DASSAULT SYSTEMES

SIMULIA

# ABAQUS 6.14

## *KEYWORDS REFERENCE GUIDE*




3DEXPERIENCE

# Abaqus Keywords

# Reference Guide

# Legal Notices

CAUTION: This documentation is intended for qualified users who will exercise sound engineering judgment and expertise in the use of the Abaqus Software. The Abaqus Software is inherently complex, and the examples and procedures in this documentation are not intended to be exhaustive or to apply to any particular situation. Users are cautioned to satisfy themselves as to the accuracy and results of their analyses.

Dassault Systèmes and its subsidiaries, including Dassault Systèmes Simulia Corp., shall not be responsible for the accuracy or usefulness of any analysis performed using the Abaqus Software or the procedures, examples, or explanations in this documentation. Dassault Systèmes and its subsidiaries shall not be responsible for the consequences of any errors or omissions that may appear in this documentation.

The Abaqus Software is available only under license from Dassault Systèmes or its subsidiary and may be used or reproduced only in accordance with the terms of such license. This documentation is subject to the terms and conditions of either the software license agreement signed by the parties, or, absent such an 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.

No part of this documentation may be reproduced or distributed in any form without prior written permission of Dassault Systèmes or its subsidiary.

The Abaqus Software is a product of Dassault Systèmes Simulia Corp., Providence, RI, USA.

© Dassault Systèmes, 2014

Abaqus, the 3DS logo, SIMULIA, CATIA, and Unified FEA are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the United States and/or other countries.

Other company, product, and service names may be trademarks or service marks of their respective owners. For additional information concerning trademarks, copyrights, and licenses, see the Legal Notices in the Abaqus 6.14 Installation and Licensing Guide.

# Preface

This section lists various resources that are available for help with using Abaqus Unified FEA software.

## Support

Both technical software support (for problems with creating a model or performing an analysis) and systems support (for installation, licensing, and hardware-related problems) for Abaqus are offered through a global network of support offices, as well as through our online support system. Regional contact information is accessible from **SIMULIA→Locations** at www.3ds.com/simulia. The online support system is accessible by selecting the **SUBMIT A REQUEST** link at www.3ds.com/support-training (http://www.3ds.com/support-training/).

### Online support

Dassault Systèmes provides a knowledge base of questions and answers, solutions to questions that we have answered, and guidelines on how to use Abaqus, Engineering Process Composer, Isight, Tosca, fe-safe, and other SIMULIA products. The knowledge base is available from www.3ds.com/support-training (http://www.3ds.com/support-training/).

By using the online support system, you can also submit new requests for support. All support/service requests are tracked. If you contact us by means outside the system to discuss an existing support problem and you know the support request number, please mention it so that we can query the knowledge base to see what the latest action has been.

## Training

All support offices offer regularly scheduled public training classes. The courses are offered in a traditional classroom form and via the Web. We also provide training seminars at customer sites. All training classes and seminars include workshops to provide as much practical experience with Abaqus as possible. For a schedule and descriptions of available classes, see the **Training Classes** link at www.3ds.com/simulia or call your support office.

## Feedback

We welcome any suggestions for improvements to Abaqus software, the support program, or documentation. We will ensure that any enhancement requests you make are considered for future releases. If you wish to make a suggestion about the service or products, refer to www.3ds.com/simulia. Complaints should be made by contacting your support office or by visiting **SIMULIA→Quality Assurance** at www.3ds.com/simulia.

## Contents

**A**


*ACOUSTIC FLOW VELOCITY 1.1
*ACOUSTIC MEDIUM 1.2
*ACOUSTIC WAVE FORMULATION 1.3
*ADAPTIVE MESH 1.4
*ADAPTIVE MESH CONSTRAINT 1.5
*ADAPTIVE MESH CONTROLS 1.6
*ADAPTIVE MESH REFINEMENT 1.7
*ADJUST 1.8
*AMPLITUDE 1.9
*ANISOTROPIC HYPERELASTIC 1.10
*ANNEAL 1.11
*ANNEAL TEMPERATURE 1.12
*AQUA 1.13
*ASSEMBLY 1.14
*ASYMMETRIC-AXISYMMETRIC 1.15
*AXIAL 1.16


**B**


*BASE MOTION 2.1
*BASELINE CORRECTION 2.2
*BEAM ADDED INERTIA 2.3
*BEAM FLUID INERTIA 2.4
*BEAM GENERAL SECTION 2.5
*BEAM SECTION 2.6
*BEAM SECTION GENERATE 2.7
*BEAM SECTION OFFSET 2.8
*BIAXIAL TEST DATA 2.9
*BLOCKAGE 2.10
*BOND 2.11
*BOUNDARY 2.12
*BRITTLE CRACKING 2.13
*BRITTLE FAILURE 2.14
*BRITTLE SHEAR 2.15
*BUCKLE 2.16
*BUCKLING ENVELOPE 2.17
*BUCKLING LENGTH 2.18

*BUCKLING REDUCTION FACTORS 2.19
*BULK VISCOSITY 2.20

## C

*C ADDED MASS 3.1
*CAP CREEP 3.2
*CAP HARDENING 3.3
*CAP PLASTICITY 3.4
*CAPACITY 3.5
*CAST IRON COMPRESSION HARDENING 3.6
*CAST IRON PLASTICITY 3.7
*CAST IRON TENSION HARDENING 3.8
*CAVITY DEFINITION 3.9
*CECHARGE 3.10
*CECURRENT 3.11
*CENTROID 3.12
*CFD 3.13
*CFILM 3.14
*CFLOW 3.15
*CFLUX 3.16
*CHANGE FRICTION 3.17
*CHARACTERISTIC LENGTH 3.18
*CLAY HARDENING 3.19
*CLAY PLASTICITY 3.20
*CLEARANCE 3.21
*CLOAD 3.22
*COHESIVE BEHAVIOR 3.23
*COHESIVE SECTION 3.24
*COMBINED TEST DATA 3.25
*COMPLEX FREQUENCY 3.26
*COMPOSITE MODAL DAMPING 3.27
*CONCRETE 3.28
*CONCRETE COMPRESSION DAMAGE 3.29
*CONCRETE COMPRESSION HARDENING 3.30
*CONCRETE DAMAGED PLASTICITY 3.31
*CONCRETE TENSION DAMAGE 3.32
*CONCRETE TENSION STIFFENING 3.33
*CONDUCTIVITY 3.34
*CONNECTOR BEHAVIOR 3.35
*CONNECTOR CONSTITUTIVE REFERENCE 3.36
*CONNECTOR DAMAGE EVOLUTION 3.37
*CONNECTOR DAMAGE INITIATION 3.38

*CONNECTOR DAMPING 3.39
*CONNECTOR DERIVED COMPONENT 3.40
*CONNECTOR ELASTICITY 3.41
*CONNECTOR FAILURE 3.42
*CONNECTOR FRICTION 3.43
*CONNECTOR HARDENING 3.44
*CONNECTOR LOAD 3.45
*CONNECTOR LOCK 3.46
*CONNECTOR MOTION 3.47
*CONNECTOR PLASTICITY 3.48
*CONNECTOR POTENTIAL 3.49
*CONNECTOR SECTION 3.50
*CONNECTOR STOP 3.51
*CONNECTOR UNIAXIAL BEHAVIOR 3.52
*CONSTRAINT CONTROLS 3.53
*CONTACT 3.54
*CONTACT CLEARANCE 3.55
*CONTACT CLEARANCE ASSIGNMENT 3.56
*CONTACT CONTROLS 3.57
*CONTACT CONTROLS ASSIGNMENT 3.58
*CONTACT DAMPING 3.59
*CONTACT EXCLUSIONS 3.60
*CONTACT FILE 3.61
*CONTACT FORMULATION 3.62
*CONTACT INCLUSIONS 3.63
*CONTACT INITIALIZATION ASSIGNMENT 3.64
*CONTACT INITIALIZATION DATA 3.65
*CONTACT INTERFERENCE 3.66
*CONTACT OUTPUT 3.67
*CONTACT PAIR 3.68
*CONTACT PERMEABILITY 3.69
*CONTACT PRINT 3.70
*CONTACT PROPERTY ASSIGNMENT 3.71
*CONTACT RESPONSE 3.72
*CONTACT STABILIZATION 3.73
*CONTOUR INTEGRAL 3.74
*CONTROLS 3.75
*CONWEP CHARGE PROPERTY 3.76
*CORRELATION 3.77
*CO-SIMULATION 3.78
*CO-SIMULATION CONTROLS 3.79
*CO-SIMULATION REGION 3.80

*COUPLED TEMPERATURE-DISPLACEMENT 3.81
*COUPLED THERMAL-ELECTRICAL 3.82
*COUPLING 3.83
*CRADIATE 3.84
*CREEP 3.85
*CREEP STRAIN RATE CONTROL 3.86
*CRUSHABLE FOAM 3.87
*CRUSHABLE FOAM HARDENING 3.88
*CYCLED PLASTIC 3.89
*CYCLIC 3.90
*CYCLIC HARDENING 3.91
*CYCLIC SYMMETRY MODEL 3.92

**D**

*D ADDED MASS 4.1
*D EM POTENTIAL 4.2
*DAMAGE EVOLUTION 4.3
*DAMAGE INITIATION 4.4
*DAMAGE STABILIZATION 4.5
*DAMPING 4.6
*DAMPING CONTROLS 4.7
*DASHPOT 4.8
*DEBOND 4.9
*DECHARGE 4.10
*DECURRENT 4.11
*DEFORMATION PLASTICITY 4.12
*DENSITY 4.13
*DEPVAR 4.14
*DESIGN GRADIENT 4.15
*DESIGN PARAMETER 4.16
*DESIGN RESPONSE 4.17
*DETONATION POINT 4.18
*DFLOW 4.19
*DFLUX 4.20
*DIAGNOSTICS 4.21
*DIELECTRIC 4.22
*DIFFUSIVITY 4.23
*DIRECT CYCLIC 4.24
*DISCRETE SECTION 4.25
*DISPLAY BODY 4.26
*DISTRIBUTING 4.27
*DISTRIBUTING COUPLING 4.28

*DISTRIBUTION 4.29
*DISTRIBUTION TABLE 4.30
*DLOAD 4.31
*DOMAIN DECOMPOSITION 4.32
*DRAG CHAIN 4.33
*DRUCKER PRAGER 4.34
*DRUCKER PRAGER CREEP 4.35
*DRUCKER PRAGER HARDENING 4.36
*DSA CONTROLS 4.37
*DSECHARGE 4.38
*DSECURRENT 4.39
*DSFLOW 4.40
*DSFLUX 4.41
*DSLOAD 4.42
*DYNAMIC 4.43
*DYNAMIC TEMPERATURE-DISPLACEMENT 4.44

**E**

*EL FILE 5.1
*EL PRINT 5.2
*ELASTIC 5.3
*ELCOPY 5.4
*ELECTRICAL CONDUCTIVITY 5.5
*ELECTROMAGNETIC 5.6
*ELEMENT 5.7
*ELEMENT MATRIX OUTPUT 5.8
*ELEMENT OPERATOR OUTPUT 5.9
*ELEMENT OUTPUT 5.10
*ELEMENT RESPONSE 5.11
*ELGEN 5.12
*ELSET 5.13
*EMBEDDED ELEMENT 5.14
*EMISSIVITY 5.15
*END ASSEMBLY 5.16
*END INSTANCE 5.17
*END LOAD CASE 5.18
*END PART 5.19
*END STEP 5.20
*ENERGY EQUATION SOLVER 5.21
*ENERGY FILE 5.22
*ENERGY OUTPUT 5.23
*ENERGY PRINT 5.24

*ENRICHMENT 5.25
*ENRICHMENT ACTIVATION 5.26
*EOS 5.27
*EOS COMPACTION 5.28
*EPJOINT 5.29
*EQUATION 5.30
*EULERIAN BOUNDARY 5.31
*EULERIAN MESH MOTION 5.32
*EULERIAN SECTION 5.33
*EXPANSION 5.34
*EXTREME ELEMENT VALUE 5.35
*EXTREME NODE VALUE 5.36
*EXTREME VALUE 5.37

**F**

*FABRIC 6.1
*FAIL STRAIN 6.2
*FAIL STRESS 6.3
*FAILURE RATIOS 6.4
*FASTENER 6.5
*FASTENER PROPERTY 6.6
*FIELD 6.7
*FILE FORMAT 6.8
*FILE OUTPUT 6.9
*FILM 6.10
*FILM PROPERTY 6.11
*FILTER 6.12
*FIXED MASS SCALING 6.13
*FLOW 6.14
*FLEXIBLE BODY 6.15
*FLUID BEHAVIOR 6.16
*FLUID BOUNDARY 6.17
*FLUID BULK MODULUS 6.18
*FLUID CAVITY 6.19
*FLUID DENSITY 6.20
*FLUID EXCHANGE 6.21
*FLUID EXCHANGE ACTIVATION 6.22
*FLUID EXCHANGE PROPERTY 6.23
*FLUID EXPANSION 6.24
*FLUID FLUX 6.25
*FLUID INFLATOR 6.26
*FLUID INFLATOR ACTIVATION 6.27

*FLUID INFLATOR MIXTURE 6.28
*FLUID INFLATOR PROPERTY 6.29
*FLUID LEAKOFF 6.30
*FLUID SECTION 6.31
*FOUNDATION 6.32
*FRACTURE CRITERION 6.33
*FRAME SECTION 6.34
*FREQUENCY 6.35
*FRICTION 6.36

**G**

*GAP 7.1
*GAP CONDUCTANCE 7.2
*GAP ELECTRICAL CONDUCTANCE 7.3
*GAP FLOW 7.4
*GAP HEAT GENERATION 7.5
*GAP RADIATION 7.6
*GAS SPECIFIC HEAT 7.7
*GASKET BEHAVIOR 7.8
*GASKET CONTACT AREA 7.9
*GASKET ELASTICITY 7.10
*GASKET SECTION 7.11
*GASKET THICKNESS BEHAVIOR 7.12
*GEL 7.13
*GEOSTATIC 7.14
*GLOBAL DAMPING 7.15

**H**

*HEADING 8.1
*HEAT GENERATION 8.2
*HEAT TRANSFER 8.3
*HEATCAP 8.4
*HOURGLASS STIFFNESS 8.5
*HYPERELASTIC 8.6
*HYPERFOAM 8.7
*HYPOELASTIC 8.8
*HYSTERESIS 8.9

**I**

*IMPEDANCE 9.1
*IMPEDANCE PROPERTY 9.2

*IMPERFECTION 9.3
*IMPORT 9.4
*IMPORT CONTROLS 9.5
*IMPORT ELSET 9.6
*IMPORT NSET 9.7
*INCIDENT WAVE 9.8
*INCIDENT WAVE FLUID PROPERTY 9.9
*INCIDENT WAVE INTERACTION 9.10
*INCIDENT WAVE INTERACTION PROPERTY 9.11
*INCIDENT WAVE PROPERTY 9.12
*INCIDENT WAVE REFLECTION 9.13
*INCLUDE 9.14
*INCREMENTATION OUTPUT 9.15
*INELASTIC HEAT FRACTION 9.16
*INERTIA RELIEF 9.17
*INITIAL CONDITIONS 9.18
*INSTANCE 9.19
*INTEGRATED OUTPUT 9.20
*INTEGRATED OUTPUT SECTION 9.21
*INTERFACE 9.22
*ITS 9.23

**J**

*JOINT 10.1
*JOINT ELASTICITY 10.2
*JOINT PLASTICITY 10.3
*JOINTED MATERIAL 10.4
*JOULE HEAT FRACTION 10.5

**K**

*KAPPA 11.1
*KINEMATIC 11.2
*KINEMATIC COUPLING 11.3

**L**

*LATENT HEAT 12.1
*LOAD CASE 12.2
*LOADING DATA 12.3
*LOW DENSITY FOAM 12.4

**M**


*MAGNETIC PERMEABILITY 13.1
*MAGNETOSTATIC 13.2
*MAP SOLUTION 13.3
*MASS 13.4
*MASS ADJUST 13.5
*MASS DIFFUSION 13.6
*MASS FLOW RATE 13.7
*MATERIAL 13.8
*MATRIX 13.9
*MATRIX ASSEMBLE 13.10
*MATRIX CHECK 13.11
*MATRIX GENERATE 13.12
*MATRIX INPUT 13.13
*MATRIX OUTPUT 13.14
*MEDIA TRANSPORT 13.15
*MEMBRANE SECTION 13.16
*MODAL DAMPING 13.17
*MODAL DYNAMIC 13.18
*MODAL FILE 13.19
*MODAL OUTPUT 13.20
*MODAL PRINT 13.21
*MODEL CHANGE 13.22
*MOHR COULOMB 13.23
*MOHR COULOMB HARDENING 13.24
*MOISTURE SWELLING 13.25
*MOLECULAR WEIGHT 13.26
*MOMENTUM EQUATION SOLVER 13.27
*MONITOR 13.28
*MOTION 13.29
*MPC 13.30
*MULLINS EFFECT 13.31
*M1 13.32
*M2 13.33


**N**


*NCOPY 14.1
*NFILL 14.2
*NGEN 14.3
*NMAP 14.4
*NO COMPRESSION 14.5

*NO TENSION 14.6
*NODAL ENERGY RATE 14.7
*NODAL THICKNESS 14.8
*NODE 14.9
*NODE FILE 14.10
*NODE OUTPUT 14.11
*NODE PRINT 14.12
*NODE RESPONSE 14.13
*NONLINEAR BH 14.14
*NONSTRUCTURAL MASS 14.15
*NORMAL 14.16
*NSET 14.17

**O**

*ORIENTATION 15.1
*ORNL 15.2
*OUTPUT 15.3

**P, Q**

*PARAMETER 16.1
*PARAMETER DEPENDENCE 16.2
*PARAMETER SHAPE VARIATION 16.3
*PART 16.4
*PERIODIC 16.5
*PERIODIC MEDIA 16.6
*PERMANENT MAGNETIZATION 16.7
*PERMEABILITY 16.8
*PHYSICAL CONSTANTS 16.9
*PIEZOELECTRIC 16.10
*PIPE-SOIL INTERACTION 16.11
*PIPE-SOIL STIFFNESS 16.12
*PLANAR TEST DATA 16.13
*PLASTIC 16.14
*PLASTIC AXIAL 16.15
*PLASTIC M1 16.16
*PLASTIC M2 16.17
*PLASTIC TORQUE 16.18
*POROUS BULK MODULI 16.19
*POROUS ELASTIC 16.20
*POROUS FAILURE CRITERIA 16.21
*POROUS METAL PLASTICITY 16.22
*POST OUTPUT 16.23

*POTENTIAL 16.24
*PREPRINT 16.25
*PRESSURE EQUATION SOLVER 16.26
*PRESSURE PENETRATION 16.27
*PRESSURE STRESS 16.28
*PRESTRESS HOLD 16.29
*PRE-TENSION SECTION 16.30
*PRINT 16.31
*PSD-DEFINITION 16.32

**R**

*RADIATE 17.1
*RADIATION FILE 17.2
*RADIATION OUTPUT 17.3
*RADIATION PRINT 17.4
*RADIATION SYMMETRY 17.5
*RADIATION VIEW FACTOR 17.6
*RANDOM RESPONSE 17.7
*RATE DEPENDENT 17.8
*RATIOS 17.9
*REACTION RATE 17.10
*REBAR 17.11
*REBAR LAYER 17.12
*REFLECTION 17.13
*RELEASE 17.14
*RESPONSE SPECTRUM 17.15
*RESTART 17.16
*RETAINED NODAL DOFS 17.17
*RIGID BODY 17.18
*RIGID SURFACE 17.19
*ROTARY INERTIA 17.20

**S**

*SECTION CONTROLS 18.1
*SECTION FILE 18.2
*SECTION ORIGIN 18.3
*SECTION POINTS 18.4
*SECTION PRINT 18.5
*SELECT CYCLIC SYMMETRY MODES 18.6
*SELECT EIGENMODES 18.7
*SFILM 18.8
*SFLOW 18.9

*SHEAR CENTER 18.10
*SHEAR FAILURE 18.11
*SHEAR RETENTION 18.12
*SHEAR TEST DATA 18.13
*SHELL GENERAL SECTION 18.14
*SHELL SECTION 18.15
*SHELL TO SOLID COUPLING 18.16
*SIMPEDANCE 18.17
*SIMPLE SHEAR TEST DATA 18.18
*SLIDE LINE 18.19
*SLOAD 18.20
*SOILS 18.21
*SOLID SECTION 18.22
*SOLUBILITY 18.23
*SOLUTION TECHNIQUE 18.24
*SOLVER CONTROLS 18.25
*SORPTION 18.26
*SPECIFIC HEAT 18.27
*SPECTRUM 18.28
*SPRING 18.29
*SRADIATE 18.30
*STATIC 18.31
*STEADY STATE CRITERIA 18.32
*STEADY STATE DETECTION 18.33
*STEADY STATE DYNAMICS 18.34
*STEADY STATE TRANSPORT 18.35
*STEP 18.36
*SUBCYCLING 18.37
*SUBMODEL 18.38
*SUBSTRUCTURE COPY 18.39
*SUBSTRUCTURE DELETE 18.40
*SUBSTRUCTURE DIRECTORY 18.41
*SUBSTRUCTURE GENERATE 18.42
*SUBSTRUCTURE LOAD CASE 18.43
*SUBSTRUCTURE MATRIX OUTPUT 18.44
*SUBSTRUCTURE PATH 18.45
*SUBSTRUCTURE PROPERTY 18.46
*SURFACE 18.47
*SURFACE BEHAVIOR 18.48
*SURFACE FLAW 18.49
*SURFACE INTERACTION 18.50
*SURFACE OUTPUT 18.51

*SURFACE PROPERTY 18.52
*SURFACE PROPERTY ASSIGNMENT 18.53
*SURFACE SECTION 18.54
*SURFACE SMOOTHING 18.55
*SWELLING 18.56
*SYMMETRIC MODEL GENERATION 18.57
*SYMMETRIC RESULTS TRANSFER 18.58
*SYSTEM 18.59

**T**

*TEMPERATURE 19.1
*TENSILE FAILURE 19.2
*TENSION CUTOFF 19.3
*TENSION STIFFENING 19.4
*THERMAL EXPANSION 19.5
*TIE 19.6
*TIME POINTS 19.7
*TORQUE 19.8
*TORQUE PRINT 19.9
*TRACER PARTICLE 19.10
*TRANSFORM 19.11
*TRANSPORT EQUATION SOLVER 19.12
*TRANSPORT VELOCITY 19.13
*TRANSVERSE SHEAR STIFFNESS 19.14
*TRIAXIAL TEST DATA 19.15
*TRS 19.16
*TURBULENCE MODEL 19.17

**U**

*UEL PROPERTY 20.1
*UNDEX CHARGE PROPERTY 20.2
*UNIAXIAL 20.3
*UNIAXIAL TEST DATA 20.4
*UNLOADING DATA 20.5
*USER DEFINED FIELD 20.6
*USER ELEMENT 20.7
*USER MATERIAL 20.8
*USER OUTPUT VARIABLES 20.9

**V**

*VARIABLE MASS SCALING 21.1

*VIEW FACTOR OUTPUT 21.2
*VISCO 21.3
*VISCOELASTIC 21.4
*VISCOSITY 21.5
*VISCOUS 21.6
*VOID NUCLEATION 21.7
*VOLUMETRIC TEST DATA 21.8

**W, X, Y, Z**

*WAVE 22.1
*WIND 22.2

## 1.0 BROWSING THE Abaqus Keywords Reference Guide

This guide describes all of the input options that are available in Abaqus.

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 "Defining a model in Abaqus," Section 1.3.1 of the Abaqus Analysis User's Guide.

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 "Defining an assembly," Section 2.10.1 of the Abaqus Analysis User's Guide.

The **Abaqus/CAE** field indicates where within Abaqus/CAE you can locate the user interface related to the keyword. You can also refer to the online HTML version of Appendix A, "Keyword support," of the Abaqus/CAE User's Guide, 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," Section 3.2.15 of the Abaqus Analysis User's Guide) 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," Section 3.2.16 of the Abaqus Analysis User's Guide.

# 1. A

## 1.1 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**Reference:**

- "Acoustic, shock, and coupled acoustic-structural analysis," Section 6.10.1 of the Abaqus Analysis User's Guide

**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 parameter:**

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," Section 34.1.2 of the Abaqus Analysis User's Guide).

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," Section 1.2.2 of the Abaqus Analysis User's Guide, 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.

## 1.2 *ACOUSTIC MEDIUM: Specify an acoustic medium.

This option is used to define the properties of an acoustic medium used with 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

**Reference:**

- "Acoustic medium," Section 26.3.1 of the Abaqus Analysis User's Guide

**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 (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.

VOLUMETRIC DRAG

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

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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 or POROUS MODEL=MIKI:**

First (and only) line:

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

## 1.3 *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

**Reference:**

- "Acoustic and shock loads," Section 34.4.6 of the Abaqus Analysis User's Guide

**Optional parameter:**

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.**

## 1.4 *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," Section 12.2.2 of the Abaqus Analysis User's Guide
- "ALE adaptive meshing and remapping in Abaqus/Explicit," Section 12.2.3 of the Abaqus Analysis User's Guide
- "Defining ALE adaptive mesh domains in Abaqus/Standard," Section 12.2.6 of the Abaqus Analysis User's Guide
- "ALE adaptive meshing and remapping in Abaqus/Standard," Section 12.2.7 of the Abaqus Analysis User's Guide
- *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 acoustic analysis or when 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. 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 1.

**There are no data lines associated with this option.**

## 1.5 *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," Section 12.2.2 of the Abaqus Analysis User's Guide
- "Defining ALE adaptive mesh domains in Abaqus/Standard," Section 12.2.6 of the Abaqus Analysis User's Guide
- "UMESHMOTION," Section 1.1.43 of the Abaqus User Subroutines Reference Guide
- *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," Section 34.1.2 of the Abaqus Analysis User's Guide).

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.

## 1.6 *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," Section 12.2.2 of the Abaqus Analysis User's Guide
- "ALE adaptive meshing and remapping in Abaqus/Explicit," Section 12.2.3 of the Abaqus Analysis User's Guide
- "Defining ALE adaptive mesh domains in Abaqus/Standard," Section 12.2.6 of the Abaqus Analysis User's Guide
- "ALE adaptive meshing and remapping in Abaqus/Standard," Section 12.2.7 of the Abaqus Analysis User's Guide
- *ADAPTIVE MESH

**Required parameter:**

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, $\alpha_C$. 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, $\theta_I$, in degrees ($0^\circ \leq \theta_I \leq 180^\circ$). This angle is used to detect geometric edges and corners. The default value is $\theta_I = 30^\circ$. Setting $\theta_I = 180^\circ$ 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, $\theta_C$, in degrees ($5^\circ \leq \theta_C \leq 85^\circ$). The default value is $\theta_C = 60^\circ$.

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 $\theta_C$. 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 $\theta_C$.

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, $\theta_T$, in degrees ($0^\circ \leq \theta_T \leq 180^\circ$). This angle is used to determine when geometric edges and corners should be deactivated to allow remeshing across them. The default value is $\theta_T = 30^\circ$. Setting $\theta_T = 0^\circ$ 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.

## 1.7 *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.

**Product:** Abaqus/Explicit

**Type:** Model data

**Level:** Model

**References:**

- "Eulerian analysis," Section 14.1.1 of the Abaqus Analysis User's Guide
- "Defining adaptive mesh refinement in the Eulerian domain," Section 14.1.4 of the Abaqus Analysis User's Guide
- *EULERIAN SECTION

**Required parameter:**

ELSET

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

**Optional parameter:**

LEVEL

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

COARSENING

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.

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.

**Data lines to define the criteria used in the mesh refinement:**

First line:

1. Refinement criteria label (see “Defining adaptive mesh refinement in the Eulerian domain,” Section 14.1.4 of the Abaqus Analysis User’s Guide).
2. Value of the refinement criteria if applicable.

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

## 1.8 *ADJUST: Adjust user-specified nodal coordinates to lie on a given surface.

This option is used to adjust user-specified nodal coordinates so that the nodes lie on a given surface.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Part, Part instance, Assembly

**Abaqus/CAE:** Interaction module

**Reference:**

- "Adjusting nodal coordinates," Section 2.1.6 of the Abaqus Analysis User's Guide

**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 parameter:**

ORIENTATION

Set this parameter equal to the name of an orientation definition (see "Orientations," Section 2.2.5 of the Abaqus Analysis User's Guide) 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.**

## 1.9 *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/CFD 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.

**Reference:**

- "Amplitude curves," Section 34.1.2 of the Abaqus Analysis User's Guide

**Required parameter:**

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," Section 34.1.2 of the Abaqus Analysis User's Guide.

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.

Parameter settings of BUBBLE, SOLUTION DEPENDENT, USER, and ACTUATOR are not available in Abaqus/CFD analyses.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Guide, 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.

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," Section 1.2.2 of the Abaqus Analysis User's Guide, 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. This parameter is not available in Abaqus/CFD.

**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.
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.
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. $\omega$, 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. $\omega_1$.
5. $\omega_2$.

**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, $\gamma$.
7. Density of charge material, $\rho_c$.
8. Mass of charge material, $m_c$.
9. Depth magnitude of charge material, $d_I$.

Second line:

1. Fluid mass density, $\rho_f$.
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, $\eta$. 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, $T_{\text{final}}$.
2. Maximum number of time steps for the bubble simulation, $N_{\text{steps}}$. The bubble amplitude simulation ceases when the number of steps reaches $N_{\text{steps}}$ or the time duration, $T_{\text{final}}$, is reached. The default is 1500.
3. Relative step size control parameter, $\Omega_{\text{rel}}$. The default is $1 \times 10^{-11}$.
4. Absolute step size control parameter, $X_{\text{abs}}$. The default is $1 \times 10^{-11}$.
5. Step size control exponent, $\beta$. The step size, $\Delta t$, 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.**

## 1.10 *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," Section 22.5.3 of the Abaqus Analysis User's Guide
- "UANISOHYPER_STRAIN," Section 1.1.21 of the Abaqus User Subroutines Reference Guide
- "UANISOHYPER_INV," Section 1.1.20 of the Abaqus User Subroutines Reference Guide
- "VUANISOHYPER_STRAIN," Section 1.2.10 of the Abaqus User Subroutines Reference Guide
- "VUANISOHYPER_INV," Section 1.2.9 of the Abaqus User Subroutines Reference Guide

**Optional, mutually exclusive parameters:**

FUNG-ANISOTROPIC

Include this parameter to use the generalized Fung anisotropic strain energy potential.

FUNG-ORTHOTROPIC

Include this parameter to use the generalized Fung orthotropic strain energy potential.

HOLZAPFEL

Include this parameter to use the Holzapfel-Gasser-Ogden strain energy potential.

USER

Include this parameter 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 the USER parameter is included:**

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" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

LOCAL DIRECTIONS

This parameter can only be used in combination with an invariant-based strain energy potential, such as HOLZAPFEL or 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 $M \geq N$. If $M > N$, the first $N$ directions will be used.

If the HOLZAPFEL strain energy potential is used, at least one local direction must be specified.

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 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 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. $c$. (Units of $\mathrm{FL}^{-2}$.)
7. $D$. (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 $\mathrm{FL}^{-2}$.)
3. $D$. (Units of $\mathrm{F}^{-1}\mathrm{L}^{2}$.)
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 model:**

First line:

1. $C_{10}$. (Units of $\mathrm{FL}^{-2}$.)
2. $D$. (Units of $\mathrm{F}^{-1}\mathrm{L}^{2}$.)
3. $k_1$. (Units of $\mathrm{FL}^{-2}$.)
4. $k_2$.
5. Fiber dispersion parameter $\kappa$ ($0 \leq \kappa \leq 1/3$).

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 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.

## 1.11 *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

**Reference:**

- "Annealing procedure," Section 6.12.1 of the Abaqus Analysis User's Guide

**Optional parameter:**

TEMPERATURE

Set this parameter equal to the temperature, $\theta$, 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.**

## 1.12 *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," Section 23.2.5 of the Abaqus Analysis User's Guide
- *PLASTIC

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

**Data lines to define the annealing temperature:**

First line:

1. Value of the annealing temperature, $\Theta$.
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 $\Theta$ on field variables.

## 1.13 *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.

**Product:** Abaqus/Aqua

**Type:** Model data

**Level:** Model

**Reference:**

- "Abaqus/Aqua analysis," Section 6.11.1 of the Abaqus Analysis User's Guide

**Optional parameter:**

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Guide, 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,” Section 1.2.1 of the Abaqus Analysis User’s Guide, for a description of how to define a property as a function of multiple independent variables.

