of nodes in the cross-section of the workpiece. Increasing this value will decrease the amount of mass scaling used.

EXTRUDED LENGTH

Set this parameter equal to the average element length in the rolling direction.

FEED RATE

Set this parameter equal to the estimated average velocity of the workpiece in the rolling direction at steady-state conditions.

There are no data lines associated with this option.

*VIEW FACTOR OUTPUT: Write radiation view factors to the results file in cavity radiation heat transfer analysis.

This option is used to write cavity radiation element view factor matrices to the results file. This option is available only for heat transfer analysis including cavity radiation.

Products: Abaqus/Standard

Type: History data

Level: Step

References:

- *Cavity Radiation in Abaqus/Standard*
- *About Output*

Required parameters:

CAVITY

Set this parameter equal to the name of the cavity for which this output request is being made.

Optional parameters:

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be written at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

There are no data lines associated with this option.

*VISCO: Transient, static, stress/displacement analysis with time-dependent material response (creep, swelling, and viscoelasticity).

This option is used to obtain a transient static response in an analysis with time-dependent material behavior (creep, swelling, and viscoelasticity).

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- *Quasi-Static Analysis*
- *Rate-Dependent Plasticity: Creep and Swelling*

Optional parameters:

ALLSDTOL

Include this parameter to indicate that an adaptive automatic damping algorithm will be activated in this step. Set this parameter equal to the maximum allowable ratio of the stabilization energy to the total strain energy. The initial damping factor is specified via the STABILIZE parameter or the FACTOR parameter. This damping factor will then be adjusted through the step based on the convergence history and the value of ALLSDTOL. If this parameter is set equal to zero, the adaptive automatic damping algorithm is not activated; a constant damping factor will be used throughout the step. If this parameter is included without a specified value, the default value is 0.05. If this parameter is omitted but the STABILIZE parameter is included with the default value of dissipated energy fraction, the adaptive automatic damping algorithm will be activated automatically with ALLSDTOL=0.05.

This parameter must be used in conjunction with the STABILIZE parameter (see *Solving Nonlinear Problems*).

CETOL

Set this parameter equal to the maximum difference in the creep strain increment calculated from the creep strain rates based on conditions at the beginning and on conditions at the end of the increment, thus controlling the accuracy of the creep integration. In general, the tolerance can be calculated by choosing an acceptable stress error tolerance and dividing by a typical elastic modulus. If the model consists of a linear viscoelastic material characterized by the **VISCOELASTIC* option, a looser tolerance of the order of the elastic strains can be used. If CETOL is omitted, fixed time increments are used.

CONTINUE

Set CONTINUE=NO (default) to specify that this step will not carry over the damping factors from the results of the preceding general step. In this case the initial damping factors will be recalculated based on the declared damping intensity and on the solution of the first increment of the step or can be specified directly.

Set CONTINUE=YES to specify that this step will carry over the damping factors from the end of the immediately preceding general step.

This parameter must be used in conjunction with the ALLSDTOL and the STABILIZE parameters.

CREEP

Set CREEP=EXPLICIT to use explicit integration for creep behavior throughout the step, which may sometimes be computationally less expensive. The time increment will be limited by the accuracy tolerance (CETOL) and by the stability limit of the forward difference operator. See *Rate-Dependent Plasticity: Creep and Swelling* for details on the integration scheme.

FACTOR

Set this parameter equal to the damping factor to be used in the automatic damping algorithm (see *Solving Nonlinear Problems*) if the problem is expected to be unstable due to local instabilities and the damping factor calculated by Abaqus/Standard is not suitable. This parameter must be used in conjunction with the STABILIZE parameter and overrides the automatic calculation of the damping factor based on a value of the dissipated energy fraction.

STABILIZE

Include this parameter to use automatic stabilization if the problem is expected to be unstable due to local instabilities. Set this parameter equal to the dissipated energy fraction of the automatic damping algorithm (see *Solving Nonlinear Problems*). If this parameter is omitted, the stabilization algorithm is not activated. If this parameter is included without a specified value, the default value of the dissipated energy fraction is 2×10^{-4} and the adaptive automatic damping algorithm will be activated by default with ALLSDTOL=0.05 in this step; set ALLSDTOL=0 to deactivate the adaptive automatic damping algorithm. If the FACTOR parameter is used, any value of the dissipated energy fraction will be overridden by the damping factor.

Data line for a transient, quasi-static analysis:

First (and only) line:

1. Suggested initial time increment. This value should be reasonably correct (in agreement with the strain change allowed with the CETOL parameter) but may be reduced if automatic time incrementation is being used.
2. Time period of the step.
3. Minimum time increment allowed. If Abaqus/Standard finds it needs a smaller time increment than this value, the analysis is terminated. If this entry is zero, a default value of the smaller of the suggested initial time increment or 10^{-5} times the total time period is assumed.
4. Maximum time increment allowed. Only used for automatic time incrementation. If this value is not specified, no upper limit is imposed.

*VISCOELASTIC: Specify dissipative behavior for use with elasticity.

This option is used to generalize a material's elastic response to include viscoelasticity. The viscoelasticity can be defined as a function of frequency for steady-state small-vibration analyses, as a function of reduced time for time-dependent analyses, or by specifying a creep law for nonlinear viscoelastic analyses.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Parallel Rheological Framework*
- *Time Domain Viscoelasticity*
- *Frequency Domain Viscoelasticity*
- *UCREEPNETWORK*
- *VUCREEPNETWORK*
- **COMBINED TEST DATA*
- **NETWORK STIFFNESS RATIO*
- **SHEAR TEST DATA*
- **TRS*
- **VOLUMETRIC TEST DATA*

Required, mutually exclusive parameters:

FREQUENCY

This parameter applies only to frequency domain procedures in Abaqus/Standard analyses.

Use this parameter to choose the frequency domain definition. In this case the material's long-term elasticity must be defined using the **ELASTIC* or the **HYPERELASTIC* option. If a frequency-domain viscoelastic response is specified for a time domain procedure, the viscoelastic response is ignored and the material response is based on the long-term elastic moduli.

Set FREQUENCY=CREEP TEST DATA to define the frequency domain response using a Prony series representation of the relaxation moduli, if the Prony series parameters are to be computed from shear and volumetric creep test data.

Set FREQUENCY=FORMULA to define the dissipative material parameters by the power law formulæ.

Set FREQUENCY=PRONY to define the frequency domain response using a Prony series representation of the relaxation moduli by specifying the Prony series parameters.

Set FREQUENCY=RELAXATION TEST DATA to define the frequency domain response using a Prony series representation of the relaxation moduli, if the Prony series parameters are to be computed from shear and volumetric relaxation test data.

Set FREQUENCY=TABULAR to provide tabular definition of the frequency domain response.

NONLINEAR

Include this parameter to define a nonlinear viscoelastic network within the parallel rheological framework.

TIME

This parameter applies only to time domain procedures.

Use this parameter to choose the time domain definition. In this case the material's elasticity must be defined using the **ELASTIC*, the **HYPERELASTIC*, or the **HYPERFOAM* option. If a time-domain viscoelastic response is specified for a frequency domain procedure, the viscoelastic response is ignored and the material response is based on the instantaneous elastic moduli.

Set TIME=CREEP TEST DATA if the Prony series parameters are to be computed by Abaqus from data taken from shear and volumetric creep tests.

Set TIME=FREQUENCY DATA if the Prony series parameters are to be computed by Abaqus from frequency-dependent cyclic test data.

Set TIME=PRONY to define a linear, viscoelastic material by giving the parameters of the Prony series representation of the relaxation moduli.

Set TIME=RELAXATION TEST DATA if the Prony series parameters are to be computed by Abaqus from data taken from shear and volumetric relaxation tests.

Required parameters when the NONLINEAR parameter is included:

LAW

Set LAW=BB POWER LAW to choose the preferred form of the Bergstrom-Boyce power law. It is recommended that you use this parameter setting rather than LAW=BERGSTROM-BOYCE.

Set LAW=BERGSTROM-BOYCE to choose a Bergstrom-Boyce law. This parameter setting is superseded by LAW=BB POWER LAW.

Set LAW=DARVEAUX to choose a Darveaux law.

Set LAW=MODIFIED DARVEAUX to choose a modified Darveaux law.

Set LAW=HYPERB to choose a hyperbolic-sine law.

Set LAW=POWER LAW to choose a power law. It is recommended that you use this parameter setting rather than LAW=STRAIN.

Set LAW=STRAIN to choose a strain hardening law. This parameter setting is superseded by LAW=POWER LAW.

Set LAW=USER to input the creep law using user subroutine *UCREEPNETWORK* or user subroutine *VUCREEPNETWORK*.

NETWORKID

Set this parameter equal to the network id. This number must be greater than or equal to 1 and smaller than or equal to the number of networks. The network ids must consist of consecutive integers.

SRATIO

Set this parameter equal to the stiffness ratio for the network. The sum of the ratios for the networks must be smaller than or equal to 1. If the sum is smaller than 1, an additional elastic network is defined with the value of the stiffness ratio such that the condition is satisfied.

Optional parameters:

DEPENDENCIES

This parameter can be used only when the NONLINEAR parameter is included.

Set this parameter equal to the number of field variable dependencies included in the definition of the creep constants in addition to temperature. If this parameter is omitted, it is assumed that the creep constants have no dependencies or depend only on temperature. See *Specifying Field Variable Dependence* for more information.

PRELOAD

This parameter applies only to Abaqus/Standard analyses.

This parameter can be used only in conjunction with FREQUENCY=TABULAR to specify the nature of preload used for defining frequency-domain viscoelastic material properties or effective thickness-direction gasket properties.

Set PRELOAD=UNIAXIAL to specify that the frequency-domain viscoelastic material properties correspond to a uniaxial test.

Set PRELOAD=VOLUMETRIC to specify that the frequency-domain viscoelastic material properties correspond to a volumetric test. This setting is not meaningful when used with gasket elements to define effective thickness-direction properties.

PROPERTIES

This parameter applies only to Abaqus/Standard analyses and can be used only when LAW=USER is used.

Set this parameter equal to the number of property values needed as data in user subroutine *UCREEPNETWORK* or user subroutine *VUCREEPNETWORK*. The default is PROPERTIES=0.

TYPE

This parameter cannot be used in conjunction with the NONLINEAR parameter.

Use this parameter to define whether the **VISCOELASTIC* option is being used to define continuum material properties or effective thickness-direction gasket properties.

Set TYPE=ISOTROPIC (default, and only option for Abaqus/Explicit) to define continuum material properties. This choice is appropriate when the viscoelastic material model is used for any continuum, structural, or special-purpose elements whose material response is modeled using continuum material properties (examples of such special-purpose elements include modeling cohesive elements with a continuum response—see *Modeling of an Adhesive Layer of Finite Thickness* or gasket elements with a material response—see *Defining the Gasket Behavior Using a Material Model*). In Abaqus/Explicit this parameter should also be used for the definition of viscoelastic properties for cohesive elements with elastic traction-separation behavior (*Modeling Viscoelastic Traction-Separation Behavior in Abaqus/Explicit*).

Set TYPE=TRANSVERSELY ISOTROPIC to define a linear, transversely isotropic, viscoelastic material. This option can be used only with TIME=PRONY or FREQUENCY=PRONY.