## 1.14 *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/CFD Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Assembly module

**References:**

- *END ASSEMBLY
- "Defining an assembly," Section 2.10.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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.**

## 1.15 *ASYMMETRIC-AXISYMMETRIC: Define areas of integration for contact elements used with CAXA*n* or SAXA*n* 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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance, Assembly

**References:**

- "Contact interaction analysis: overview," Section 36.1.1 of the Abaqus Analysis User's Guide
- "Contact modeling if asymmetric-axisymmetric elements are present," Section 36.3.10 of the Abaqus Analysis User's Guide
- *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.**

## 1.16 *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," Section 29.3.7 of the Abaqus Analysis User's Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

**2. B**

## 2.1 *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," Section 6.3.5 of the Abaqus Analysis User's Guide
- "Transient modal dynamic analysis," Section 6.3.7 of the Abaqus Analysis User's Guide
- "Mode-based steady-state dynamic analysis," Section 6.3.8 of the Abaqus Analysis User's Guide
- "Random response analysis," Section 6.3.11 of the Abaqus Analysis User's Guide

**Required parameter:**

DOF

Set this parameter equal to the direction (1–6, including rotations) for which the base motion is being defined. This direction is always a global direction.

Set this parameter equal to 8 for the acoustic degree of freedom, which is for use for secondary base motions only in the SIM-based architecture.

**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.

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.

## 2.2 *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," Section 34.1.2 of the Abaqus Analysis User's Guide

**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.

## 2.3 *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," Section 29.3.3 of the Abaqus Analysis User's Guide
- "Beam section behavior," Section 29.3.5 of the Abaqus Analysis User's Guide

**Optional parameters:**

ALPHA

Set this parameter equal to the $\alpha_R$ 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, $\alpha$ (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.

## 2.4 *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," Section 29.3.5 of the Abaqus Analysis User's Guide
- "Acoustic, shock, and coupled acoustic-structural analysis," Section 6.10.1 of the Abaqus Analysis User's Guide
- "Loading due to an incident dilatational wave field," Section 6.3.1 of the Abaqus Theory Guide

**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.

## 2.5 *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. In this case the beam section geometry and material descriptions are combined; no *MATERIAL reference is associated with this option.

**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," Section 29.3.7 of the Abaqus Analysis User's Guide
- "Beam modeling: overview," Section 29.3.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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.

**Optional parameters:**

DEPENDENCIES

This parameter cannot be used when SECTION=NONLINEAR GENERAL or SECTION=MESHED.

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.

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 will enforce incompressible behavior of the element.

For PIPE elements with SECTION=PIPE, this parameter will also be 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," Section 29.3.9 of the Abaqus Analysis User's Guide). The following cross-sections are available:

- 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 hollow, circular section.
- RECT, for a solid, rectangular section.
- TRAPEZOID, for a trapezoidal section.

TAPER

This parameter is relevant only for Abaqus/Standard analyses.
Include this parameter to define a general beam section with a tapered cross-section.

ZERO

This parameter cannot be used when SECTION=MESHED.
Set this parameter equal to the reference temperature for thermal expansion ($\theta^0$), 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, $\Gamma_0$. (Only needed in Abaqus/Standard when the section is associated with open-section beam elements.)
7. Warping constant, $\Gamma_W$. (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," Section 29.3.4 of the Abaqus Analysis User's Guide, for details.

Third line:

1. Young's modulus, $E$.
2. Torsional shear modulus, $G$. (Not used for beams in a plane.)

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 node A):

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, $\Gamma_0$. (Only needed in Abaqus/Standard when the section is associated with open-section beam elements.)
7. Warping constant, $\Gamma_W$. (Only needed in Abaqus/Standard when the section is associated with open-section beam elements.)

Second line (properties of node B):

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, $\Gamma_0$. (Only needed in Abaqus/Standard when the section is associated with open-section beam elements.)
7. Warping constant, $\Gamma_W$. (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," Section 29.3.4 of the Abaqus Analysis User's Guide, for details.

Fourth line:

1. Young's modulus, $E$.
2. Torsional shear modulus, $G$. (Not used for beams in a plane.)
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," Section 29.3.4 of the Abaqus Analysis User's Guide, 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," Section 29.3.4 of the Abaqus Analysis User's Guide, for details.

Second line:

The entries on this line and the following line consist of the beam section properties that result from the two-dimensional meshed cross-section generation procedure. The properties are written to the file *jobname*`.bsp` during the cross-section generation and are typically read into a subsequent beam analysis using the *INCLUDE option. See "Meshed beam cross-sections," Section 10.6.1 of the Abaqus Analysis User's Guide, for details.

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:

1. Total mass of the section per unit length, $(\rho A)$.
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, CIRC, 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," Section 29.3.9 of the Abaqus Analysis User's Guide, 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. See "Beam element cross-section orientation," Section 29.3.4 of the Abaqus Analysis User's Guide, for details.

Third line:

1. Young's modulus, $E$.
2. Torsional shear modulus, $G$. (Not used for beams in a plane.)
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, HEX, I, L, PIPE, RECT, and TRAPEZOID sections if the TAPER parameter is included:**

First line (properties of node A):

1. Beam section geometric data. Values should be given as specified in "Beam cross-section library," Section 29.3.9 of the Abaqus Analysis User's Guide, for the chosen section type.
2. Etc.

Second line (properties of node B):

1. Beam section geometric data. Values should be given as specified in "Beam cross-section library," Section 29.3.9 of the Abaqus Analysis User's Guide, 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," Section 29.3.4 of the Abaqus Analysis User's Guide, for details.

Fourth line:

1. Young's modulus, $E$.
2. Torsional shear modulus, $G$. (Not used for beams in a plane.)
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=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," Section 29.3.4 of the Abaqus Analysis User's Guide, for details.

Fourth line:

1. Young's modulus, $E$.
2. Torsional shear modulus, $G$. (Not used for beams in a plane.)
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.

## 2.6 *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," Section 29.3.6 of the Abaqus Analysis User's Guide
- "Beam modeling: overview," Section 29.3.1 of the Abaqus Analysis User's Guide
- "Pipes and pipebends with deforming cross-sections: elbow elements," Section 29.5.1 of the Abaqus Analysis User's Guide

**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," Section 29.3.9 of the Abaqus Analysis User's Guide). 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:**

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," Section 29.3.9 of the Abaqus Analysis User's Guide).

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," Section 29.5.1 of the Abaqus Analysis User's Guide.

**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," Section 29.3.9 of the Abaqus Analysis User's Guide, 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. See "Beam element cross-section orientation," Section 29.3.4 of the Abaqus Analysis User's Guide, 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," Section 29.3.9 of the Abaqus Analysis User's Guide.
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," Section 29.3.9 of the Abaqus Analysis User's Guide. 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," Section 29.3.9 of the Abaqus Analysis User's Guide. 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," Section 29.3.4 of the Abaqus Analysis User's Guide, 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 $R = 0$.

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.

## 2.7 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Meshed beam cross-sections," Section 10.6.1 of the Abaqus Analysis User's Guide
- "Beam section behavior," Section 29.3.5 of the Abaqus Analysis User's Guide
- "Using a general beam section to define the section behavior," Section 29.3.7 of the Abaqus Analysis User's Guide

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

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

This option can be used only in conjunction with the *BEAM GENERAL SECTION option for the standard cross-sections available in the beam cross-section library (See "Beam element cross-section orientation," Section 29.3.4 of the Abaqus Analysis User's Guide, for details). It is used to define an offset for the cross-section origin from the beam axis with respect to the local (1, 2) axis system.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance

**References:**

- "Using a general beam section to define the section behavior," Section 29.3.7 of the Abaqus Analysis User's Guide
- *BEAM GENERAL 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.

## 2.9 *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," Section 22.5.1 of the Abaqus Analysis User's Guide
- "Hyperelastic behavior in elastomeric foams," Section 22.5.2 of the Abaqus Analysis User's Guide
- "Mullins effect," Section 22.6.1 of the Abaqus Analysis User's Guide
- "Energy dissipation in elastomeric foams," Section 22.6.2 of the Abaqus Analysis User's Guide
- *HYPERELASTIC
- *HYPERFOAM
- *MULLINS EFFECT

### Using biaxial test data to define a hyperelastic material

**Optional parameter:**

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:**

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 (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, $\epsilon_B$.

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 (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, $\epsilon_B$.
3. Nominal lateral strain, $\epsilon_3$. Not needed if the POISSON parameter is specified on the *HYPERELASTIC option or if the *VOLUMETRIC TEST DATA option is used.
4. Temperature, $\theta$.
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, $\epsilon_B$.

3. Nominal transverse strain, $\epsilon_3$. 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 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, $\epsilon_B$.

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

## 2.10 *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.

**Product:** Abaqus/Explicit

**Type:** Model or history data

**Level:** Model, Step

**References:**

- "Fluid exchange definition," Section 11.5.3 of the Abaqus Analysis User's Guide
- "Mechanical contact properties: overview," Section 37.1.1 of the Abaqus Analysis User's Guide
- "Contact blockage," Section 37.1.4 of the Abaqus Analysis User's Guide
- *SURFACE INTERACTION
- *FLUID EXCHANGE ACTIVATION

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

## 2.11 *BOND: Define bonds and bonding properties.

This option is used to define breakable bonds that initially tie two contact boundaries to each other. This option must be used in conjunction with the *SURFACE INTERACTION option.

**Product:** Abaqus/Explicit

**Type:** History data

**Level:** Step

**References:**

- "Breakable bonds," Section 37.1.9 of the Abaqus Analysis User's Guide
- *SURFACE INTERACTION

**There are no parameters associated with this option.**

**Data lines to define spot welds with the time to failure model:**

First line:

1. Name of bonded node set.
2. Maximum uniaxial normal force, $F_f^n$. This value must be nonzero and positive.
3. Maximum uniaxial shear force, $F_f^s$. This value must be nonzero and positive.
4. Initial bead size, $d_b$.
5. Time to failure, $T_f$. If $T_f$ is nonzero, the breakage displacements $u_f^n$ and $u_f^s$ must be left blank.

Repeat this data line as often as necessary to define spot welds using the time to failure model.

**Data lines to define spot welds with the damaged model:**

First line:

1. Name of bonded node set.
2. Maximum uniaxial normal force, $F_f^n$. This value must be nonzero and positive.
3. Maximum uniaxial shear force, $F_f^s$. This value must be nonzero and positive.
4. Initial bead size, $d_b$.
5. Blank space.
6. Normal breakage displacement, $u_f^n$. If $u_f^n$ is nonzero, the time to failure $T_f$ must be left blank.
7. Shear breakage displacement, $u_f^s$ (default value is $u_f^n$). If $u_f^s$ is nonzero, the time to failure $T_f$ must be left blank.

Repeat this data line as often as necessary to define spot welds using the damaged model.

## 2.12 *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/CFD 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:**

- "Defining a model in Abaqus," Section 1.3.1 of the Abaqus Analysis User's Guide
- "Boundary conditions in Abaqus/Standard and Abaqus/Explicit," Section 34.3.1 of the Abaqus Analysis User's Guide
- "Boundary conditions in Abaqus/CFD," Section 34.3.2 of the Abaqus Analysis User's Guide
- "DISP," Section 1.1.4 of the Abaqus User Subroutines Reference Guide
- "VDISP," Section 1.2.1 of the Abaqus User Subroutines Reference Guide
- "Natural frequency extraction," Section 6.3.5 of the Abaqus Analysis User's Guide
- "Node-based submodeling," Section 10.2.2 of the Abaqus Analysis User's Guide
- "Modeling discontinuities as an enriched feature using the extended finite element method," Section 10.7.1 of the Abaqus Analysis User's Guide
- "Defining ALE adaptive mesh domains in Abaqus/Explicit," Section 12.2.2 of the Abaqus Analysis User's Guide

### 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 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," Section 34.1.2 of the Abaqus Analysis User's Guide).

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," Section 6.1.2 of the Abaqus Analysis User's Guide). 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 or an Abaqus/CFD 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," Section 34.1.2 of the Abaqus Analysis User's Guide.

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 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. This parameter is ignored if it is used in the first step of an analysis.

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,” Section 1.2.1 of the Abaqus Analysis User’s Guide, for the syntax of such names.

OP

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.

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 in Abaqus/Standard and Abaqus/Explicit," Section 34.3.1 of the Abaqus Analysis User's Guide, for details. In Abaqus/CFD this type is used to prescribe mesh displacements.

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 Abaqus/Standard and Abaqus/Explicit," Section 34.3.1 of the Abaqus Analysis User's Guide. In this case the default variation is STEP. Velocity histories are not used in Abaqus/CFD.

Set TYPE=ACCELERATION to give an acceleration history. Acceleration histories should not be used in Abaqus/CFD or 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 applies only to Abaqus/Standard and Abaqus/Explicit analyses and 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 in Abaqus/Standard and Abaqus/Explicit," Section 34.3.1 of the Abaqus Analysis User's Guide). 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," Section 1.2.2 of the Abaqus Analysis User's Guide.
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 and Abaqus/Explicit," Section 34.3.1 of the Abaqus Analysis User's Guide). 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," Section 1.2.2 of the Abaqus Analysis User's Guide.
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. In Abaqus/Standard 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," Section 1.2.2 of the Abaqus Analysis User's Guide.
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. In Abaqus/Standard 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 parameter:**

BASE NAME

This parameter is used to define a secondary base and can be used only in a frequency extraction step ("Natural frequency extraction," Section 6.3.5 of the Abaqus Analysis User's Guide). Set this parameter equal to the name of a secondary base ("Dynamic analysis procedures: overview," Section 6.3.1 of the Abaqus Analysis User's Guide). 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," Section 1.2.2 of the Abaqus Analysis User's Guide.
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 linear perturbation procedures," Section 6.1.3 of the Abaqus Analysis User's Guide). 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," Section 1.2.2 of the Abaqus Analysis User's Guide.
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.

## 2.13 *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," Section 23.6.2 of the Abaqus Analysis User's Guide
- *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 "Using the DEPENDENCIES parameter to define field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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, $\sigma_{tu}^{I}$, 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, $\sigma_{t}^{I}$. (Units of $\mathrm{FL}^{-2}$.)

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, $\sigma_t^I$. (Units of $\mathrm{FL}^{-2}$.)
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, $\sigma_{tu}^I$. (Units of $\mathrm{FL}^{-2}$.)
2. Mode I fracture energy, $G_f^I$. (Units of $\mathrm{FL}^{-1}$.)
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.

## 2.14 *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," Section 23.6.2 of the Abaqus Analysis User's Guide
- *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 "Using the DEPENDENCIES parameter to define field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

**Data lines if TYPE=STRAIN (default) is used on the *BRITTLE CRACKING option:**

First line:

1. Direct cracking failure strain, $\left(e_{nn}^{ck}\right)_f$.
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.

## 2.15 *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," Section 23.6.2 of the Abaqus Analysis User's Guide
- *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 "Using the DEPENDENCIES parameter to define field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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 $e^{ck}_{max}$ for the power law shear retention model.

**Data lines if the TYPE=RETENTION FACTOR parameter is included (default):**

First line:

1. Shear retention factor, $\rho$.
2. Crack opening strain, $e^{ck}_{nn}$.

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. $e^{ck}_{max}$.
2. $\boldsymbol{p}$.
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.

## 2.16 *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

**Reference:**

- "Eigenvalue buckling prediction," Section 6.2.3 of the Abaqus Analysis User's Guide

**Optional parameter:**

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.
3. Number of vectors used in the iteration. This number is usually determined by Abaqus/Standard but can be changed 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. If this field is left blank, no minimum is set.

3. Maximum eigenvalue of interest. 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.

## 2.17 *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=$\sigma^0$ option with or without the PINNED parameter.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance

**References:**

- "Frame section behavior," Section 29.4.2 of the Abaqus Analysis User's Guide
- *FRAME SECTION

**There are no parameters associated with this option.**

**Data line to define the buckling strut envelope:**

First (and only) data line:

1. $\xi$, coefficient defining $P_y = \xi\sigma^0 A$ (default value 0.95).
2. $\gamma$, coefficient defining the isotropic hardening slope $\gamma EA$ (default value 0.02).
3. $\alpha_0$, coefficient defining $\alpha = \alpha_0 + \alpha_1 \frac{L}{D}$ (default value 0.03).
4. $\alpha_1$, coefficient defining $\alpha = \alpha_0 + \alpha_1 \frac{L}{D}$ (default value 0.004).
5. $\kappa$, coefficient defining compressive force for discontinuity in buckling envelope (default value 0.28).
6. $\beta$, buckling envelope slope coefficient (default value 0.02).
7. $\zeta$, coefficient defining the force axis intercept point (default value $\min(1.0, \frac{5.8}{\xi}(\frac{t}{D})^{0.7})$).

In the above data line $A$ is the cross-section area, $\sigma^0$ 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.

## 2.18 *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=$\sigma^0$ 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=$\sigma^0$ option, with or without the PINNED parameter.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance

**References:**

- "Frame section behavior," Section 29.4.2 of the Abaqus Analysis User's Guide
- *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.

## 2.19 *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=$\sigma^0$ 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=$\sigma^0$ option.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance, Model

**References:**

- "Frame elements," Section 29.4.1 of the Abaqus Analysis User's Guide
- "Frame section behavior," Section 29.4.2 of the Abaqus Analysis User's Guide
- "Buckling strut response for frame elements," Section 3.9.3 of the Abaqus Theory Guide

**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.

## 2.20 *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

**Reference:**

- "Explicit dynamic analysis," Section 6.3.3 of the Abaqus Analysis User's Guide

**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.

**3. C**

## 3.1 *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.

**Product:** Abaqus/Aqua

**Type:** History data

**Level:** Step

**Reference:**

- "Abaqus/Aqua analysis," Section 6.11.1 of the Abaqus Analysis User's Guide

**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.

## 3.2 *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," Section 23.3.2 of the Abaqus Analysis User's Guide
- *CAP PLASTICITY
- *CAP HARDENING
- "CREEP," Section 1.1.1 of the Abaqus User Subroutines Reference Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

LAW

Set LAW=STRAIN (default) to choose a strain hardening power law.
Set LAW=TIME to choose a time hardening power law.
Set LAW=SINGHM to choose a Singh-Mitchell type law.
Set LAW=USER to input the creep law using user subroutine **CREEP**.

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 or LAW=SINGHM 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. 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. $\alpha$. (Units of $F^{-1}L^{2}$.)
3. $m$.
4. $t_1$. (Units of T.)
5. Temperature.
6. First field variable.
7. 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 set of data lines as often as necessary to define the dependence of the creep constants on temperature and other predefined field variables.

## 3.3 *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," Section 23.3.2 of the Abaqus Analysis User's Guide
- *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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 3.4 *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," Section 23.3.2 of the Abaqus Analysis User's Guide
- *CAP HARDENING
- *CAP CREEP

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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, $\beta$, 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 $0.0001 \leq R \leq 1000.0$).
4. Initial cap yield surface position on the volumetric inelastic strain axis, $\varepsilon^{in}_{vol}|_0$.

5. Transition surface radius parameter, $\alpha$. 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, $\alpha$ 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 $0.778 \leq K \leq 1.0$. If this field is left blank or a value of 0.0 is entered, the default of 1.0 is used. If creep 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.

## 3.5 *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," Section 11.5.2 of the Abaqus Analysis User's Guide
- "Inflator definition," Section 11.5.4 of the Abaqus Analysis User's Guide
- *FLUID BEHAVIOR
- *FLUID CAVITY

**Required parameter:**

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 parameter:**

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 JMOLE$^{-1}$K$^{-4}$.)
5. $\tilde{e}$, the fifth molar heat capacity coefficient. (Units of JMOLE$^{-1}$K.)

**Data lines for TYPE=TABULAR:**

First line:

1. Molar heat capacity, $\tilde{c}_p$, at constant pressure. (Units of JMOLE$^{-1}$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.

## 3.6 *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," Section 23.2.10 of the Abaqus Analysis User's Guide
- *CAST IRON PLASTICITY
- *CAST IRON TENSION HARDENING

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

**Data lines to define compression hardening:**

First line:

1. Yield stress in compression, $\sigma_c$.
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.

## 3.7 *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," Section 23.2.10 of the Abaqus Analysis User's Guide
- *CAST IRON COMPRESSION HARDENING
- *CAST IRON TENSION HARDENING

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

**Data lines to define the plastic "Poisson's ratio":**

First line:

1. Value of the plastic "Poisson's ratio," $\nu_{pl}$, where $-1.0 < \nu_{pl} \leq 0.5$. (Dimensionless.) If no value is provided, a default value of $0.04$ is assumed.
2. Temperature, $\theta$.
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 $\nu_{pl}$ on temperature and field variables.

## 3.8 *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," Section 23.2.10 of the Abaqus Analysis User's Guide
- *CAST IRON COMPRESSION HARDENING
- *CAST IRON PLASTICITY

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

**Data lines to define tension hardening:**

First line:

1. Yield stress in uniaxial tension, $\sigma_t$.
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.

## 3.9 *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," Section 41.1.1 of the Abaqus Analysis User's Guide
- *SURFACE
- *SURFACE PROPERTY

**Required parameter:**

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.

## 3.10 *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

**Reference:**

- "Piezoelectric analysis," Section 6.7.2 of the Abaqus Analysis User's Guide

**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," Section 6.1.2 of the Abaqus Analysis User's Guide).

OP

Set OP=MOD (default) for existing *CECHARGEs to remain, with this option modifying existing electric charges or defining additional electric charges.

Set OP=NEW if all existing *CECHARGEs 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.

## 3.11 *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 and coupled thermal-electrical-structural analyses.

**Products:** Abaqus/Standard Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Load module

**References:**

- "Coupled thermal-electrical analysis," Section 6.7.3 of the Abaqus Analysis User's Guide
- "Fully coupled thermal-electrical-structural analysis," Section 6.7.4 of the Abaqus Analysis User's Guide

**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," Section 34.1.2 of the Abaqus Analysis User's Guide). 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," Section 6.1.2 of the Abaqus Analysis User's Guide).

OP

Set OP=MOD (default) for existing *CECURRENTs to remain, with this option modifying existing concentrated currents or defining additional concentrated currents.

Set OP=NEW if all existing *CECURRENTs 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.

## 3.12 *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 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," Section 29.3.7 of the Abaqus Analysis User's Guide
- "Meshed beam cross-sections," Section 10.6.1 of the Abaqus Analysis User's Guide

**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.

## 3.13 *CFD: Computational fluid dynamic analysis.

This option is used to control the transport of momentum, energy, temperature, species, and other scalar variables in both transient and steady-state fluid flow problems.

**Products:** Abaqus/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Step module

**Reference:**

- "Incompressible fluid dynamic analysis," Section 6.6.2 of the Abaqus Analysis User's Guide

**Required parameter:**

INCOMPRESSIBLE NAVIER STOKES

Include this parameter to specify an incompressible flow analysis.

**Optional parameters:**

ENERGY EQUATION

Set ENERGY EQUATION=NO ENERGY (default) to specify an isothermal flow problem.

Set ENERGY EQUATION=TEMPERATURE to specify a thermal (heat) transport problem with temperature as the primary transport scalar variable.

INCREMENTATION

This parameter is valid only for a transient flow problem.

Set INCREMENTATION=FIXED CFL (default) to set a fixed Courant-Friedrichs-Lewy (CFL) number for time marching. Abaqus/CFD automatically determines the stable time step based on the specified value of the CFL number.

Set INCREMENTATION=FIXED STEP SIZE to set a fixed time step for time marching.

STEADY STATE

This parameter is valid only for a steady-state flow problem.

**Data lines for INCREMENTATION=FIXED CFL:**

First line:

1. Suggested initial time increment (default=0.01). If the estimated value of the CFL number is greater than the specified CFL or the maximum allowable time increment, the specified value will be overridden by a time increment determined automatically by Abaqus/CFD for numerical stability purposes.
2. Time period of the step (default=1).
3. Scale factor for increasing the stable time step (default=0.025).
4. Suggested CFL number (default=0.45).
5. Interval to check stable time step (default=1).
6. Maximum allowable time increment (default value is specified as the time period of the step).

Second line:

1. Specified tolerance for the pressure projection scheme used for the initial divergence-free projection and for pressure equation subcycling near a steady-state solution (default=$1 \times 10^{-10}$).
2. The time weight $\theta$ for the viscous/diffusive terms in the momentum and scalar transport equations ($0 \leq \theta \leq 1$). Enter 0.5 (default) for the second-order accurate Crank-Nicolson method, 0.6667 for the Galerkin method, or 1 for the first-order backward-Euler method.
3. Blank space.
4. The time weight $\theta$ for the advective terms in the momentum and scalar transport equations ($0 \leq \theta \leq 1$). Enter 0.5 (default) for the second-order accurate Crank-Nicolson method, 0.6667 for the Galerkin method, or 1 for the first-order backward-Euler method.
5. The time weight $\theta$ for the treatment of boundary conditions ($0 \leq \theta \leq 1$). Enter 0.5 (default) for the second-order accurate Crank-Nicolson method, 0.6667 for the Galerkin method, or 1 for the first-order backward-Euler method.

**Data lines for INCREMENTATION=FIXED STEP SIZE:**

First line:

1. Time increment (default=0.01).
2. Time period of the step (default=1).

Second line:

1. Specified tolerance for the pressure projection scheme used for the initial divergence-free projection and for pressure equation subcycling near a steady-state solution (default=$1 \times 10^{-10}$).
2. The time weight $\theta$ for the viscous/diffusive terms in the momentum and scalar transport equations ($0 \leq \theta \leq 1$). Enter 0.5 (default) for the second-order accurate Crank-Nicolson method, 0.6667 for the Galerkin method, or 1 for the first-order backward-Euler method.
3. Blank space.

4. The time weight $\theta$ for the advective terms in the momentum and scalar transport equations ($0 \leq \theta \leq 1$). Enter 0.5 (default) for the second-order accurate Crank-Nicolson method, 0.6667 for the Galerkin method, or 1 for the first-order backward-Euler method.
5. The time weight $\theta$ for the treatment of boundary conditions ($0 \leq \theta \leq 1$). Enter 0.5 (default) for the second-order accurate Crank-Nicolson method, 0.6667 for the Galerkin method, or 1 for the first-order backward-Euler method.

**Data line for steady-state flow:**

First (and only) line:

1. Number of nonlinear iterations (increments). The default is 10000.

## 3.14 *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 fully coupled thermal-stress analysis. In Abaqus/Standard it is also used to provide film coefficients and sink temperatures at any node in the model 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:** Interaction module

**References:**

- "Thermal loads," Section 34.4.4 of the Abaqus Analysis User's Guide
- "FILM," Section 1.1.6 of the Abaqus User Subroutines Reference Guide

**Optional parameters:**

AMPLITUDE

Set this parameter equal to the name of the *AMPLITUDE option that gives the variation of the sink temperature, $\theta^0$, 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," Section 6.1.2 of the Abaqus Analysis User's Guide). 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 (which are available only in Abaqus/Standard), the sink temperature amplitude is defined in user subroutine **FILM** and AMPLITUDE references are used only to modify the sink temperature passed into the user subroutine.

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 (which are available only in Abaqus/Standard), the film coefficient amplitude is defined in user subroutine **FILM** and FILM AMPLITUDE references are used only to modify the film coefficient passed into the user subroutine.

OP

Set OP=MOD (default) for existing *CFILMs to remain, with this option modifying existing films or defining additional films.

Set OP=NEW if all existing *CFILMs applied to the model should be removed.

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.

USER

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to indicate that any nonzero film coefficients prescribed through this option will be defined in user subroutine **FILM**. If this parameter is used, any film coefficient and sink temperature values defined by the data lines of the option (and possibly modified by the AMPLITUDE and FILM AMPLITUDE parameters) are ignored and can be redefined in subroutine **FILM**.

**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, $\theta^0$. (Units of $\theta$.) For nonuniform film coefficients the sink temperature must be defined in user subroutine **FILM**. If given, this value will be modified by the 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**. 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.

## 3.15 *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," Section 6.8.1 of the Abaqus Analysis User's Guide
- "Geostatic stress state," Section 6.8.2 of the Abaqus Analysis User's Guide
- "Pore fluid flow," Section 34.4.7 of the Abaqus Analysis User's Guide
- "Modeling discontinuities as an enriched feature using the extended finite element method," Section 10.7.1 of the Abaqus Analysis User's Guide

**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," Section 6.1.2 of the Abaqus Analysis User's Guide).

OP

Set OP=MOD (default) for existing *CFLOWs to remain, with this option modifying existing concentrated flows or defining additional concentrated flows.

Set OP=NEW if all existing *CFLOWs 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.

**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.

## 3.16 *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

**Reference:**

- "Thermal loads," Section 34.4.4 of the Abaqus Analysis User's Guide

**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," Section 34.1.2 of the Abaqus Analysis User's Guide).

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," Section 6.1.2 of the Abaqus Analysis User's Guide). 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 *CFLUXs to remain, with this option modifying existing fluxes or defining additional fluxes.

Set OP=NEW if all existing *CFLUXs applied to the model should be removed.

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.

## 3.17 *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:**

- "Mechanical contact properties: overview," Section 37.1.1 of the Abaqus Analysis User's Guide
- "Frictional behavior," Section 37.1.5 of the Abaqus Analysis User's Guide
- "Connector behavior," Section 31.2.1 of the Abaqus Analysis User's Guide
- *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," Section 34.1.2 of the Abaqus Analysis User's Guide).

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," Section 6.1.2 of the Abaqus Analysis User's Guide). 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.**

## 3.18 *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," Section 21.1.2 of the Abaqus Analysis User's Guide). If this option is not specified, Abaqus computes the characteristic element length using the geometric mean–based definition.

**Product:** Abaqus/Explicit

**Type:** Model data

**Level:** Model

**Reference:**

- "VUCHARLENGTH," Section 1.2.11 of the Abaqus User Subroutines Reference Guide

**Optional parameter:**

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.

## 3.19 *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," Section 23.3.4 of the Abaqus Analysis User's Guide
- *CLAY PLASTICITY

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

**Data lines to define hardening for Cam-clay plasticity:**

First line:

1. Value of the hydrostatic pressure stress at yield, $p_c$ .
2. Volumetric plastic strain (positive for compression ).
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.

## 3.20 *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," Section 23.3.4 of the Abaqus Analysis User's Guide
- *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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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, $\lambda$ (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, $\varepsilon^{pl}_{vol}|_0$, corresponding to $p_c|_0$ 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. $\beta$, 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.

## 3.21 *CLEARANCE: Specify a particular initial clearance value and a contact direction for the slave nodes on a surface.

This option is used to define initial clearance values and/or contact directions precisely at contact slave 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 (*CONTACT PAIR, SMALL SLIDING). 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," Section 39.1.2 of the Abaqus Analysis User's Guide
- "Common difficulties associated with contact modeling using contact pairs in Abaqus/Explicit," Section 39.2.2 of the Abaqus Analysis User's Guide
- "Adjusting initial surface positions and specifying initial clearances in Abaqus/Standard contact pairs," Section 36.3.5 of the Abaqus Analysis User's Guide
- "Adjusting initial surface positions and specifying initial clearances for contact pairs in Abaqus/Explicit," Section 36.5.4 of the Abaqus Analysis User's Guide

**Required parameters:**

CPSET

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the name of the contact pair set name to associate these clearance data with the appropriate contact pairs.

MASTER

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the name of the master surface of the contact pair.

SLAVE

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the name of the slave surface of the contact pair.

**Required, mutually exclusive parameters:**

TABULAR

Include this parameter to specify the slave 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 slave 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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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.

**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 slave node and the corresponding closest point on the master surface. The specification of the normal is optional. If the normal is given, it should be in the direction of

the master surface's outward normal. If the normal is not given, Abaqus calculates it from the geometry of the master surface (see "Common difficulties associated with contact modeling in Abaqus/Standard," Section 39.1.2 of the Abaqus Analysis User's Guide, and "Common difficulties associated with contact modeling using contact pairs in Abaqus/Explicit," Section 39.2.2 of the Abaqus Analysis User's Guide).

**Data lines if both the TABULAR parameter and the BOLT parameter are included without the INPUT parameter (see Figure 3.21–1 and Figure 3.21–2):**

First line:

1. Half-thread angle, α, (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 "Adjusting initial surface positions and specifying initial clearances in Abaqus/Standard contact pairs," Section 36.3.5 of the Abaqus Analysis User's Guide, and "Specifying initial clearance values precisely" in "Adjusting initial surface positions and specifying initial clearances for contact pairs in Abaqus/Explicit," Section 36.5.4 of the Abaqus Analysis User's Guide).

**To define a clearance value by using the VALUE parameter:**

No data lines are used with this option when the VALUE parameter is specified.