Set TYPE=ORTHOTROPIC to define a linear,orthotropic, viscoelastic material. This option can be used only with TIME=PRONY or FREQUENCY=PRONY.

Set TYPE=TRACTION to define effective thickness-direction gasket properties. This option is supported only for gasket elements whose behavior is modeled directly using a gasket behavior model (*Defining the Gasket Behavior Using a Material Model*).

Optional parameters when test data are given to define time domain viscoelasticity with TIME=CREEP TEST DATA, TIME=RELAXATION TEST DATA, or TIME=FREQUENCY DATA or

when test data are given to define frequency domain viscoelasticity with FREQUENCY=CREEP TEST DATA or FREQUENCY=RELAXATION TEST DATA:

ERRTOL

Set this parameter equal to the allowable average root-mean-square error of the data points in the least-squares fit. The default is 0.01 (1%).

NMAX

Set this parameter equal to the maximum number of terms *N* in the Prony series. Abaqus will perform the least-squares fit from $N = 1$ to $N =$ NMAX until convergence is achieved for the lowest *N* with respect to ERRTOL. The default and maximum value is 13.

Data line to define continuum material properties for FREQUENCY=FORMULA:

First (and only) line:

1. Real part of g_1^*. $(g^*(\omega) = g_1^* f^{-a})$
2. Imaginary part of g_1^*.
3. Value of *a*.
4. Real part of k_1^*. $(k^*(\omega) = k_1^* f^{-b})$ If the material is incompressible, this value is ignored.
5. Imaginary part of k_1^*. If the material is incompressible, this value is ignored.
6. Value of *b*. If the material is incompressible, this value is ignored.

Data lines to define continuum material properties for FREQUENCY=TABULAR without the PRELOAD parameter or for TIME=FREQUENCY DATA:

First line:

1. Real part of ωg^*. $(\omega \Re(g^*) = G_\ell / G_\infty)$
2. Imaginary part of ωg^*. $(\omega \Im(g^*) = 1 - G_s / G_\infty)$
3. Real part of ωk^*. $(\omega \Re(k^*) = \mathrm{K}_\ell / \mathrm{K}_\infty)$ If the material is incompressible, this value is ignored.
4. Imaginary part of ωk^*. $(\omega \Im(k^*) = 1 - \mathrm{K}_s / \mathrm{K}_\infty)$ If the material is incompressible, this value is ignored.
5. Frequency, *f*, in cycles per time.

Repeat this data line as often as necessary to define the dissipative part of the material behavior.

Data lines to define continuum material properties for FREQUENCY=TABULAR, PRELOAD=UNIAXIAL:

First line:

1. Uniaxial loss modulus.
2. Uniaxial storage modulus.
3. Frequency, *f*, in cycles per time.
4. Uniaxial nominal strain (defines the level of uniaxial preload).

Repeat this data line as often as necessary to define the uniaxial loss and storage moduli as functions of frequency and preload.

Data lines to define continuum material properties for FREQUENCY=TABULAR, PRELOAD=VOLUMETRIC:

First line:

1. Bulk loss modulus.
2. Bulk storage modulus.
3. Frequency, *f*, in cycles per time.
4. Volume ratio, *J* (current volume/original volume; defines the level of volumetric preload).

Repeat this data line as often as necessary to define the bulk loss and storage moduli as functions of frequency and preload.

Data lines to specify continuum material properties with the Prony series parameters directly using TIME=PRONY or FREQUENCY=PRONY with TYPE=ISOTROPIC:

First line:

1. $\bar{g}_1^P$, the modulus ratio in the first term in the Prony series expansion of the shear relaxation modulus.
2. $\bar{k}_1^P$, the modulus ratio in the first term in the Prony series expansion of the bulk relaxation modulus.
3. τ_1, the relaxation time for the first term in the Prony series expansion.

Repeat this data line as often as necessary to define the second, third, etc. terms in the Prony series. There is no restriction on the number of terms in the Prony series.

Data lines to specify continuum material properties with the Prony series parameters directly using TIME=PRONY or FREQUENCY=PRONY with TYPE=TRANSVERSELY ISOTROPIC:

First line:

1. $\bar{r}^P_{(1,1111)}$, the modulus ratio in the first term in the Prony series expansion of the D_{1111} component of the relaxation modulus.
2. $\bar{r}^P_{(1,2222)}$, the modulus ratio in the first term in the Prony series expansion of the D_{2222} component of the relaxation modulus.
3. $\bar{r}^P_{(1,1122)}$, the modulus ratio in the first term in the Prony series expansion of the D_{1122} component of the relaxation modulus.
4. $\bar{r}^P_{(1,2233)}$, the modulus ratio in the first term in the Prony series expansion of the D_{2233} component of the relaxation modulus.
5. $\bar{r}^P_{(1,1212)}$, the modulus ratio in the first term in the Prony series expansion of the D_{1212} component of the relaxation modulus.
6. τ_1, the relaxation time for the first term in the Prony series expansion.

Repeat this data line as often as necessary to define the second, third, etc. terms in the Prony series. There is no restriction on the number of terms in the Prony series.

Data lines to specify continuum material properties with the Prony series parameters directly using TIME=PRONY or FREQUENCY=PRONY, TYPE=ORTHOTROPIC:

First line:

1. $\bar{r}^P_{(1,1111)}$, the modulus ratio in the first term in the Prony series expansion of the D_{1111} component of the relaxation modulus.
2. $\bar{r}^P_{(1,1122)}$, the modulus ratio in the first term in the Prony series expansion of the D_{1122} component of the relaxation modulus.
3. $\bar{r}^P_{(1,2222)}$, the modulus ratio in the first term in the Prony series expansion of the D_{2222} component of the relaxation modulus.
4. $\bar{r}^P_{(1,1133)}$, the modulus ratio in the first term in the Prony series expansion of the D_{1133} component of the relaxation modulus.
5. $\bar{r}^P_{(1,2233)}$, the modulus ratio in the first term in the Prony series expansion of the D_{2233} component of the relaxation modulus.
6. $\bar{r}^P_{(1,3333)}$, the modulus ratio in the first term in the Prony series expansion of the D_{3333} component of the relaxation modulus.
7. $\bar{r}^P_{1,1212}$, the modulus ratio in the first term in the Prony series expansion of the D_{1212} component of the relaxation modulus.
8. $\bar{r}^P_{(1,1313)}$, the modulus ratio in the first term in the Prony series expansion of the D_{1313} component of the relaxation modulus.

Second line:

1. $\bar{r}^P_{(1,2323)}$, the modulus ratio in the first term in the Prony series expansion of the D_{2323} component of the relaxation modulus.
2. τ_1, the relaxation time for the first term in the Prony series expansion.

Repeat this set of data lines as often as necessary to define the second, third, etc. terms in the Prony series. There is no restriction on the number of terms in the Prony series.

Data lines to define viscoelastic properties for cohesive elements with elastic traction-separation behavior using TIME=PRONY in Abaqus/Explicit:

First line:

1. $\bar{g}^P_1$, the modulus ratio in the first term in the Prony series expansion of the shear traction relaxation modulus.
2. $\bar{k}^P_1$, the modulus ratio in the first term in the Prony series expansion of the normal traction relaxation modulus.
3. τ_1, the relaxation time for the first term in the Prony series expansion.

Repeat this data line as often as necessary to define the second, third, etc. terms in the Prony series. There is no restriction on the number of terms in the Prony series.

Data lines to define viscoelastic properties for cohesive elements with elastic traction-separation behavior using TIME=FREQUENCY DATA in Abaqus/Explicit:

First line:

1. Real part of ωg^*. $(\omega \Re(g^*) = G_\ell / G_\infty)$
2. Imaginary part of ωg^*. $(\omega \Im(g^*) = 1 - G_s / G_\infty)$
3. Real part of ωk^*. $(\omega \Re(k^*) = \mathrm{K}_\ell / \mathrm{K}_\infty)$
4. Imaginary part of ωk^*. $(\omega \Im(k^*) = 1 - \mathrm{K}_s / \mathrm{K}_\infty)$
5. Frequency, *f*, in cycles per time.

Repeat this data line as often as necessary to define the dissipative part of the material behavior.

To specify viscoelastic behavior via test data:

No data lines are used with this option when either TIME=CREEP TEST DATA or TIME=RELAXATION TEST DATA is specified. The test data are given by the **SHEAR TEST DATA* and the **VOLUMETRIC TEST DATA* options or by the **COMBINED TEST DATA* option.

Data lines to define effective thickness-direction gasket properties for PRELOAD=UNIAXIAL:

First line:

1. Effective thickness-direction loss modulus.
2. Effective thickness-direction storage modulus.
3. Frequency, *f*, in cycles per time.
4. Closure (defines the level of preload).

Repeat this data line as often as necessary to define the effective thickness-direction gasket loss and storage moduli as functions of frequency and preload.

Data lines to define effective thickness-direction gasket properties if PRELOAD=UNIAXIAL is not included:

First line:

1. Real part of ωk^*. $(\omega \Re(k^*) = k_\ell / k_\infty)$, where k^* represents the complex effective thickness direction dynamic stiffness.
2. Imaginary part of ωk^*. $(\omega \Im(k^*) = 1 - k_s / k_\infty)$, where k^* represents the complex effective thickness direction dynamic stiffness.
3. Frequency, *f*, in cycles per time.

Repeat this data line as often as necessary to define the normalized effective thickness-direction gasket loss and storage moduli as functions of frequency.

Data lines for LAW=BB POWER LAW:

First line:

1. q_0. (Units of FL^{-2}.)

2. m.
3. C.
4. E, a constant used for regularizing the creep strain rate near the undeformed state. The value of E should be non-negative. If this field is left blank, the default value of 0.01 is used.
5. $\dot{\varepsilon}_0$. The default is 1.0. (Units of T^{-1}.)
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the creep constants on temperature and other predefined field variables.

Data lines for LAW=BERGSTROM-BOYCE:

First line:

1. Creep parameter, A.
2. Effective stress exponent, m.
3. Creep strain exponent, C.
4. E, a constant used for regularizing the creep strain rate near the undeformed state. The value of E should be non-negative. If this field is left blank, the default value of 0.01 is used.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the creep constants on temperature and other predefined field variables.

Data lines for LAW=DARVEAUX and LAW=MODIFIED DARVEAUX:

First line:

1. C_{ss}. (Units of T^{-1}.)
2. α. (Units of $F^{-1}L^2$.)
3. n.
4. ϵ_T.
5. B.
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc. up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the creep constants on predefined field variables.

Data lines for LAW=HYPERB:

First line:

1. A. (Units of T^{-1}.)
2. B. (Units of $F^{-1}L^2$.)
3. n.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc. up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the creep constants on predefined field variables.

Data lines for LAW=POWER LAW:

First line:

1. q_0. (Units of FL^{-2}.)
2. n.
3. m.
4. a. The default is 0.0.
5. $\dot{\varepsilon}_0$. The default is 1.0. (Units of T^{-1}.)
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the creep constants on temperature and other predefined field variables.

Data lines for LAW=STRAIN:

First line:

1. A. (Units of $F^{-n}L^{2n}T^{-1-m}$.)
2. n.
3. m.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.

2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the creep constants on temperature and other predefined field variables.

Data lines for LAW=USER:

No data lines are needed if the PROPERTIES parameter is omitted or set to 0. Otherwise, first line:

1. Give the properties, eight per line.

Repeat this data line as often as necessary to define the material properties.

*VISCOSITY: Specify material shear viscosity.

This option is used to specify the shear viscosity of the material. When used in Abaqus/Explicit, it must be used in conjunction with the **EOS* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Viscosity*
- *Equation of State*
- *VUVISCOSITY*
- **EOS*
- **TRS*

Optional parameters:

DEFINITION

Use this parameter to choose the shear viscosity of the material.

Set DEFINITION=BINGHAM PLASTIC to define the Bingham plastic viscous shear behavior. This behavior is valid only for fluid pipe and fluid pipe connector elements in Abaqus/Standard.

Set DEFINITION=CARREAU-YASUDA (Abaqus/Explicit only) to define the Carreau-Yasuda viscous shear behavior.

Set DEFINITION=CROSS (Abaqus/Explicit only) to define the Cross viscous shear behavior.

Set DEFINITION=ELLIS-METER (Abaqus/Explicit only) to define the Ellis-Meter viscous shear behavior.

Set DEFINITION=HERSCHEL-BULKLEY to define the Herschel-Bulkley viscous shear behavior. This behavior is valid for Abaqus/Explicit and is valid only for fluid pipe and fluid pipe connector elements in Abaqus/Standard.

Set DEFINITION=NEWTONIAN (default) to define Newtonian (linear) viscous shear behavior. This behavior is valid for Abaqus/Explicit and is valid only for fluid pipe and fluid pipe connector elements in Abaqus/Standard.

Set DEFINITION=POWELL-EYRING (Abaqus/Explicit only) to define the Powell-Eyring viscous shear behavior.

Set DEFINITION=POWER LAW to define the power law viscous shear behavior. This behavior is valid for Abaqus/Explicit and is valid only for fluid pipe and fluid pipe connector elements in Abaqus/Standard.

Set DEFINITION=TABULAR (Abaqus/Explicit only) to define the non-Newtonian viscous shear behavior in tabular form.

Set DEFINITION=USER (Abaqus/Explicit only) to define the viscous shear behavior in user subroutine *VUVISCOSITY*.

DEPENDENCIES

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the number of field variable dependencies included in the definition of the viscosity, in addition to temperature. If this parameter is omitted, it is assumed that the viscosity depends only on temperature.

FLUID ID

This parameter applies only to Abaqus/Standard analyses.

Set FLUID ID equal to the fluid identifier when multi-fluid flow is defined.

This parameter is valid only for DEFINITION= BINGHAM PLASTIC, DEFINITION=NEWTONIAN, DEFINITION= POWER LAW, and DEFINITION=HERSCHEL-BULKLEY.

PROPERTIES

This parameter can be used only if DEFINITION=USER is specified.

Set this parameter equal to the number of property values needed as data in user subroutine *VUVISCOSITY*. The default value is 0.

SLURRY

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to define viscosity as a function of the slurry concentration.

This parameter is valid only for DEFINITION= BINGHAM PLASTIC, DEFINITION=NEWTONIAN, DEFINITION= POWER LAW, and DEFINITION=HERSCHEL-BULKLEY.

Data lines to define the Bingham plastic viscous shear behavior (DEFINITION=BINGHAM PLASTIC) without the SLURRY parameter:

First line:

1. Shear viscosity at low shear rates, η_0. (Units of $FL^{-2}T$.)
2. Yield shear stress, τ_0. (Units of FL^{-2}.)
3. Consistency index, k. (Units of $FL^{-2}T$.)
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the coefficients of the Bingham plastic viscous shear model as a function of temperature and other predefined field variables.

Data lines to define the Bingham plastic viscous shear behavior (DEFINITION=BINGHAM PLASTIC) with the SLURRY parameter:

First line:

1. Shear viscosity at low shear rates, η_0. (Units of $FL^{-2}T$.)
2. Yield shear stress, τ_0. (Units of FL^{-2}.)

3. Consistency index, k. (Units of $FL^{-2}T$.)
4. Slurry concentration, C_v.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the coefficients of the Bingham plastic viscous shear model as a function of temperature and other predefined field variables.

Data lines to define the Carreau-Yasuda viscous shear behavior (DEFINITION=CARREAU-YASUDA):

First line:

1. Shear viscosity at low shear rates (Newtonian limit), η_0. (Units of $FL^{-2}T$.)
2. Shear viscosity at large shear rates, η_∞. (Units of $FL^{-2}T$.)
3. Time constant, λ. (Units of T.)
4. Flow behavior index, n.
5. a.
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the coefficients of the Carreau-Yasuda viscosity model as a function of temperature and other predefined field variables.

Data lines to define the Cross viscous shear behavior (DEFINITION=CROSS):

First line:

1. Shear viscosity at low shear rates (Newtonian limit), η_0. (Units of $FL^{-2}T$.)
2. Shear viscosity at large shear rates, η_∞. (Units of $FL^{-2}T$.)
3. Time constant, λ. (Units of T.)
4. Flow behavior index, n.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the coefficients of the Cross viscosity model as a function of temperature and other predefined field variables.

Data lines to define the Ellis-Meter viscous shear behavior (DEFINITION=ELLIS-METER):

First line:

1. Shear viscosity at low shear rates (Newtonian limit), η_0. (Units of $FL^{-2}T$.)
2. Shear viscosity at large shear rates, η_∞. (Units of $FL^{-2}T$.)
3. Effective shear stress, $\tau_{1/2}$, at which the viscosity is 50% between the Newtonian limit, η_0, and the infinite shear viscosity, η_∞. (Units of FL^{-2}.)
4. Flow behavior index, n.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the coefficients of the Ellis-Meter viscosity model as a function of temperature and other predefined field variables.

Data lines to define the Herschel-Bulkley viscous shear behavior (DEFINITION=HERSCHEL-BULKLEY) without the SLURRY parameter:

First line:

1. Shear viscosity at low shear rates, η_0. (Units of $FL^{-2}T$.)
2. Yield shear stress, τ_0. (Units of FL^{-2}.)
3. Consistency index, k. (Units of $FL^{-2}T^n$.)
4. Flow behavior index, n.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Data lines to define the Herschel-Bulkley viscous shear behavior (DEFINITION=HERSCHEL-BULKLEY) with the SLURRY parameter:

First line:

1. Shear viscosity at low shear rates, η_0. (Units of $FL^{-2}T$.)
2. Yield shear stress, τ_0. (Units of FL^{-2}.)
3. Consistency index, k. (Units of $FL^{-2}T^n$.)
4. Flow behavior index, n.
5. Slurry concentration, C_v.
6. Temperature.
7. First field variable.
8. Second field variable.

9. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the coefficients of the Herschel-Bulkley viscosity model as a function of temperature and other predefined field variables.

Data lines to define Newtonian viscous shear behavior (DEFINITION=NEWTONIAN) without the SLURRY parameter:

First line:

1. Viscosity. (Units of $FL^{-2}T$.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the viscosity as a function of temperature and other predefined field variables.

Data lines to define Newtonian viscous shear behavior (DEFINITION=NEWTONIAN) with the SLURRY parameter:

First line:

1. Viscosity. (Units of $FL^{-2}T$.)
2. Slurry concentration, C_v.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the viscosity as a function of temperature and other predefined field variables.

Data lines to define the Powell-Eyring viscous shear behavior (DEFINITION=POWELL-EYRING):

First line:

1. Shear viscosity at low shear rates (Newtonian limit), η_0. (Units of $FL^{-2}T$.)
2. Shear viscosity at large shear rates, η_∞. (Units of $FL^{-2}T$.)
3. Time constant, λ. (Units of T.)
4. Temperature.
5. First field variable.

6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the coefficients of the Powell-Eyring viscosity model as a function of temperature and other predefined field variables.

Data lines to define the power law viscous shear behavior (DEFINITION=POWER LAW) without the SLURRY parameter:

First line:

1. Consistency index, k. (Units of $FL^{-2}T^{n}$.)
2. Flow behavior index, n.
3. Maximum viscosity, η_{max}. (Units of $FL^{-2}T$.)
4. Minimum viscosity, η_{min}. (Units of $FL^{-2}T$.)
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the coefficients of the power law viscosity model as a function of temperature and other predefined field variables.

Data lines to define the power law viscous shear behavior (DEFINITION=POWER LAW) with the SLURRY parameter:

First line:

1. Consistency index, k. (Units of $FL^{-2}T^{n}$.)
2. Flow behavior index, n.
3. Maximum viscosity, η_{max}. (Units of $FL^{-2}T$.)
4. Minimum viscosity, η_{min}. (Units of $FL^{-2}T$.)
5. Slurry concentration, C_v.
6. Temperature.
7. First field variable.
8. Second field variable.
9. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the coefficients of the power law viscosity model as a function of temperature and other predefined field variables.

Data lines to define the viscous shear behavior in tabular form (DEFINITION=TABULAR):

First line:

1. Viscosity, η. (Units of $FL^{-2}T$.)
2. Effective shear strain rate, $\dot{\gamma}$. (Units of T^{-1}.)
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the viscosity as a function of effective shear strain rate, temperature, and other predefined field variables.

Data lines to define the material properties for the user-defined viscosity model (DEFINITION=USER):

No data lines are needed if the PROPERTIES parameter is omitted or set to 0. Otherwise, first line:

1. Give the material properties, eight per line.

Repeat this data line as often as necessary to define the material properties.

*VISCOUS: Specify viscous material properties for the two-layer viscoplastic model.

This option is used to define the viscous properties for the two-layer viscoplastic material model. It must be used in conjunction with the **ELASTIC* and **PLASTIC* options.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Two-Layer Viscoplasticity*
- **ELASTIC*
- **PLASTIC*
- **POTENTIAL*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the material properties, in addition to temperature. If this parameter is omitted, it is assumed that the material properties depend only on temperature. See *Material Data Definition* for more information.

LAW

Set LAW=STRAIN (default) to choose a strain hardening law. This parameter setting is superseded by LAW=POWER LAW.

Set LAW=TIME to choose a time hardening law. This parameter setting is superseded by LAW=TIME POWER LAW.

Set LAW=USER to input the creep law using user subroutine *CREEP*.

Set LAW=ANAND to choose an Anand law.

Set LAW=DARVEAUX to choose a Darveaux law.

Set LAW=DOUBLE POWER to choose a double power law.

Set LAW=POWER LAW to choose a power law. It is recommended that you use this parameter setting rather than LAW=STRAIN.

Set LAW=TIME POWER LAW to choose a time power law. It is recommended that you use this parameter setting rather than LAW=TIME.

TIME

This parameter is relevant only when LAW=TIME or LAW=TIME POWER LAW.

Set TIME=CREEP to use creep time.

Set TIME=TOTAL (default) to use total time.

Data lines for LAW=TIME or LAW=STRAIN:

First line:

1. A. (Units of $F^{-n}L^{2n}T^{-1-m}$.)
2. n.
3. m.
4. f.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the viscous constants on temperature and other predefined field variable.

Data lines for LAW=USER:

First line:

1. f.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc. up to seven field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than seven):

1. Eighth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the viscous constants on temperature and other predefined field variables.