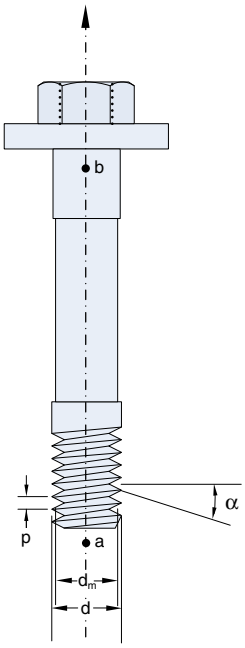



**Figure 3.21–1** Thread geometry.

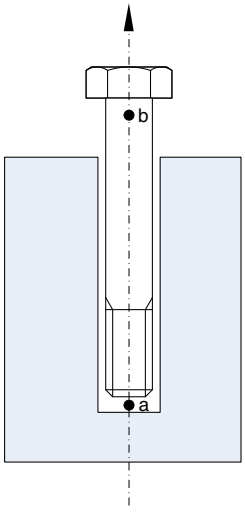



**Figure 3.21–2** Points *a* and *b* on the centerline of the bolt and bolt-hole assembly.

## 3.22 *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 a fluid reference pressure for incompressible flow in an Abaqus/CFD analysis and to specify concentrated buoyancy, drag, and inertia loads in an Abaqus/Aqua analysis.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE Abaqus/Aqua

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Load module

**References:**

- "Concentrated loads," Section 34.4.2 of the Abaqus Analysis User's Guide
- "Abaqus/Aqua analysis," Section 6.11.1 of the Abaqus Analysis User's Guide
- "Analysis of models that exhibit cyclic symmetry," Section 10.4.3 of the Abaqus Analysis User's Guide
- "Defining ALE adaptive mesh domains in Abaqus/Explicit," Section 12.2.2 of the Abaqus Analysis User's Guide

### 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," Section 6.1.2 of the Abaqus Analysis User's Guide). 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," Section 6.3.11 of the Abaqus Analysis User's Guide), 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 *CLOADs to remain, with this option modifying existing concentrated loads or defining additional concentrated loads.

Set OP=NEW if all existing *CLOADs 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 a fluid reference pressure in Abaqus/CFD

**Optional parameters:**

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the magnitude of the fluid reference pressure 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,” Section 6.1.2 of the Abaqus Analysis User’s Guide).

OP

Set OP=MOD (default) for existing *CLOADs to remain, with this option modifying existing fluid reference pressures.

Set OP=NEW if all existing *CLOADs applied to the model should be removed.

**Data lines to define a fluid reference pressure for incompressible flow in Abaqus/CFD:**

First line:

1. Node number or node set label.
2. Hydrostatic pressure type label, HP.
3. Reference magnitude for load.

Repeat this data line as often as necessary to define reference pressures; however, only the last specified hydrostatic pressure load in a given fluid domain is applied.

## 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," Section 6.1.2 of the Abaqus Analysis User's Guide).

OP

Set OP=MOD (default) for existing *CLOADs to remain, with this option modifying existing concentrated loads or defining additional concentrated loads.

Set OP=NEW if all existing *CLOADs 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:

5. $X$-direction cosine of the outward normal to the exposed area, pointing into the fluid, in the initial configuration.
6. $Y$-direction cosine of the outward normal to the exposed area, pointing into the fluid, in the initial configuration.
7. $Z$-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" in "Abaqus/Aqua analysis," Section 6.11.1 of the Abaqus Analysis User's Guide. Gravity waves do not affect the buoyancy loading when any external fluid property is overridden.

8. Density of the fluid outside the element.
9. Free surface elevation of the fluid outside the element.
10. 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, $\Delta A$.
5. Drag coefficient, $C_n$.
6. Structural velocity factor, $\alpha_R$. 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.

## 3.23 *COHESIVE BEHAVIOR: Specify surface-based cohesive behavior properties.

This option is used to define surface-based 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:**

- "Surface-based cohesive behavior," Section 37.1.10 of the Abaqus Analysis User's Guide
- *SURFACE INTERACTION

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

ELIGIBILITY

Set ELIGIBILITY=CURRENT CONTACTS (default) to define cohesive behavior not only for all nodes of the slave surface that are in contact with the master surface at the start of a step, but also for slave nodes that are not initially in contact but may come in contact during the course of a step.

Set ELIGIBILITY=ORIGINAL CONTACTS to restrict cohesive behavior to only those nodes of the slave surface that are in contact with the master surface at the start of a step.

Set ELIGIBILITY=SPECIFIED CONTACTS to restrict cohesive behavior to a subset of slave nodes defined using *INITIAL CONDITIONS, TYPE=CONTACT. This parameter value is available only for Abaqus/Standard analyses.

REPEATED CONTACTS

Include this parameter to modify the default post-failure behavior when progressive damage has been defined. By default, cohesive behavior is not enforced for nodes on the slave 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 slave 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. $K_{nn}$.
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. $K_{nn}$.
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.

## 3.24 *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:**

- "Cohesive elements: overview," Section 32.5.1 of the Abaqus Analysis User's Guide
- "Defining the constitutive response of cohesive elements using a continuum approach," Section 32.5.5 of the Abaqus Analysis User's Guide

**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," Section 27.1.4 of the Abaqus Analysis User's Guide). 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, $\mu$, for viscous regularization.

ORIENTATION

Set this parameter equal to the name given for the *ORIENTATION option ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Guide) 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.

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.

## 3.25 *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," Section 22.7.1 of the Abaqus Analysis User's Guide
- *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 $1 - \sum_{i=1}^{N} \bar{g}_i^P = g_R(\infty)$.

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 $1 - \sum_{i=1}^{N} \bar{k}_i^P = k_R(\infty)$.

**Data lines to specify creep test data:**

First line:

1. Normalized shear compliance $j_S(t)$, $(j_S(t) \geq 1)$.
2. Normalized volumetric (bulk) compliance $j_K(t)$, $(j_K(t) \geq 1)$.

3. Time $t$, $(t > 0)$.

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 $g_R(t)$, $(0 \leq g_R(t) \leq 1)$.
2. Normalized volumetric (bulk) modulus $k_R(t)$, $(0 \leq k_R(t) \leq 1)$.
3. Time $t$, $(t > 0)$.

Repeat the above data line as often as necessary to give the modulus-time data.

## 3.26 *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

**Reference:**

- "Complex eigenvalue extraction," Section 6.3.6 of the Abaqus Analysis User's Guide

**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.

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 written to the output database (`.odb`) file. 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 ($S \geq 0$). The eigenvalues with the imaginary part closest to this point are extracted. The default value is zero.

## 3.27 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Material damping," Section 26.1.1 of the Abaqus Analysis User's Guide
- "Dynamic analysis procedures: overview," Section 6.3.1 of the Abaqus Analysis User's Guide

**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, $\xi_M$, for mass matrices included in this element or element set.
3. Fraction of critical damping, $\xi_K$, 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.

## 3.28 *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," Section 23.6.1 of the Abaqus Analysis User's Guide; "Defining rebar as an element property," Section 2.2.4 of the Abaqus Analysis User's Guide; 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," Section 2.2.4 of the Abaqus Analysis User's Guide).

The *BRITTLE CRACKING option is used to analyze concrete in an Abaqus/Explicit analysis (see "Cracking model for concrete," Section 23.6.2 of the Abaqus Analysis User's Guide).

**Products:** Abaqus/Standard Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "Concrete smeared cracking," Section 23.6.1 of the Abaqus Analysis User's Guide
- *TENSION STIFFENING
- *SHEAR RETENTION
- *FAILURE RATIOS

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 3.29 *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 *CONCRETE TENSION DAMAGE option can be used to specify tensile stiffness degradation damage.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "Concrete damaged plasticity," Section 23.6.3 of the Abaqus Analysis User's Guide
- *CONCRETE DAMAGED PLASTICITY
- *CONCRETE TENSION STIFFENING
- *CONCRETE COMPRESSION HARDENING
- *CONCRETE TENSION DAMAGE

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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 $w_t$ $(0 \leq w_t \leq 1)$ 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.

## 3.30 *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 *CONCRETE TENSION DAMAGE and/or *CONCRETE COMPRESSION DAMAGE options can be used to specify tensile and/or compressive stiffness degradation damage.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "Concrete damaged plasticity," Section 23.6.3 of the Abaqus Analysis User's Guide
- *CONCRETE DAMAGED PLASTICITY
- *CONCRETE TENSION STIFFENING
- *CONCRETE TENSION DAMAGE
- *CONCRETE COMPRESSION DAMAGE

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

**Data lines to define compressive hardening:**

First line:

1. Yield stress in compression, $\sigma_c$. (Units of $\mathrm{FL}^{-2}$.)
2. Inelastic (crushing) strain, $\tilde{\varepsilon}_c^{in}$.
3. Inelastic (crushing) strain rate, $\dot{\tilde{\varepsilon}}_c^{in}$. (Units of $\mathrm{T}^{-1}$.)
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, $\sigma_{c0}$.

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.

## 3.31 *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 *CONCRETE TENSION DAMAGE and/or the *CONCRETE COMPRESSION DAMAGE options can be used to specify tensile and/or compressive stiffness degradation damage.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "Concrete damaged plasticity," Section 23.6.3 of the Abaqus Analysis User's Guide
- *CONCRETE TENSION STIFFENING
- *CONCRETE COMPRESSION HARDENING
- *CONCRETE TENSION DAMAGE
- *CONCRETE COMPRESSION DAMAGE

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

**Data lines to define concrete damaged plasticity flow potential, yield surface, and viscosity parameters:**

First line:

1. Dilation angle, $\psi$, in the $p$–$q$ plane. Give the value in degrees.

2. Flow potential eccentricity, $\epsilon$. 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. $\sigma_{b0}/\sigma_{c0}$, 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, $q_{(\mathrm{TM})}$, to that on the compressive meridian, $q_{(\mathrm{CM})}$, at initial yield for any given value of the pressure invariant $\boldsymbol{p}$ 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, $\mu$, 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.

## 3.32 *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 *CONCRETE COMPRESSION DAMAGE option can be used to specify compressive stiffness degradation damage.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "Concrete damaged plasticity," Section 23.6.3 of the Abaqus Analysis User's Guide
- *CONCRETE DAMAGED PLASTICITY
- *CONCRETE TENSION STIFFENING
- *CONCRETE COMPRESSION HARDENING
- *CONCRETE COMPRESSION DAMAGE

**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 $w_c$ $(0 \leq w_c \leq 1)$ 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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 3.33 *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 *CONCRETE TENSION DAMAGE and/or *CONCRETE COMPRESSION DAMAGE options can be used to specify tensile and/or compressive stiffness degradation damage.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "Concrete damaged plasticity," Section 23.6.3 of the Abaqus Analysis User's Guide
- *CONCRETE DAMAGED PLASTICITY
- *CONCRETE COMPRESSION HARDENING
- *CONCRETE TENSION DAMAGE
- *CONCRETE COMPRESSION DAMAGE

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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, $\sigma_{t0}$, and the fracture energy, $G_f$.

**Data lines if the TYPE=STRAIN parameter is included (default):**

First line:

1. Remaining direct stress after cracking, $\sigma_t$. (Units of $\mathrm{FL}^{-2}$.)
2. Direct cracking strain, $\tilde{\varepsilon}_t^{ck}$.
3. Direct cracking strain rate, $\dot{\tilde{\varepsilon}}_t^{ck}$. (Units of $\mathrm{T}^{-1}$.)
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, $\sigma_{t0}$.

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, $\sigma_t$. (Units of $\mathrm{FL}^{-2}$.)
2. Direct cracking displacement, $u_t^{ck}$. (Units of L.)
3. Direct cracking displacement rate, $\dot{u}_t^{ck}$. (Units of $\mathrm{LT}^{-1}$.)
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, $\sigma_{t0}$. (Units of $\mathrm{FL}^{-2}$.)
2. Fracture energy, $G_f$. (Units of $\mathrm{FL}^{-1}$.)
3. Direct cracking displacement rate, $\dot{u}_t^{ck}$. (Units of $\mathrm{LT}^{-1}$.)
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.

## 3.34 *CONDUCTIVITY: Specify thermal conductivity.

This option is used to specify a material's thermal conductivity.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**Reference:**

- "Conductivity," Section 26.2.2 of the Abaqus Analysis User's Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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 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. Abaqus/CFD supports only isotropic conductivity without field-dependent variants.

**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 $\mathrm{JT^{-1}L^{-1}}\ \theta^{-1}$.)
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 $\mathrm{JT^{-1}L^{-1}}\theta^{-1}$.)
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.

## 3.35 *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," Section 31.2.1 of the Abaqus Analysis User's Guide
- *CONNECTOR SECTION

**Required parameter:**

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.**

## 3.36 *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," Section 31.2.1 of the Abaqus Analysis User's Guide
- *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.

## 3.37 *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-type library," Section 31.1.5 of the Abaqus Analysis User's Guide
- "Connector behavior," Section 31.2.1 of the Abaqus Analysis User's Guide
- "Connector damage behavior," Section 31.2.7 of the Abaqus Analysis User's Guide
- *CONNECTOR BEHAVIOR
- *CONNECTOR DAMAGE INITIATION
- *CONNECTOR POTENTIAL

**Required parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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," Section 31.2.1 of the Abaqus Analysis User's Guide, 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," Section 31.2.1 of the Abaqus Analysis User's Guide, for a description of the connector relative motions.

2. Exponential law parameter, $\alpha$ (see "Connector damage behavior," Section 31.2.7 of the Abaqus Analysis User's Guide).
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," Section 31.2.1 of the Abaqus Analysis User's Guide, 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.

## 3.38 *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-type library," Section 31.1.5 of the Abaqus Analysis User's Guide
- "Connector behavior," Section 31.2.1 of the Abaqus Analysis User's Guide
- "Connector damage behavior," Section 31.2.7 of the Abaqus Analysis User's Guide
- *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-type library," Section 31.1.5 of the Abaqus Analysis User's Guide, 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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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" in "Connector plastic behavior," Section 31.2.6 of the Abaqus Analysis User's Guide, 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.

## 3.39 *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," Section 31.2.1 of the Abaqus Analysis User's Guide
- "Connector damping behavior," Section 31.2.3 of the Abaqus Analysis User's Guide
- *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 steady-state dynamic direct and subspace projection analyses and to steady-state and transient mode-based analyses that support nondiagonal damping in Abaqus/Standard. If TYPE=STRUCTURAL, only linear damping behavior is permitted.

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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 $\text{FTL}^{-1}$.)
2. $C_{12}$. (Units of $\text{FTL}^{-1}$.)
3. $C_{22}$. (Units of $\text{FTL}^{-1}$.)
4. $C_{13}$. (Units of $\text{FTL}^{-1}$.)
5. $C_{23}$. (Units of $\text{FTL}^{-1}$.)
6. $C_{33}$. (Units of $\text{FTL}^{-1}$.)
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 $\mathrm{FTL}^{-1}$.)
2. $C_{12}$. (Units of $\mathrm{FTL}^{-1}$.)
3. $C_{22}$. (Units of $\mathrm{FTL}^{-1}$.)
4. $C_{13}$. (Units of $\mathrm{FTL}^{-1}$.)
5. $C_{23}$. (Units of $\mathrm{FTL}^{-1}$.)
6. $C_{33}$. (Units of $\mathrm{FTL}^{-1}$.)

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 $\text{FTL}^{-1}$.)

2. $C_{21}$. (Units of $\text{FTL}^{-1}$.)
3. $C_{31}$. (Units of $\text{FTL}^{-1}$.)
4. $C_{41}$. (Units of FT.)
5. $C_{51}$. (Units of FT.)
6. $C_{61}$. (Units of FT.)
7. $C_{12}$. (Units of $\text{FTL}^{-1}$.)
8. $C_{22}$. (Units of $\text{FTL}^{-1}$.)

Second line:

1. $C_{32}$. (Units of $\text{FTL}^{-1}$.)
2. $C_{42}$. (Units of FT.)
3. $C_{52}$. (Units of FT.)
4. $C_{62}$. (Units of FT.)
5. $C_{13}$. (Units of $\text{FTL}^{-1}$.)
6. $C_{23}$. (Units of $\text{FTL}^{-1}$.)
7. $C_{33}$. (Units of $\text{FTL}^{-1}$.)
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 $\text{FTL}^{-1}$.)
2. $C_{21}$. (Units of $\text{FTL}^{-1}$.)
3. $C_{31}$. (Units of $\text{FTL}^{-1}$.)
4. $C_{41}$. (Units of FT.)
5. $C_{51}$. (Units of FT.)
6. $C_{61}$. (Units of FT.)
7. $C_{12}$. (Units of $\text{FTL}^{-1}$.)
8. $C_{22}$. (Units of $\text{FTL}^{-1}$.)

Second line:

1. $C_{32}$. (Units of $\text{FTL}^{-1}$.)
2. $C_{42}$. (Units of FT.)
3. $C_{52}$. (Units of FT.)
4. $C_{62}$. (Units of FT.)
5. $C_{13}$. (Units of $\text{FTL}^{-1}$.)

6. $C_{23}$. (Units of $\text{FTL}^{-1}$.)
7. $C_{33}$. (Units of $\text{FTL}^{-1}$.)
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. $s_{66}$.

## 3.40 *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," Section 31.2.1 of the Abaqus Analysis User's Guide
- "Connector functions for coupled behavior," Section 31.2.4 of the Abaqus Analysis User's Guide
- *CONNECTOR BEHAVIOR
- *CONNECTOR FRICTION
- *CONNECTOR POTENTIAL

**Required parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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 ($\alpha_1$) that multiplies the first component identified on the first data line.
2. Scaling constant ($\alpha_2$) 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 ($\alpha_1$) that multiplies the first component identified on the second data line.
2. Scaling constant ($\alpha_2$) 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.

## 3.41 *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," Section 31.2.1 of the Abaqus Analysis User's Guide
- "Connector elastic behavior," Section 31.2.2 of the Abaqus Analysis User's Guide
- *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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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 $\mathrm{FL}^{-1}$.)
2. $D_{21}$. (Units of $\mathrm{FL}^{-1}$.)
3. $D_{31}$. (Units of $\mathrm{FL}^{-1}$.)
4. $D_{41}$. (Units of F.)
5. $D_{51}$. (Units of F.)
6. $D_{61}$. (Units of F.)
7. $D_{12}$. (Units of $\mathrm{FL}^{-1}$.)
8. $D_{22}$. (Units of $\mathrm{FL}^{-1}$.)

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 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.

## 3.42 *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," Section 31.2.1 of the Abaqus Analysis User's Guide
- "Connection-type library," Section 31.1.5 of the Abaqus Analysis User's Guide
- *CONNECTOR BEHAVIOR

**Required parameter:**

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-type library," Section 31.1.5 of the Abaqus Analysis User's Guide, for connector components of relative motion definitions.

**Optional parameter:**

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.

## 3.43 *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-type library," Section 31.1.5 of the Abaqus Analysis User's Guide
- "Connector behavior," Section 31.2.1 of the Abaqus Analysis User's Guide
- "Connector friction behavior," Section 31.2.5 of the Abaqus Analysis User's Guide
- *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-type library," Section 31.1.5 of the Abaqus Analysis User's Guide.

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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-type library," Section 31.1.5 of the Abaqus Analysis User's Guide.

2. Second friction parameter.
3. Etc., up to as many friction parameters discussed in "Connection-type library," Section 31.1.5 of the Abaqus Analysis User's Guide.

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.

## 3.44 *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," Section 31.2.1 of the Abaqus Analysis User's Guide
- "Connector plastic behavior," Section 31.2.6 of the Abaqus Analysis User's Guide
- "Models for metals subjected to cyclic loading," Section 23.2.2 of the Abaqus Analysis User's Guide
- *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 $\gamma$ 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," Section 31.2.1 of the Abaqus Analysis User's Guide) 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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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 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=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 $\gamma$. Set $\gamma$=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.

## 3.45 *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

**Reference:**

- "Connector actuation," Section 31.1.3 of the Abaqus Analysis User's Guide

**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," Section 6.1.2 of the Abaqus Analysis User's Guide). 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," Section 6.3.11 of the Abaqus Analysis User's Guide), 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 LOADs to remain, with this option modifying existing connector loads or defining additional connector loads.

Set OP=NEW if all existing *CONNECTOR LOADs applied to the model should be removed. New connector loads can be defined.

**Optional, mutally 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.

## 3.46 *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," Section 31.2.1 of the Abaqus Analysis User's Guide
- "Connection-type library," Section 31.1.5 of the Abaqus Analysis User's Guide
- *CONNECTOR BEHAVIOR

**Required parameter:**

COMPONENT

Set this parameter equal to the component number on which a locking criterion is based. See "Connection-type library," Section 31.1.5 of the Abaqus Analysis User's Guide, 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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 3.47 *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-type library," Section 31.1.5 of the Abaqus Analysis User's Guide
- "DISP," Section 1.1.4 of the Abaqus User Subroutines Reference Guide

**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," Section 34.1.2 of the Abaqus Analysis User's Guide).

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," Section 6.1.2 of the Abaqus Analysis User's Guide). 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," Section 34.1.2 of the Abaqus Analysis User's Guide.

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-type library," Section 31.1.5 of the Abaqus Analysis User's Guide, 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.

## 3.48 *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," Section 31.2.1 of the Abaqus Analysis User's Guide
- "Connector plastic behavior," Section 31.2.6 of the Abaqus Analysis User's Guide
- *CONNECTOR BEHAVIOR
- *CONNECTOR HARDENING
- *CONNECTOR POTENTIAL

**Optional parameter:**

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.**

## 3.49 *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-type library," Section 31.1.5 of the Abaqus Analysis User's Guide
- "Connector behavior," Section 31.2.1 of the Abaqus Analysis User's Guide
- "Connector functions for coupled behavior," Section 31.2.4 of the Abaqus Analysis User's Guide
- *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, $\beta$. $\beta$ 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 $\alpha$. 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). 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.

## 3.50 *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," Section 31.1.2 of the Abaqus Analysis User's Guide
- "Connection-type library," Section 31.1.5 of the Abaqus Analysis User's Guide
- *CONNECTOR BEHAVIOR

**Required parameter:**

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," Section 27.1.4 of the Abaqus Analysis User's Guide) 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, $\mu$, 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-type library," Section 31.1.5 of the Abaqus Analysis User's Guide. 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). In Abaqus/Standard the default value is 0.0. In Abaqus/Explicit the contact angle is computed automatically if it is not specified.

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, $\beta_s$ (default value is 1.0).

Omit the third line if no data are specified.

## 3.51 *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," Section 31.2.1 of the Abaqus Analysis User's Guide
- *CONNECTOR BEHAVIOR

**Required parameter:**

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.

## 3.52 *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," Section 31.2.1 of the Abaqus Analysis User's Guide
- "Connector uniaxial behavior," Section 31.2.10 of the Abaqus Analysis User's Guide
- *CONNECTOR BEHAVIOR
- *LOADING DATA
- *UNLOADING DATA

**Required parameter:**

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.**

## 3.53 *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.*

**Product:** Abaqus/Standard

**Type:** Model or history data

**Level:** Model, Step

**References:**

- "Overconstraint checks," Section 35.6.1 of the Abaqus Analysis User's Guide
- "Connectors: overview," Section 31.1.1 of the Abaqus Analysis User's Guide
- "Mesh tie constraints," Section 35.3.1 of the Abaqus Analysis User's Guide
- "Common difficulties associated with contact modeling in Abaqus/Standard," Section 39.1.2 of the Abaqus Analysis User's Guide

**Optional and mutually exclusive parameters (model data only):**

DELETE SLAVE

Include this parameter to delete contact elements associated with tied slave 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.**

## 3.54 *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:**

- "Defining general contact interactions in Abaqus/Standard," Section 36.2.1 of the Abaqus Analysis User's Guide
- "Defining general contact interactions in Abaqus/Explicit," Section 36.4.1 of the Abaqus Analysis User's Guide

**Optional parameter:**

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.**

## 3.55 *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.

**Product:** Abaqus/Explicit

**Type:** Model data

**Level:** Model

**References:**

- "Controlling initial contact status for general contact in Abaqus/Explicit," Section 36.4.4 of the Abaqus Analysis User's Guide
- *CONTACT
- *CONTACT CLEARANCE ASSIGNMENT
- *DISTRIBUTION

**Required parameter:**

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 slave nodes or to the name of a nodal distribution (see "Distribution definition," Section 2.8.1 of the Abaqus Analysis User's Guide). The clearance values must be non-negative for slave 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 slave 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 slave node. The default for structural elements (e.g., shell elements) is the thickness associated with the slave 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 slave 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 slave node. The default for structural elements (e.g., shell elements) is the thickness associated with the slave 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 slave nodes to be included in the clearance specification. The specified clearance will be enforced at all slave nodes in this node set irrespective of whether they are above or below their respective master 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.**

## 3.56 *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.

**Product:** Abaqus/Explicit

**Type:** Model or history data

**Level:** Model, Step

**References:**

- "Controlling initial contact status for general contact in Abaqus/Explicit," Section 36.4.4 of the Abaqus Analysis User's Guide
- *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:

4. Blank, the "word" MASTER, or the "word" SLAVE 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 master-slave. A setting of MASTER or SLAVE specifies the behavior of the first surface in a pure master-slave interaction.

Repeat this data line as often as necessary. If the contact clearance assignments overlap, the last assignment applies in the overlap region.

## 3.57 *CONTACT CONTROLS: Specify additional controls for contact.

This option is used to provide additional optional solution controls for models involving contact between bodies. 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:**

- "Adjusting contact controls in Abaqus/Standard," Section 36.3.6 of the Abaqus Analysis User's Guide
- "Defining contact pairs in Abaqus/Explicit," Section 36.5.1 of the Abaqus Analysis User's Guide
- "Contact controls for contact pairs in Abaqus/Explicit," Section 36.5.5 of the Abaqus Analysis User's Guide
- *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:**

MASTER

Set this parameter equal to the master surface name to apply the controls to a specific contact pair. This parameter must be used in conjunction with the SLAVE 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 SLAVE and MASTER parameters. When this parameter is used in conjunction with the SLAVE and MASTER parameters, the controls applied to the specific contact pair are removed.

SLAVE

Set this parameter equal to the slave surface name to apply the controls to a specific contact pair. This parameter must be used in conjunction with the MASTER 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 SLAVE and MASTER 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.

**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 parameter:

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 will not be 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 master 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.**

## 3.58 *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. It must be used in conjunction with the *CONTACT option.

**Product:** Abaqus/Explicit

**Type:** Model or history data

**Level:** Model, Step

**References:**

- "Contact controls for general contact in Abaqus/Explicit," Section 36.4.5 of the Abaqus Analysis User's Guide
- *CONTACT

**Required, mutually exclusive parameters:**

AUTOMATIC OVERCLOSURE RESOLUTION

Include this parameter to store offsets instead of adjusting nodes during initial overclosure resolution between surface pairs in the general contact domain.

CONTACT THICKNESS REDUCTION

Set CONTACT THICKNESS REDUCTION=SELF to limit automatic contact thickness reductions to only regions of potential self-contact and the perimeters of shell surfaces.

Set CONTACT THICKNESS REDUCTION=NOPERIMSELF to limit automatic contact thickness reductions to only regions of potential self-contact.

NODAL EROSION

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

Set ROTATIONAL TERMS=NONE (default) to ignore the effects of shell and beam thickness offsets for frictional contact.

Set ROTATIONAL TERMS=STRUCTURAL for frictional contact to account for the incremental rotation of shell and beam thickness offsets in slip increment calculations and to apply a moment to nodes offset from the contact interface due to shell and beam thicknesses.

TYPE

Set TYPE=ENHANCED EDGE TRACKING (default) to activate the default tracking algorithm for edge-to-edge contact.

Set TYPE=EDGE TRACKING to activate an alternative tracking algorithm for edge-to-edge contact.

Set TYPE=FOLD TRACKING to activate the nondefault tracking algorithm for node-to-face contact.

Set TYPE=FOLD INVERSION CHECK to activate the fold inversion check.

Set TYPE=SCALE PENALTY to assign a scale factor to the default penalty stiffnesses.

**Optional parameter:**

SEEDING

This parameter 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.

**No data lines are used with this option when the NODAL EROSION parameter is specified.**

**Data lines for TYPE=FOLD TRACKING:**

First line:

1. The name of the surface whose nodes will be tracked using the nondefault node-to-face tracking algorithm. 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=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.

## 3.59 *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:**

- "Mechanical contact properties: overview," Section 37.1.1 of the Abaqus Analysis User's Guide
- "Contact damping," Section 37.1.3 of the Abaqus Analysis User's Guide

**Required parameter:**

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 parameter:**

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.

2. Clearance at which the damping coefficient is zero, $c_0$.
3. 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 $\eta = 0.0$.

## 3.60 *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 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:**

- "Defining general contact interactions in Abaqus/Standard," Section 36.2.1 of the Abaqus Analysis User's Guide
- "Defining general contact interactions in Abaqus/Explicit," Section 36.4.1 of the Abaqus Analysis User's Guide
- *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.

## 3.61 *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.

**Reference:**

- "Output to the data and results files," Section 4.1.2 of the Abaqus Analysis User's Guide

**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.

MASTER

Set this parameter equal to the name of the master 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.

SLAVE

Set this parameter equal to the name of the slave 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," Section 4.2.1 of the Abaqus Analysis User's Guide.

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.

## 3.62 *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:**

- "Numerical controls for general contact in Abaqus/Standard," Section 36.2.6 of the Abaqus Analysis User's Guide
- "Contact formulation for general contact in Abaqus/Explicit," Section 38.2.1 of the Abaqus Analysis User's Guide
- *CONTACT

**Required parameter:**

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=MASTER SLAVE ROLES to control master-slave roles for specific interactions in Abaqus/Standard. This setting does not apply for Abaqus/Explicit.

Set TYPE=PURE MASTER-SLAVE to specify that a contact interaction should use pure master-slave 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.

**Optional parameter:**

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 master-slave roles 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 "word" BALANCED, the "word" SLAVE, or the "word" MASTER. A balanced master-slave formulation is used if BALANCED is specified; otherwise, a pure master-slave formulation is used with SLAVE or MASTER indicating the desired behavior of the first surface.

Repeat this data line as often as necessary.

**Data lines to assign pure master-slave 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" SLAVE (default) or the "word" MASTER. 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.

**There are no data lines to control the edge-to-edge (beam-to-beam) contact formulations in Abaqus/Standard.**

## 3.63 *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 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:**

- "Defining general contact interactions in Abaqus/Standard," Section 36.2.1 of the Abaqus Analysis User's Guide
- "Defining general contact interactions in Abaqus/Explicit," Section 36.4.1 of the Abaqus Analysis User's Guide
- *CONTACT

**Optional parameter:**

ALL EXTERIOR

Include this parameter to specify self-contact for a default unnamed, all-inclusive surface that includes all element-based surface facets and, in Abaqus/Explicit only, 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.

## 3.64 *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 in Abaqus/Standard. It must be used in conjunction with the *CONTACT and *CONTACT INITIALIZATION DATA options.

**Products:** Abaqus/Standard Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Interaction module

**References:**

- "Controlling initial contact status in Abaqus/Standard," Section 36.2.4 of the Abaqus Analysis User's Guide
- *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.

Repeat this data line as often as necessary. If the contact initialization method assignments overlap, the last assignment applies in the overlap region.

## 3.65 *CONTACT INITIALIZATION DATA: Define contact initialization methods for general contact.

This option is used to define contact initialization methods for Abaqus/Standard. The contact initialization method is applied to a contact interaction using the *CONTACT INITIALIZATION ASSIGNMENT option.

**Products:** Abaqus/Standard Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Interaction module

**References:**

- "Controlling initial contact status in Abaqus/Standard," Section 36.2.4 of the Abaqus Analysis User's Guide
- "Common difficulties associated with contact modeling in Abaqus/Standard," Section 39.1.2 of the Abaqus Analysis User's Guide
- *CONTACT
- *CONTACT INITIALIZATION ASSIGNMENT

**Required parameter:**

NAME

Set this parameter equal to a label that will be 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.

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:**

MINIMUM DISTANCE

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.

**There are no data lines associated with this option.**

## 3.66 *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," Section 36.3.4 of the Abaqus Analysis User's Guide
- "Adjusting initial surface positions and specifying initial clearances in Abaqus/Standard contact pairs," Section 36.3.5 of the Abaqus Analysis User's Guide

**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. Slave surface name.
2. Master surface name. It must be distinct from the slave surface name; self-contact is not allowed with this option.

If the SHRINK parameter is included, no additional data are required. Otherwise:

3. Reference allowable interference, $v$.
4. $X$-direction cosine of the shift direction vector (optional).
5. $Y$-direction cosine of the shift direction vector (optional).
6. $Z$-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:

2. Reference allowable interference, $v$.
3. $X$-direction cosine of the shift direction vector (optional).
4. $Y$-direction cosine of the shift direction vector (optional).
5. $Z$-direction cosine of the shift direction vector (optional).

Repeat this data line as often as necessary to specify additional element sets containing contact elements.

## 3.67 *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," Section 4.1.3 of the Abaqus Analysis User's Guide
- *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.

NSET

Set this parameter equal to the name of the node set for which this output request is being made. This parameter is valid only for nodes defined under *BOND, and only the BONDSTAT and BONDLOAD output variables may be requested.

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:**

MASTER

Set this parameter equal to the name of a master 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.

SLAVE

Set this parameter equal to the name of a slave 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," Section 4.2.1 of the Abaqus Analysis User's Guide, and "Abaqus/Explicit output variable identifiers," Section 4.2.2 of the Abaqus Analysis User's Guide.