Data lines for LAW=ANAND:

First line:

1. S_1. (Units of FL^{-2}.)
2. $\boldsymbol{Q/R}$. (Units of $\boldsymbol{\theta}$.)
3. A. (Units of T^{-1}.)
4. $\boldsymbol{\xi}$.
5. m.
6. A_0. (Units of FL^{-2}.)
7. $\hat{\boldsymbol{s}}$. (Units of FL^{-2}.)
8. n.

Second line:

1. a.
2. S_2. (Units of $FL^{-2}\boldsymbol{\theta^{-1}}$.)
3. S_3. (Units of $FL^{-2}\boldsymbol{\theta^{-2}}$.)

4. A_1. (Units of $FL^{-2}\theta^{-1}$.)
5. A_2. (Units of $FL^{-2}\theta^{-2}$.)
6. A_3. (Units of $FL^{-2}T$.)
7. A_4. (Units of $FL^{-2}T^2$.)
8. *f*.

Data lines for LAW=DARVEAUX:

First line:

1. C_{ss}. (Units of T^{-1}.)
2. Q/R. (Units of θ.)
3. α. (Units of $F^{-1}L^2$.)
4. *n*.
5. ϵ_T.
6. *B*.
7. *f*.

Data lines for LAW=DOUBLE POWER:

First line:

1. A_1. (Units of T^{-1}.)
2. B_1. (Units of θ.)
3. C_1.
4. A_2. (Units of T^{-1}.)
5. B_2. (Units of θ.)
6. C_2.
7. σ_0. (Units of FL^{-2}.)
8. *f*.

Data lines for LAW=POWER LAW:

First line:

1. q_0. Units of FL^{-2}.)
2. *n*.
3. *m*.
4. Blank space.
5. $\dot{\varepsilon}_0$. The default is 1.0. (Units of T^{-1}.)
6. *f*.
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the viscous constants on temperature and other predefined field variables.

Data lines for LAW=TIME POWER LAW:

First line:

1. q_0. (Units of FL^{-2}.)
2. n.
3. m.
4. $\dot{\varepsilon}_0$. The default is 1.0. (Units of T^{-1}.)
5. f.
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the viscous constants on temperature and other predefined field variables.

*VOID NUCLEATION: Define the nucleation of voids in a porous material.

This option is used to model the nucleation of voids in a porous material. It can be used only with the **POROUS METAL PLASTICITY* option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Porous Metal Plasticity*
- **POROUS METAL PLASTICITY*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies, in addition to temperature, that define the normal distribution of the nucleation strain. If this parameter is omitted, the constants defining the normal distribution may depend only on temperature. See *Material Data Definition* for more information.

Data lines to define void nucleation:

First line:

1. ε_N, mean value of the nucleation-strain normal distribution.
2. s_N, standard deviation of the nucleation-strain normal distribution.
3. f_N, volume fraction of nucleating voids.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of ε_N, s_N, and f_N on temperature and other predefined field variables.

*VOLUMETRIC TEST DATA: Provide volumetric test data.

This option can be used only in conjunction with the *HYPERELASTIC option, the *HYPERFOAM option, or the *VISCOELASTIC option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *Hyperelastic Behavior of Rubberlike Materials*
- *Hyperelastic Behavior in Elastomeric Foams*
- *Time Domain Viscoelasticity*
- *HYPERELASTIC
- *HYPERFOAM
- *VISCOELASTIC

Hyperelastic material model

For hyperelastic materials, you can use the *VOLUMETRIC TEST DATA option to provide volumetric loading test data that includes user-defined material compressibility. Alternatively, you can include compressibility by:

- using the POISSON parameter on the *HYPERELASTIC option, or
- in the case of the Marlow and Valanis-Landel models, by specifying the nominal lateral strain under the *UNIAXIAL TEST DATA option.

If none of these options is used to specify volumetric behavior, Abaqus/Standard assumes that the material is incompressible, while Abaqus/Explicit selects a default value for the compressibility.

See the discussion in *Compressibility* before using this option in Abaqus/Explicit.

Optional parameters:

SMOOTH

Include this parameter to apply a smoothing filter to the stress-strain data. If the parameter is omitted, no smoothing is performed.

Set this parameter equal to the number n such that $2n + 1$ is equal to the total number of data points in the moving window through which a cubic polynomial is fit using the least-squares method. n should be larger than 1. The default is SMOOTH=3.

Optional parameter when the *VOLUMETRIC TEST DATA option is used in conjunction with the *HYPERELASTIC, MARLOW option or the *HYPERELASTIC, VALANIS-LANDEL option:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the test data. If this parameter is omitted, it is assumed that the test data depend only on temperature.

See "Using the DEPENDENCIES parameter to define field variable dependence" in *Material Data Definition* for more information.

Data lines to specify volumetric test data for hyperelasticity other than the Marlow and the Valanis-Landel models (the volume ratios must be arranged in either ascending or descending order if the SMOOTH parameter is used):

First line:

1. Pressure, p.
2. Volume ratio, J (current volume/original volume).

Repeat this data line as often as necessary.

Data lines to specify volumetric test data for the Marlow and the Valanis-Landel models (the volume ratios must be arranged in descending order if the SMOOTH parameter is used):

First line:

1. Pressure, p.
2. Volume ratio, J.
3. Temperature, θ.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the test data as a function of temperature and other predefined field variables. The volume ratios must be given in descending order.

Hyperfoam material model

There are no parameters associated with this option.

Data lines to specify volumetric test data for a hyperfoam:

First line:

1. Pressure, p.
2. Volume ratio, J (current volume/original volume).

Repeat this data line as often as necessary.

Viscoelastic material model

Optional parameters:

VOLINF

To specify creep test data, set this parameter equal to the value of the long-term, normalized volumetric compliance, $j_K(\infty)$.

To specify relaxation test data, set this parameter equal to the value of the long-term, normalized volumetric modulus $k_R(\infty)$.

The volumetric compliance is related to the volumetric modulus by $j_K(\infty) = 1/k_R(\infty)$. The fitting procedure will use this value in the constraint $1 - \sum_{i=1}^{N} \bar{k}_i^P = k_R(\infty)$.

Data lines to specify volumetric creep test data for a viscoelastic material:
First line:

1. Normalized volumetric (bulk) compliance $j_K(t), \quad (j_K(t) \geq 1)$.
2. Time t, $\quad (t > 0)$.

Repeat this data line as often as necessary to give the compliance-time data.

Data lines to specify volumetric relaxation test data for a viscoelastic material:
First line:

1. Normalized volumetric (bulk) modulus $k_R(t), \quad (0 \leq k_R(t) \leq 1)$.
2. Time t, $\quad (t > 0)$.

Repeat this data line as often as necessary to give the modulus-time data.

W, X, Y, Z

This section describes each Abaqus input option that begins with W, X, Y, or Z, including the intended use, Abaqus product support, whether it is model or history data, and the relevant parameters and data lines.

In this section:

- **WAVE*
- **WEAR SURFACE PROPERTIES*
- **WIND*

*WAVE: Define gravity waves for use in immersed structure calculations.

This option is used to define gravity waves for use in applying loads.

Products: Abaqus/Aqua

Type: Model data

Level: Model

References:

- *Abaqus/Aqua Analysis*
- *VWAVE*
- *UWAVE*

Optional parameters:

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See *Input Syntax Rules* for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

TYPE

Set TYPE=STOKES (default) to use Stokes fifth-order wave theory.

Set TYPE=AIRY to use Airy (linearized) wave theory.

Set TYPE=GRIDDED (Abaqus/Standard only) to use gridded data to define the fluid particle velocities, accelerations, free surface elevation, and dynamic pressure.

Set TYPE=USER to allow user-defined waves and update fluid variables like velocity, acceleration, free surface elevation, pressure, and pressure gradients.

Optional parameter for TYPE=AIRY:

WAVE PERIOD

Include this parameter to indicate that the second field in the data line specifies the wave period, τ_N. If this parameter is omitted, the second field in the data line specifies the wavelength, λ_N.

Required parameter for TYPE=GRIDDED:

DATA FILE

Set this parameter equal to the name of the file containing the gridded data. The file must be a sequential, binary-format file containing records in the format described in *Abaqus/Aqua Analysis*.

Optional parameters for TYPE=GRIDDED:

MINIMUM

Set this parameter equal to the elevation below which point the structure is fully immersed for all time *t*. If this parameter is omitted, the elevation of the structure is compared against the instantaneous free surface to check for fluid surface penetration.

QUADRATIC

Include this parameter to indicate that quadratic interpolation of the wave data is used to determine information between grid points. If this parameter is omitted, linear interpolation is used.

Optional parameter for TYPE=USER:

STOCHASTIC

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to make the intermediate configuration available to user subroutine *UWAVE*. Set this parameter equal to a random number seed for use in stochastic analysis. If this parameter is omitted or is included without a value, a default value of 0.0 is used for the random number seed. This value is passed into user subroutine *UWAVE*. It is not used otherwise by Abaqus/Aqua.

PROPERTIES

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the number of constant properties required for the user-defined wave. The default value is zero. This value is passed into user subroutine *VWAVE* as the integer argument `NPROPS`, whereas the property values are passed in as the real array `PROPS`.

DEPVAR

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the number of state variables required for user-defined waves. The default value is zero. This value is passed into user subroutine *VWAVE* as the integer argument `NSTATEVAR`. The state variables are stored at the nodes of the elements on which the Abaqus/Aqua loads are applied and passed into user subroutine *VWAVE* as the real array argument `STATEVAR`. You must update the state variables in the user subroutine. They are initialized to zero at the beginning of each step.

Data line to define Stokes fifth-order waves (TYPE=STOKES):

First (and only) line:

1. Wave height, H.
2. Period of wave, τ.
3. Phase angle, ϕ, in degrees.
4. Direction cosines giving the direction of travel of the wave, d. In three-dimensional cases both x- and y-components are needed; in two-dimensional cases only the x-component is used (in that case +1 means the wave travels in the direction of x increasing, and −1 means a component traveling in the direction of x decreasing).

Data lines to define Airy waves (TYPE=AIRY):

First line:

1. Wave component amplitude, a_N.
2. Wavelength of this component, λ_N (default), or wave period of this component, τ_N (if the parameter WAVE PERIOD is used).
3. Phase angle of this component, ϕ_N, in degrees.
4. x-direction cosine defining the direction of the vector d_N (the direction of travel for this wave component). In three-dimensional cases both x- and y-components are needed; in two-dimensional cases only the x-component is used (in that case +1.0 means the wave travels in the direction of x increasing, and −1.0 means a component traveling in the direction of x decreasing).

5. y-direction cosine defining the direction of the vector $\boldsymbol{d_N}$ (the direction of travel for this wave component). This component is not needed in two-dimensional cases.

Repeat this data line as often as necessary to define multiple wave trains; one line per wave component.

Data line to define gridded wave data (TYPE=GRIDDED):

First (and only) line:

1. Global x-coordinate of the origin of the wave data grid.
2. Global y-coordinate of the origin of the wave data grid.
3. Global z-coordinate of the origin of the wave data grid.
4. Direction cosine between the local x-axis of the wave data grid and the global x-axis. (Plus or minus one for two-dimensional analysis.)
5. Direction cosine between the local x-axis of the wave data grid and the global y-axis. (Zero for two-dimensional analysis.)