Repeat this data line as often as necessary to define the list of variables to be written to the output database.

## 3.68 *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:**

- "Defining contact pairs in Abaqus/Standard," Section 36.3.1 of the Abaqus Analysis User's Guide
- "Adjusting initial surface positions and specifying initial clearances in Abaqus/Standard contact pairs," Section 36.3.5 of the Abaqus Analysis User's Guide
- "Defining tied contact in Abaqus/Standard," Section 36.3.7 of the Abaqus Analysis User's Guide
- "Adjusting contact controls in Abaqus/Standard," Section 36.3.6 of the Abaqus Analysis User's Guide
- "Contact formulations in Abaqus/Standard," Section 38.1.1 of the Abaqus Analysis User's Guide
- "Smoothing contact surfaces in Abaqus/Standard," Section 38.1.3 of the Abaqus Analysis User's Guide
- "Common difficulties associated with contact modeling in Abaqus/Standard," Section 39.1.2 of the Abaqus Analysis User's Guide
- "Defining contact pairs in Abaqus/Explicit," Section 36.5.1 of the Abaqus Analysis User's Guide
- "Contact formulations for contact pairs in Abaqus/Explicit," Section 38.2.2 of the Abaqus Analysis User's Guide
- "Adjusting initial surface positions and specifying initial clearances for contact pairs in Abaqus/Explicit," Section 36.5.4 of the Abaqus Analysis User's Guide

### Defining contacting surfaces in an Abaqus/Standard analysis

**Required parameter:**

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. This parameter is required for TIED contact.

EXTENSION ZONE

Set this parameter equal to a fraction of the end segment or facet edge length by which the master 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 slave surface must penetrate the master 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 slave 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 slave 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 slave 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 slave 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).

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 master 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 slave 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 slave node projects onto the master 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 slave surface name.
2. The master surface name. If the master surface name is omitted or is the same as the slave surface name, Abaqus/Standard assumes that self-contact is defined.
3. Optional orientation name to specify the local tangent directions on the slave surface.
4. Optional orientation name to specify the local tangent directions on the master 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 to 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.

## 3.69 *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 surface interaction model will have no resistance to fluid flow across a contact interface (corresponding to infinite permeability) while contact is active and for clearances up to a default tolerance distance.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Model

**References:**

- "Pore fluid contact properties," Section 37.4.1 of the Abaqus Analysis User's Guide
- "Coupled pore fluid diffusion and stress analysis," Section 6.8.1 of the Abaqus Analysis User's Guide
- *SURFACE INTERACTION
- *SOILS

**Optional parameters:**

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, $k$, 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.

## 3.70 *CONTACT PRINT: Define print requests for contact variables.

This option is used to provide tabular printed output of contact variables for contact surface pairs.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**Reference:**

- "Output to the data and results files," Section 4.1.2 of the Abaqus Analysis User's Guide

**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.

MASTER

Set this parameter equal to the name of the master 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.

SLAVE

Set this parameter equal to the name of the slave 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," Section 4.2.1 of the Abaqus Analysis User's Guide.

Repeat this data line as often as necessary: each line defines a table. If this line is omitted, the default variables will be output.

## 3.71 *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," Section 36.2.3 of the Abaqus Analysis User's Guide
- "Assigning contact properties for general contact in Abaqus/Explicit," Section 36.4.3 of the Abaqus Analysis User's Guide
- *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. 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 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.

Repeat this data line as often as necessary. If the contact property assignments overlap, the last assignment applies in the overlap region.

## 3.72 *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.

**Product:** Abaqus/Design

**Type:** History data

**Level:** Step

**References:**

- "Design sensitivity analysis," Section 19.1.1 of the Abaqus Analysis User's Guide
- *DESIGN RESPONSE

**Optional parameters:**

MASTER

Set this parameter equal to the name of the master 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.

SLAVE

Set this parameter equal to the name of the slave 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 "Design sensitivity analysis," Section 19.1.1 of the Abaqus Analysis User's Guide.

Repeat this data line as often as necessary to define the contact responses whose sensitivities are to be written to the output database.

## 3.73 *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," Section 36.2.5 of the Abaqus Analysis User's Guide
- *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 factor by which Abaqus/Standard will reduce the contact stabilization coefficient per increment. The default value is 0.1 for the interactions specified on the data lines of this option.

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 the RESET parameter is 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.

**Data line if SCALE FACTOR=USER ADAPTIVE:**

First line:

1. Up to eight scale values in descending order.

## 3.74 *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. Initial stresses are not included in the evaluation of the* ***J****-integrals, the stress intensity factors, and the* ***T****-stress (see "Contour integral evaluation," Section 11.4.2 of the Abaqus Analysis User's Guide, for details).*

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," Section 11.4.2 of the Abaqus Analysis User's Guide
- "Modeling discontinuities as an enriched feature using the extended finite element method," Section 10.7.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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 that will be used 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 will be formed along the first nodes of the crack front node sets. (The first node will be 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.

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.

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.

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.

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 that the crack is modeled as an enriched feature with the extended finite element 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.**

## 3.75 *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," Section 7.2.2 of the Abaqus Analysis User's Guide, 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/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Step module

**References:**

- "Convergence and time integration criteria: overview," Section 7.2.1 of the Abaqus Analysis User's Guide
- "Commonly used control parameters," Section 7.2.2 of the Abaqus Analysis User's Guide
- "Convergence criteria for nonlinear problems," Section 7.2.3 of the Abaqus Analysis User's Guide
- "Time integration accuracy in transient problems," Section 7.2.4 of the Abaqus Analysis User's Guide

**Required, mutually exclusive parameters:**

ANALYSIS

This parameter applies only to Abaqus/Standard analyses.

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

This parameter applies only to Abaqus/Standard analyses.

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=FSI to set the parameters that will be used in deforming-mesh problems involving moving boundaries or deforming geometries in Abaqus/CFD as well as an Abaqus/CFD to Abaqus/Standard or to Abaqus/Explicit co-simulation. This parameter setting can be used only in Abaqus/CFD analyses.

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=VCCT LINEAR SCALING to set the $\beta$ parameter that will be used with linear scaling for a VCCT debonding analysis.

**Optional parameters:**

DISTORTION CONTROL

This parameter applies only to Abaqus/CFD analyses when TYPE=FSI, MESH SMOOTHING=EXPLICIT.

Set DISTORTION CONTROL=OFF (default) to deactivate a constraint that acts to prevent negative element volumes in the CFD meshing movement.

Set DISTORTION CONTROL=ON to activate a constraint that acts to prevent negative element volumes in the CFD meshing movement. The DISTORTION CONTROL parameter cannot prevent elements from being distorted due to temporal instabilities and physically unrealistic deformation.

FIELD

This parameter can be used only with PARAMETERS=FIELD in Abaqus/Standard.

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=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=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=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.

MESH SMOOTHING

This parameter applies only to Abaqus/CFD analyses when TYPE=FSI.

Set MESH SMOOTHING=IMPLICIT (default) to choose the implicit algorithm to control mesh smoothing.

Set MESH SMOOTHING=EXPLICIT to choose the explicit algorithm to control mesh smoothing.

**Data line for TYPE=FSI, MESH SMOOTHING=IMPLICIT:**

First (and only) line:

1. Number of iterations between convergence checks.
2. Maximum number of iterations.
3. FSI penalty scale factor, $\beta$. Default $\beta = 1.0$.
4. Solid/fluid density ratio.
5. Scale factor for deformable mesh stiffness.
6. Linear convergence criterion.

**Data line for TYPE=FSI, MESH SMOOTHING=EXPLICIT:**

First (and only) line:

1. Minimum number of mesh smoothing increments.
2. Maximum number of mesh smoothing increments.
3. FSI penalty scale factor, $\beta$. Default $\beta = 1.0$.
4. Solid/fluid density ratio.
5. Scale factor for deformable mesh stiffness.

**Data lines for PARAMETERS=FIELD:**

First line:

1. $R_n^\alpha$, convergence criterion for the ratio of the largest residual to the corresponding average flux norm for convergence. Default $R_n^\alpha = 5 \times 10^{-3}$.
2. $C_n^\alpha$, 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.

5. $R_P^\alpha$, alternative residual convergence criterion to be used after $I_P^\alpha$ iterations. Default $R_P^\alpha = 2 \times 10^{-2}$.
6. $\epsilon^\alpha$, criterion for zero flux compared to $\tilde{q}^\alpha$. Default $\epsilon^\alpha = 10^{-5}$.
7. $C_\epsilon^\alpha$, 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 $C_\epsilon^\alpha = 10^{-3}$.
8. $R_l^\alpha$, convergence criterion for the ratio of the largest 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. $\epsilon_l^\alpha$, 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. $\epsilon_d^\alpha$, 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. $f^{ls}_{s}$, residual reduction factor at which line searching terminates. Default $f^{ls}_{s} = 0.25$.

5. $\eta^{ls}$, 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 discontinutities) 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.

3. $I_P$, number of consecutive equilibrium iterations (without severe discontinuities) after which the residual tolerance $R_p$ is used instead of $R_n$. Default $I_P = 9$.
4. $I_C$, upper limit on the number of consecutive equilibrium iterations (without severe discontinuities), based on prediction of the logarithmic rate of convergence. Default $I_C = 16$.
5. $I_L$, number of consecutive equilibrium iterations (without severe discontinuities) above which the size of the next increment will be reduced. Default $I_L = 10$.
6. $I_G$, maximum number of consecutive equilibrium iterations (without severe discontinuities) allowed in consecutive increments for the time increment to be increased. Default $I_G = 4$.
7. $I_S$, 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.
8. $I_A$, maximum number of cutbacks allowed for an increment. Default $I_A = 5$.
9. $I_J$, 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.
10. $I_T$, 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 $I_T = 10$.
11. $I_S^c$, 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.
12. $I_J^c$, 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.
13. $I_A^c$, maximum number of allowed contact augmentations if the augmented Lagrange contact constraint enforcement method is specified. Default $I_A^c = 6$.

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. $CR_n^\alpha$, stabilized state detection criterion for the ratio of the largest residual coefficient on any terms in the Fourier series to the corresponding average flux norm. Default $CR_n^\alpha = 5 \times 10^{-3}$.
3. $CU_n^\alpha$, 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. $CR_0^\alpha$, plastic ratchetting detection criterion for the ratio of the largest 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. $CU_0^\alpha$, plastic ratchetting 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=VCCT LINEAR SCALING:**

First (and only) line:

1. $\beta$ parameter. Default $\beta = 0.9$.

## 3.76 *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.

**Product:** Abaqus/Explicit

**Type:** Model data

**Level:** Model

**References:**

- "Acoustic and shock loads," Section 34.4.6 of the Abaqus Analysis User's Guide
- *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 ($N/m^2$).

## 3.77 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Random response analysis," Section 6.3.11 of the Abaqus Analysis User's Guide
- *PSD-DEFINITION
- "UCORR," Section 1.1.22 of the Abaqus User Subroutines Reference Guide

**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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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.

## 3.78 *CO-SIMULATION: Identify the analysis program for co-simulation with Abaqus.

This option is used to identify the analysis program for co-simulation with Abaqus and the co-simulation controls used to define the coupling and rendezvousing scheme.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Interaction module

**References:**

- "Co-simulation: overview," Section 17.1.1 of the Abaqus Analysis User's Guide
- "Preparing an Abaqus analysis for co-simulation," Section 17.2.1 of the Abaqus Analysis User's Guide
- "Structural-to-structural co-simulation," Section 17.3.1 of the Abaqus Analysis User's Guide
- "Fluid-to-structural and conjugate heat transfer co-simulation," Section 17.3.2 of the Abaqus Analysis User's Guide
- *CO-SIMULATION CONTROLS
- *CO-SIMULATION REGION

**Required 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," Section 1.2.1 of the Abaqus Analysis User's Guide), except that it cannot begin with a number.

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. This parameter should also be used in an Abaqus/CFD to Abaqus/Standard or Abaqus/CFD to Abaqus/Explicit co-simulation.

Set PROGRAM=ABAQUS for exchange of data with another Abaqus analysis in an Abaqus/Standard to Abaqus/Explicit co-simulation.

Set PROGRAM=DIRECT for exchange of data between Abaqus and certain third-party analysis programs. For more information, refer to the appropriate third-party program documentation.

Set PROGRAM=MPCCI for exchange of data between Abaqus and the Mesh-based parallel Code Coupling Interface (MpCCI), which in turn can exchange data with third-party analysis programs supporting MpCCI.

**Optional parameter:**

CONTROLS

This parameter is required when PROGRAM=ABAQUS or PROGRAM=MPCCI.

Set this parameter equal to the name of the co-simulation controls to be used to define the coupling and rendezvousing scheme.

For PROGRAM=DIRECT or PROGRAM=MULTIPHYSICS, the co-simulation controls are defined in the co-simulation configuration file.

**There are no data lines associated with this option.**

## 3.79 *CO-SIMULATION CONTROLS: Specify the coupling and rendezvousing scheme for co-simulation.

This option is used to specify the coupling and rendezvousing scheme for co-simulation. It must be used in conjunction with the *CO-SIMULATION option to identify the analysis program for which the co-simulation controls are specified. This option is required for co-simulation when PROGRAM=ABAQUS or PROGRAM=MPCCI is used on the *CO-SIMULATION option.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Interaction module

**References:**

- "Co-simulation: overview," Section 17.1.1 of the Abaqus Analysis User's Guide
- "Preparing an Abaqus analysis for co-simulation," Section 17.2.1 of the Abaqus Analysis User's Guide
- "Structural-to-structural co-simulation," Section 17.3.1 of the Abaqus Analysis User's Guide
- "Fluid-to-structural and conjugate heat transfer co-simulation," Section 17.3.2 of the Abaqus Analysis User's Guide
- *CO-SIMULATION
- *CO-SIMULATION REGION

### Specifying the coupling and rendezvousing scheme for *CO-SIMULATION, PROGRAM=ABAQUS

**Required parameter:**

NAME

Set this parameter equal to the label that will be used to identify the co-simulation controls. All co-simulation control names in the same input file must be unique.

**Optional parameters:**

FACTORIZATION FREQUENCY

This parameter is valid when used with the TIME INCREMENTATION=SUBCYCLE parameter.

Set FACTORIZATION FREQUENCY=EXPLICIT INCREMENT (default) to specify factoring of the interface matrix every Abaqus/Explicit increment.

Set FACTORIZATION FREQUENCY=STANDARD INCREMENT to specify factoring of the interface matrix once per Abaqus/Standard increment.

TIME INCREMENTATION

Set TIME INCREMENTATION=SUBCYCLE (default) to allow Abaqus to take one or more increments to reach the next target time to exchange data with the external program. This setting is valid only in an Abaqus/Standard or Abaqus/Explicit analysis.

Set TIME INCREMENTATION=LOCKSTEP to force Abaqus to use only one increment to reach the next target time.

**There are no data lines associated with this option.**

## Specifying the coupling and rendezvousing scheme for *CO-SIMULATION, PROGRAM=MPCCI

**Required parameters:**

NAME

Set this parameter equal to the label that will be used to identify the co-simulation controls. All co-simulation control names in the same input file must be unique.

STEP SIZE

Set this parameter equal to a value that defines the constant coupling step size to be used throughout the coupled simulation.

Set STEP SIZE=IMPORT for Abaqus to import a coupling step size from the external program for the next coupling step.

Set STEP SIZE=EXPORT for Abaqus to export a coupling step size to the external program for the next coupling step.

Set STEP SIZE=MAX for Abaqus to select the maximum coupling step size based on the suggested coupling step size of Abaqus and the external program.

Set STEP SIZE=MIN for Abaqus to select the minimum coupling step size based on the suggested coupling step size of Abaqus and the external program.

**There are no data lines associated with this option.**

## 3.80 *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/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Interaction module

**References:**

- "Co-simulation: overview," Section 17.1.1 of the Abaqus Analysis User's Guide
- "Preparing an Abaqus analysis for co-simulation," Section 17.2.1 of the Abaqus Analysis User's Guide
- *CO-SIMULATION

### Defining a co-simulation region for *CO-SIMULATION, PROGRAM=MULTIPHYSICS or PROGRAM=MPCCI

**Required, mutually exclusive parameters:**

EXPORT

When TYPE=NODE, SURFACE, or VOLUME, include this parameter to specify fields and the accompanying region for export to the coupled analysis.

When TYPE=DISCRETE, include this parameter to specify sensor variables for export.

IMPORT

When TYPE=NODE, SURFACE, or VOLUME, include this parameter to specify fields and the accompanying region for import from the coupled analysis.

When TYPE=DISCRETE, include this parameter to specify actuator variables for import.

**Optional parameter:**

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.
Set TYPE=DISCRETE to define coupling through sensors and actuators.

**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," Section 17.2.1 of the Abaqus Analysis User's Guide.
3. Etc., up to seven field identifiers.

Repeat this data line as often as necessary to define more than seven field identifiers for a given region. Repeat the option to define import and/or export fields. When PROGRAM=MULTIPHYSICS, only a single node set can be declared for the co-simulation.

**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," Section 17.2.1 of the Abaqus Analysis User's Guide.
3. Etc., up to seven field identifiers.

Repeat this data line as often as necessary to define more than seven field identifiers for a given region. Repeat the option to define import and/or export fields. When PROGRAM=MULTIPHYSICS, only a single surface can be declared for the co-simulation.

**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," Section 17.2.1 of the Abaqus Analysis User's Guide.
3. Etc., up to seven field identifiers.

Repeat this data line as often as necessary to define more than seven field identifiers for a given region. Repeat the option to define import and/or export fields. When PROGRAM=MULTIPHYSICS, only a single element set can be declared for the co-simulation.

**Data lines for TYPE=DISCRETE:**

First line:

1. The name of the first sensor or actuator.
2. The name of the second sensor or actuator.
3. Etc., up to seven sensor or actuator identifiers.

Repeat this data line as often as necessary to define more than seven sensor or actuator definitions. The data line can refer either to names of sensors only or to names of actuators only, according to the IMPORT or EXPORT parameter setting.

## Defining a co-simulation region for *CO-SIMULATION, PROGRAM=ABAQUS

**Optional parameter:**

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.

## 3.81 *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," Section 6.5.3 of the Abaqus Analysis User's Guide
- "Rate-dependent plasticity: creep and swelling," Section 23.2.4 of the Abaqus Analysis User's Guide

**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," Section 7.1.1 of the Abaqus Analysis User's Guide).

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 may 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," Section 23.2.4 of the Abaqus Analysis User's Guide 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 specify that electrical conduction will be modeled 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 the electrical conduction will not be 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," Section 7.1.1 of the Abaqus Analysis User's Guide) 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," Section 7.1.1 of the Abaqus Analysis User's Guide). 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 $2 \times 10^{-4}$ 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 overriden 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

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. This parameter is meaningful only when the material response is time dependent (creep and swelling). If both this parameter and the DELTMX parameter are omitted in a transient analysis, fixed time increments will be 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 will restrict the time step to ensure that this value is not exceeded at any node during any increment of the step. If both this and the CETOL parameter are omitted in a transient analysis, fixed time increments will be used, with a constant time increment equal to the initial time 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.

## 3.82 *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

**Reference:**

- "Coupled thermal-electrical analysis," Section 6.7.3 of the Abaqus Analysis User's Guide

**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.

## 3.83 *COUPLING: Define a surface-based coupling constraint.

This option is used to impose a kinematic or distributing coupling constraint between a reference node and a group of nodes located on a surface. It must be used in conjunction with the *KINEMATIC or the *DISTRIBUTING option.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Part, Part instance, Assembly

**Abaqus/CAE:** Interaction module

**References:**

- "Coupling constraints," Section 35.3.2 of the Abaqus Analysis User's Guide
- "Element-based surface definition," Section 2.3.2 of the Abaqus Analysis User's Guide
- "Node-based surface definition," Section 2.3.3 of the Abaqus Analysis User's Guide

**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.

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

Set this parameter equal to the name given to the *ORIENTATION definition ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Guide) 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.**

## 3.84 *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 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:** Interaction module

**Reference:**

- "Thermal loads," Section 34.4.4 of the Abaqus Analysis User's Guide

**Optional parameters:**

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," Section 6.1.2 of the Abaqus Analysis User's Guide). 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.

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, $\theta^0$. (Units of $\theta$.)
4. Emissivity, $\epsilon$.

Repeat this data line as often as necessary to define radiation conditions.

## 3.85 *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," Section 23.2.4 of the Abaqus Analysis User's Guide
- "Anisotropic yield/creep," Section 23.2.6 of the Abaqus Analysis User's Guide
- "CREEP," Section 1.1.1 of the Abaqus User Subroutines Reference Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

LAW

Set LAW=STRAIN (default) to choose a strain-hardening power law.
Set LAW=TIME to choose a time-hardening 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.

TIME

This parameter is relevant only when LAW=TIME is used.

Set TIME=CREEP to use creep time in the time-hardening relation.
Set TIME=TOTAL (default) to use total time in the time-hardening relation.

**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. $\triangle H$. (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 $\mathrm{FL}^{-2}$.)
2. $Q/R$. (Units of $\theta$.)
3. $A$. (Units of $\mathrm{T}^{-1}$.)
4. $\xi$.
5. $m$.
6. $A_0$. (Units of $\mathrm{FL}^{-2}$.)
7. $\hat{s}$. (Units of $\mathrm{FL}^{-2}$.)
8. $n$.

Second line:

1. $a$.
2. $S_2$. (Units of $\mathrm{FL}^{-2}\theta^{-1}$.)
3. $S_3$. (Units of $\mathrm{FL}^{-2}\theta^{-2}$.)
4. $A_1$. (Units of $\mathrm{FL}^{-2}\theta^{-1}$.)
5. $A_2$. (Units of $\mathrm{FL}^{-2}\theta^{-2}$.)
6. $A_3$. (Units of $\mathrm{FL}^{-2}\mathrm{T}$.)
7. $A_4$. (Units of $\mathrm{FL}^{-2}\mathrm{T}^2$.)

**Data lines for LAW=DARVEAUX:**

First line:

1. $C_{\mathrm{ss}}$. (Units of $\mathrm{T}^{-1}$.)
2. $Q/R$. (Units of $\theta$.)
3. $\alpha$. (Units of $\mathrm{F}^{-1}\mathrm{L}^2$.)
4. $n$.
5. $\epsilon_T$.
6. $B$.

**Data lines for LAW=DOUBLE POWER:**

First line:

1. $A_1$. (Units of $\mathrm{T}^{-1}$.)
2. $B_1$. (Units of $\theta$.)
3. $C_1$.
4. $A_2$. (Units of $\mathrm{T}^{-1}$.)
5. $B_2$. (Units of $\theta$.)

6. $C_2$.
7. $\sigma_0$. (Units of $FL^{-2}$.)

## 3.86 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**Reference:**

- "Rate-dependent plasticity: creep and swelling," Section 23.2.4 of the Abaqus Analysis User's Guide

**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," Section 34.1.2 of the Abaqus Analysis User's Guide).

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," Section 21.1.2 of the Abaqus Analysis User's Guide) 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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 3.87 *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," Section 23.3.5 of the Abaqus Analysis User's Guide
- *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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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; $0 \leq k < 3$. Enter the ratio of initial yield stress in uniaxial compression to initial yield stress in hydrostatic compression.
2. $\nu_p$, plastic Poisson's ratio; $-1 < \nu_p \leq 0.5$.
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.

## 3.88 *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," Section 23.3.5 of the Abaqus Analysis User's Guide
- *CRUSHABLE FOAM

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

**Data lines to define crushable foam hardening:**

First line:

1. $\sigma_c$, 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.

## 3.89 *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," Section 23.2.12 of the Abaqus Analysis User's Guide

**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.

## 3.90 *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," Section 41.1.1 of the Abaqus Analysis User's Guide
- *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 3.90–1. 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 3.90–2. 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 3.90–1).
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 3.90–2).
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 $k$.

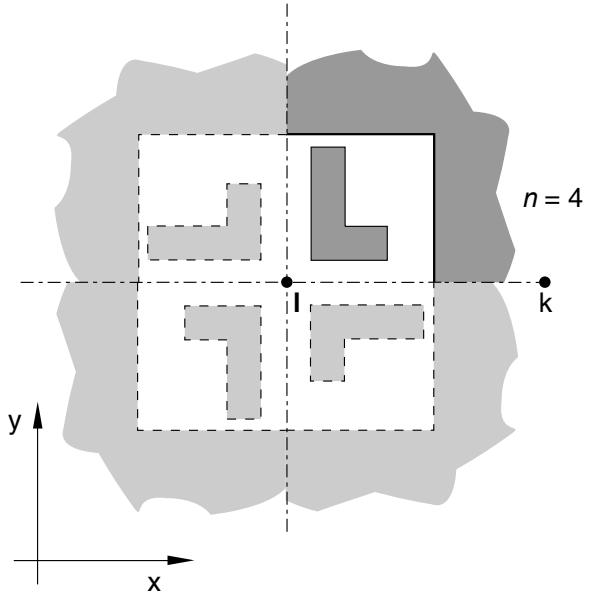



**Figure 3.90–1** *CYCLIC, TYPE=POINT option.

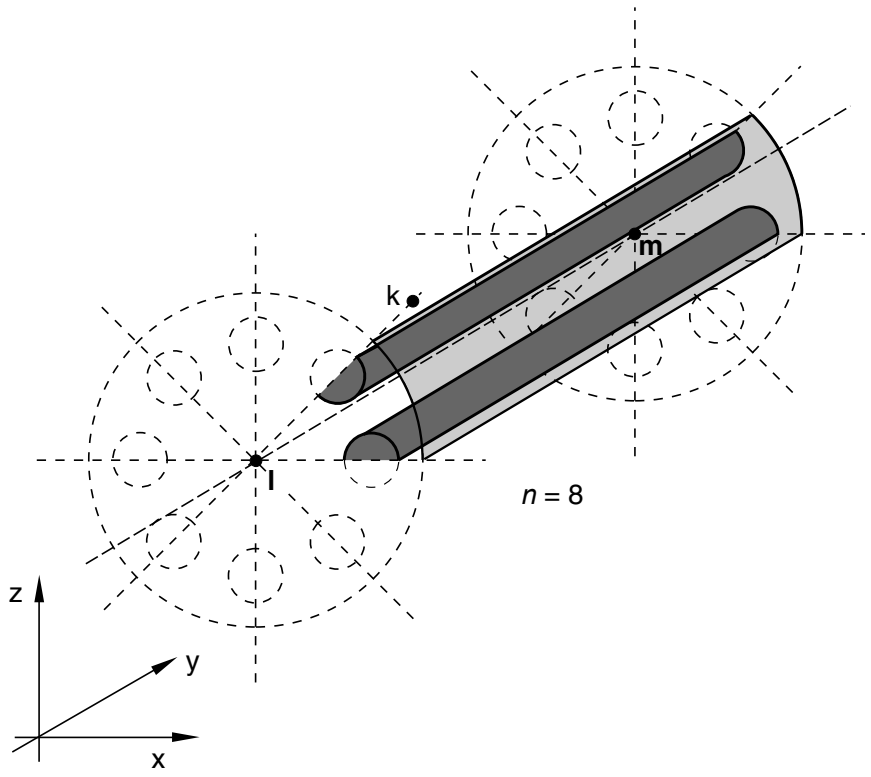



**Figure 3.90–2** *CYCLIC, TYPE=AXIS option.

## 3.91 *CYCLIC HARDENING: Specify the size of the elastic range for the combined hardening model.

This option is used to define the evolution of the elastic domain for the nonlinear isotropic/kinematic hardening model. It can be used only in conjunction with the *PLASTIC option. The elastic domain 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," Section 23.2.2 of the Abaqus Analysis User's Guide
- *PLASTIC
- "UHARD," Section 1.1.36 of the Abaqus User Subroutines Reference Guide
- "VUHARD," Section 1.2.17 of the Abaqus User Subroutines Reference Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

PARAMETERS

Include this parameter to provide the material parameters $Q_\infty$ and $b$ directly.

USER

Include this parameter to define 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 parameter if neither PARAMETERS nor USER is included:**

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, $Q_\infty$.
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.

## 3.92 *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/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Interaction module

**References:**

- "Analysis of models that exhibit cyclic symmetry," Section 10.4.3 of the Abaqus Analysis User's Guide
- *SELECT CYCLIC SYMMETRY MODES
- *TIE

**Required parameter:**

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.

4. $X$-coordinate of the second point defining the cyclic symmetry axis.
5. $Y$-coordinate of the second point defining the cyclic symmetry axis.
6. $Z$-coordinate of the second point defining the cyclic symmetry axis.

**4. D**

## 4.1 *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.

**Product:** Abaqus/Aqua

**Type:** History data

**Level:** Step

**Reference:**

- "Abaqus/Aqua analysis," Section 6.11.1 of the Abaqus Analysis User's Guide

**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.

## 4.2 *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

**Reference:**

- "Eddy current analysis," Section 6.7.5 of the Abaqus Analysis User's Guide
- "Magnetostatic analysis," Section 6.7.6 of the Abaqus Analysis User's Guide

**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 POTENTIALs 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 POTENTIALs 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," Section 2.2.5 of the Abaqus Analysis User's Guide) 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," Section 2.2.5 of the Abaqus Analysis User's Guide) 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.

## 4.3 *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," Section 24.2.3 of the Abaqus Analysis User's Guide
- "Damage evolution and element removal for fiber-reinforced composites," Section 24.3.3 of the Abaqus Analysis User's Guide
- "Damage evolution for ductile materials in low-cycle fatigue," Section 24.4.3 of the Abaqus Analysis User's Guide
- "Defining the constitutive response of cohesive elements using a traction-separation description," Section 32.5.6 of the Abaqus Analysis User's Guide
- "Surface-based cohesive behavior," Section 37.1.10 of the Abaqus Analysis User's Guide
- "Modeling discontinuities as an enriched feature using the extended finite element method," Section 10.7.1 of the Abaqus Analysis User's Guide

**Required parameter:**

TYPE

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 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.

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

FAILURE INDEX

This parameter can be used only in conjunction with the user-defined damage initiation criterion.

Set this parameter equal to the corresponding failure mechanism specified in the user-defined 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.

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.

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.

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" in "Defining the constitutive response of cohesive elements using a traction-separation description," Section 32.5.6 of the Abaqus Analysis User's Guide, or "Defining damage evolution data as a tabular function of mode mix" in "Surface-based cohesive behavior," Section 37.1.10 of the Abaqus Analysis User's Guide, for further details.

Set MODE MIX RATIO=ENERGY (default for surface-based cohesive behavior in Abaqus/Standard and for materials associated with cohesive elements) 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. This option can be used when MIXED MODE BEHAVIOR=POWER LAW or BK. This option is not available for surface-based cohesive behavior in Abaqus/Explicit.

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. This option can be used when MIXED MODE BEHAVIOR=POWER LAW or BK.

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.

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=DISPLACEMENT, SOFTENING=LINEAR without the MIXED MODE BEHAVIOR parameter:**

First line:

1. Effective total or plastic displacement at failure, measured from the time of 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 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 parameter:**

First line:

1. Fracture energy.
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=DISPLACEMENT, SOFTENING=LINEAR, MIXED MODE BEHAVIOR=TABULAR:**

First line:

1. Total displacement at failure, measured from the time of damage initiation.
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=ENERGY, SOFTENING=LINEAR, MIXED MODE BEHAVIOR=TABULAR:**

First line:

1. Fracture energy.
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=DISPLACEMENT, SOFTENING=EXPONENTIAL without the MIXED MODE BEHAVIOR parameter:**

First line:

1. Effective total or plastic displacement at failure, measured from the time of damage initiation.
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=ENERGY, SOFTENING=EXPONENTIAL without the MIXED MODE BEHAVIOR parameter:**

First line:

1. Fracture energy.
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=DISPLACEMENT, SOFTENING=EXPONENTIAL, MIXED MODE BEHAVIOR=TABULAR:**

First line:

1. Total displacement at failure, measured from the time of damage initiation.
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=ENERGY, SOFTENING=EXPONENTIAL, MIXED MODE BEHAVIOR=TABULAR:**

First line:

1. Fracture energy.
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=DISPLACEMENT, SOFTENING=TABULAR without the MIXED MODE BEHAVIOR parameter:**

First line:

1. Damage variable.
2. Effective total or plastic displacement, measured from the time of damage initiation.
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, MIXED MODE BEHAVIOR=TABULAR:**

First line:

1. Damage variable.
2. Effective total displacement, measured from the time of damage initiation.
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=ENERGY, SOFTENING=LINEAR or EXPONENTIAL, MIXED MODE BEHAVIOR=POWER LAW or BK:**

First line:

1. Normal mode fracture energy.
2. Shear mode fracture energy for failure in the first shear direction.
3. Shear mode fracture energy for failure in 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 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 fiber-reinforced materials:**

First line:

1. Fracture energy of the lamina in the longitudinal tensile direction.
2. Fracture energy of the lamina in the longitudinal compressive direction.
3. Fracture energy of the lamina in the transverse tensile direction.
4. Fracture energy of the lamina in the transverse compressive direction.
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=HYSTERESIS ENERGY in a low-cycle fatigue analysis:**

First line:

1. Material constant $c_3$. (Units of $L/CYCLE/F^{c_4}L^{-2c_4}$)
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.

## 4.4 *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. It 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," Section 24.2.2 of the Abaqus Analysis User's Guide
- "Damage initiation for fiber-reinforced composites," Section 24.3.2 of the Abaqus Analysis User's Guide
- "Damage initiation for ductile materials in low-cycle fatigue," Section 24.4.2 of the Abaqus Analysis User's Guide
- "Defining the constitutive response of cohesive elements using a traction-separation description," Section 32.5.6 of the Abaqus Analysis User's Guide
- "Modeling discontinuities as an enriched feature using the extended finite element method," Section 10.7.1 of the Abaqus Analysis User's Guide
- "Surface-based cohesive behavior," Section 37.1.10 of the Abaqus Analysis User's Guide
- "UDMGINI," Section 1.1.26 of the Abaqus User Subroutines Reference Guide

### Defining damage initiation as a material property

**Required parameter:**

CRITERION

Set CRITERION=DUCTILE to specify a damage initiation criterion based on the ductile failure strain.

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=HASHIN to specify damage initiation criteria based on the Hashin analysis.

Set CRITERION=HYSTERESIS ENERGY to specify damage initiation criteria based on the inelastic hysteresis energy dissipated per stabilized cycle in a low-cycle fatigue analysis.

Set CRITERION=JOHNSON COOK to specify a damage initiation criterion based on the Johnson-Cook failure strain.

Set CRITERION=MAXE to specify a damage initiation criterion based on the maximum nominal strain for cohesive elements or enriched elements.

Set CRITERION=MAXS to specify a damage initiation criterion based on the maximum nominal stress criterion for cohesive elements or enriched elements.

Set CRITERION=MAXPE to specify a damage initiation criterion based on the maximum principal strain for enriched elements.

Set CRITERION=MAXPS to specify a damage initiation criterion based on the maximum principal stress criterion for enriched elements.

Set CRITERION=MK to specify a damage initiation criterion based on a Marciniak-Kuczynski analysis.

Set CRITERION=MSFLD to specify a damage initiation criterion based on the Müschenborn and Sonne forming limit diagram.

Set CRITERION=QUADE to specify a damage initiation based on the quadratic separation-interaction criterion for cohesive elements or enriched elements.

Set CRITERION=QUADS to specify a damage initiation based on the quadratic traction-interaction criterion for cohesive elements or enriched elements.

Set CRITERION=SHEAR to specify a damage initiation criterion based on the shear failure strain.

Set CRITERION=USER to specify a user-defined damage initiation criterion for enriched elements.

**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 $\alpha = 0.0$.

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 $\alpha$.

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. This parameter cannot be used with CRITERION=JOHNSON COOK.

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.

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, $f_{eq}^{\text{crit}}$. 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, $f_{nn}^{\text{crit}}$. 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, $f_{nt}^{\text{crit}}$. 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.

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.

NORMAL DIRECTION

This parameter can be used only in conjunction with CRITERION=MAXE, CRITERION=MAXS, CRITERION=QUADE, or CRITERION=QUADS for enriched elements in Abaqus/Standard.

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.

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 $\omega$ 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 $0.002$ ($0.2$ %).

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 for enriched elements in Abaqus/Standard.

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). 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 used to obtain the crack propagation direction. The default value is three times the typical element characteristic length in the enriched region.

TOLERANCE

This parameter can be used only in conjunction with CRITERION=MAXPE, CRITERION=MAXPS, CRITERION=MAXE, CRITERION=MAXS, CRITERION=QUADE, CRITERION=QUADS, or CRITERION=USER for enriched elements in Abaqus/Standard.

Set this parameter equal to the tolerance within which the damage initiation criterion must be satisfied. The default is 0.05.

**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=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=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=HYSTERESIS ENERGY:**

First line:

1. Material constant $c_1$. (Units of $CYCLE/F^{c_2}L^{-2c_2}$)
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.

**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, $\theta_{\text{melt}}$.
7. Transition temperature, $\theta_{\text{transition}}$.
8. Reference strain rate, $\dot{\varepsilon}_0$.

**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, $\alpha$.
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 $\alpha$, 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.

**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=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.

**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 parameter:**

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 parameter:**

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=QUADU or CRITERION=MAXU:**

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=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.

## 4.5 *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 surface-based traction-separation behavior, you can use this option in conjunction with the *DAMAGE INITIATION, CRITERION=MAXS, MAXE, QUADS, or QUADE and *DAMAGE EVOLUTION options.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "Viscous regularization" in "Damage evolution and element removal for fiber-reinforced composites," Section 24.3.3 of the Abaqus Analysis User's Guide
- "Surface-based cohesive behavior," Section 37.1.10 of the Abaqus Analysis User's Guide
- "Modeling discontinuities as an enriched feature using the extended finite element method," Section 10.7.1 of the Abaqus Analysis User's Guide

**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 line to define viscosity coefficients for surfaced-based traction-separation behavior or cohesive behavior in enriched elements:**

First (and only) line:

1. Viscosity coefficient.

## 4.6 *DAMPING: Specify material damping.

*WARNING: The use of stiffness proportional material damping in Abaqus/Explicit may reduce the stable time increment dramatically and can lead to longer analysis times. See "Material damping," Section 26.1.1 of the Abaqus Analysis User's Guide.*

This option is used to provide material damping for mode-based analyses and for direct-integration dynamic analysis 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 may also be defined as step data using the *GLOBAL DAMPING option and may come from damper elements like connectors and dashpots.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Part, Part instance

**Abaqus/CAE:** Property module

**References:**

- "Material damping," Section 26.1.1 of the Abaqus Analysis User's Guide
- "Dynamic analysis procedures: overview," Section 6.3.1 of the Abaqus Analysis User's Guide
- "Explicit dynamic analysis," Section 6.3.3 of the Abaqus Analysis User's Guide

**Optional parameters:**

ALPHA

Set this parameter equal to the $\alpha_R$ factor to create Rayleigh mass proportional damping in the following procedures:

*DYNAMIC (Abaqus/Standard or Abaqus/Explicit)

*COMPLEX FREQUENCY

*STEADY STATE DYNAMICS, DIRECT

*STEADY STATE DYNAMICS, SUBSPACE PROJECTION

*STEADY STATE DYNAMICS that allows nondiagonal damping

*MODAL DYNAMIC that allows nondiagonal damping

This parameter is ignored in mode-based procedures that use Lanczos or subspace iteration eigenvalue extraction that is not based on the SIM architecture. The default is ALPHA=0. (Units of $T^{-1}$.)

In Abaqus/Explicit set ALPHA=TABULAR to specify that the mass proportional damping is dependent on temperature and/or field variables.

BETA

Set this parameter equal to the $\beta_R$ factor to create Rayleigh stiffness proportional damping in the following procedures:

*DYNAMIC (Abaqus/Standard or Abaqus/Explicit)

*COMPLEX FREQUENCY

*STEADY STATE DYNAMICS, DIRECT

*STEADY STATE DYNAMICS, SUBSPACE PROJECTION

*STEADY STATE DYNAMICS that allows nondiagonal damping

*MODAL DYNAMIC that allows nondiagonal damping

This parameter is ignored in mode-based procedures that use Lanczos or subspace iteration eigenvalue extraction that is not based on the SIM architecture. The default is BETA=0. (Units of T.)

In Abaqus/Explicit 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 modal-based procedures that follow subspace iteration eigenvalue extraction or eigenvalue extraction using the Lanczos eigensolver that does not use the SIM architecture, except for *STEADY STATE DYNAMICS, SUBSPACE PROJECTION. Use the *MODAL DAMPING, VISCOUS=COMPOSITE (or MODAL=COMPOSITE) option to activate composite modal damping.

The default is COMPOSITE=0.

DEPENDENCIES

This parameter applies only to Abaqus/Explicit analyses when ALPHA=TABULAR and/or BETA=TABULAR.

Set this parameter equal to the number of field variables included in the definition of the $\alpha_R$ and/or $\beta_R$ factors, in addition to temperature. If this parameter is omitted, it is assumed that Rayleigh damping is constant or depends only on temperature. See “Specifying field variable dependence” in “Material data definition,” Section 21.1.2 of the Abaqus Analysis User’s Guide, for more information.

STRUCTURAL

Set this parameter equal to the $s$ factor to create imaginary stiffness proportional damping in the following procedures:

*FREQUENCY, DAMPING PROJECTION=ON

*STEADY STATE DYNAMICS, DIRECT

*STEADY STATE DYNAMICS, SUBSPACE PROJECTION

*STEADY STATE DYNAMICS that allows nondiagonal damping

*MODAL DYNAMIC that allows nondiagonal damping

*COMPLEX FREQUENCY that uses the SIM architecture

This parameter is ignored in mode-based procedures that use Lanczos or subspace iteration eigenvalue extraction that is not based on the SIM architecture.

The default is STRUCTURAL=0.

**Data lines to define temperature and/or field variable-dependent mass proportional damping (ALPHA=TABULAR) in Abaqus/Explicit:**

First line:

1. $\alpha_R$. (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 temperature and/or field variable-dependent stiffness proportional damping (BETA=TABULAR) in Abaqus/Explicit:**

First line:

1. $\beta_R$. (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 both temperature and/or field variable-dependent mass and stiffness proportional damping (both ALPHA=TABULAR and BETA=TABULAR) in Abaqus/Explicit:**

First line:

1. $\alpha_R$. (Units of $T^{-1}$.)
2. $\beta_R$. (Units of T.)
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 alpha and beta damping as a function of temperature and other predefined field variables.

## 4.7 *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 *DAMPING; at the element level using *SPRING, COMPLEX STIFFNESS or *CONNECTOR DAMPING; 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.

The *DAMPING CONTROLS option is also used to define the type and the source of substructure damping under the *SUBSTRUCTURE PROPERTY option in all analysis procedures using substructures that take damping into account. The rules for using this option within a substructure property definition are the same as the rules for using it within a step definition (see "Defining substructure damping" in "Using substructures," Section 10.1.1 of the Abaqus Analysis User's Guide, 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," Section 26.1.1 of the Abaqus Analysis User's Guide
- "Damping in dynamic analysis" in "Dynamic analysis procedures: overview," Section 6.3.1 of the Abaqus Analysis User's Guide
- *FREQUENCY

**Optional parameters:**

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.**

## 4.8 *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," Section 32.2.1 of the Abaqus Analysis User's Guide
- "Flexible joint element," Section 32.3.1 of the Abaqus Analysis User's Guide
- "Tube support elements," Section 32.8.1 of the Abaqus Analysis User's Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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," Section 2.2.5 of the Abaqus Analysis User's Guide).

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," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 4.9 *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," Section 11.4.3 of the Abaqus Analysis User's Guide
- *FRACTURE CRITERION

**Required parameters:**

MASTER

Set this parameter equal to the name of the master surface of the contact pair used in the crack propagation analysis.

SLAVE

Set this parameter equal to the name of the slave surface of the contact pair used in the crack propagation analysis.

**Optional parameters:**

DEBONDING FORCE

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.

This parameter is relevant only when TYPE=VCCT or TYPE= ENHANCED VCCT is used on the *FRACTURE CRITERION option.

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.

## 4.10 *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

**Reference:**

- "Piezoelectric analysis," Section 6.7.2 of the Abaqus Analysis User's Guide

**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," Section 6.1.2 of the Abaqus Analysis User's Guide).

OP

Set OP=MOD (default) for existing *DECHARGEs to remain, with this option defining electric charges to be added or modified. Set OP=NEW if all existing *DECHARGEs 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 Part VI, “Elements,” of the Abaqus Analysis User’s Guide).
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.

## 4.11 *DECURRENT: Specify distributed current densities in an electromagnetic analysis.

This option is used to input distributed current densities in coupled thermal-electrical and coupled thermal-electrical-structural 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:**

- "Coupled thermal-electrical analysis," Section 6.7.3 of the Abaqus Analysis User's Guide
- "Fully coupled thermal-electrical-structural analysis," Section 6.7.4 of the Abaqus Analysis User's Guide
- "Eddy current analysis," Section 6.7.5 of the Abaqus Analysis User's Guide
- "Magnetostatic analysis," Section 6.7.6 of the Abaqus Analysis User's Guide

**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," Section 34.1.2 of the Abaqus Analysis User's Guide). 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," Section 6.1.2 of the Abaqus Analysis User's Guide).

OP

Set OP=MOD (default) for existing *DECURRENTs to remain, with this option defining distributed current densities to be added or modified.

Set OP=NEW if all existing *DECURRENTs 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 or coupled thermal-electrical-structural analyses:**

First line:

1. Element number or element set label.
2. Distributed current density type label (see Part VI, "Elements," of the Abaqus Analysis User's Guide).
3. Reference current density magnitude. (Units of $CL^{-2}T^{-1}$ for surface current densities and $CL^{-3}T^{-1}$ for body current sources.)

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.

7. Name of the orientation option ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Guide) 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.

## 4.12 *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

**Reference:**

- "Deformation plasticity," Section 23.2.13 of the Abaqus Analysis User's Guide

**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, $\nu$.
3. Yield stress, $\sigma^0$.
4. Exponent, $n$.
5. Yield offset, $\alpha$.
6. Temperature.

Repeat this data line as often as necessary to define the dependence of the deformation plasticity parameters on temperature.

## 4.13 *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," Section 2.8.1 of the Abaqus Analysis User's Guide).

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**Reference:**

- "Density," Section 21.2.1 of the Abaqus Analysis User's Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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," Section 2.8.1 of the Abaqus Analysis User's Guide.

It is assumed that the density is constant in Abaqus/CFD analyses.

PORE FLUID

This parameter applies only to Abaqus/Standard analyses.

Include this parameter if the density of the pore fluid in a porous medium is being defined.

**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 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 $ML^{-3}$.

## 4.14 *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 key and a description can be given for some or all of the solution-dependent state variables allocated by this option. 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 key has been specified under this option will consist of the string "SDV_," followed by the specified key. Similarly, the descriptions specified under this option will be used in the corresponding field descriptions written to the output database.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "User subroutines: overview," Section 18.1.1 of the Abaqus Analysis User's Guide
- "User-defined mechanical material behavior," Section 26.7.1 of the Abaqus Analysis User's Guide
- "Finite element conversion to SPH particles," Section 15.2.2 of the Abaqus Analysis User's Guide

**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," Section 15.2.2 of the Abaqus Analysis User's Guide).

DELETE

Set this parameter equal to the state variable number controlling the element deletion flag (see "User-defined mechanical material behavior," Section 26.7.1 of the Abaqus Analysis User's Guide).

**Data line to specify the number of solution-dependent state variables:**

First line:

1. Number of solution-dependent state variables required at each integration point.

**Optional data lines to specify output descriptions for select solution-dependent state variables:**

Second line:

1. Index of the solution-dependent state variable for which an output key and a description are being given. This value is 1 for the first solution-dependent state variable.
2. The output variable key. The key is treated as a label; therefore, it must adhere to the conventions for labels (see "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Guide), 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," Section 1.2.1 of the Abaqus Analysis User's Guide), with the exception that case will be preserved.

Repeat this data line for each solution-dependent state variable for which an output key and a description are being defined. If an output key and a description are not given for a solution-dependent state variable, the default output identifier SDV*n* and description "Solution-dependent state variables" will be used.

## 4.15 *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.)

**Product:** Abaqus/Design

**Type:** Model data

**Level:** Part, Part instance, Assembly, Model, Step

**References:**

- "Design sensitivity analysis," Section 19.1.1 of the Abaqus Analysis User's Guide
- *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.

## 4.16 *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.

**Product:** Abaqus/Design

**Type:** Model data

**Level:** Model

**References:**

- "Design sensitivity analysis," Section 19.1.1 of the Abaqus Analysis User's Guide
- *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.

## 4.17 *DESIGN RESPONSE: Specify responses for 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.

**Product:** Abaqus/Design

**Type:** History data

**Level:** Step

**References:**

- "Design sensitivity analysis," Section 19.1.1 of the Abaqus Analysis User's Guide
- *CONTACT RESPONSE
- *ELEMENT RESPONSE
- *NODE RESPONSE

**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.

## 4.18 *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," Section 25.2.1 of the Abaqus Analysis User's Guide

**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," Section 1.2.2 of the Abaqus Analysis User's Guide). The default is 0.

Repeat this data line as often as necessary to define each detonation point.

## 4.19 *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/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Load module

**References:**

- "Pore fluid flow," Section 34.4.7 of the Abaqus Analysis User's Guide
- "DFLOW," Section 1.1.2 of the Abaqus User Subroutines Reference Guide

**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," Section 6.1.2 of the Abaqus Analysis User's Guide). Amplitude references are ignored for flows defined in user subroutine **DFLOW**.

OP

Set OP=MOD (default) for existing *DFLOWs to remain, with this option modifying existing flows or defining additional flows.

Set OP=NEW if all existing *DFLOWs 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 Part VI, "Elements," of the Abaqus Analysis User's Guide).

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 Part VI, "Elements," of the Abaqus Analysis User's Guide).
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.

## 4.21 *DFLUX: Specify distributed fluxes in heat transfer, computational fluid dynamics, or mass diffusion 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. In Abaqus/CFD this option is used to specify distributed volumetric heat sources.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Load module

**References:**

- "Thermal loads," Section 34.4.4 of the Abaqus Analysis User's Guide
- "DFLUX," Section 1.1.3 of the Abaqus User Subroutines Reference Guide

**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," Section 34.1.2 of the Abaqus Analysis User's Guide).

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," Section 6.1.2 of the Abaqus Analysis User's Guide). If this parameter is omitted in an Abaqus/Explicit or Abaqus/CFD analysis, the reference magnitude is applied immediately at the beginning of the step.

For nonuniform fluxes of type BFNU and S*n*NU (which are 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 *DFLUXs to remain, with this option modifying existing fluxes or defining additional fluxes.

Set OP=NEW if all existing *DFLUXs 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 Part VI, “Elements,” of the Abaqus Analysis User’s Guide).
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.

## 4.22 *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 all parameter values remain the same during the analysis until they are redefined explicitly in the beginning of the next step.

**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," Section 6.3.3 of the Abaqus Analysis User's Guide
- "Output and diagnostics for ALE adaptive meshing in Abaqus/Explicit," Section 12.2.5 of the Abaqus Analysis User's Guide
- "Contact diagnostics in an Abaqus/Explicit analysis," Section 39.2.1 of the Abaqus Analysis User's Guide
- "Linear elastic behavior," Section 22.2.1 of the Abaqus Analysis User's Guide
- "Hyperelastic behavior of rubberlike materials," Section 22.5.1 of the Abaqus Analysis User's Guide
- *CONTACT CONTROLS

### 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 (default) to write all of 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 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 slaves being on opposite sides of master surfaces in the initial configuration.

Set DETECT CROSSED SURFACES=OFF to suppress this diagnostic.

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 may cause the analysis to run for a longer time. It is currently available only for Mises plasticity.

Set PLASTICITY=OFF to suppress all of 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 parameter:**

NONHYBRID INCOMPRESSIBLE

If a model has nearly incompressible elastic or hyperelastic materials with an effective initial Poisson's ratio greater than 0.495 (i.e., the ratio of the initial bulk modulus to the initial shear modulus is greater than 100) used in nonbybrid 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.

**There are no data lines associated with this option.**

## 4.23 *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

**Reference:**

- "Piezoelectric behavior," Section 26.5.2 of the Abaqus Analysis User's Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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, $\theta$.
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 $\mathrm{C}\varphi^{-1}\mathrm{L}^{-1}$.)
2. $D_{22}^{\varphi(\varepsilon)}$.
3. $D_{33}^{\varphi(\varepsilon)}$.
4. Temperature, $\theta$.
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 $\mathrm{C}\varphi^{-1}\mathrm{L}^{-1}$.)
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, $\theta$.
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.

## 4.24 *DIFFUSIVITY: Specify mass diffusivity.

This option is used to define the mass diffusivity of a material diffusing through a base material. 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," Section 26.4.1 of the Abaqus Analysis User's Guide
- *SOLUBILITY

**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 may still depend on concentration and temperature. See "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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 diffusivity (TYPE=ISO):**

First line:

1. Diffusivity, $D$. (Units of $L^2T^{-1}$.)
2. Concentration, $c$.
3. Temperature, $\theta$.
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 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, $\theta$.
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 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, $\theta$.

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.

## 4.25 *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," Section 6.2.6 of the Abaqus Analysis User's Guide
- "Low-cycle fatigue analysis using the direct cyclic approach," Section 6.2.7 of the Abaqus Analysis User's Guide
- *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.

## 4.26 *DISCRETE SECTION: Specify element properties for discrete elements.

This option is used to define properties of discrete elements.

**Product:** Abaqus/Explicit

**Type:** Model data

**Level:** Part, Part instance

**Reference:**

- "Discrete element method," Section 15.1.1 of the Abaqus Analysis User's Guide

**Required parameters:**

ELSET

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

DENSITY

Set this parameter equal to a numerical value or to the name of a distribution (see "Distribution definition," Section 2.8.1 of the Abaqus Analysis User's Guide) to be used with these elements.

SHAPE

This parameter is used to specify the shape of the discrete element.
Set this parameter equal to SPHERE (default).

**Optional parameter:**

ALPHA

Set this parameter equal to the value of the mass proportional damping factor for discrete elements. The default value is 0.0.

**Data line for discrete elements:**

First (and only) line:

1. Enter the name of the distribution for the radius of the discrete elements or the value of the radius.

## 4.27 *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

**Reference:**

- "Display body definition," Section 2.9.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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).

## 4.28 *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," Section 35.3.2 of the Abaqus Analysis User's Guide
- *COUPLING

**Optional parameters:**

COUPLING

Set this parameter equal to the coupling method used to couple the displacement and rotation of the reference node to the average motion of the surface nodes within the influence radius.

Set COUPLING=CONTINUUM (default) to couple the displacement and rotation of each attachment point to the average displacement of the surface nodes within the influence radius.

Set COUPLING=STRUCTURAL to couple the displacement and rotation of each attachment point to the average displacement and rotation of the surface nodes within the influence radius. This parameter value is available only in three-dimensional analyses.

WEIGHTING METHOD

Defines an optional weighting method to modify the default weight distribution at the coupling nodes.

Set WEIGHTING METHOD=UNIFORM to select a uniform weight distribution equal to 1.0. This is the default.

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,” Section 1.2.2 of the Abaqus Analysis User’s Guide, 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).

## 4.29 *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. A DCOUP* element, together with the *DISTRIBUTING COUPLING option, must be used if a point mass at the reference node needs to be distributed as well.

**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.

**Reference:**

- "Distributing coupling elements," Section 32.4.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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 parameter:**

MASS

Set this parameter equal to the mass to be distributed to the coupling 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.

## 4.30 *DISTRIBUTION: Define spatial distributions.

This option is used to define a spatial distribution.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** Model data

**Level:** Part, Part instance, Assembly, Model

**Abaqus/CAE:** Property module

**References:**

- "Distribution definition," Section 2.8.1 of the Abaqus Analysis User's Guide
- *CONTACT CLEARANCE
- *DENSITY
- *DISTRIBUTION TABLE
- *ELASTIC
- *EXPANSION
- *FLUID BOUNDARY
- *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=NONE to define a distribution used with a 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 parameter:**

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Guide, 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.

**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 or membrane 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 or membrane 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 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. $\nu$.

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. $\nu_{12}$.
6. $\nu_{13}$.
7. $\nu_{23}$.
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. $\nu_{12}$.
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. $\alpha$.

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. $\alpha_{11}$.
3. $\alpha_{22}$.
4. $\alpha_{33}$. (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. $\alpha_{11}$.
3. $\alpha_{22}$.
4. $\alpha_{33}$. (Not used for plane stress case.)
5. $\alpha_{12}$.
6. $\alpha_{13}$.
7. $\alpha_{23}$.

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 pressure that varies with the total volume of fluid crossing a surface in Abaqus/CFD:**

First line when specifying pressure first:

1. Pressure.
2. Volume.

First line when specifying volume first:

1. Volume.
2. Pressure.

Repeat this data line as often as necessary to define the data for pressure and volume.

## 4.31 *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/CFD Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "Distribution definition," Section 2.8.1 of the Abaqus Analysis User's Guide
- *DISTRIBUTION

**Required parameter:**

NAME
Set this parameter equal to a label that will be used to refer to the distribution table.

**Data line to define a distribution table for shell thickness or initial contact clearance:**

First (and only) line:

1. The "word" LENGTH.

**Data line to define a distribution table for shell offset:**

First (and only) line:

1. The "word" RATIO.

**Data lines to define a distribution table for shell stiffness:**

First line:

1. The "word" SHELLSTIFF1.
2. The "word" SHELLSTIFF1.
3. The "word" SHELLSTIFF1.
4. The "word" SHELLSTIFF1.
5. The "word" SHELLSTIFF1.
6. The "word" SHELLSTIFF1.
7. The "word" SHELLSTIFF2.

Second line:

1. The “word” SHELLSTIFF2.
2. The “word” SHELLSTIFF2.
3. The “word” SHELLSTIFF3.
4. The “word” SHELLSTIFF2.
5. The “word” SHELLSTIFF2.
6. The “word” SHELLSTIFF2.
7. The “word” SHELLSTIFF3.
8. The “word” SHELLSTIFF3.

Third line:

1. The “word” SHELLSTIFF2.
2. The “word” SHELLSTIFF2.
3. The “word” SHELLSTIFF2.
4. The “word” SHELLSTIFF3.
5. The “word” SHELLSTIFF3.
6. The “word” SHELLSTIFF3.

**Data line to define a distribution table for the coordinates of points $a$ and $b$ used to define a local coordinate system:**

First (and only ) line:

1. The “word” COORD3D.
2. The “word” COORD3D.

**Data line to define a distribution table for an additional rotation angle used to define a local coordinate system:**

First (and only) line:

1. The “word” ANGLE.

**Data line to define a distribution table for isotropic elasticity:**

First (and only) line:

1. The “word” MODULUS.
2. The “word” RATIO.

**Data lines to define a distribution table for orthotropic elasticity:**

First line:

1. The “word” MODULUS.
2. The “word” MODULUS.
3. The “word” MODULUS.
4. The “word” MODULUS.
5. The “word” MODULUS.
6. The “word” MODULUS.
7. The “word” MODULUS.

Second line:

1. The “word” MODULUS.
2. The “word” MODULUS.

**Data lines to define a distribution table for orthotropic elasticity with engineering constants:**

First line:

1. The “word” MODULUS.
2. The “word” MODULUS.
3. The “word” MODULUS.
4. The “word” RATIO.
5. The “word” RATIO.
6. The “word” RATIO.
7. The “word” MODULUS.

Second line:

1. The “word” MODULUS.
2. The “word” MODULUS.

**Data line to define a distribution table for orthotropic elasticity in plane stress:**

First (and only) line:

1. The “word” MODULUS.
2. The “word” MODULUS.
3. The “word” RATIO.
4. The “word” MODULUS.
5. The “word” MODULUS.
6. The “word” MODULUS.

**Data lines to define a distribution table for anisotropic elasticity:**

First line:

1. The “word” MODULUS.
2. The “word” MODULUS.
3. The “word” MODULUS.
4. The “word” MODULUS.
5. The “word” MODULUS.
6. The “word” MODULUS.
7. The “word” MODULUS.

Second line:

1. The “word” MODULUS.
2. The “word” MODULUS.
3. The “word” MODULUS.
4. The “word” MODULUS.
5. The “word” MODULUS.
6. The “word” MODULUS.
7. The “word” MODULUS.
8. The “word” MODULUS.

Third line:

1. The “word” MODULUS.
2. The “word” MODULUS.
3. The “word” MODULUS.
4. The “word” MODULUS.
5. The “word” MODULUS.
6. The “word” MODULUS.

**Data line to define a distribution table for mass density:**

First (and only) line:

1. The “word” DENSITY.

**Data line to define a distribution table for isotropic thermal expansion:**

First (and only) line:

1. The “word” EXPANSION.

**Data line to define a distribution table for orthotropic thermal expansion:**

First (and only) line:

1. The “word” EXPANSION.
2. The “word” EXPANSION.
3. The “word” EXPANSION.

**Data line to define a distribution table for anisotropic thermal expansion:**

First (and only) line:

1. The “word” EXPANSION.
2. The “word” EXPANSION.
3. The “word” EXPANSION.
4. The “word” EXPANSION.
5. The “word” EXPANSION.
6. The “word” EXPANSION.

**Data line to define a distribution table for pressure that varies with the total volume of fluid crossing a surface in Abaqus/CFD:**

First (and only) line when specifying pressure first:

1. The “word” PRESSURE.
2. The “word” VOLUME.

First (and only) line when specifying volume first:

1. The “word” VOLUME.
2. The “word” PRESSURE.

## 4.32 *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 or to apply general body, buoyancy, or porous drag force loading in Abaqus/CFD.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE Abaqus/Aqua

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Load module

**References:**

- "Distributed loads," Section 34.4.3 of the Abaqus Analysis User's Guide
- "DLOAD," Section 1.1.5 of the Abaqus User Subroutines Reference Guide
- "Defining ALE adaptive mesh domains in Abaqus/Explicit," Section 12.2.2 of the Abaqus Analysis User's Guide
- "Analysis of models that exhibit cyclic symmetry," Section 10.4.3 of the Abaqus Analysis User's Guide
- "Abaqus/Aqua analysis," Section 6.11.1 of the Abaqus Analysis User's Guide

### 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," Section 6.1.2 of the Abaqus Analysis User's Guide). If this parameter is

omitted in an Abaqus/Explicit or an Abaqus/CFD 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 *DLOADs to remain, with this option modifying existing distributed loads or defining additional distributed loads.

Set OP=NEW if all existing *DLOADs 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," Section 2.2.5 of the Abaqus Analysis User's Guide) 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 Part VI, “Elements,” of the Abaqus Analysis User’s Guide).
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 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.
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.
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.
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 in an Abaqus/Standard or Abaqus/Explicit analysis, all elements in the model that have mass contributions (including point mass elements) are automatically included in an element set called **`_Whole_Model_Gravity_Elset`** and the gravity load is applied to all elements in this element set. If this field is left blank in an Abaqus/CFD analysis, the gravity load is applied to all user-defined element sets.
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. For Abaqus/CFD gravity loading defines the gravity vector used with a Boussinesq-type body force in buoyancy driven flow (see "Specifying gravity loading" in "Distributed loads," Section 34.4.3 of the Abaqus Analysis User's Guide).

Repeat this data line as often as necessary to define gravity loading for different elements or element sets.

**Data lines to define porous drag load (Abaqus/CFD only):**

First line:

1. Element number or element set label.
2. Distributed load type label PDBF.
3. Value of the porosity, which can be modified by the use of the *AMPLITUDE option.

Repeat this data line as often as necessary to define porous drag body force loads 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," Section 6.1.2 of the Abaqus Analysis User's Guide). 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 *DLOADs to remain, with this option modifying existing loads or defining additional loads.

Set OP=NEW if all existing *DLOADs 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:

5. Density of fluid inside the element.
6. Effective inner diameter of the element.
7. 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" in "Abaqus/Aqua analysis," Section 6.11.1 of the Abaqus Analysis User's Guide. Gravity waves do not affect the buoyancy loading when any external fluid property is overridden.

8. Density of the fluid outside the element.
9. Free surface elevation of the fluid outside the element.
10. 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, $\alpha_R$. 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, $\alpha_R$. 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, $\Delta A$.
5. Drag coefficient, $C$.
6. Structural velocity factor, $\alpha_R$. 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.

## 4.33 *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.

**Product:** Abaqus/Explicit

**Type:** Model data

**Level:** Assembly

**Reference:**

- "Parallel execution in Abaqus/Explicit," Section 3.5.3 of the Abaqus Analysis User's Guide

**Optional parameter:**

ELSET

Set this parameter equal to the name of the element set to define a domain decomposition region. This element set will be split into the user-specified number of parallel domains. If this parameter is omitted, this domain decomposition region will include all elements not included in other domain decomposition regions.

**Data lines to define constraints on domain decomposition:**

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.

## 4.34 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance

**Reference:**

- "Drag chains," Section 32.11.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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, $\ell$.
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).

## 4.35 *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," Section 23.3.1 of the Abaqus Analysis User's Guide
- *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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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, $\epsilon$. 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," Section 23.3.1 of the Abaqus Analysis User's Guide).

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, $\beta$, 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, $\psi$, 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, $\beta$, at high confining pressure in the $p$–$q$ plane. Give the value in degrees.
2. Initial hydrostatic tension strength, $p_t|_0$. (Units of $\mathrm{FL}^{-2}$.)
3. Not used.
4. Dilation angle, $\psi$, 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, $\psi$, 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, $\psi$, 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.