Data lines to define frequency versus wave amplitude data for stochastic user wave theory (TYPE=USER) in an Abaqus/Standard analysis:

First line:

1. Frequency (in cycles per time, for stochastic analysis only).
2. Wave amplitude.

Repeat this data line as often as necessary to define the wave spectrum. These data pairs are passed into user subroutine *UWAVE*. They are not used otherwise by Abaqus/Aqua.

Data lines to define wave theory (TYPE=USER) in an Abaqus/Explicit analysis:

First line:

1. Enter the properties required for the user-defined wave kinematics, separated by commas. These values will be passed into *VWAVE* as the real-array argument `PROPS`.

Repeat this data line as often as necessary to include all properties, with a maximum of eight values per line.

*WEAR SURFACE PROPERTIES: Specify wear properties to activate wear distance accumulation at contact surface nodes.

This option is used to specify wear properties to activate wear distance accumulation at contact surface nodes. It must appear immediately after the **SURFACE PROPERTY* option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Assembly

References:

- *Contact Wear*
- **SURFACE PROPERTY*
- **SURFACE PROPERTY ASSIGNMENT*

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the wear coefficient in addition to wear distance, contact pressure, and temperature. If this parameter is omitted, it is assumed that the wear coefficients have no dependencies or depend only on wear distance, contact pressure, and temperature. See *Material Data Definition*

FRIC COEF DEPENDENT

Set this parameter equal to YES if the Archard's wear equation depends explicitly on the friction coefficient.

Set this parameter equal to NO (default) if the Archard's wear equation does not depend on the friction coefficient.

REFERENCE STRESS

This parameter is required if UNITLESS WEAR COEF=YES.

Set this parameter equal to the magnitude of hardness of the material forming the wear surface.

UNITLESS WEAR COEF

Set this parameter equal to YES if the wear coefficient is dimensionless.

Set this parameter equal to NO (default) if the wear coefficient has dimensions of inverse of stress.

Data lines to define the wear coefficient:

First line:

1. Wear coefficient, φ_{wear} (or alternatives $\tilde{\varphi}_{wear}$, $\hat{\varphi}_{wear}$, or, $\overline{\varphi}_{wear}$ depending on the specific form of the Archard's wear equation).
2. Surface wear distance, W. If this value is omitted, the wear coefficient is assumed to be independent of the wear distance.
3. Contact pressure, p . If this value is omitted, the wear coefficient is assumed to be independent of the contact pressure.
4. Surface temperature, T . If this value is omitted, the wear coefficient is assumed to be independent of the surface temperature.

5. Surface field variable, f^1.
6. Surface field variable, f^2.
7. Etc., up to four field variables

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four).:

1. Surface field variable, f^5.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the wear coefficient as a function of the surface wear distance, contact pressure, surface temperature, and predefined surface field variables.

*WIND: Define wind velocity profile for wind loading.

This option is used to define a wind velocity profile for use in applying loads.

Products: Abaqus/Aqua

Type: Model data

Level: Model

References:

• *Abaqus/Aqua Analysis*

There are no parameters associated with this option.

Data line to define the wind velocity profile:

First (and only) line:

1. Air density, ρ_a.
2. Reference height for wind profile, z_0.
3. Local x-direction velocity constant, c_x (default value 1.0).
4. Local y-direction velocity constant, c_y (default value 1.0).
5. x-direction cosine defining the direction of the unit vector $\hat{\mathbf{i}}$ (the wind local x-axis) with respect to the global coordinate system. In three-dimensional cases both x- and y-components are needed; in two-dimensional cases only the x-component is used.
6. y-direction cosine defining the direction of the unit vector $\hat{\mathbf{i}}$ (the wind local x-axis) with respect to the global coordinate system. This component is not needed in two-dimensional cases.
7. α (default value 1/7).

Product Index

This index lists the sections in this guide that apply to a particular Abaqus product.

In this section:

- *Abaqus/Standard*
- *Abaqus/Explicit*
- *Abaqus/CAE*
- *Abaqus/AMS*
- *Abaqus/Aqua*
- *Abaqus/Design*

Abaqus/Standard

This section lists the topics in this guide that apply to Abaqus/Standard.

A

ACOUSTIC CONTRIBUTION
ACOUSTIC FLOW VELOCITY
ACOUSTIC MEDIUM
ACOUSTIC WAVE FORMULATION
ACTIVATE ELEMENTS
ADAPTIVE MESH
ADAPTIVE MESH CONSTRAINT
ADAPTIVE MESH CONTROLS
ADJUST
ALLOWABLE STRESS
AMPLITUDE
ANISOTROPIC HYPERELASTIC
ANNEAL TEMPERATURE
ASSEMBLY
ASYMMETRIC-AXISYMMETRIC
AXIAL

B

BASE MOTION
BASELINE CORRECTION
BEAM ADDED INERTIA
BEAM FLUID INERTIA
BEAM GENERAL SECTION
BEAM SECTION
BEAM SECTION GENERATE
BEAM SECTION OFFSET
BIAXIAL TEST DATA
BOUNDARY
BUCKLE
BUCKLING ENVELOPE
BUCKLING LENGTH

BUCKLING REDUCTION FACTORS

C

CAP CREEP
CAP HARDENING
CAP PLASTICITY
CAST IRON COMPRESSION HARDENING
CAST IRON PLASTICITY
CAST IRON TENSION HARDENING
CAVITY DEFINITION
CECHARGE
CECURRENT
CENTROID
CFILM
CFLOW
CFLUX
CHANGE FRICTION
CLAY HARDENING
CLAY PLASTICITY
CLEARANCE
CLOAD
COHESIVE BEHAVIOR
COHESIVE SECTION
COMBINED TEST DATA
COMPLEX FREQUENCY
COMPOSITE MODAL DAMPING
CONCENTRATION TENSOR
CONCRETE
CONCRETE COMPRESSION DAMAGE
CONCRETE COMPRESSION HARDENING
CONCRETE DAMAGED PLASTICITY
CONCRETE TENSION DAMAGE
CONCRETE TENSION STIFFENING
CONDUCTIVITY
CONNECTOR BEHAVIOR
CONNECTOR CONSTITUTIVE REFERENCE

CONNECTOR DAMAGE EVOLUTION
CONNECTOR DAMAGE INITIATION
CONNECTOR DAMPING
CONNECTOR DERIVED COMPONENT
CONNECTOR ELASTICITY
CONNECTOR FAILURE
CONNECTOR FRICTION
CONNECTOR HARDENING
CONNECTOR LOAD
CONNECTOR LOCK
CONNECTOR MOTION
CONNECTOR PLASTICITY
CONNECTOR POTENTIAL
CONNECTOR SECTION
CONNECTOR STOP
CONSTITUENT
CONSTRAINT CONTROLS
CONTACT
CONTACT CONTROLS
CONTACT CONTROLS ASSIGNMENT
CONTACT DAMPING
CONTACT EXCLUSIONS
CONTACT FILE
CONTACT FORMULATION
CONTACT INCLUSIONS
CONTACT INITIALIZATION ASSIGNMENT
CONTACT INITIALIZATION DATA
CONTACT INTERFERENCE
CONTACT OUTPUT
CONTACT PAIR
CONTACT PERMEABILITY
CONTACT PRINT
CONTACT PROPERTY ASSIGNMENT
CONTACT STABILIZATION
CONTOUR INTEGRAL
CONTROLS

CORRELATION
CO-SIMULATION
CO-SIMULATION REGION
COUPLED CONSTITUTIVE RESPONSE
COUPLED TEMPERATURE-DISPLACEMENT
COUPLED THERMAL-ELECTRICAL
COUPLED THERMAL-ELECTROCHEMICAL
COUPLING
CRADIATE
CREEP
CREEP STRAIN RATE CONTROL
CRUSHABLE FOAM
CRUSHABLE FOAM HARDENING
CURE GLASS TRANSITION TEMPERATURE
CURE HEAT GENERATION
CURE KINETICS
CURE MAX CONVERSION
CURE SHRINKAGE
CYCLED PLASTIC
CYCLIC
CYCLIC HARDENING
CYCLIC SYMMETRY MODEL

D

D EM POTENTIAL
DAMAGE EVOLUTION
DAMAGE INITIATION
DAMAGE STABILIZATION
DAMPING
DAMPING CONTROLS
DASHPOT
DEBOND
DECHARGE
DECURRENT
DEFORMATION PLASTICITY
DENSITY

DEPVAR
DESIGN RESPONSE
DETONATION POINT
DFLOW
DFLUX
DIAGNOSTICS
DIELECTRIC
DIFFUSIVITY
DIRECT CYCLIC
DISPLAY BODY
DISTRIBUTING
DISTRIBUTING COUPLING
DISTRIBUTION
DISTRIBUTION TABLE
DLOAD
DRAG CHAIN
DRUCKER PRAGER
DRUCKER PRAGER CREEP
DRUCKER PRAGER HARDENING
DSECHARGE
DSECURRENT
DSFLOW
DSFLUX
DSLOAD
DYNAMIC

E

EIGENSTRAIN
EL FILE
EL PRINT
ELASTIC
ELCOPY
ELECTRIC MACHINE LOAD
ELECTRIC MACHINE PROPERTY
ELECTRICAL CONDUCTIVITY
ELECTRICAL RESISTIVITY

ELECTROMAGNETIC
ELEMENT
ELEMENT MATRIX OUTPUT
ELEMENT OPERATOR OUTPUT
ELEMENT OUTPUT
ELEMENT PROGRESSIVE ACTIVATION
ELEMENT RECOVERY MATRIX
ELEMENT RESPONSE
ELEMENT SOLUTION-DEPENDENT VARIABLES
ELEMENT USER OUTPUT VARIABLES
ELGEN
ELSET
EMBEDDED ELEMENT
EMISSIVITY
END ASSEMBLY
END INSTANCE
END LOAD CASE
END PART
END STEP
ENERGY FILE
ENERGY OUTPUT
ENERGY PRINT
ENRICHMENT
ENRICHMENT ACTIVATION
EOS COMPACTION
EPJOINT
EQUATION
EQUIVALENT RADIATED SURFACE PROPERTIES
EVENT SERIES
EVENT SERIES TYPE
EXPANSION
EXTERNAL FIELD

F

FAIL STRAIN
FAIL STRESS

FAILURE RATIOS
FASTENER
FASTENER PROPERTY
FATIGUE
FIBER DISPERSION
FIELD
FIELD IMPORT
FIELD MAPPER CONTROLS
FIELD OPERATIONS
FILE FORMAT
FILM
FILM PROPERTY
FLEXIBLE BODY
FLOW
FLUID BEHAVIOR
FLUID BULK MODULUS
FLUID CAVITY
FLUID DENSITY
FLUID EXCHANGE
FLUID EXCHANGE PROPERTY
FLUID EXPANSION
FLUID FLUX
FLUID ID FIELD
FLUID LEAKOFF
FLUID PIPE CONNECTOR LOSS
FLUID PIPE CONNECTOR SECTION
FLUID PIPE CONNECTOR THERMAL LOSS
FLUID PIPE CONNECTOR WALL
FLUID PIPE FLOW LOSS
FLUID PIPE SECTION
FLUID PIPE THERMAL
FLUID PIPE WALL
FOUNDATION
FRACTURE CRITERION
FRAME SECTION
FREQUENCY