## 4.36 *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," Section 23.3.1 of the Abaqus Analysis User's Guide
- *DRUCKER PRAGER
- *DRUCKER PRAGER HARDENING
- "CREEP," Section 1.1.1 of the Abaqus User Subroutines Reference Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

LAW

Set LAW=STRAIN (default) to choose a strain-hardening power law.
Set LAW=TIME to choose a time-hardening power law.
Set LAW=SINGHM to choose a Singh-Mitchell type law.
Set LAW=USER to input the creep law using user subroutine **`CREEP`**.

TIME

This parameter is relevant only when LAW=TIME or LAW=SINGHM 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 $\mathrm{F}^{-n}\mathrm{L}^{2n}\mathrm{T}^{-1-m}$.)
2. $n$.
3. $m$.
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 creep constants on temperature and other predefined field variables.

**Data lines for LAW=SINGHM:**

First line:

1. $A$. (Units of $\mathrm{T}^{-1}$.)
2. $\alpha$. (Units of $\mathrm{F}^{-1}\mathrm{L}^{2}$.)
3. $m$.
4. $t_1$. (Units of T.)
5. Temperature.
6. First field variable.
7. 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 set of data lines as often as necessary to define the dependence of the creep constants on temperature and other predefined field variables.

## 4.37 *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," Section 23.3.1 of the Abaqus Analysis User's Guide
- *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," Section 21.1.2 of the Abaqus Analysis User's Guide, 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, $\sigma_c$, as a function of uniaxial compression plastic strain, $\bar{\varepsilon}^{pl} = |\varepsilon_{11}^{pl}|$.

Set TYPE=TENSION to define the hardening behavior by giving the uniaxial tension yield stress, $\sigma_t$, as a function of uniaxial tension plastic strain, $\bar{\varepsilon}^{pl} = \varepsilon^{pl}_{11}$.

Set TYPE=SHEAR to define the hardening behavior by giving the cohesion, $d = \frac{\sqrt{3}}{2}\tau(1+\frac{1}{K})$, as a function of equivalent shear plastic strain, $\bar{\varepsilon}^{pl} = \gamma^{pl}/\sqrt{3}$, where $\tau$ is the yield stress in shear, $K$ is the ratio of flow stress in triaxial tension to the flow stress in triaxial compression, and $\gamma^{pl}$ 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.

## 4.38 *DSA CONTROLS: Set DSA solution controls.

This option can be used to control the accuracy or efficiency of the DSA computations.

**Product:** Abaqus/Design

**Type:** Model or history data

**Level:** Model, Step

**Reference:**

- "Design sensitivity analysis," Section 19.1.1 of the Abaqus Analysis User's Guide

**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=$1.0 \times 10^{-4}$.

**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.

## 4.39 *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

**Reference:**

- "Piezoelectric analysis," Section 6.7.2 of the Abaqus Analysis User's Guide

**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," Section 6.1.2 of the Abaqus Analysis User's Guide).

OP

Set OP=MOD (default) for existing *DSECHARGEs to remain, with this option defining electric charges to be added or modified. Set OP=NEW if all existing *DSECHARGEs 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.

## 4.40 *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, eddy current, and magnetostatic analyses.

**Products:** Abaqus/Standard Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Load module

**References:**

- "Coupled thermal-electrical analysis," Section 6.7.3 of the Abaqus Analysis User's Guide
- "Fully coupled thermal-electrical-structural analysis," Section 6.7.4 of the Abaqus Analysis User's Guide
- "Eddy current analysis," Section 6.7.5 of the Abaqus Analysis User's Guide
- "Magnetostatic analysis," Section 6.7.6 of the Abaqus Analysis User's Guide

**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," Section 34.1.2 of the Abaqus Analysis User's Guide). 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," Section 6.1.2 of the Abaqus Analysis User's Guide).

OP

Set OP=MOD (default) for existing *DSECURRENTs to remain, with this option defining distributed current densities to be added or modified.

Set OP=NEW if all existing *DSECURRENTs 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 or coupled thermal-electrical-structural analyses:**

First line:

1. Surface name.
2. Distributed current density type label CS.
3. Reference surface current density magnitude. (Units of $CL^{-2}T^{-1}$.)

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.

7. Name of the orientation option ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Guide) 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.

## 4.41 *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," Section 34.4.7 of the Abaqus Analysis User's Guide
- "DFLOW," Section 1.1.2 of the Abaqus User Subroutines Reference Guide

**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," Section 6.1.2 of the Abaqus Analysis User's Guide). Amplitude references are ignored for flows defined in user subroutine **DFLOW**.

OP

Set OP=MOD (default) for existing *DSFLOWs to remain, with this option modifying existing flows or defining additional flows.

Set OP=NEW if all existing *DSFLOWs 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.

## 4.42 *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. In Abaqus/CFD it is used for thermal energy (heat) transport and conjugate heat transfer analyses.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Load module

**References:**

- "Thermal loads," Section 34.4.4 of the Abaqus Analysis User's Guide
- "DFLUX," Section 1.1.3 of the Abaqus User Subroutines Reference Guide

**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," Section 34.1.2 of the Abaqus Analysis User's Guide).

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," Section 6.1.2 of the Abaqus Analysis User's Guide). If this parameter is omitted in an Abaqus/Explicit or Abaqus/CFD 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 *DSFLUXs to remain, with this option modifying existing fluxes or defining additional fluxes.

Set OP=NEW if all existing *DSFLUXs 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.

## 4.43 *DSLOAD: Specify distributed surface loads.

This option is used to prescribe distributed surface loading.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Load module

**References:**

- "Distributed loads," Section 34.4.3 of the Abaqus Analysis User's Guide
- "DLOAD," Section 1.1.5 of the Abaqus User Subroutines Reference Guide
- "Analysis of models that exhibit cyclic symmetry," Section 10.4.3 of the Abaqus Analysis User's Guide
- "Submodeling: overview," Section 10.2.1 of the Abaqus Analysis User's Guide
- "Surface-based submodeling," Section 10.2.3 of the Abaqus Analysis User's Guide
- "Modeling discontinuities as an enriched feature using the extended finite element method," Section 10.7.1 of the Abaqus Analysis User's Guide

### 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," Section 6.1.2 of the Abaqus Analysis User's Guide). 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 *DSLOADs to remain, with this option modifying existing distributed loads or defining additional distributed loads.

Set OP=NEW if all existing *DSLOADs 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,” Section 2.2.5 of the Abaqus Analysis User’s Guide) 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 will be 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 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.

## 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 linear perturbation procedures,” Section 6.1.3 of the Abaqus Analysis User’s Guide).

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 stresses. By default, Abaqus/Standard uses the solution at the last increment of the selected step.

OP

Set OP=MOD (default) for existing *DSLOADs to remain, with this option modifying existing distributed loads or defining additional distributed loads.

Set OP=NEW if all existing *DSLOADs 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.

## 4.44 *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," Section 6.3.2 of the Abaqus Analysis User's Guide
- "Explicit dynamic analysis," Section 6.3.3 of the Abaqus Analysis User's Guide
- "Adiabatic analysis," Section 6.5.4 of the Abaqus Analysis User's Guide

### 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, $\alpha$, 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, $\beta$, 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, $\gamma$, 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.

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 (i.e., 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 $\omega_{\max}$ 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 parameter:**

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," Section 23.1.1 of the Abaqus Analysis User's Guide). 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.

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. $\Delta t_{max}$, 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. $\Delta t$, 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.

## 4.45 *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

**Reference:**

- "Fully coupled thermal-stress analysis," Section 6.5.3 of the Abaqus Analysis User's Guide

**Required parameter:**

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. $\Delta t_{max}$, 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. $\Delta t$, 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.

## 5. E

## 5.1 *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," Section 4.1.2 of the Abaqus Analysis User's Guide
- *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 output format," Section 5.1.2 of the Abaqus Analysis User's Guide, 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," Section 6.3.5 of the Abaqus Analysis User's Guide) and for eigenvalue buckling estimation ("Eigenvalue buckling prediction," Section 6.2.3 of the Abaqus Analysis User's Guide). 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," Section 6.3.5 of the Abaqus Analysis User's Guide) and for eigenvalue buckling estimation ("Eigenvalue buckling prediction," Section 6.2.3 of the Abaqus Analysis User's Guide). 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 Part VI, "Elements," of the Abaqus Analysis User's Guide. 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," Section 4.2.1 of the Abaqus Analysis User's Guide.

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," Section 4.2.2 of the Abaqus Analysis User's Guide.

Repeat this data line as often as necessary to define the list of variables to be output to the selected results file.

## 5.2 *EL PRINT: Define data file requests for element variables.

This option is used to provide tabular printed output of element variables (stresses, strains, etc.).

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**Reference:**

- "Output to the data and results files," Section 4.1.2 of the Abaqus Analysis User's Guide

**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," Section 6.3.5 of the Abaqus Analysis User's Guide), complex eigenvalue extraction ("Complex eigenvalue extraction," Section 6.3.6 of the Abaqus Analysis User's Guide), and for eigenvalue buckling estimation ("Eigenvalue buckling prediction," Section 6.2.3 of the Abaqus Analysis User's Guide). 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 Part VI, "Elements," of the Abaqus Analysis User's Guide. 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," Section 4.2.1 of the Abaqus Analysis User's Guide. 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.

## 5.3 *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," Section 2.8.1 of the Abaqus Analysis User's Guide).

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**Reference:**

- "Linear elastic behavior," Section 22.2.1 of the Abaqus Analysis User's Guide

**Optional parameters:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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," Section 2.8.1 of the Abaqus Analysis User's Guide.

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=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," Section 3.2.37 of the Abaqus Analysis User's Guide for more information.

Set TYPE=TRACTION to define orthotropic shear behavior for warping elements or uncoupled traction behavior for cohesive elements.

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," Section 2.8.1 of the Abaqus Analysis User's Guide.

**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 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. $\nu_{12}$.
5. $\nu_{13}$.
6. $\nu_{23}$.
7. $G_{12}$.
8. $G_{13}$.

Second line:

1. $G_{23}$.
2. Temperature, $\theta$.
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, $\nu$.
3. Temperature, $\theta$.
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. $\nu_{12}$.
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 $\mathrm{FL}^{-2}$.)
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 warping elements or uncoupled traction behavior for cohesive elements (TYPE=TRACTION):**

First line (only line for defining orthotropic shear behavior for warping elements; in this case the data cannot be defined as functions of temperature and/or field variables):

1. $E$ for warping elements; $E_{nn}$ for cohesive elements.
2. $G_1$ for warping elements; $E_{ss}$ for cohesive elements.
3. $G_2$ for 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 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.

## 5.4 *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/CFD Abaqus/CAE

**Type:** Model data

**Level:** Part, Part instance

**Abaqus/CAE:** Not applicable; copying portions of sketches and instancing of parts serve similar purposes.

**Reference:**

- "Element definition," Section 2.2.1 of the Abaqus Analysis User's Guide

**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.**

## 5.5 *ELECTRICAL CONDUCTIVITY: Specify electrical conductivity.

This option is used to define electrical conductivity for coupled thermal-electrical and coupled thermal-electrical-structural elements in coupled thermal-electrical and coupled thermal-electrical-structural analyses. 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," Section 26.5.1 of the Abaqus Analysis User's Guide
- "Coupled thermal-electrical analysis," Section 6.7.3 of the Abaqus Analysis User's Guide
- "Fully coupled thermal-electrical-structural analysis," Section 6.7.4 of the Abaqus Analysis User's Guide
- "Eddy current analysis," Section 6.7.5 of the Abaqus Analysis User's Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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. $\sigma^E_{11}$. (Units of $CT^{-1}L^{-1}\varphi^{-1}$.)
2. $\sigma^E_{22}$.
3. $\sigma^E_{33}$.
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. $\sigma^E_{11}$. (Units of $CT^{-1}L^{-1}\varphi^{-1}$.)
2. $\sigma^E_{22}$.
3. $\sigma^E_{33}$.
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. $\sigma^E_{11}$. (Units of $CT^{-1}L^{-1}\varphi^{-1}$.)
2. $\sigma^E_{12}$.
3. $\sigma^E_{22}$.
4. $\sigma^E_{13}$.
5. $\sigma^E_{23}$.
6. $\sigma^E_{33}$.
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. $\sigma^E_{11}$. (Units of $CT^{-1}L^{-1}\varphi^{-1}$.)
2. $\sigma^E_{12}$.
3. $\sigma^E_{22}$.
4. $\sigma^E_{13}$.
5. $\sigma^E_{23}$.
6. $\sigma^E_{33}$.
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.

## 5.6 *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

**Reference:**

- "Electromagnetic analysis procedures," Section 6.7.1 of the Abaqus Analysis User's Guide
- "Eddy current analysis," Section 6.7.5 of the Abaqus Analysis User's Guide

**Required parameter:**

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.

## 5.7 *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/CFD Abaqus/CAE

**Type:** Model data

**Level:** Part, Part instance, Assembly

**Abaqus/CAE:** Mesh module

**Reference:**

- "Element definition," Section 2.2.1 of the Abaqus Analysis User's Guide

**Required parameter:**

TYPE

Set this parameter equal to the element type, as defined in Part VI, "Elements," of the Abaqus Analysis User's Guide.

For user elements specify the U*n* type identification (see "User-defined elements," Section 32.15.1 of the Abaqus Analysis User's Guide). The *USER ELEMENT option must also appear in the same input file.

For substructures specify the Z*n* type identification (see "Using substructures," Section 10.1.1 of the Abaqus Analysis User's Guide).

**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. Set this parameter equal to the name (with no extension) of the substructure library on which the substructure resides. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Guide, for the syntax of such library names. If no name is specified, the default name is used (see "Using substructures," Section 10.1.1 of the Abaqus Analysis User's Guide).

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Guide, 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 needed in the connectivity (see "Element definition," Section 2.2.1 of the Abaqus Analysis User's Guide, 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, the connectivity of the entire gasket or cohesive element must be specified on the data lines (see "Defining the gasket element's initial geometry," Section 32.6.4 of the Abaqus Analysis User's Guide, and "Defining the cohesive element's initial geometry," Section 32.5.4 of the Abaqus Analysis User's Guide).

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 Part VI, "Elements," of the Abaqus Analysis User's Guide.

Continuation lines (only needed 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 (maximum 80 characters).

## 5.8 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**Reference:**

- "Output," Section 4.1.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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 output format,” Section 5.1.2 of the Abaqus Analysis User’s Guide.

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,” Section 32.15.1 of the Abaqus Analysis User’s Guide). 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.**

## 5.9 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**Reference:**

- "Generating thermal matrices," Section 10.3.2 of the Abaqus Analysis User's Guide

**Optional parameters:**

ASSEMBLE

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.**

## 5.10 *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/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Step module

**References:**

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Guide
- *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:**

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.

If this parameter and the EXTERIOR parameter are omitted, output will be 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.

If this parameter and the ELSET parameter are omitted, output will be written for all the elements in the model.

POSITION

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). In Abaqus/CFD element output always uses POSITION=CENTROIDAL.

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 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 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 needed 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," Section 4.2.1 of the Abaqus Analysis User's Guide, "Abaqus/Explicit output variable identifiers," Section 4.2.2 of the Abaqus Analysis User's Guide, and "Abaqus/CFD output variable identifiers," Section 4.2.3 of the Abaqus Analysis User's Guide.

Repeat the second data line as often as necessary to define the list of variables to be output to the output database.

## 5.11 *ELEMENT RESPONSE: Define element responses for design sensitivity analysis.

This option is used to write element response sensitivities calculated at the integration points to the output database. It must be used in conjunction with the *DESIGN RESPONSE option.

**Product:** Abaqus/Design

**Type:** History data

**Level:** Step

**References:**

- "Design sensitivity analysis," Section 19.1.1 of the Abaqus Analysis User's Guide
- *DESIGN RESPONSE

**Optional parameter:**

ELSET

Set this parameter equal to the name of the element set for which this sensitivity output is being made.

**Data lines to request element 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 "Design sensitivity analysis," Section 19.1.1 of the Abaqus Analysis User's Guide.

Repeat this data line as often as necessary to define the element responses whose sensitivities are to be written to the output database.

## 5.12 *ELGEN: Incremental element generation.

This option is used to generate elements incrementally.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** Model data

**Level:** Part, Part instance

**Abaqus/CAE:** Not applicable; elements are generated when you mesh the model.

**Reference:**

- "Element definition," Section 2.2.1 of the Abaqus Analysis User's Guide

**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 master element, will be assigned.

**Data lines to generate elements incrementally:**

First line:

1. Master element number.
2. Number of elements to be defined in the first row generated, including the master 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 master row to define a layer of elements.

5. Number of rows to be defined, including the master row. The default is 1.
6. Increment in node numbers of corresponding nodes from row to row.
7. Increment in element numbers of corresponding elements from row to row.

If necessary, copy this newly created master layer to define a block of elements.

8. Number of layers to be defined, including the master layer. The default is 1.

9. Increment in node numbers of corresponding nodes from layer to layer.
10. Increment in element numbers of corresponding elements from layer to layer.

Repeat this data line as often as necessary. Each line will generate *N1* × *N2* × *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.

## 5.13 *ELSET: Assign elements to an element set.

This option is used to assign elements to an element set.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** Model or history data

**Level:** Part, Part instance, Assembly, Model, Step

**Abaqus/CAE:** Set toolset

**Reference:**

- "Element definition," Section 2.2.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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.

## 5.14 *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

**Reference:**

- "Embedded elements," Section 35.4.1 of the Abaqus Analysis User's Guide

**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.

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.

## 5.15 *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," Section 41.1.1 of the Abaqus Analysis User's Guide
- *SURFACE PROPERTY

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

**Data lines to define the emissivity of a surface:**

First line:

1. Emissivity, $\epsilon$.
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.

## 5.16 *END ASSEMBLY: End the definition of an assembly.

This option is used to end an assembly definition.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Assembly module

**References:**

- "Defining an assembly," Section 2.10.1 of the Abaqus Analysis User's Guide
- *ASSEMBLY

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

## 5.17 *END INSTANCE: End the definition of an instance.

This option is used to end an instance definition.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD 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:**

- "Defining an assembly," Section 2.10.1 of the Abaqus Analysis User's Guide
- *INSTANCE

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

## 5.18 *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," Section 6.1.4 of the Abaqus Analysis User's Guide
- *LOAD CASE

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

## 5.19 *END PART: End the definition of a part.

This option is used to end a part definition.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Part module

**References:**

- "Defining an assembly," Section 2.10.1 of the Abaqus Analysis User's Guide
- *PART

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

## 5.20 *END STEP: End the definition of a step.

This option is used to end a step definition.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** History data

**Level:** Model

**Abaqus/CAE:** Step module

**References:**

- "Defining an analysis," Section 6.1.2 of the Abaqus Analysis User's Guide
- *STEP

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

## 5.21 *ENERGY EQUATION SOLVER: Specify the linear solver and parameters for solving the conduction equations in an Abaqus/CFD analysis.

This option is used to enable linear solver parameters for solving the conduction equations in Abaqus/CFD. It must be used in conjunction with the *CFD or *HEAT TRANSFER options.

**Products:** Abaqus/CFD

**Type:** History data

**Level:** Step

**References:**

- "Incompressible fluid dynamic analysis," Section 6.6.2 of the Abaqus Analysis User's Guide
- "Uncoupled heat transfer analysis," Section 6.5.2 of the Abaqus Analysis User's Guide
- *CFD
- *HEAT TRANSFER

**Optional parameters:**

CONVERGENCE

Set CONVERGENCE=ON to write convergence information to the log file.
Set CONVERGENCE=OFF (default) to suppress convergence information.

DIAGNOSTICS

Set DIAGNOSTICS=ON to write diagnostic information about the solver to the log file.
Set DIAGNOSTICS=OFF (default) to suppress diagnostic information.

TYPE

Set TYPE=DSCG to enable the Diagonal Scaled Conjugate Gradient linear solver.
Set TYPE=SSORCG to enable the Symmetric Successive Over Relaxation Conjugate Gradient linear solver.
Set TYPE=DSGMRES to enable the Diagonally Scaled Generalized Minimum Residual linear solver.
Set TYPE=DSFGMRES to enable the Diagonally Scaled Flexible Generalized Minimum Residual linear solver.
Set TYPE=AMG to enable the Algebraic Multigrid linear solver.
The default is TYPE=DSCG if the node-centered solver is selected and TYPE=DSFGMRES if the element-centered solver is selected.

**Data line for TYPE=DSCG and TYPE=SSORCG:**

First (and only) line:

1. Maximum number of iterations (default=250).
2. Interval to check linear convergence (default=2).
3. Linear convergence criterion (default=$10^{-5}$).
4. Under-relaxation factor (default=0.7). This value is valid only for a steady-state analysis.

**Data line for TYPE=DSGMRES and TYPE=DSFGMRES:**

First (and only) line:

1. Maximum number of iterations (default=50).
2. Interval to check linear convergence (default=2).
3. Linear convergence criterion (default=$10^{-5}$).
4. Under-relaxation factor (default=0.7). This value is valid only for a steady-state analysis.

**Data lines for TYPE=AMG:**

First line:

1. Maximum number of iterations (default=250).
2. Interval to check linear convergence (default=2).
3. Linear convergence criterion (default=$10^{-5}$).
4. Under-relaxation factor (default=0.7). This value is valid only for a steady-state analysis.

Second line:

1. Specify the algebraic multigrid smoother as CHEBYCHEV, ILU, ICC, or SSOR (default=ICC).
2. Number of pre-coarsening smoothing steps (default=1).
3. Number of post-coarsening smoothing steps (default=1).
4. Specify the Krylov solver as CG, BCGS, or FGMRES (default=CG).

Third line:

1. Specify the algebraic multigrid cycle as V or W (default=V).

## 5.22 *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," Section 4.1.2 of the Abaqus Analysis User's Guide
- *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.**

## 5.23 *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/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Step module

**References:**

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Guide
- *OUTPUT

**Optional parameters:**

ELSET

Set this parameter equal to the name of the element set for which this output request is being made.

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.

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).

**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," Section 4.2 of the Abaqus Analysis User's Guide.

Repeat this data line as often as necessary to define the energy variables to be written to the output database.

## 5.24 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**Reference:**

- "Output to the data and results files," Section 4.1.2 of the Abaqus Analysis User's Guide

**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.**

## 5.25 *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

**Reference:**

- "Modeling discontinuities as an enriched feature using the extended finite element method," Section 10.7.1 of the Abaqus Analysis User's Guide

**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 three times the typical element characteristic length 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.**

## 5.26 *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

**Reference:**

- "Modeling discontinuities as an enriched feature using the extended finite element method," Section 10.7.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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.

TYPE

Set this parameter equal to the type of enriched feature specified on the *ENRICHMENT option. Currently, only TYPE=PROPAGATION CRACK (default) is supported.

**There are no data lines associated with this option.**

## 5.27 *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/CFD Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**Reference:**

- "Equation of state," Section 25.2.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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, $\boldsymbol{R}$. (Units of $\mathrm{JM}^{-1}\theta^{-1}$.)
2. The ambient pressure, $p_A$ (Units of $\mathrm{FL}^{-2}$). 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 $\mathrm{FL}^{-2}$.)
2. $B_s$. (Units of $\mathrm{FL}^{-2}$.)
3. $\omega_s$. (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 $\mathrm{FL}^{-2}$.)
2. $B_g$. (Units of $\mathrm{FL}^{-2}$.)
3. $\omega_g$. (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 $\mathrm{LT}^{-1}$.)
2. $\boldsymbol{A}$. (Units of $\mathrm{FL}^{-2}$.)
3. $\boldsymbol{B}$. (Units of $\mathrm{FL}^{-2}$.)
4. $\omega$. (Dimensionless.)
5. $R_1$. (Dimensionless.)
6. $R_2$. (Dimensionless.)
7. Detonation energy density, $E_0$. (Units of $\mathrm{JM}^{-1}$.)
8. Pre-detonation bulk modulus, $K_{pd}$. (Units of $\mathrm{FL}^{-2}$.)

**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 $\mathrm{FL}^{-2}$.)
2. $f_2$. (Dimensionless.)
3. Volumetric strain $\varepsilon_{\mathrm{vol}}$. (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. $c_0$. (Units of $\mathrm{LT}^{-1}$.)
2. $s$. (Dimensionless.)
3. $\Gamma_0$. (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.

## 5.28 *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," Section 25.2.1 of the Abaqus Analysis User's Guide
- *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 $\mathrm{LT}^{-1}$.)
2. Value of the porosity of the unloaded (virgin) material, $n_0$ . (Dimensionless.)
3. Pressure required to initialize plastic behavior, $p_e$. (Units of $\mathrm{FL}^{-2}$.)
4. Compaction pressure at which all pores are crushed, $p_S$. (Units of $\mathrm{FL}^{-2}$.)

## 5.29 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance

**References:**

- "Elastic-plastic joints," Section 32.10.1 of the Abaqus Analysis User's Guide
- *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," Section 2.2.5 of the Abaqus Analysis User's Guide) that gives the orientation of the local system in the joint.

**Optional parameter:**

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. $\theta$, 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.

## 5.30 *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

**Reference:**

- "Linear constraint equations," Section 35.2.1 of the Abaqus Analysis User's Guide

**Optional parameter:**

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Guide, 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.

## 5.31 *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

**Reference:**

- "Defining Eulerian boundaries," Section 14.1.2 of the Abaqus Analysis User's Guide

**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=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.

## 5.32 *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," Section 14.1.1 of the Abaqus Analysis User's Guide
- "Eulerian mesh motion," Section 14.1.3 of the Abaqus Analysis User's Guide

**Required parameter:**

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," Section 2.2.5 of the Abaqus Analysis User's Guide) 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 $\infty$ (default): maximum scaling of the bounding box in local direction 1.
2. Value between 1.0 and $\infty$ (default): maximum scaling of the bounding box in local direction 2.
3. Value between 1.0 and $\infty$ (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.

## 5.33 *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," Section 14.1.1 of the Abaqus Analysis User's Guide
- "Eulerian elements," Section 32.14.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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," Section 27.1.4 of the Abaqus Analysis User's Guide) 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.

## 5.34 *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," Section 2.8.1 of the Abaqus Analysis User's Guide). In an Abaqus/CFD analysis this option is used to define thermal expansion for computing bouyancy forces.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "Thermal expansion," Section 26.1.2 of the Abaqus Analysis User's Guide
- "Field expansion," Section 26.1.3 of the Abaqus Analysis User's Guide
- "UEXPAN," Section 1.1.30 of the Abaqus User Subroutines Reference Guide

**Optional parameters:**

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," Section 2.8.1 of the Abaqus Analysis User's Guide).

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.

PORE FLUID

This parameter applies only to Abaqus/Standard analyses.

Include this parameter if the thermal expansion of the pore fluid in a porous medium is being defined. The thermal expansion of a fluid must be isotropic, so TYPE=ORTHO and TYPE=ANISO cannot be used if this parameter is included.

TYPE

Set TYPE=ISO (default) to define isotropic expansion. The only option that is available in an Abaqus/CFD analysis is TYPE=ISO.

Set TYPE=ORTHO to define orthotropic expansion.

Set TYPE=ANISO to define fully anisotropic expansion in an Abaqus/Standard analysis.

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," Section 3.2.37 of the Abaqus Analysis User's Guide for more information.

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," Section 2.8.1 of the Abaqus Analysis User's Guide.

USER

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to indicate that user subroutine **`UEXPAN`** will be used to define increments of thermal strain. The TYPE parameter should be used to indicate the level of anisotropy of thermal expansion. The PORE FLUID parameter can also be used to indicate that the thermal expansion of the pore fluid is being defined.

The DEPENDENCIES and ZERO parameters are not relevant if this parameter is used.

ZERO

If the thermal expansion is temperature- or field-variable-dependent, set this parameter equal to the value of $\theta^0$. The default is ZERO=0.

This parameter is not relevant if the USER parameter is included.

In Abaqus/CFD this parameter should be set equal to the reference temperature used for the Boussinesq approximation of buoyancy forces.

**Data lines to define isotropic thermal expansion coefficients (TYPE=ISO with USER parameter omitted):**

First line:

1. $\alpha$ in Abaqus/Standard or Abaqus/Explicit analysis, or $\beta$ in Abaqus/CFD analysis. (Units of $\theta^{-1}$.)
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. $\alpha_{11}$. (Units of $\theta^{-1}$.)
2. $\alpha_{22}$.
3. $\alpha_{33}$. (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 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 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. $\alpha_{11}$. (Units of $\theta^{-1}$.)
2. $\alpha_{22}$.
3. $\alpha_{33}$. (Not used for plane stress and shell cases.)
4. $\alpha_{12}$.
5. $\alpha_{13}$. (Not used for plane stress and shell cases.)
6. $\alpha_{23}$. (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 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 $\theta^{-1}$ 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.

**Data lines to define isotropic field expansion coefficients (TYPE=ISO with USER parameter omitted):**

First line:

1. $\alpha_f$. (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. $\alpha_{f11}$. (Units of $FV_n^{-1}$.)
2. $\alpha_{f22}$.
3. $\alpha_{f33}$.
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. $\alpha_{f_{11}}$. (Units of $FV_n^{-1}$.)
2. $\alpha_{f_{22}}$.
3. $\alpha_{f_{33}}$. (Not used for plane stress case.)
4. $\alpha_{f_{12}}$.
5. $\alpha_{f_{13}}$.
6. $\alpha_{f_{23}}$.
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 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.

## 5.35 *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.

**Product:** Abaqus/Explicit

**Type:** History data

**Level:** Step

**References:**

- "Explicit dynamic analysis," Section 6.3.3 of the Abaqus Analysis User's Guide
- *EXTREME VALUE

**Required parameter:**

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 parameter:**

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," Section 4.2.2 of the Abaqus Analysis User's Guide.
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.

## 5.36 *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.

**Product:** Abaqus/Explicit

**Type:** History data

**Level:** Step

**References:**

- "Explicit dynamic analysis," Section 6.3.3 of the Abaqus Analysis User's Guide
- *EXTREME VALUE

**Required parameter:**

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 parameter:**

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," Section 4.2.2 of the Abaqus Analysis User's Guide.
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.

## 5.37 *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.

**Product:** Abaqus/Explicit

**Type:** History data

**Level:** Step

**References:**

- "Explicit dynamic analysis," Section 6.3.3 of the Abaqus Analysis User's Guide
- *EXTREME ELEMENT VALUE
- *EXTREME NODE VALUE

**Optional parameter:**

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.**

**6. F**

## 6.1 *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.

**Product:** Abaqus/Explicit

**Type:** Model data

**Level:** Model

**References:**

- "Fabric material behavior," Section 23.4.1 of the Abaqus Analysis User's Guide
- "VFABRIC," Section 1.2.4 of the Abaqus User Subroutines Reference Guide
- *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.

## 6.2 *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," Section 22.2.3 of the Abaqus Analysis User's Guide
- *ELASTIC

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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, $S_{\varepsilon}$.
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.

## 6.3 *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," Section 22.2.3 of the Abaqus Analysis User's Guide
- *ELASTIC

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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, $\overset{*}{f}$ ($-1.0 \leq \overset{*}{f} \leq 1.0$). This value is used only for the Tsai-Wu theory and is ignored if $\sigma_{biax}$ is given. The default is zero.
7. Biaxial stress limit, $\sigma_{biax}$. 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.

## 6.4 *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," Section 23.6.1 of the Abaqus Analysis User's Guide
- *CONCRETE
- *TENSION STIFFENING
- *SHEAR RETENTION

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 6.5 *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," Section 35.3.4 of the Abaqus Analysis User's Guide
- *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

Set this parameter equal to the coupling method used to couple the displacement and rotation of each fastening point to the average motion of the surface nodes within the radius of influence from the fastener projection point.

Set COUPLING=CONTINUUM (default) to couple the displacement and rotation of each fastening point to the average displacement of the surface nodes within the radius of influence.

Set COUPLING=STRUCTURAL to couple the displacement and rotation of each fastening point to the average 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,” Section 2.2.5 of the Abaqus Analysis User’s Guide) 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,” Section 1.2.2 of the Abaqus Analysis User’s Guide) 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.

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.

## 6.6 *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," Section 35.3.4 of the Abaqus Analysis User's Guide
- *FASTENER

**Required parameter:**

NAME

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

**Optional parameter:**

MASS

Set this parameter equal to the additional mass that will be distributed to the fastener nodes.

**Data lines to specify the fastener properties:**

First line:

1. Radius, $r$.

Second line:

1. First degree of freedom constrained. See "Conventions," Section 1.2.2 of the Abaqus Analysis User's Guide, 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).

## 6.7 *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

**Type:** History data

**Level:** Step

**References:**

- "Predefined fields," Section 34.6.1 of the Abaqus Analysis User's Guide
- "UFIELD," Section 1.1.32 of the Abaqus User Subroutines Reference Guide
- "VUFIELD," Section 1.2.14 of the Abaqus User Subroutines Reference Guide

**Optional parameter:**

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," Section 34.1.2 of the Abaqus Analysis User's Guide).

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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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. The default is to linearly ramp the field variable back to its initial conditions. 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.

**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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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," Section 34.6.1 of the Abaqus Analysis User's Guide.

**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 $n_1$-direction for beams or gradient through the thickness for shells.
4. Gradient in the $n_2$-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.

**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.

## 6.8 *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.

**Reference:**

- "Output," Section 4.1.1 of the Abaqus Analysis User's Guide

**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.**

## 6.9 *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," Section 4.1.2 of the Abaqus Analysis User's Guide
- *EL FILE
- *ENERGY FILE
- *NODE FILE

**Required parameter:**

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 parameter:**

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.**

## 6.10 *FILM: Define film coefficients and associated sink temperatures.

This option is used to provide film coefficients and sink temperatures for fully coupled thermal-stress analysis. In Abaqus/Standard it is also used to provide film coefficients and sink temperatures 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:** Interaction module

**References:**

- "Thermal loads," Section 34.4.4 of the Abaqus Analysis User's Guide
- "FILM," Section 1.1.6 of the Abaqus User Subroutines Reference Guide

**Optional parameters:**

AMPLITUDE

Set this parameter equal to the name of the *AMPLITUDE option that gives the variation of the sink temperature, $\theta^0$, 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," Section 6.1.2 of the Abaqus Analysis User's Guide). 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 (which are available only in Abaqus/Standard), the sink temperature amplitude is defined in user subroutine **FILM**, and AMPLITUDE references are used only to modify the sink temperature passed into the user subroutine.

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 (which are available only in Abaqus/Standard), the film coefficient amplitude is defined in user subroutine **FILM**, and FILM AMPLITUDE references are used only to modify the film coefficient passed into the user subroutine.

OP

Set OP=MOD (default) for existing *FILMs to remain, with this option modifying existing films or defining additional films.

Set OP=NEW if all existing *FILMs 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.

**Data lines to define sink temperatures and film coefficients:**

First line:

1. Element number or element set label.
2. Film type label (see Part VI, “Elements,” of the Abaqus Analysis User’s Guide).
3. Reference sink temperature value, $\theta^0$. (Units of $\theta$.) For nonuniform film coefficients the sink temperature must be defined in user subroutine **FILM**. If given, this value will be modified by the AMPLITUDE reference and passed into the user subroutine.
4. Reference film coefficient value, $h$ (units of $\mathrm{JT^{-1}L^{-2}\theta^{-1}}$), or name of the film property table defined with the *FILM PROPERTY option. Nonuniform film coefficients must be defined

in user subroutine **FILM**. 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.

## 6.11 *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," Section 34.4.4 of the Abaqus Analysis User's Guide
- *FILM
- *CFILM

**Required parameter:**

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 parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 6.12 *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," Section 4.1.3 of the Abaqus Analysis User's Guide
- *OUTPUT

**Required parameter:**

NAME

Set this parameter equal to a label that will be used to refer to this filter and/or operator.

**Optional parameters:**

HALT

Include this parameter if you want the analysis to stop when the value specified with the LIMIT parameter is reached.

This parameter must be used in conjunction with the LIMIT parameter.

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.

Set INVARIANT=INTERMP to apply the filtering to the intermediate principal stress.

Set INVARIANT=MINP to apply the filtering to the minimum principal stress.

See "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Guide, 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, $\epsilon$.
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.

## 6.13 *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," Section 11.6.1 of the Abaqus Analysis User's Guide
- "Output," Section 4.1.1 of the Abaqus Analysis User's Guide

**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.**

## 6.14 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Pore fluid flow," Section 34.4.7 of the Abaqus Analysis User's Guide
- "FLOW," Section 1.1.7 of the Abaqus User Subroutines Reference Guide

**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," Section 6.1.2 of the Abaqus Analysis User's Guide). 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 *FLOWs to remain, with this option modifying existing flows or defining additional flows.

Set OP=NEW if all existing *FLOWs 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 Part VI, "Elements," of the Abaqus Analysis User's Guide).
3. Reference pore pressure value, $u_w^\infty$. (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 Part VI, "Elements," of the Abaqus Analysis User's Guide).
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 Part VI, "Elements," of the Abaqus Analysis User's Guide).
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, $u_w^\infty$, 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.

## 6.15 *FLEXIBLE BODY: Generate a flexible body from a substructure.

This option is used to generate a flexible body from an Abaqus/Standard substructure. Abaqus/Standard generates several flexible body types that can be used 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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Defining substructures," Section 10.1.2 of the Abaqus Analysis User's Guide
- *SUBSTRUCTURE GENERATE

**Optional parameter:**

TYPE

Set TYPE=ADAMS to generate flexible body entities for the Adams flexible body dynamics solver from MSC.Software Corporation.

Set TYPE=EXCITE to generate the CON6 flexible body entities for the EXCITE flexible body dynamics solver from AVL.

Set TYPE=GENERIC (default) to generate a generic flexible body. A generic flexible body can be converted into Adams, SIMPACK, or EXCITE flexible bodies by the postprocessing programs.

Set TYPE=SID to generate the Standard Input Data representation of the flexible body.

Set TYPE=SIMPACK to generate flexible body entities for the SIMPACK flexible body dynamics solver from SIMPACK AG.

**There are no data lines associated with this option.**

## 6.16 *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

**Reference:**

- "Fluid cavity definition," Section 11.5.2 of the Abaqus Analysis User's Guide

**Required parameter:**

NAME

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

**Optional parameter:**

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.**

## 6.17 *FLUID BOUNDARY: Specify boundary conditions for fluid flow analyses.

This option is used to specify boundary conditions for fluid flow analyses. Boundary conditions can be specified as spatially varying or constant with an associated variation in time.

**Products:** Abaqus/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Load module

**References:**

- "Incompressible fluid dynamic analysis," Section 6.6.2 of the Abaqus Analysis User's Guide
- "Boundary conditions in Abaqus/CFD," Section 34.3.2 of the Abaqus Analysis User's Guide
- *CFD

**Required parameter:**

TYPE

Set TYPE=ELEMENT to specify distributed boundary values at element faces.
Set TYPE=NODE to specify distributed boundary values at nodes.
Set TYPE=PHYSICAL to specify boundary values based on a physical type.
Set TYPE=SURFACE to specify boundary values at a surface.

**Required, mutually exclusive parameters for use with TYPE=PHYSICAL:**

PRESSUREOUTLET

Include this parameter to specify a pressure outlet boundary condition.

SYMMETRY

Include this parameter to specify a symmetry boundary condition.

VELOCITYINLET

Include this parameter to specify a velocity inlet boundary condition.

WALL

Include this parameter to specify a wall boundary condition.

**Required parameter if TYPE=PHYSICAL:**

SURFACE

Set this parameter equal to the name of the surface on which to apply the boundary condition.

**Optional parameters:**

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the variation of the boundary condition magnitude during the step ("Amplitude curves," Section 34.1.2 of the Abaqus Analysis User's Guide).

If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step. This parameter cannot be used when TYPE=PHYSICAL.

DISTRIBUTION

Set this parameter equal to a label that defines the distribution used in conjunction with the PVDEP boundary type ("Distribution definition," Section 2.8.1 of the Abaqus Analysis User's Guide). This parameter is ignored if the boundary type is not PVDEP. This parameter cannot be used when TYPE=PHYSICAL.

OP

Set OP=MOD (default) for existing *FLUID BOUNDARYs to remain, with this option modifying existing boundary conditions or defining additional boundary conditions.

Set OP=NEW if all existing *FLUID BOUNDARYs applied to the model should be removed. New boundary conditions can be defined.

SLIP

The parameter can be used only if the WALL parameter is included. Set SLIP=NO (default) to indicate a no-slip wall boundary condition. Set SLIP=YES to indicate a slip wall boundary condition.

**Data lines for TYPE=ELEMENT:**

First line:

1. Element number or element set label.
2. Distributed boundary condition type label VELX*n*, VELY*n*, VELZ*n*, TEMP*n*, TURBKE*n*, TURBEPS*n*, TURBOMEGA*n*, or TURBNU*n* (see "Active degrees of freedom" in "Boundary conditions in Abaqus/CFD," Section 34.3.2 of the Abaqus Analysis User's Guide).
3. Boundary condition magnitude.

Repeat this data line as often as necessary to specify fixed boundary conditions at different elements and degrees of freedom.

**Data lines for TYPE=NODE:**

First line:

1. Node number or node set label.
2. Distributed boundary condition type label P (see “Active degrees of freedom” in “Boundary conditions in Abaqus/CFD,” Section 34.3.2 of the Abaqus Analysis User’s Guide).
3. Boundary condition magnitude.

Repeat this data line as often as necessary to specify fixed boundary conditions at different nodes and degrees of freedom.

**Data line for TYPE=PHYSICAL if the WALL and SLIP=NO parameters are included:**

First (and only) line:

1. Boundary condition type label TEMP (see “Active degrees of freedom” in “Boundary conditions in Abaqus/CFD,” Section 34.3.2 of the Abaqus Analysis User’s Guide).
2. Boundary condition magnitude or distribution name.
3. Amplitude name (if required).

**Data lines for TYPE=PHYSICAL if the WALL and SLIP=YES parameters are included:**

First line:

1. Boundary condition type label VELX, VELY, VELZ, VELXNU, VELYNU, VELZNU, TEMP, TURBKE, TURBEPS, TURBNU, TURBOMEGA, TURBINTENSITY, TURBLENGTHSCALE, TURBVELOCITYSCALE, or TURBVISCOSITYSCALE (see “Active degrees of freedom” in “Boundary conditions in Abaqus/CFD,” Section 34.3.2 of the Abaqus Analysis User’s Guide).
2. Boundary condition magnitude or distribution name.
3. Amplitude name (if required).

Repeat this data line as often as necessary to specify boundary conditions at the slip wall.

**Data lines for TYPE=PHYSICAL if the PRESSUREOUTLET parameter is included:**

First line:

1. Boundary condition type label P, PNU, TEMP, TURBKE, TURBEPS, TURBNU, TURBOMEGA, TURBINTENSITY, TURBLENGTHSCALE, TURBVELOCITYSCALE, TURBVISCOSITYSCALE, or PASSIVEOUTFLOW (see “Active degrees of freedom” in “Boundary conditions in Abaqus/CFD,” Section 34.3.2 of the Abaqus Analysis User’s Guide).
2. Boundary condition magnitude or distribution name. This value is ignored for PASSIVEOUTFLOW.
3. Amplitude name (if required). This value is ignored for PASSIVEOUTFLOW.

Repeat this data line as often as necessary to specify boundary conditions at the pressure outlet.

**Data lines for TYPE=PHYSICAL if the VELOCITYINLET parameter is included:**

First line:

1. Boundary condition type label VELX, VELY, VELZ, VELN, VELXNU, VELYNU, VELZNU, TEMP, TURBKE, TURBEPS, TURBNU, TURBOMEGA, TURBINTENSITY, TURBLENGTHSCALE, TURBVELOCITYSCALE, or TURBVISCOSITYSCALE (see "Active degrees of freedom" in "Boundary conditions in Abaqus/CFD," Section 34.3.2 of the Abaqus Analysis User's Guide).
2. Boundary condition magnitude or distribution name.
3. Amplitude name (if required).

Repeat this data line as often as necessary to specify boundary conditions at the velocity inlet.

**Data lines for TYPE=SURFACE:**

First line:

1. Surface label.
2. Boundary condition type label VELX, VELY, VELZ, VELXNU, VELYNU, VELZNU, P, PNU, TEMP, TURBKE, TURBEPS, TURBNU, TURBOMEGA, DIST, PVDEP, or PASSIVEOUTFLOW (see "Active degrees of freedom" in "Boundary conditions in Abaqus/CFD," Section 34.3.2 of the Abaqus Analysis User's Guide).
3. Boundary condition magnitude when the boundary type is not PVDEP. Initial volume when the boundary type is PVDEP and the DISTRIBUTION parameter is included. This value is ignored for PASSIVEOUTFLOW.

Repeat this data line as often as necessary to specify boundary conditions at different surfaces and degrees of freedom.

**There are no data lines if the SYMMETRY parameter is specified.**

## 6.18 *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," Section 11.5.2 of the Abaqus Analysis User's Guide
- *FLUID BEHAVIOR

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 6.19 *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

**Reference:**

- "Fluid cavity definition," Section 11.5.2 of the Abaqus Analysis User's Guide

**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

Set this parameter equal to the name of the surface forming the boundary of the fluid cavity. This parameter is required if the ADDED VOLUME parameter is omitted.

**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.

## 6.20 *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," Section 11.5.2 of the Abaqus Analysis User's Guide
- *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, $\rho_R$.

## 6.21 *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," Section 11.5.3 of the Abaqus Analysis User's Guide
- *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.

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.

**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.

## 6.22 *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.

**Product:** Abaqus/Explicit

**Type:** History data

**Level:** Step

**References:**

- "Fluid cavity definition," Section 11.5.2 of the Abaqus Analysis User's Guide
- "Fluid exchange definition," Section 11.5.3 of the Abaqus Analysis User's Guide
- *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," Section 34.1.2 of the Abaqus Analysis User's Guide.

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.

## 6.23 *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," Section 11.5.3 of the Abaqus Analysis User's Guide
- *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.

## 6.24 *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," Section 11.5.2 of the Abaqus Analysis User's Guide
- *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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

ZERO

Set this parameter equal to the value of $\theta_0$. The default is ZERO=0.

**Data lines to define the thermal expansion coefficient:**

First line:

1. Mean coefficient of thermal expansion, $\alpha$.
2. Temperature, $\theta$.
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 $\alpha$ as a function of $\theta$ and field variables.

## 6.25 *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

**Reference:**

- "Fluid exchange definition," Section 11.5.3 of the Abaqus Analysis User's Guide

**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," Section 34.1.2 of the Abaqus Analysis User's Guide). 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 $MT^{-1}$.)

## 6.26 *FLUID INFLATOR: Define a fluid inflator.

This option is used to define a fluid inflator to model deployment of an airbag.

**Product:** Abaqus/Explicit

**Type:** Model data

**Level:** Part, Part instance, Assembly

**References:**

- "Inflator definition," Section 11.5.4 of the Abaqus Analysis User's Guide
- *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.

## 6.27 *FLUID INFLATOR ACTIVATION: Activate fluid inflator definitions.

This option is used to activate fluid inflator definitions.

**Product:** Abaqus/Explicit

**Type:** History data

**Level:** Step

**References:**

- "Inflator definition," Section 11.5.4 of the Abaqus Analysis User's Guide
- *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.

## 6.28 *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.

**Product:** Abaqus/Explicit

**Type:** Model data

**Level:** Part, Part instance

**References:**

- "Fluid cavity definition," Section 11.5.2 of the Abaqus Analysis User's Guide
- "Inflator definition," Section 11.5.4 of the Abaqus Analysis User's Guide
- *FLUID BEHAVIOR
- *FLUID INFLATOR PROPERTY

**Required parameter:**

NUMBER SPECIES

Set this parameter equal to the number of gas species used for this inflator.

**Optional parameter:**

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.

## 6.29 *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.

**Product:** Abaqus/Explicit

**Type:** Model data

**Level:** Part, Part instance

**References:**

- "Inflator definition," Section 11.5.4 of the Abaqus Analysis User's Guide
- *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 parameter:**

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.

## 6.30 *FLUID LEAKOFF: Define fluid leak-off coefficients for pore pressure cohesive elements.

This option is used to define leak-off coefficients for pore pressure cohesive 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," Section 32.5.7 of the Abaqus Analysis User's Guide
- "UFLUIDLEAKOFF," Section 1.1.34 of the Abaqus User Subroutines Reference Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

USER

Include this parameter to indicate that user subroutine **UFLUIDLEAKOFF** will be used to define the fluid leak-off coefficients.

**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.
2. Fluid leak-off coefficient at bottom element surface.
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.

**There are no data lines when the USER parameter is included.**

## 6.31 *FLUID SECTION: Specify element properties for fluid and porous media elements.

This option is used to define the properties of fluid elements and porous media elements.

**Products:** Abaqus/CFD Abaqus/CAE

**Type:** Model data

**Level:** Part, Part instance

**Abaqus/CAE:** Property module

**References:**

- "Fluid (continuum) elements," Section 28.2.1 of the Abaqus Analysis User's Guide
- "Fluid element library," Section 28.2.2 of the Abaqus Analysis User's Guide

**Required parameter:**

ELSET

Set this parameter equal to the name of the element set containing the elements for which the material behavior is being defined.

**Optional parameter:**

TYPE

Set TYPE=SINGLE FLUID (default) for single-phase fluid flow.
Set TYPE=POROUS MEDIA for heat transfer analysis involving porous media.

**Data line to define fluid elements for single-phase fluid flow:**

First (and only) line:

1. Enter the name of the material to be used.

**Data lines to define porous media elements for heat transfer analysis:**

First line:

1. Enter the name of the solid (matrix) material to be used.

Second line:

1. Enter the name of the fluid material to be used.

## 6.32 *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

**Reference:**

- "Element foundations," Section 2.2.2 of the Abaqus Analysis User's Guide

**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.

## 6.33 *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:** Interaction module

**References:**

- "Crack propagation analysis," Section 11.4.3 of the Abaqus Analysis User's Guide
- "Modeling discontinuities as an enriched feature using the extended finite element method," Section 10.7.1 of the Abaqus Analysis User's Guide
- *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 slave 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 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.

**Optional parameters:**

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 “Specifying field variable dependence” in “Material data definition,” Section 21.1.2 of the Abaqus Analysis User’s Guide, for more information.

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=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.

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.

SYMMETRY

Include this parameter to compare the opening between the slave 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.

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 VCCT criterion is satisfied for an unstable crack problem.

If this parameter is included without a specified value, the default value is infinity.

This parameter is meaningful only when used with the *DEBOND option in Abaqus/Standard.

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 and TYPE=VCCT, the default is TOLERANCE=0.2.

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, $\sigma^f$.
2. Shear failure stress, $\tau_1^f$.
3. Shear failure stress, $\tau_2^f$. (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, $\delta_c$.
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, $G^P_{IIC}$.
6. Mode III critical energy release rate for crack propagation, $G^P_{IIIC}$.
7. Exponent, $\eta$.
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, $G^P_{IC}$.
5. Mode II critical energy release rate for crack propagation, $G^P_{IIC}$.
6. Mode III critical energy release rate for crack propagation, $G^P_{IIIC}$.
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. Third field variable.
6. Fourth field variable.
7. Fifth field variable.
8. Sixth field variable.

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) for MIXED MODE BEHAVIOR=BK or REEDER:**

First line:

1. Material constant for fatigue crack initiation, $c_1$.
2. Material constant for fatigue crack initiation, $c_2$.
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, $\eta$.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Third field variable.
7. Fourth field variable.
8. Fifth field variable.

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) for MIXED MODE BEHAVIOR=POWER:**

First line:

1. Material constant for fatigue crack initiation, $c_1$.

2. Material constant for fatigue crack initiation, $c_2$.
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 the VCCT criterion (TYPE=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, $\eta$.
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) 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.

## 6.34 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance

**References:**

- "Frame elements," Section 29.4.1 of the Abaqus Analysis User's Guide
- "Frame section behavior," Section 29.4.2 of the Abaqus Analysis User's Guide

**Required parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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," Section 29.3.9 of the Abaqus Analysis User's Guide). 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 ($\theta^0$), 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," Section 29.4.1 of the Abaqus Analysis User's Guide, 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," Section 29.3.9 of the Abaqus Analysis User's Guide, 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," Section 29.4.1 of the Abaqus Analysis User's Guide, 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.

## 6.35 *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 Abaqus/AMS

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Step module

**Reference:**

- "Natural frequency extraction," Section 6.3.5 of the Abaqus Analysis User's Guide

**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, the projection is not performed.

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 multi-level 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 eignensolver and for the Lanczos eigensolver when they are used without the SIM parameter.

Set NORMALIZATION=MASS to normalize the eigenvectors with respect to the structure's mass matrix (the eigenvectors are scaled so that the generalized mass for each vector is unity). Mass normalization is the default and only available option for the AMS eigensolver. Mass normalization is switched on for both the Lanczos eigensolver and the subspace iteration eigensolver when they are used in conjunction with the 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 this parameter is omitted, Abaqus/Standard will evaluate the stiffness associated with frequency-dependent springs and dashpots at zero frequency and will 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 residual modes are to be computed.

SIM

This parameter is relevant only for the Lanczos and subspace iteration eigensolvers.

Include this parameter to indicate that subsequent mode-based linear dynamic analysis steps should use the high-performance versions based on the SIM software architecture. The SIM parameter is turned on by default 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 eigenvectors will be computed. If this parameter is omitted, eigenvectors will be computed at all nodes.

**Data line for a natural frequency extraction when EIGENSOLVER=LANCZOS:**

First (and only) line:

1. Number of eigenvalues to be calculated. This field can be left blank if the maximum frequency of interest is provided and the evaluation of all the eigenvalues in the given range is desired. The number of requested eigenmodes must be provided 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 this field is left blank, no minimum is set.
3. Maximum frequency of interest, in cycles/time. If this field is left blank, no maximum is set. This value is required if the first field was left blank.
4. Shift point, in squared cycles per time (positive or negative). The eigenvalues closest to this point will be extracted.
5. Block size. If this entry is omitted, a default value, which is usually appropriate, is created.
6. Maximum number of block Lanczos steps within each Lanczos run. If this entry is omitted, a default value, which is usually appropriate, is created.
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. This factor is supported only when using the SIM architecture, and the maximum frequency of interest is provided. The acoustic range factor must be greater than 0. The default value is 1.0.

**Data lines for a natural frequency extraction when EIGENSOLVER=AMS:**

First line:

1. Number of eigenvalues to be calculated. If this field is left 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 this field is left blank, no minimum is set.
3. Maximum frequency of interest, in cycles/time.
4. $AMS_{\text{cutoff}_1}$, the first AMS parameter. $AMS_{\text{cutoff}_1}$ is a cutoff frequency for substructure eigenproblems, defined as a multiplier of the maximum frequency of interest. The default value is 5.
5. $AMS_{\text{cutoff}_2}$, the second AMS parameter. $AMS_{\text{cutoff}_2}$ is the first cutoff frequency used to define a starting subspace in the reduced eigensolution phase, defined as a multiplier of the maximum frequency of interest. $AMS_{\text{cutoff}_2} < AMS_{\text{cutoff}_1}$. The default value is 1.7.
6. $AMS_{\text{cutoff}_3}$, the third AMS parameter. $AMS_{\text{cutoff}_3}$ is the second cutoff frequency used to define a starting subspace in the reduced eigensolution phase, defined as a multiplier of the maximum frequency of interest. $1.0 < AMS_{\text{cutoff}_3} < AMS_{\text{cutoff}_1}$. The default value is 1.1.
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. The acoustic range factor must be greater than 0. The default value is 1.0.

No additional data lines are needed if default residual modes are sufficient or residual modes are not requested. Otherwise, subsequent lines:

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. This field can be left blank if residual modes for only one degree of freedom are requested.

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. This user-specified maximum frequency is increased automatically by 12.5% to help capture closely-spaced modes. Abaqus/Standard will also report all eigenvalues that converge in the same iteration as those in the specified range, even if their frequencies are more than 12.5% above the maximum frequency specified by the user. If this field is left blank, no maximum is set.

Abaqus/Standard will extract frequencies until either of the above limits is reached.

3. Shift point, in squared cycles per time (positive or negative). The eigenvalues closest to this point will be extracted.
4. Number of vectors used in the iteration. If this entry is omitted, a default value, which is usually appropriate, is created. 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. Thus, if the user knows that a particular type of eigenproblem converges slowly, providing more vectors by using this option might reduce the analysis cost.
5. Maximum number of iterations. The default is 30.

## 6.36 *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," Section 31.2.1 of the Abaqus Analysis User's Guide
- "Mechanical contact properties: overview," Section 37.1.1 of the Abaqus Analysis User's Guide
- "Frictional behavior," Section 37.1.5 of the Abaqus Analysis User's Guide
- "FRIC," Section 1.1.8 of the Abaqus User Subroutines Reference Guide
- "FRIC_COEF," Section 1.1.9 of the Abaqus User Subroutines Reference Guide
- "VFRIC," Section 1.2.5 of the Abaqus User Subroutines Reference Guide
- "VFRIC_COEF," Section 1.2.6 of the Abaqus User Subroutines Reference Guide
- "VFRICTION," Section 1.2.7 of the Abaqus User Subroutines Reference Guide
- *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 ($\gamma_i$) 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

This parameter applies only to Abaqus/Standard analyses and cannot be used when friction is defined for connector elements.

Include this parameter if anisotropic friction is being defined.

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 “Specifying field variable dependence” in “Material data definition,” Section 21.1.2 of the Abaqus Analysis User’s Guide, 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 subroutine **FRIC** in an Abaqus/Standard analysis or for user subroutines **VFRIC** and **VFRICTION** 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 and TAUMAX parameters cannot be used with this parameter.

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, $\bar{\tau}_{max}$; that is, the maximum achievable value of the equivalent shear stress. If no value is given or TAUMAX=0 in an Abaqus/Standard analysis, there is no limit on the equivalent shear stress. A value of zero is not allowed in an Abaqus/Explicit analysis.

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 parameters are omitted:**

First line:

1. Friction coefficient, $\mu$.
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, $\bar{\theta}$, 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, $\bar{f}^1$.
6. Average value of the second field variable, $\bar{f}^2$.
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, $\bar{f}^5$.
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 parameter is used and the USER, ROUGH, and EXPONENTIAL DECAY parameters are omitted:**

First line:

1. Friction coefficient in the first slip direction, $\mu_1$.
2. Friction coefficient in the second slip direction, $\mu_2$.
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, $\bar{\theta}$, 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, $\bar{f}^1$.
7. Average value of the second field variable, $\bar{f}^2$.
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, $\bar{f}^4$.
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 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, $\mu_s$.
2. Kinetic friction coefficient, $\mu_k$.
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, $\mu_1$. This value corresponds to the static friction coefficient.

Second line:

1. Friction coefficient for the second data point, $\mu_2$. 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, $\mu_\infty$. 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 $\mu_\infty$ such that $(\mu_2 - \mu_\infty)/(\mu_1 - \mu_\infty) = 0.05$.

**Data line when the ROUGH parameter is used:**

There are no data lines in this case.

**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.

**Data line when the USER parameter is used without the PROPERTIES parameter:**

There are no data lines in this case.

# 7. G

## 7.1 *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.

**Reference:**

- "Gap contact elements," Section 40.2.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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.

2. Global $X$-direction cosine of the contact direction.
3. Global $Y$-direction cosine of the contact direction.
4. Global $Z$-direction cosine of the contact direction.
5. 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.

## 7.2 *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.

**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," Section 37.2.1 of the Abaqus Analysis User's Guide
- *GAP
- *INTERFACE
- *SURFACE INTERACTION
- "GAPCON," Section 1.1.10 of the Abaqus User Subroutines Reference Guide

**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.

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, $k$. (Units of $JT^{-1}L^{-2}\,\theta^{-1}$.)
2. Gap clearance, $d$, or gap pressure, $p$.
3. Average temperature, $\bar{\theta}$.

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, $\bar{f}_1$.
6. Average value of the second field variable, $\bar{f}_2$.
7. Etc.

Repeat this data line as often as necessary to define the dependence of gap conductance on gap clearance, gap pressure, average surface temperature, average mass flow rate, and any predefined field variables. At least two data lines must be specified. When gap conductance is defined as a function of clearance, the value of the conductance drops to zero immediately after the last data point; therefore, there is no heat conductance when the clearance is greater than the value corresponding to the last data point.

**Data lines to define the gap conductance (*k*) by a user subroutine:**

There are no data lines when the USER parameter is used. Instead, define the gap conductance in user subroutine **GAPCON**.

## 7.3 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Model

**References:**

- "Fully coupled thermal-stress analysis," Section 6.5.3 of the Abaqus Analysis User's Guide
- "Fully coupled thermal-electrical-structural analysis," Section 6.7.4 of the Abaqus Analysis User's Guide
- "Electrical contact properties," Section 37.3.1 of the Abaqus Analysis User's Guide
- *SURFACE INTERACTION
- "GAPELECTR," Section 1.1.11 of the Abaqus User Subroutines Reference Guide

**Optional parameters:**

DEPENDENCIES

Set this parameter equal to the number of field variables on which $\sigma_g$ depends.

PRESSURE

Include this parameter to indicate that $\sigma_g$ is a function of the contact pressure between the surfaces, $p$. Omit this parameter to define $\sigma_g$ as a function of the clearance, $d$, between the surfaces.

USER

Include this parameter to define $\sigma_g$ 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:**

First line:

1. Electrical conductivity, $\sigma_g$. (Units of $\mathrm{CT^{-1}L^{-2}}\varphi^{-1}$.)
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.

## 7.4 *GAP FLOW: Define constitutive parameters for tangential flow in pore pressure cohesive elements.

This option is used to define tangential flow constitutive parameters for pore pressure cohesive 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

**Reference:**

- "Defining the constitutive response of fluid within the cohesive element gap," Section 32.5.7 of the Abaqus Analysis User's Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

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.

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.

**Data lines to define the pore fluid viscosity $(\mu)$ (TYPE=NEWTONIAN):**

First line:

1. $\mu$.
2. Temperature, $\theta$.
3. First field variable.

4. Second 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 variation.

**Data lines to define the consistency, $K$, and exponent, $\alpha$ (TYPE=POWER LAW):**

First line:

1. $K$.
2. $\alpha$.
3. Temperature, $\theta$.
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 variation.

## 7.5 *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," Section 37.1.5 of the Abaqus Analysis User's Guide
- "Thermal contact properties," Section 37.2.1 of the Abaqus Analysis User's Guide
- "Electrical contact properties," Section 37.3.1 of the Abaqus Analysis User's Guide
- *GAP
- *INTERFACE
- *SURFACE INTERACTION

**There are no parameters associated with this option.**

**Data line to define the gap heat generation:**

First (and only) line:

1. $\eta$, 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 slave surface is weighted by $f$, and the heat flux into the master surface is weighted by $1 - f$. By default, the heat is distributed equally between the two surfaces, $f = 0.5$.

## 7.6 *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.

**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," Section 37.2.1 of the Abaqus Analysis User's Guide
- *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, $\epsilon_A$.
2. Emissivity, $\epsilon_B$.

Second line:

1. Effective view factor, $F$. $(0 \leq F \leq 1)$
2. Gap clearance, $d$.

Repeat this data line as often as necessary to define the dependence of the view factor on gap clearance.

## 7.7 *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

**Reference:**

- "Equation of state," Section 25.2.1 of the Abaqus Analysis User's Guide

**Optional parameter:**

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.

## 7.8 *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

**Reference:**

- "Defining the gasket behavior directly using a gasket behavior model," Section 32.6.6 of the Abaqus Analysis User's Guide

**Required parameter:**

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.**

## 7.9 *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

**Reference:**

- "Defining the gasket behavior directly using a gasket behavior model," Section 32.6.6 of the Abaqus Analysis User's Guide

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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, $\theta$.
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.

## 7.10 *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

**Reference:**

- "Defining the gasket behavior directly using a gasket behavior model," Section 32.6.6 of the Abaqus Analysis User's Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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, $\theta$.
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, $\nu$.
3. Temperature, $\theta$.
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.

## 7.11 *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:**

- "Gasket elements: overview," Section 32.6.1 of the Abaqus Analysis User's Guide
- "Defining the gasket behavior using a material model," Section 32.6.5 of the Abaqus Analysis User's Guide
- "Defining the gasket behavior directly using a gasket behavior model," Section 32.6.6 of the Abaqus Analysis User's Guide

**Required parameter:**

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," Section 2.2.5 of the Abaqus Analysis User's Guide) 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.

## 7.12 *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

**Reference:**

- "Defining the gasket behavior directly using a gasket behavior model," Section 32.6.6 of the Abaqus Analysis User's Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

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. (This value cannot be negative.)
3. Temperature, $\theta$.
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. (This value cannot be negative.)
3. Temperature, $\theta$.
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. (This value must be positive.)
3. Plastic closure. (This value must be positive.)
4. Temperature, $\theta$.
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. (This value must be positive.)
3. Plastic closure. (This value must be positive.)
4. Temperature, $\theta$.
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. (This value cannot be negative.)
3. Maximum closure reached while loading the gasket. (This value must be positive.)
4. Temperature, $\theta$.
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. (This value cannot be negative.)
3. Maximum closure reached while loading. (This value must be positive.)
4. Temperature, $\theta$.
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.

## 7.13 *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

**Reference:**

- "Swelling gel," Section 26.6.5 of the Abaqus Analysis User's Guide

**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, $r_a^{\text{dry}}$. (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, $\tau_1$. (Units of T.)

## 7.14 *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

**Reference:**

- "Geostatic stress state," Section 6.8.2 of the Abaqus Analysis User's Guide

**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.

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.