FRICTION

G

GAP

GAP CONDUCTANCE

GAP CONVECTION

GAP DIFFUSIVITY

GAP ELECTRICAL CONDUCTANCE

GAP FLOW

GAP HEAT GENERATION

GAP RADIATION

GASKET BEHAVIOR

GASKET CONTACT AREA

GASKET ELASTICITY

GASKET SECTION

GASKET THICKNESS BEHAVIOR

GEL

GEOSTATIC

GLOBAL DAMPING

GLOBAL RESPONSE

H

HEADING

HEAT GENERATION

HEAT TRANSFER

HEATCAP

HOURGLASS STIFFNESS

HYPERELASTIC

HYPERFOAM

HYPOELASTIC

HYSTERESIS

I

IMPEDANCE

IMPEDANCE PROPERTY

IMPERFECTION

IMPORT

IMPORT CONTROLS
IMPORT ELSET
IMPORT NSET
IMPORT SURFACE
INCIDENT WAVE
INCIDENT WAVE FLUID PROPERTY
INCIDENT WAVE INTERACTION
INCIDENT WAVE INTERACTION PROPERTY
INCIDENT WAVE PROPERTY
INCIDENT WAVE REFLECTION
INCLUDE
INCREMENTATION OUTPUT
INELASTIC HEAT FRACTION
INERTIA RELIEF
INITIAL CONDITIONS
INSTANCE
INTEGRATED OUTPUT
INTEGRATED OUTPUT SECTION
INTERFACE
INTERFACE REACTION
INTERNAL FLUX GENERATION
ISOTROPIZATION PARAMETERS
ITS

J

JOINT
JOINT ELASTICITY
JOINT PLASTICITY
JOINTED MATERIAL
JOULE HEAT FRACTION

K

KAPPA
KINEMATIC
KINEMATIC COUPLING

L

LATENT HEAT
LOAD CASE
LOW DENSITY FOAM

M

M1
M2
MAGNETIC PERMEABILITY
MAGNETOSTATIC
MANIFEST
MAP SOLUTION
MASS
MASS DIFFUSION
MASS FLOW RATE
MATERIAL
MATRIX
MATRIX ASSEMBLE
MATRIX CHECK
MATRIX EXPORT
MATRIX GENERATE
MATRIX INPUT
MATRIX OUTPUT
MEAN FIELD DAMAGE
MEAN FIELD HOMOGENIZATION
MEMBRANE SECTION
MODAL DAMPING
MODAL DYNAMIC
MODAL FILE
MODAL OUTPUT
MODAL PRINT
MODEL CHANGE
MOHR COULOMB
MOHR COULOMB HARDENING
MOISTURE SWELLING
MOLECULAR WEIGHT

MONITOR

MOTION

MPC

MULLINS EFFECT

MULTIPHYSICS LOAD

N

NCOPY

NETWORK STIFFNESS RATIO

NFILL

NGEN

NMAP

NO COMPRESSION

NO TENSION

NODAL ENERGY RATE

NODAL THICKNESS

NODE

NODE FILE

NODE OUTPUT

NODE PRINT

NONLINEAR BH

NONSTRUCTURAL MASS

NORMAL

NSET

O

ONE-STEP INVERSE

OPERATOR OUTPUT

ORIENTATION

ORNL

OUTPUT

P, Q

PAPERBOARD HARDENING

PAPERBOARD PLASTICITY

PAPERBOARD THICKNESS COMPRESSION ELASTIC

PAPERBOARD THICKNESS COMPRESSION PLASTICITY

PAPERBOARD TRANSVERSE SHEAR PLASTICITY
PARAMETER
PARAMETER DEPENDENCE
PARAMETER SHAPE VARIATION
PARAMETER TABLE
PARAMETER TABLE TYPE
PART
PERFECTLY MATCHED LAYER
PERIODIC
PERMANENT MAGNETIZATION
PERMEABILITY
PHYSICAL CONSTANTS
PIEZOELECTRIC
PIEZOELECTRIC DAMPING
PIEZORESISTIVITY
PIPE-SOIL INTERACTION
PIPE-SOIL STIFFNESS
PLANAR TEST DATA
PLASTIC
PLASTIC AXIAL
PLASTIC M1
PLASTIC M2
PLASTIC TORQUE
PLASTICITY CORRECTION
PML COEFFICIENT
PORE FLUID PRESSURE
POROUS BULK MODULI
POROUS ELASTIC
POROUS ELECTRODE THEORY
POROUS METAL PLASTICITY
POST OUTPUT
POTENTIAL
PREPRINT
PRESSURE PENETRATION
PRESSURE STRESS
PRESTRESS HOLD

PRE-TENSION SECTION

PRINT

PROPERTY TABLE

PROPERTY TABLE TYPE

PSD-DEFINITION

R

RADIATE

RADIATION FILE

RADIATION OUTPUT

RADIATION PRINT

RADIATION SYMMETRY

RADIATION VIEW FACTOR

RANDOM RESPONSE

RATE DEPENDENT

RATIOS

REBAR

REBAR LAYER

REBAR LINE

REDUCED BASIS GENERATE

REFLECTION

RELEASE

RESPONSE SPECTRUM

RESTART

RETAINED NODAL DOFS

RIGID BODY

RIGID SURFACE

ROTARY INERTIA

S

SCALE MASS

SCALE STIFFNESS

SCALE STRESS DESIGN

SCALE THERMAL CONDUCTIVITY

SECTION CONTROLS

SECTION CURVATURE

SECTION FILE

SECTION INERTIA
SECTION ORIGIN
SECTION POINTS
SECTION PRINT
SECTION STIFFNESS
SECTION TANGENT
SELECT CYCLIC SYMMETRY MODES
SELECT EIGENMODES
SFILM
SFLOW
SHEAR CENTER
SHEAR RETENTION
SHEAR TEST DATA
SHELL GENERAL SECTION
SHELL SECTION
SHELL TO SOLID COUPLING
SIMPEDANCE
SIMPLE SHEAR TEST DATA
SLIDE LINE
SLOAD
SOFT ROCK HARDENING
SOFT ROCK PLASTICITY
SOFTENING REGULARIZATION
SOILS
SOLID ELECTROLYTE THEORY
SOLID SECTION
SOLUBILITY
SOLUTION TECHNIQUE
SOLVER CONTROLS
SORPTION
SPECIFIC HEAT
SPECTRUM
SPRING
SRADIATE
STATIC
STEADY STATE DYNAMICS

STEADY STATE TRANSPORT
STEP
STEP CONTROL
STEP CYCLING
STEP CYCLING CONTROL
SUBMODEL
SUBMODEL CONDITIONS
SUBMODEL CUT
SUBSTRUCTURE CHANGE
SUBSTRUCTURE DAMPING
SUBSTRUCTURE DAMPING CONTROLS
SUBSTRUCTURE GENERATE
SUBSTRUCTURE LOAD CASE
SUBSTRUCTURE MATRIX OUTPUT
SUBSTRUCTURE MODAL DAMPING
SUBSTRUCTURE OUTPUT
SUBSTRUCTURE PATH
SUBSTRUCTURE PROPERTY
SUPERELASTIC
SUPERELASTIC HARDENING
SUPERELASTIC HARDENING MODIFICATIONS
SURFACE
SURFACE BEHAVIOR
SURFACE FLAW
SURFACE INTERACTION
SURFACE PROPERTY
SURFACE PROPERTY ASSIGNMENT
SURFACE SECTION
SURFACE SMOOTHING
SWELLING
SYMMETRIC MODEL GENERATION
SYMMETRIC RESULTS TRANSFER
SYSTEM

T

TABLE COLLECTION

TEMPERATURE
TENSION CUTOFF
TENSION STIFFENING
THERMAL EXPANSION
TIE
TIME POINTS
TORQUE
TORQUE PRINT
TRANSFORM
TRANSPORT VELOCITY
TRANSVERSE SHEAR
TRANSVERSE SHEAR STIFFNESS
TRIAXIAL TEST DATA
TRS

U

UEL PROPERTY
UNDEX CHARGE PROPERTY
UNIAXIAL TEST DATA
UNIFORM
UNIT SYSTEM
USER DEFINED FIELD
USER ELEMENT
USER MATERIAL
USER OUTPUT VARIABLES

V

VIEW FACTOR OUTPUT
VISCO
VISCOELASTIC
VISCOSITY
VISCOUS
VOID NUCLEATION
VOLUMETRIC TEST DATA

W, X, Y, Z

WEAR SURFACE PROPERTIES

Abaqus/Explicit

This section lists the topics in this guide that apply to Abaqus/Explicit.

A

ACOUSTIC MEDIUM

ACOUSTIC WAVE FORMULATION

ADAPTIVE MESH

ADAPTIVE MESH CONSTRAINT

ADAPTIVE MESH CONTROLS

ADAPTIVE MESH REFINEMENT

ADJUST

AMPLITUDE

ANISOTROPIC HYPERELASTIC

ANNEAL

ANNEAL TEMPERATURE

ASSEMBLY

AXIAL

B

BEAM ADDED INERTIA

BEAM FLUID INERTIA

BEAM GENERAL SECTION

BEAM SECTION

BEAM SECTION OFFSET

BIAXIAL TEST DATA

BLOCKAGE

BOUNDARY

BRITTLE CRACKING

BRITTLE FAILURE

BRITTLE SHEAR

BULK VISCOSITY

C

CAP HARDENING

CAP PLASTICITY

CAPACITY

CAST IRON COMPRESSION HARDENING
CAST IRON PLASTICITY
CAST IRON TENSION HARDENING
CENTROID
CFILM
CFLUX
CHARACTERISTIC LENGTH
CLAY HARDENING
CLAY PLASTICITY
CLEARANCE
CLOAD
CLUSTER MASS INERTIA TABLE
COHESIVE BEHAVIOR
COHESIVE SECTION
COMBINATORIAL RULE
COMBINED TEST DATA
CONCRETE COMPRESSION DAMAGE
CONCRETE COMPRESSION HARDENING
CONCRETE DAMAGED PLASTICITY
CONCRETE FAILURE
CONCRETE TENSION DAMAGE
CONCRETE TENSION STIFFENING
CONDUCTIVITY
CONNECTOR BEHAVIOR
CONNECTOR CONSTITUTIVE REFERENCE
CONNECTOR DAMAGE EVOLUTION
CONNECTOR DAMAGE INITIATION
CONNECTOR DAMPING
CONNECTOR DERIVED COMPONENT
CONNECTOR ELASTICITY
CONNECTOR FAILURE
CONNECTOR FRICTION
CONNECTOR HARDENING
CONNECTOR LOAD
CONNECTOR LOCK
CONNECTOR MOTION

CONNECTOR PLASTICITY
CONNECTOR POTENTIAL
CONNECTOR SECTION
CONNECTOR STOP
CONNECTOR UNIAXIAL BEHAVIOR
CONSTITUENT
CONTACT
CONTACT CLEARANCE
CONTACT CLEARANCE ASSIGNMENT
CONTACT CONTROLS
CONTACT CONTROLS ASSIGNMENT
CONTACT DAMPING
CONTACT EXCLUSIONS
CONTACT FORMULATION
CONTACT INCLUSIONS
CONTACT INITIALIZATION ASSIGNMENT
CONTACT INITIALIZATION DATA
CONTACT MASS SCALING
CONTACT OUTPUT
CONTACT PAIR
CONTACT PROPERTY ASSIGNMENT
CONWEP CHARGE PROPERTY
CO-SIMULATION
CO-SIMULATION REGION
COUPLING
CRADIATE
CRUSH STRESS
CRUSH STRESS VELOCITY FACTOR
CRUSHABLE FOAM
CRUSHABLE FOAM HARDENING
CYCLIC HARDENING
CYCLIC SYMMETRY MODEL

D

DAMAGE EVOLUTION
DAMAGE INITIATION

DAMAGE STABILIZATION
DAMPING
DASHPOT
DENSITY
DEPVAR
DETONATION POINT
DFLUX
DIAGNOSTICS
DISCRETE ELASTICITY
DISCRETE SECTION
DISPLAY BODY
DISTRIBUTING
DISTRIBUTION
DISTRIBUTION TABLE
DLOAD
DOMAIN DECOMPOSITION
DRUCKER PRAGER
DRUCKER PRAGER HARDENING
DSFLUX
DSLOAD
DYNAMIC
DYNAMIC TEMPERATURE-DISPLACEMENT

E

EL FILE
ELASTIC
ELCOPY
ELEMENT
ELEMENT OUTPUT
ELGEN
ELSET
EMBEDDED ELEMENT
END ASSEMBLY
END INSTANCE
END PART
END STEP

ENERGY FILE
ENERGY OUTPUT
EOS
EOS COMPACTION
EQUATION
EULERIAN BOUNDARY
EULERIAN MESH MOTION
EULERIAN SECTION
EXPANSION
EXTERNAL FIELD
EXTREME ELEMENT VALUE
EXTREME NODE VALUE
EXTREME VALUE

F

FABRIC
FAIL STRAIN
FAIL STRESS
FASTENER
FASTENER PROPERTY
FIBER DISPERSION
FIELD
FIELD IMPORT
FIELD MAPPER CONTROLS
FIELD OPERATIONS
FILE OUTPUT
FILM
FILM PROPERTY
FILTER
FIXED MASS SCALING
FLUID BEHAVIOR
FLUID BULK MODULUS
FLUID CAVITY
FLUID DENSITY
FLUID EXCHANGE
FLUID EXCHANGE ACTIVATION

FLUID EXCHANGE PROPERTY
FLUID EXPANSION
FLUID FLUX
FLUID INFLATOR
FLUID INFLATOR ACTIVATION
FLUID INFLATOR MIXTURE
FLUID INFLATOR PROPERTY
FRACTURE CRITERION
FRICTION

G

GAP CONDUCTANCE
GAP HEAT GENERATION
GAP RADIATION
GAS SPECIFIC HEAT

H

HEADING
HEAT GENERATION
HEATCAP
HYPERELASTIC
HYPERFOAM

I

IMPEDANCE
IMPEDANCE PROPERTY
IMPERFECTION
IMPORT
IMPORT CONTROLS
IMPORT ELSET
IMPORT NSET
IMPORT SURFACE
INCIDENT WAVE
INCIDENT WAVE FLUID PROPERTY
INCIDENT WAVE INTERACTION
INCIDENT WAVE INTERACTION PROPERTY
INCIDENT WAVE PROPERTY

INCIDENT WAVE REFLECTION
INCLUDE
INCREMENTATION OUTPUT
INELASTIC HEAT FRACTION
INITIAL CONDITIONS
INSTANCE
INTEGRATED OUTPUT
INTEGRATED OUTPUT SECTION

K

KINEMATIC

L

LATENT HEAT
LOADING DATA
LOW DENSITY FOAM

M

M1
M2
MASS
MASS ADJUST
MATERIAL
MEAN FIELD DAMAGE
MEAN FIELD HOMOGENIZATION
MEDIA TRANSPORT
MEMBRANE SECTION
MOHR COULOMB
MOHR COULOMB HARDENING
MOLECULAR WEIGHT
MONITOR
MPC
MULLINS EFFECT

N

NCOPY
NETWORK STIFFNESS RATIO
NFILL

NGEN
NMAP
NO COMPRESSION
NO TENSION
NODAL ENERGY RATE
NODAL THICKNESS
NODE
NODE FILE
NODE OUTPUT
NONSTRUCTURAL MASS
NORMAL
NSET

O

ORIENTATION
OUTPUT

P, Q

PAPERBOARD HARDENING
PAPERBOARD PLASTICITY
PAPERBOARD THICKNESS COMPRESSION ELASTIC
PAPERBOARD THICKNESS COMPRESSION PLASTICITY
PAPERBOARD TRANSVERSE SHEAR PLASTICITY
PARAMETER
PARAMETER DEPENDENCE
PARAMETER SHAPE VARIATION
PARAMETER TABLE
PART
PARTICLE GENERATOR
PARTICLE GENERATOR FLOW
PARTICLE GENERATOR INLET
PARTICLE GENERATOR MIXTURE
PARTICLE OUTLET
PARTICLE OUTLET FLOW
PERIODIC MEDIA
PHYSICAL CONSTANTS
PLANAR TEST DATA

PLASTIC
PLY FABRIC FAILURE
PLY FABRIC HARDENING
PORE FLUID PRESSURE
POROUS BULK MODULI
POROUS FAILURE CRITERIA
POROUS METAL PLASTICITY
POTENTIAL
PREPRINT
PRINT
PROBABILITY DENSITY FUNCTION
PROPERTY TABLE

R

RADIATE
RATE DEPENDENT
REACTION RATE
REBAR
REBAR LAYER
REBAR LINE
RESTART
RIGID BODY
ROTARY INERTIA

S

SECTION CONTROLS
SECTION INERTIA
SECTION POINTS
SECTION STIFFNESS
SFILM
SHEAR CENTER
SHEAR FAILURE
SHEAR TEST DATA
SHELL GENERAL SECTION
SHELL SECTION
SHELL TO SOLID COUPLING
SIMPEDANCE

SIMPLE SHEAR TEST DATA
SOFT ROCK HARDENING
SOFT ROCK PLASTICITY
SOFTENING REGULARIZATION
SOLID SECTION
SPECIFIC HEAT
SPH SURFACE BEHAVIOR
SPRING
SRADIATE
STEADY STATE CRITERIA
STEADY STATE DETECTION
STEP
SUBCYCLING
SUBMODEL
SUPERELASTIC
SUPERELASTIC HARDENING
SUPERELASTIC HARDENING MODIFICATIONS
SURFACE
SURFACE BEHAVIOR
SURFACE INTERACTION
SURFACE PROPERTY
SURFACE PROPERTY ASSIGNMENT
SURFACE SECTION
SURFACE TENSION
SURFACE TENSION CONTACT ANGLE
SYSTEM

T

TABLE COLLECTION
TEMPERATURE
TENSILE FAILURE
TENSION CUTOFF
THERMAL EXPANSION
TIE
TIME POINTS
TORQUE

TRACER PARTICLE

TRANSFORM

TRANSVERSE SHEAR

TRANSVERSE SHEAR STIFFNESS

TRIAXIAL TEST DATA

TRS

U

UEL PROPERTY

UNDEX CHARGE PROPERTY

UNIAXIAL

UNIAXIAL TEST DATA

UNIFORM

UNIT SYSTEM

UNLOADING DATA

USER DEFINED FIELD

USER ELEMENT

USER MATERIAL

V

VARIABLE MASS SCALING

VISCOELASTIC

VISCOSITY

VOID NUCLEATION

VOLUMETRIC TEST DATA

W, X, Y, Z

WEAR SURFACE PROPERTIES

Abaqus/CAE

This section lists the topics in this guide that apply to Abaqus/CAE.

A

ACOUSTIC MEDIUM

ACOUSTIC WAVE FORMULATION

ADAPTIVE MESH

ADAPTIVE MESH CONSTRAINT

ADAPTIVE MESH CONTROLS

ADJUST

AMPLITUDE

ANISOTROPIC HYPERELASTIC

ANNEAL

ANNEAL TEMPERATURE

ASSEMBLY

B

BASELINE CORRECTION

BEAM FLUID INERTIA

BEAM GENERAL SECTION

BEAM SECTION

BEAM SECTION OFFSET

BIAXIAL TEST DATA

BOUNDARY

BRITTLE CRACKING

BRITTLE FAILURE

BRITTLE SHEAR

BUCKLE

BULK VISCOSITY

C

CAP CREEP

CAP HARDENING

CAP PLASTICITY

CAPACITY

CAST IRON COMPRESSION HARDENING

CAST IRON PLASTICITY
CAST IRON TENSION HARDENING
CAVITY DEFINITION
CECHARGE
CECURRENT
CENTROID
CFILM
CFLOW
CFLUX
CHANGE FRICTION
CLAY HARDENING
CLAY PLASTICITY
CLEARANCE
CLOAD
COHESIVE BEHAVIOR
COHESIVE SECTION
COMBINED TEST DATA
COMPLEX FREQUENCY
CONCRETE
CONCRETE COMPRESSION DAMAGE
CONCRETE COMPRESSION HARDENING
CONCRETE DAMAGED PLASTICITY
CONCRETE TENSION DAMAGE
CONCRETE TENSION STIFFENING
CONDUCTIVITY
CONNECTOR BEHAVIOR
CONNECTOR CONSTITUTIVE REFERENCE
CONNECTOR DAMAGE EVOLUTION
CONNECTOR DAMAGE INITIATION
CONNECTOR DAMPING
CONNECTOR DERIVED COMPONENT
CONNECTOR ELASTICITY
CONNECTOR FAILURE
CONNECTOR FRICTION
CONNECTOR HARDENING
CONNECTOR LOAD

CONNECTOR LOCK
CONNECTOR MOTION
CONNECTOR PLASTICITY
CONNECTOR POTENTIAL
CONNECTOR SECTION
CONNECTOR STOP
CONNECTOR UNIAXIAL BEHAVIOR
CONTACT
CONTACT CONTROLS
CONTACT DAMPING
CONTACT EXCLUSIONS
CONTACT FILE
CONTACT FORMULATION
CONTACT INCLUSIONS
CONTACT INITIALIZATION ASSIGNMENT
CONTACT INITIALIZATION DATA
CONTACT INTERFERENCE
CONTACT MASS SCALING
CONTACT OUTPUT
CONTACT PAIR
CONTACT PROPERTY ASSIGNMENT
CONTACT STABILIZATION
CONTOUR INTEGRAL
CONTROLS
CO-SIMULATION
CO-SIMULATION REGION
COUPLED TEMPERATURE-DISPLACEMENT
COUPLED THERMAL-ELECTRICAL
COUPLING
CRADIATE
CREEP
CRUSHABLE FOAM
CRUSHABLE FOAM HARDENING
CYCLED PLASTIC
CYCLIC
CYCLIC HARDENING

CYCLIC SYMMETRY MODEL

D

D EM POTENTIAL
DAMAGE EVOLUTION
DAMAGE INITIATION
DAMAGE STABILIZATION
DAMPING
DAMPING CONTROLS
DASHPOT
DEBOND
DECHARGE
DECURRENT
DEFORMATION PLASTICITY
DENSITY
DEPVAR
DETONATION POINT
DFLOW
DFLUX
DIELECTRIC
DIFFUSIVITY
DIRECT CYCLIC
DISPLAY BODY
DISTRIBUTING
DISTRIBUTING COUPLING
DISTRIBUTION
DISTRIBUTION TABLE
DLOAD
DRUCKER PRAGER
DRUCKER PRAGER CREEP
DRUCKER PRAGER HARDENING
DSECHARGE
DSECURRENT
DSFLOW
DSFLUX
DSLOAD

DYNAMIC

DYNAMIC TEMPERATURE-DISPLACEMENT

E

EL FILE

ELASTIC

ELCOPY

ELECTRICAL CONDUCTIVITY

ELECTROMAGNETIC

ELEMENT

ELEMENT OUTPUT

ELGEN

ELSET

EMBEDDED ELEMENT

EMISSIVITY

END ASSEMBLY

END INSTANCE

END LOAD CASE

END PART

END STEP

ENERGY FILE

ENERGY OUTPUT

ENRICHMENT

ENRICHMENT ACTIVATION

EOS

EOS COMPACTION

EQUATION

EULERIAN BOUNDARY

EULERIAN MESH MOTION

EULERIAN SECTION

EXPANSION

F

FAIL STRAIN

FAIL STRESS

FAILURE RATIOS

FASTENER

FASTENER PROPERTY
FIELD
FILE FORMAT
FILE OUTPUT
FILM
FILM PROPERTY
FILTER
FLUID BEHAVIOR
FLUID BULK MODULUS
FLUID CAVITY
FLUID DENSITY
FLUID EXCHANGE
FLUID EXCHANGE ACTIVATION
FLUID EXCHANGE PROPERTY
FLUID EXPANSION
FLUID INFLATOR
FLUID INFLATOR ACTIVATION
FLUID INFLATOR MIXTURE
FLUID INFLATOR PROPERTY
FLUID LEAKOFF
FOUNDATION
FRACTURE CRITERION
FREQUENCY
FRICTION

G

GAP
GAP CONDUCTANCE
GAP FLOW
GAP HEAT GENERATION
GAP RADIATION
GAS SPECIFIC HEAT
GASKET BEHAVIOR
GASKET CONTACT AREA
GASKET ELASTICITY
GASKET SECTION

GASKET THICKNESS BEHAVIOR
GEL
GEOSTATIC
GLOBAL DAMPING

H

HEADING
HEAT GENERATION
HEAT TRANSFER
HEATCAP
HOURGLASS STIFFNESS
HYPERELASTIC
HYPERFOAM
HYPOELASTIC
HYSTERESIS

I

IMPEDANCE PROPERTY
IMPERFECTION
IMPORT
INCIDENT WAVE
INCIDENT WAVE FLUID PROPERTY
INCIDENT WAVE INTERACTION
INCIDENT WAVE INTERACTION PROPERTY
INCIDENT WAVE PROPERTY
INCLUDE
INCREMENTATION OUTPUT
INELASTIC HEAT FRACTION
INERTIA RELIEF
INITIAL CONDITIONS
INSTANCE
INTEGRATED OUTPUT
INTEGRATED OUTPUT SECTION
INTERFACE
ITS

J

JOINT

JOULE HEAT FRACTION

K

KAPPA

KINEMATIC

KINEMATIC COUPLING

L

LATENT HEAT

LOAD CASE

LOW DENSITY FOAM

M

MAGNETIC PERMEABILITY

MASS

MASS DIFFUSION

MATERIAL

MEMBRANE SECTION

MODAL DAMPING

MODAL DYNAMIC

MODAL FILE

MODAL OUTPUT

MODEL CHANGE

MOHR COULOMB

MOHR COULOMB HARDENING

MOISTURE SWELLING

MOLECULAR WEIGHT

MONITOR

MPC

MULLINS EFFECT

N

NCOPY

NFILL

NGEN

NMAP

NO COMPRESSION
NO TENSION
NODAL THICKNESS
NODE
NODE FILE
NODE OUTPUT
NONSTRUCTURAL MASS
NSET

O

ORIENTATION
ORNL
OUTPUT

P, Q

PART
PERIODIC
PERMEABILITY
PHYSICAL CONSTANTS
PIEZOELECTRIC
PLANAR TEST DATA
PLASTIC
PLASTICITY CORRECTION
POROUS BULK MODULI
POROUS ELASTIC
POROUS FAILURE CRITERIA
POROUS METAL PLASTICITY
POTENTIAL
PREPRINT
PRESSURE PENETRATION
PRE-TENSION SECTION
PRINT

R

RADIATE
RADIATION FILE
RADIATION OUTPUT

RADIATION SYMMETRY
RADIATION VIEW FACTOR
RANDOM RESPONSE
RATE DEPENDENT
RATIOS
REACTION RATE
REBAR LAYER
REFLECTION
RESPONSE SPECTRUM
RESTART
RETAINED NODAL DOFS
RIGID BODY
RIGID SURFACE
ROTARY INERTIA

S

SECTION CONTROLS
SECTION POINTS
SELECT CYCLIC SYMMETRY MODES
SELECT EIGENMODES
SFILM
SHEAR CENTER
SHEAR RETENTION
SHEAR TEST DATA
SHELL GENERAL SECTION
SHELL SECTION
SHELL TO SOLID COUPLING
SIMPEDANCE
SIMPLE SHEAR TEST DATA
SLIDE LINE
SLOAD
SOFT ROCK HARDENING
SOFT ROCK PLASTICITY
SOFTENING REGULARIZATION
SOILS
SOLID SECTION

SOLUBILITY
SOLUTION TECHNIQUE
SOLVER CONTROLS
SORPTION
SPECIFIC HEAT
SPECTRUM
SPRING
SRADIATE
STATIC
STEADY STATE DYNAMICS
STEP
SUBMODEL
SUBSTRUCTURE GENERATE
SUBSTRUCTURE LOAD CASE
SUPERELASTIC
SUPERELASTIC HARDENING
SUPERELASTIC HARDENING MODIFICATIONS
SURFACE
SURFACE BEHAVIOR
SURFACE INTERACTION
SURFACE PROPERTY
SURFACE PROPERTY ASSIGNMENT
SURFACE SECTION
SURFACE SMOOTHING
SWELLING
SYSTEM

T

TEMPERATURE
TENSILE FAILURE
TENSION CUTOFF
TENSION STIFFENING
TIE
TIME POINTS
TRANSFORM
TRANSVERSE SHEAR STIFFNESS

TRIAXIAL TEST DATA

TRS

U

UEL PROPERTY

UNDEX CHARGE PROPERTY

UNIAXIAL TEST DATA

USER DEFINED FIELD

USER ELEMENT

USER MATERIAL

USER OUTPUT VARIABLES

V

VISCO

VISCOELASTIC

VISCOSITY

VISCOUS

VOID NUCLEATION

VOLUMETRIC TEST DATA

Abaqus/AMS

This section lists the topics in this guide that apply to Abaqus/AMS.

F

FREQUENCY

S

SUBSTRUCTURE DAMPING

Abaqus/Aqua

This section lists the topics in this guide that apply to Abaqus/Aqua.

A

AQUA

C

C ADDED MASS

CLOAD

D

D ADDED MASS

DLOAD

I

INITIAL CONDITIONS

S

SURFACE SECTION

W, X, Y, Z

WAVE

WIND

Abaqus/Design

This section lists the topics in this guide that apply to Abaqus/Design.

C

CONTACT RESPONSE

D

DESIGN GRADIENT

DESIGN PARAMETER

DESIGN RESPONSE

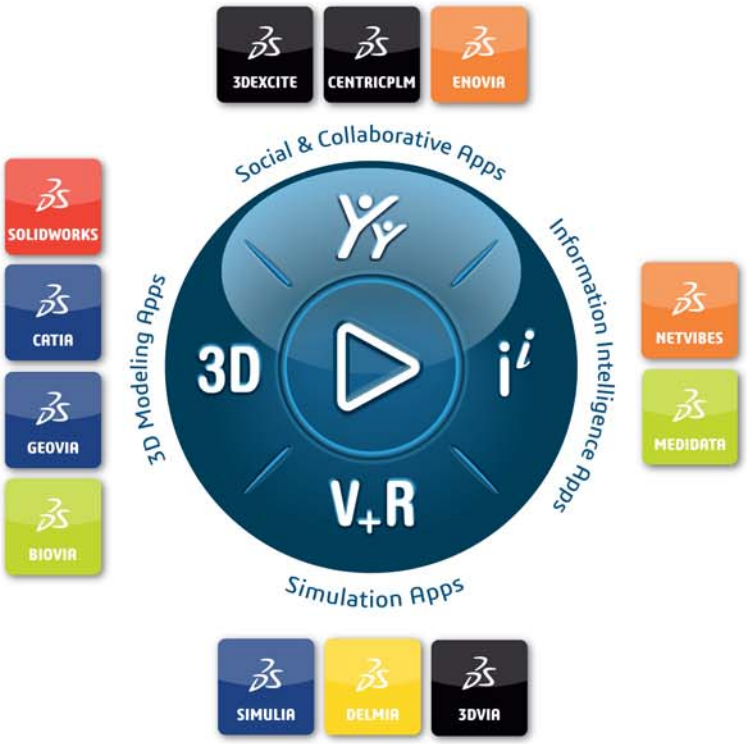
DSA CONTROLS

E

ELEMENT RESPONSE

N

NODE RESPONSE



Our **3D**EXPERIENCE® platform powers our brand applications, serving 12 industries, and provides a rich portfolio of industry solution experiences.

Dassault Systèmes is a catalyst for human progress. We provide business and people with collaborative virtual environments to imagine sustainable innovations. By creating virtual twin experiences of the real world with our **3DEXPERIENCE** platform and applications, our customers can redefine the creation, production and life-cycle-management processes of their offer and thus have a meaningful impact to make the world more sustainable. The beauty of the Experience Economy is that it is a human-centered economy for the benefit of all – consumers, patients and citizens.

Dassault Systèmes brings value to more than 300,000 customers of all sizes, in all industries, in more than 150 countries. For more information, visit **www.3ds.com**.

Europe/Middle East/Africa
Dassault Systèmes
10, rue Marcel Dassault
CS 40501
78946 Vélizy-Villacoublay Cedex
France

Asia-Pacific
Dassault Systèmes
17F, Foxconn Building,
No. 1366, Lujiazui Ring Road
Pilot Free Trade Zone, Shanghai 200120
China

Americas
Dassault Systèmes
175 Wyman Street
Waltham, Massachusetts
02451-1223
USA

DASSAULT SYSTEMES

©2025 Dassault Systèmes. All rights reserved. **3DEXPERIENCE**, the 3DS logo, the Compass icon, IFWE, 3DEXCITE, 3DVIA, BIOVIA, CATIA, CENTRIC PLM, DELMIA, ENOVIA, GEOVIA, MEDIDATA, NETVIBES, OUTSCALE, SIMULIA and SOLIDWORKS are commercial trademarks or registered trademarks of Dassault Systèmes, a European company (Societas Europaea) incorporated under French law, and registered with the Versailles trade and companies registry under number 322 306 440, or its subsidiaries in the United States and/or other countries.