## 7.15 *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," Section 26.1.1 of the Abaqus Analysis User's Guide
- "Damping in dynamic analysis" in "Dynamic analysis procedures: overview," Section 6.3.1 of the Abaqus Analysis User's Guide
- "Acoustic, shock, and coupled acoustic-structural analysis," Section 6.10.1 of the Abaqus Analysis User's Guide

**Optional parameters:**

ALPHA

Set this parameter equal to the $\alpha_{global}$ 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 $\beta_{global}$ 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.**

## 8. H

## 8.1 *HEADING: Print a heading on the output.

This option is used to define a title for the analysis.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Job module

**Reference:**

- "Defining a model in Abaqus," Section 1.3.1 of the Abaqus Analysis User's Guide

**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.

## 8.2 *HEAT GENERATION: Include volumetric heat generation in heat transfer analyses.

This option is used in a material data block to include 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**.

**Products:** Abaqus/Standard Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "Uncoupled heat transfer analysis," Section 6.5.2 of the Abaqus Analysis User's Guide
- "Fully coupled thermal-electrical-structural analysis," Section 6.7.4 of the Abaqus Analysis User's Guide
- "HETVAL," Section 1.1.13 of the Abaqus User Subroutines Reference Guide

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

## 8.3 *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/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Step module

**References:**

- "Uncoupled heat transfer analysis," Section 6.5.2 of the Abaqus Analysis User's Guide
- "Cavity radiation," Section 41.1.1 of the Abaqus Analysis User's Guide

**Required parameters to activate an Abaqus/CFD solid heat transfer analysis:**

CENTERING

Set CENTERING=ELEMENT to select an element-centered heat transfer analysis.

TYPE

Set TYPE=THERMAL FLOW to indicate a heat transfer analysis using Abaqus/CFD.

**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. If this parameter is used in an Abaqus/CFD solid heat transfer analysis, the values given on the data line are ignored.

**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 $1\times10^{-5}$ 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.

**Data line to define an Abaqus/CFD solid heat transfer analysis (TYPE=THERMAL FLOW):**

First (and only) line:

1. Fixed time increment. This value sets a fixed time increment to be used throughout the analysis.
2. Total time. This value sets the total duration of the analysis.

## 8.4 *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.

**Reference:**

- "Point capacitance," Section 30.4.1 of the Abaqus Analysis User's Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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 ($\rho c V$), 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.

## 8.5 *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," Section 27.1.4 of the Abaqus Analysis User's Guide
- *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.

## 8.6 *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," Section 22.5.1 of the Abaqus Analysis User's Guide
- "UHYPER," Section 1.1.38 of the Abaqus User Subroutines Reference Guide
- *BIAXIAL TEST DATA
- *PLANAR TEST DATA
- *UNIAXIAL TEST DATA
- *VOLUMETRIC TEST DATA

**Optional, mutually exclusive parameters:**

ARRUDA-BOYCE

Include this parameter to use the Arruda-Boyce model, also known as the eight-chain 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

This parameter applies only to Abaqus/Standard analyses.

Include this parameter if the derivatives of the strain energy potential with respect to the strain invariants are defined in user subroutine **UHYPER**.

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 parameter if the USER parameter is included:**

TYPE

This parameter applies only to Abaqus/Standard analyses.

Set TYPE=INCOMPRESSIBLE to indicate that the hyperelastic material defined by **UHYPER** is incompressible.

Set TYPE=COMPRESSIBLE to indicate that the hyperelastic material defined by **UHYPER** 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 $\beta$ 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, $\beta$ 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 parameter value is not available for frequency domain viscoelasticity in an Abaqus/Standard analysis. This is the only option available if the hyperelastic material is defined in user subroutine **UHYPER**.

Set MODULI=LONG TERM to indicate that the hyperelastic material constants define the long-term behavior. This option is not available when user subroutine **UHYPER** is used to define the hyperelastic material. 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, $\nu$, 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 if the nominal lateral strains are specified on the *UNIAXIAL TEST DATA, *BIAXIAL TEST DATA, or *PLANAR TEST DATA option.

PROPERTIES

This parameter applies only to Abaqus/Standard analyses.

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**. 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 is not relevant for the Marlow model, in which case the test data must be specified.

**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 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. $\mu$.
2. $\lambda_m$.
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 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. $\mu_1$.
2. $\alpha_1$.
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. $\mu_1$.
2. $\alpha_1$.
3. $\mu_2$.
4. $\alpha_2$.
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. $\mu_1$.
2. $\alpha_1$.
3. $\mu_2$.

4. $\alpha_2$.
5. $\mu_3$.
6. $\alpha_3$.
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 $\mu_i$ and $\alpha_i$ 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. $\mu$.
2. $\lambda_m$.
3. $a$.
4. $\beta$.
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.

## 8.7 *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," Section 22.5.2 of the Abaqus Analysis User's Guide
- *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, $\nu$, 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 $\nu_i$ are calculated from lateral strain data and/or volumetric test data.

TEST DATA INPUT

Include this parameter if the $\mu_i$, the $\alpha_i$, and the $\nu_i$ material constants are to be computed by Abaqus from data taken from simple tests on a material specimen. If this parameter is omitted, $\mu_i$, $\alpha_i$, and $\nu_i$ 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. $\mu_1$.
2. $\alpha_1$.
3. $\nu_1$.
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. $\mu_1$.
2. $\alpha_1$.
3. $\mu_2$.
4. $\alpha_2$.
5. $\nu_1$.
6. $\nu_2$.
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. $\mu_1$.
2. $\alpha_1$.
3. $\mu_2$.
4. $\alpha_2$.
5. $\mu_3$.

6. $\alpha_3$.
7. $\nu_1$.
8. $\nu_2$.

Second line if N=3:

1. $\nu_3$.
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 $\mu_i$ and $\alpha_i$ 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.

## 8.8 *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," Section 22.4.1 of the Abaqus Analysis User's Guide
- "UHYPEL," Section 1.1.37 of the Abaqus User Subroutines Reference Guide

**Optional parameter:**

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, $\nu$.
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.

## 8.9 *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

**Reference:**

- "Hysteresis in elastomers," Section 22.8.1 of the Abaqus Analysis User's Guide

**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," Section 22.8.1 of the Abaqus Analysis User's Guide.

## 9. I

## 9.1 *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:**

- "Acoustic, shock, and coupled acoustic-structural analysis," Section 6.10.1 of the Abaqus Analysis User's Guide
- "Acoustic and shock loads," Section 34.4.6 of the Abaqus Analysis User's Guide
- *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 parameter:**

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, $\epsilon$, 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.

## 9.2 *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," Section 34.4.6 of the Abaqus Analysis User's Guide
- *IMPEDANCE
- *SIMPEDANCE

**Required parameter:**

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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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," Section 34.4.6 of the Abaqus Analysis User's Guide. (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. $\mathrm{Re(Z)}$, the real part of the surface impedance. (Units of F $L^{-3}$T.)
2. $\mathrm{Im(Z)}$, 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.

## 9.3 *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

**Type:** Model data

**Level:** Model

**References:**

- "Introducing a geometric imperfection into a model," Section 11.3.1 of the Abaqus Analysis User's Guide
- "Unstable collapse and postbuckling analysis," Section 6.2.4 of the Abaqus Analysis User's Guide
- "Eigenvalue buckling prediction," Section 6.2.3 of the Abaqus Analysis User's Guide

**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 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.

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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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.

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 9.3–1.

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:

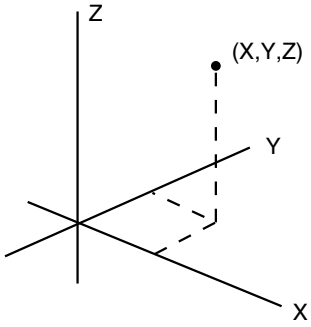
1. Set to 1.
2. Scaling factor.

**Data lines to define the imperfection if the FILE and INPUT parameters are omitted:**

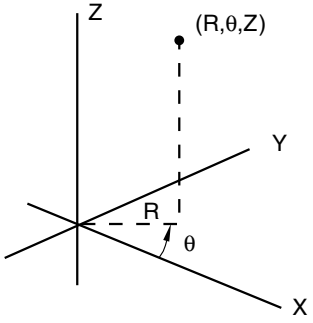
First line:

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.

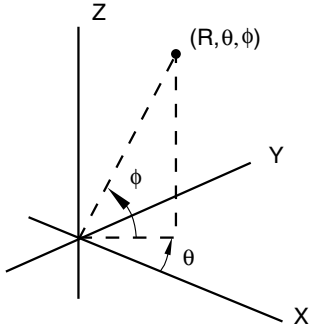
Repeat this data line as often as necessary to define the imperfection.




Rectangular Cartesian
(SYSTEM=R)
(default)




Cylindrical
(SYSTEM=C)




Spherical
(SYSTEM=S)

(θ and ϕ are given in degrees)

**Figure 9.3–1** Coordinate systems.

## 9.4 *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.

**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:**

- "Transferring results between Abaqus analyses: overview," Section 9.2.1 of the Abaqus Analysis User's Guide
- *INSTANCE

**Required parameter:**

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.

**Optional, mutually exclusive parameters:**

INCREMENT

When importing an analysis from Abaqus/Standard into Abaqus/Explicit or from one Abaqus/Standard analysis into another Abaqus/Standard analysis, 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.

INTERVAL

When importing an analysis from Abaqus/Explicit into Abaqus/Standard or from one Abaqus/Explicit analysis into another Abaqus/Explicit analysis, 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.

ITERATION

This parameter is relevant only when the results are imported from a previous direct cyclic Abaqus/Standard analysis.

When importing an analysis from Abaqus/Standard into Abaqus/Explicit or from one Abaqus/Standard analysis into another 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.

**Optional parameters:**

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 will start with no initial state or with the state as defined by the *INITIAL CONDITIONS option.

STEP

Set this parameter equal to the step on the Abaqus/Explicit state file or on the Abaqus/Standard restart file from which the analysis is being imported. If this parameter is 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.

**Data lines to specify element sets to be imported:**

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 data line.

**There are no data lines for importing a part instance.**

## 9.5 *IMPORT CONTROLS: Specify tolerances used in importing model and results data.

This option is used to specify 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," Section 9.2.2 of the Abaqus Analysis User's Guide
- *IMPORT

**Required parameter:**

NORMAL TOL

Set this parameter equal to the tolerance required for the error checking on shell normals. The default value is 0.1.

**There are no data lines associated with this option.**

## 9.6 *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 this option is omitted or is specified without any data lines, all the element sets relevant to the analysis will be imported.

**Products:** Abaqus/Standard Abaqus/Explicit

**Type:** Model data

**Level:** Model

**References:**

- "Transferring results between Abaqus/Explicit and Abaqus/Standard," Section 9.2.2 of the Abaqus Analysis User's Guide
- *IMPORT

**There are no parameters associated with this option.**

**Data lines to specify element set definitions to be imported:**

First line:

1. List of element set names for which the element set definitions are to be imported.

Repeat this data line as often as necessary to specify the element set definitions to be imported. Up to 16 element sets can be listed per line.

## 9.7 *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 this option is omitted or is specified without any data lines, all the node sets relevant to the analysis will be imported.

**Products:** Abaqus/Standard Abaqus/Explicit

**Type:** Model data

**Level:** Model

**References:**

- "Transferring results between Abaqus/Explicit and Abaqus/Standard," Section 9.2.2 of the Abaqus Analysis User's Guide
- *IMPORT

**There are no parameters associated with this option.**

**Data lines to specify node set definitions to be imported:**

First line:

1. List of node set names for which the node set definitions are to be imported.

Repeat this data line as often as necessary to specify the node set definitions to be imported. Up to 16 node sets can be listed per line.

## 9.8 *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:**

- "Acoustic, shock, and coupled acoustic-structural analysis," Section 6.10.1 of the Abaqus Analysis User's Guide
- "Acoustic and shock loads," Section 34.4.6 of the Abaqus Analysis User's Guide
- *INCIDENT WAVE PROPERTY
- *INCIDENT WAVE REFLECTION

**Required parameter:**

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," Section 34.1.2 of the Abaqus Analysis User's Guide). 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," Section 34.1.2 of the Abaqus Analysis User's Guide). 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.

## 9.9 *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," Section 26.3.1 of the Abaqus Analysis User's Guide
- "Acoustic and shock loads," Section 34.4.6 of the Abaqus Analysis User's Guide
- *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. $\rho_f$, the fluid mass density.

## 9.10 *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:**

- "Acoustic, shock, and coupled acoustic-structural analysis," Section 6.10.1 of the Abaqus Analysis User's Guide
- "Acoustic and shock loads," Section 34.4.6 of the Abaqus Analysis User's Guide
- *INCIDENT WAVE INTERACTION PROPERTY
- *INCIDENT WAVE REFLECTION

**Required parameter:**

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," Section 34.1.2 of the Abaqus Analysis User's Guide). 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," Section 34.1.2 of the Abaqus Analysis User's Guide). 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," Section 1.2.2 of the Abaqus Analysis User's Guide, for a discussion of the total time measure.
5. Magnitude scale factor. The default is 1.0.

## 9.11 *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," Section 34.4.6 of the Abaqus Analysis User's Guide
- *INCIDENT WAVE INTERACTION
- *UNDEX CHARGE PROPERTY
- *CONWEP CHARGE PROPERTY

**Required parameter:**

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 parameter:**

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. $\rho_f$, 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. $\rho_f$, 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. $\rho_f$, 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.**

## 9.12 *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," Section 34.4.6 of the Abaqus Analysis User's Guide
- *INCIDENT WAVE
- *INCIDENT WAVE FLUID PROPERTY

**Required parameter:**

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 parameter:**

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.

## 9.13 *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," Section 34.4.6 of the Abaqus Analysis User's Guide
- *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.

## 9.14 *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/CFD 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.

**Reference:**

- "Defining a model in Abaqus," Section 1.3.1 of the Abaqus Analysis User's Guide

**Required parameter:**

INPUT

Set this parameter equal to the name of the file containing the input data. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Guide, for the syntax of such file names.

**Optional parameter:**

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.**

## 9.15 *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," Section 4.1.3 of the Abaqus Analysis User's Guide
- *OUTPUT

**Optional parameter:**

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," Section 4.2.2 of the Abaqus Analysis User's Guide.

Repeat this data line as often as necessary to define the time incrementation variables to be written to the output database.

## 9.16 *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 thermo-mechanical 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," Section 6.5.4 of the Abaqus Analysis User's Guide
- "Fully coupled thermal-stress analysis," Section 6.5.3 of the Abaqus Analysis User's Guide
- "Fully coupled thermal-electrical-structural analysis," Section 6.7.4 of the Abaqus Analysis User's Guide

**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.

## 9.17 *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," Section 11.1.1 of the Abaqus Analysis User's Guide
- "Distributed loads," Section 34.4.3 of the Abaqus Analysis User's Guide

**Optional parameters:**

ORIENTATION

Set this parameter equal to the name given to the *ORIENTATION definition ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Guide) that specifies the orientation of the local system for rigid body degrees of freedom.

**Optional, mutually exclusive parameters:**

FIXED

Include this parameter (default) 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.**

## 9.18 *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/CFD Abaqus/CAE Abaqus/Aqua

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Load module

**Reference:**

- "Initial conditions in Abaqus/Standard and Abaqus/Explicit," Section 34.2.1 of the Abaqus Analysis User's Guide
- "Initial conditions in Abaqus/CFD," Section 34.2.2 of the Abaqus Analysis User's Guide

### Defining initial conditions in Abaqus/Standard and Abaqus/Explicit

**Required parameter:**

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=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 slave surface identified by a node set in an Abaqus/Standard analysis.

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=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 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 .

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=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 Part VI, “Elements,” of the Abaqus Analysis User’s Guide.

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.

Set TYPE=POROSITY to give initial porosity values for materials defined with the *EOS COMPACTION option in Abaqus/Explicit.

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**.

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=NODE REF COORDINATE to define the reference mesh (initial metric) for membrane 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=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=SOLUTION to give initial values of solution-dependent state variables. The REBAR and, in Abaqus/Standard, 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 Part VI, “Elements,” of the Abaqus Analysis User’s Guide. 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`**) 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=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 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.

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 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.

FILE

Set this parameter equal to the name of the results (`.fil`) file or output database (`.odb`) file from which initial field variable, stress, void ratio, pore pressure, or pressure stress data are to be read. This parameter must be used in conjunction with the STEP and INC parameters. For more information, see “File extensions used by Abaqus,” Section 3.6.1 of the Abaqus Analysis User’s Guide.

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=STRESS, TYPE=RATIO, or TYPE=TEMPERATURE are 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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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 1, and 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," Section 34.6.1 of the Abaqus Analysis User's Guide.

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 2.2.4 of the Abaqus Analysis User's Guide).

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.

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=STRESS, TYPE=RATIO, or TYPE=TEMPERATURE are 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.

USER

This parameter applies only to Abaqus/Standard analyses.

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 via user subroutine **SIGINI**.

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.)

**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=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. Slave surface name.
2. Master surface name.
3. Name of the node set associated with the slave surface.

Repeat this data line as often as necessary to define partially bonded surfaces.

**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, $\omega_D$ or $\omega_S$.

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, $\omega_D$ or $\omega_S$.

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, $\omega_D$ or $\omega_S$.

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, $\omega_{MSFLD}$.
3. Ratio of the principal strain rates, $\alpha$.

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=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 "Beam modeling: overview," Section 29.3.1 of the Abaqus Analysis User's Guide, as well as "Shell elements: overview," Section 29.6.1 of the Abaqus Analysis User's Guide). 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.

It may be necessary to leave blank data lines for some nodes if any other node in the model has more than seven field variable points because 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. 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.

**No data lines are required for TYPE=FIELD, VARIABLE=*n*, FILE=*file*, STEP=*step*, INC=*inc*.**

**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, $\bar{\varepsilon}^{pl}|_0$.
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 Part VI, "Elements," of the Abaqus Analysis User's Guide. 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," Section 2.2.5 of the Abaqus Analysis User's Guide).

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, $\bar{\varepsilon}^{pl}|_0$.
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.

**No data lines are required for TYPE=HARDENING, USER.**

**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, $\bar{\varepsilon}^{pl}|_0$.
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 Part VI, "Elements," of the Abaqus Analysis User's Guide. In any element for which an *ORIENTATION option applies, the backstress components must be given in the local system ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Guide).

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.

Repeat this data line as often as necessary to identify various elements or element 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 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, $\varepsilon_{11}^{pl}$.
3. Value of second plastic strain component, $\varepsilon_{22}^{pl}$.
4. Etc., up to six plastic strain components.

Give the plastic strain components as defined for this element type in Part VI, "Elements," of the Abaqus Analysis User's Guide. 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," Section 2.2.5 of the Abaqus Analysis User's Guide).

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, $\varepsilon_{11}^{pl}$.
4. Value of second plastic strain component, $\varepsilon_{22}^{pl}$.
5. Value of third plastic strain component, $\varepsilon_{12}^{pl}$.

Give the initial plastic strain components as defined for this element type in Part VI, "Elements," of the Abaqus Analysis User's Guide. In any element for which an *ORIENTATION option applies, the plastic strain components must be given in the local system ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Guide).

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.

**No data lines are required for TYPE=PORE PRESSURE, USER.**

**No data lines are required for TYPE=PORE PRESSURE, FILE=*file*, STEP=*step*, INC=*inc*.**

**No data lines are required for TYPE=PORE PRESSURE, FILE=*file*, STEP=*step*, INC=*inc*, INTERPOLATE.**

**Data lines for TYPE=PORE PRESSURE, FILE=*file*, STEP=*step*, INC=*inc*, 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.

**No data lines are required for TYPE=PRESSURE STRESS, FILE=*file*, STEP=*step*, INC=*inc*.**

**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.

**No data lines are required for TYPE=RATIO, USER.**

**No data lines are required for TYPE=RATIO, FILE=*file*, STEP=*step*, INC=*inc*.**

**Data lines for TYPE=RATIO, FILE=*file*, STEP=*step*, INC=*inc*, 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=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.

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 because 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. 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.

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 because 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. 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.

**No data lines are required for TYPE=SOLUTION, USER.**

**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, $\nu_i$.

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 Part VI, "Elements," of the Abaqus Analysis User's Guide. 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," Section 2.2.5 of the Abaqus Analysis User's Guide).

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 Part VI, "Elements," of the Abaqus Analysis User's Guide. 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," Section 2.2.5 of the Abaqus Analysis User's Guide).

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.

**No data lines are required for TYPE=STRESS, USER.**

**No data lines are required for TYPE=STRESS, FILE=*file*, STEP=*step*, INC=*inc*.**

**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," Section 29.3.6 of the Abaqus Analysis User's Guide; "Using a general beam section to define the section behavior," Section 29.3.7 of the Abaqus Analysis User's Guide; "Using a shell section integrated during the analysis to define the section behavior," Section 29.6.5 of the Abaqus Analysis User's Guide; and "Using a general shell section to define the section behavior," Section 29.6.6 of the Abaqus Analysis User's Guide). 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. 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 data line (or set of lines) as often as necessary to define initial temperatures at various nodes or node sets.

**No data lines are required for TYPE=TEMPERATURE, FILE=*file*, STEP=*step*, INC=*inc*.**

**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=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 ($0.0 < \text{EVF} \leq 1.0$). 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 may 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.

## Defining initial conditions in Abaqus/CFD

**Required parameters:**

ELEMENT AVERAGE

Include this parameter to define initial conditions for an Abaqus/CFD analysis.

TYPE

Set TYPE=DENSITY to define the initial density for a fluid analysis. For the incompressible flow procedure, if the initial density is not specified, the material density is used by default.

Set TYPE=VELOCITY to define the initial velocity field. For the incompressible flow procedure, the initial velocities can be modified to satisfy the divergence-free startup conditions.

Set TYPE=TEMPERATURE to define the initial temperature field. This setting is valid only when the energy equation has been activated for the incompressible flow procedure.

Set TYPE=TURBKE to define the initial turbulent kinetic energy field. This setting is valid only when the $k$-$\varepsilon$ or $k$-$\omega$ model has been activated for the incompressible flow procedure.

Set TYPE=TURBEPS to define the initial turbulent energy dissipation rate. This setting is valid only when the $k$-$\varepsilon$ model has been activated for the incompressible flow procedure.

Set TYPE=TURBOMEGA to define the initial specific energy dissipation rate. This setting is valid only when the $k$-$\omega$ model has been activated for the incompressible flow procedure.

Set TYPE=TURBNU to define the initial turbulent viscosity for the Spalart-Allmaras model. This setting is valid only when the Spalart-Allmaras model has been activated for the incompressible flow procedure.

Set TYPE=TURBINTENSITY to define the initial turbulence intensity. This setting is valid when the $k$-$\varepsilon$, $k$-$\omega$, or the Spalart-Allmaras model has been activated for the incompressible flow procedure.

Set TYPE=TURBVELOCITYSCALE to define the initial velocity scale. This setting is valid when the $k$-$\varepsilon$, $k$-$\omega$, or the Spalart-Allmaras model has been activated for the incompressible flow procedure.

Set TYPE=TURBVISCOSITYRATIO to define the initial eddy to molecular viscosity ratio. This setting is valid when the $k$-$\varepsilon$, $k$-$\omega$, or the Spalart-Allmaras model has been activated for the incompressible flow procedure.

Set TYPE=TURBLENGTHSCALE to define the initial turbulent length scale. This setting is valid when the $k$-$\varepsilon$, $k$-$\omega$, or the Spalart-Allmaras model has been activated for the incompressible flow procedure.

**Data line for TYPE=DENSITY:**

First (and only) line:

1. Element set name.
2. Density value.

**Data lines for TYPE=VELOCITY:**

First line:

1. Element set name.
2. Degree of freedom. Enter 1 for VELX, 2 for VELY, and 3 for VELZ. See “Active degrees of freedom” in “Boundary conditions in Abaqus/CFD,” Section 34.3.2 of the Abaqus Analysis User’s Guide, for a definition of the degrees of freedom in Abaqus/CFD.

3. Value of initial velocity.

Repeat this data line as often as necessary to define the components of the initial velocity for various element sets.

**Data line for TYPE=TEMPERATURE:**

First (and only) line:

1. Element set name.
2. Temperature value.

**Data line for TYPE=TURBKE:**

First (and only) line:

1. Element set name.
2. Turbulent kinetic energy value.

**Data line for TYPE=TURBEPS:**

First (and only) line:

1. Element set name.
2. Energy dissipation rate value.

**Data line for TYPE=TURBOMEGA:**

First (and only) line:

1. Element set name.
2. Specific energy dissipation rate value.

**Data line for TYPE=TURBNU:**

First (and only) line:

1. Element set name.
2. Spalart-Allmaras turbulent viscosity value.

**Data line for TYPE=TURBINTENSITY:**

First (and only) line:

1. Element set name.
2. Turbulence intensity.

**Data line for TYPE=TURBVELOCITYSCALE:**

First (and only) line:

1. Element set name.
2. Velocity scale.

**Data line for TYPE=TURBVISCOSITYRATIO:**

First (and only) line:

1. Element set name.
2. Eddy to molecular viscosity ratio.

**Data line for TYPE=TURBLENGTHSCALE:**

First (and only) line:

1. Element set name.
2. Turbulent length scale.

## 9.19 *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.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD 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:**

- "Defining an assembly," Section 2.10.1 of the Abaqus Analysis User's Guide
- "Transferring results between Abaqus analyses: overview," Section 9.2.1 of the Abaqus Analysis User's Guide
- *END INSTANCE
- *IMPORT

**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 from a previous analysis:**

INSTANCE

Set this parameter equal to the name of the instance to be imported from the previous analysis.

**Optional import parameter:**

LIBRARY

Set this parameter equal to the name of the previous analysis from which the instance should be imported. The previous analysis output database (`.odb`) file should reside in the current (working) directory. If the LIBRARY parameter is omitted, the job name of the previous analysis must be specified on the command line using the **oldjob** parameter (see "Abaqus/Standard, Abaqus/Explicit,

and Abaqus/CFD execution," Section 3.2.2 of the Abaqus Analysis User's Guide). If both methods are used, the command line specification will take precedence over the LIBRARY parameter.

**Data line to translate an instance that is not imported from a previous analysis:**

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 that is not imported from a previous analysis:**

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 9.19–1).
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.

**There are no data lines for an instance that is imported from a previous analysis.**

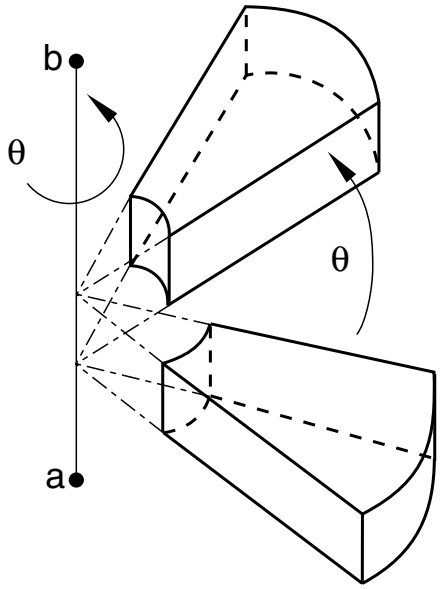



**Figure 9.19–1** Rotation of an instance.

## 9.20 *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/Explicit Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Step module

**References:**

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Guide
- "Integrated output section definition," Section 2.5.1 of the Abaqus Analysis User's Guide
- *OUTPUT
- *SURFACE
- *INTEGRATED OUTPUT SECTION

**Required, mutually exclusive parameters:**

SECTION

Set this parameter equal to the name of the *INTEGRATED OUTPUT SECTION (see "Integrated output section definition," Section 2.5.1 of the Abaqus Analysis User's Guide) over which this output request is being made.

SURFACE

Set this parameter equal to the name of the surface (see "Element-based surface definition," Section 2.3.2 of the Abaqus Analysis User's Guide) 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.

**Optional parameter:**

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/Explicit output variable identifiers," Section 4.2.2 of the Abaqus Analysis User's Guide.

Repeat this data line as often as necessary to define the list of variables to be written to the output database.

## 9.21 *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/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Part, Part instance, Assembly

**Abaqus/CAE:** Step module

**References:**

- "Integrated output section definition," Section 2.5.1 of the Abaqus Analysis User's Guide
- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Guide
- *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," Section 2.3.2 of the Abaqus Analysis User's Guide) to be associated with the integrated output section.

**Optional parameters:**

ORIENTATION

Set this parameter equal to the name of an orientation definition ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Guide) 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.**

## 9.22 *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," Section 32.13.1 of the Abaqus Analysis User's Guide
- "Tube-to-tube contact elements," Section 40.3.1 of the Abaqus Analysis User's Guide
- "Slide line contact elements," Section 40.4.1 of the Abaqus Analysis User's Guide
- "Rigid surface contact elements," Section 40.5.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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 parameter:**

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.

**Data lines for ISL21A and ISL22A elements:**

There are no data lines.

**Data lines for IRS-type elements for use with axisymmetric elements:**

There are no data lines.

**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:

2. $X$-direction cosine.
3. $Y$-direction cosine.
4. $Z$-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.

**Data lines for ASI-type elements for use with axisymmetric elements or 3D elements:**

There are no data lines.

## 9.23 *ITS: Define properties for ITS elements.

This option is used to define the properties for ITS-type elements. The *DASHPOT, *FRICTION, 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 (with the exception of friction) is available by modeling connectors.

**References:**

- "Rigid surface contact elements," Section 40.5.1 of the Abaqus Analysis User's Guide
- *DASHPOT
- *FRICTION
- *SPRING

**Required parameter:**

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.

## 10. J

## 10.1 *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," Section 32.3.1 of the Abaqus Analysis User's Guide
- *DASHPOT
- *SPRING

**Required parameter:**

ELSET

Set this parameter equal to the name of the element set containing the JOINTC elements for which properties are being defined.

**Optional parameter:**

ORIENTATION

Set this parameter equal to the name given to the *ORIENTATION definition ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Guide) 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.**

## 10.2 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance

**References:**

- "Elastic-plastic joints," Section 32.10.1 of the Abaqus Analysis User's Guide
- *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 parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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. $\nu$, 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. $\nu$, 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.

## 10.3 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance

**References:**

- "Elastic-plastic joints," Section 32.10.1 of the Abaqus Analysis User's Guide
- *EPJOINT

**Required parameter:**

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 parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

**Data lines for TYPE=SAND:**

First line:

1. $V_t$, yield strength in pure tension.
2. $\Lambda_1$, constant for yield surface shape.
3. $\Lambda_2$, constant for yield surface shape.
4. $\phi$, soil friction angle, in degrees.
5. $\gamma$, 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.

## 10.4 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Model

**Reference:**

- "Jointed material model," Section 23.5.1 of the Abaqus Analysis User's Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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, $\beta$, for this system. Give the value in degrees.
2. Dilation angle, $\psi$, 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.

## 10.5 *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," Section 6.7.3 of the Abaqus Analysis User's Guide
- "Fully coupled thermal-electrical-structural analysis," Section 6.7.4 of the Abaqus Analysis User's Guide

**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.

# 11. K

## 11.1 *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 $\kappa_s$ and $\kappa_p$, 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," Section 26.4.1 of the Abaqus Analysis User's Guide
- *DIFFUSIVITY

**Optional parameters:**

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of $\kappa_s$ or $\kappa_p$. If this parameter is omitted, $\kappa_s$ or $\kappa_p$ is assumed not to depend on any field variables but may still depend on concentration and temperature. See "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

TYPE

Set TYPE=TEMP (default) to define $\kappa_s$ (governing mass diffusion caused by temperature gradients). Set TYPE=PRESS to define $\kappa_p$ (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, $\kappa_s$. (Units of $\mathrm{F}^{1/2}\mathrm{L}^{-1}$.)
2. Concentration, $c$.
3. Temperature, $\theta$.
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 $\kappa_s$ 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, $\kappa_p$. (Units of $\mathrm{LF}^{-1/2}$.)
2. Concentration, $c$.
3. Temperature, $\theta$.
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 $\kappa_p$ as a function of concentration, temperature, and other predefined field variables.

## 11.2 *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," Section 35.3.2 of the Abaqus Analysis User's Guide
- *COUPLING

**There are no parameters associated with this option.**

**Data lines to specify the degrees of freedom to be constrained:**

First line:

1. First degree of freedom constrained. See "Conventions," Section 1.2.2 of the Abaqus Analysis User's Guide, 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).

## 11.3 *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.

**Reference:**

- "Kinematic constraints: overview," Section 35.1.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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 parameter:**

ORIENTATION

Set this parameter equal to the name given to the *ORIENTATION definition ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Guide) 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," Section 1.2.2 of the Abaqus Analysis User's Guide, 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).

## 12. L

## 12.1 *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

**Reference:**

- "Latent heat," Section 26.2.4 of the Abaqus Analysis User's Guide

**Optional parameter:**

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.

**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.

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.

## 12.2 *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," Section 6.1.4 of the Abaqus Analysis User's Guide
- *END LOAD CASE

**Required parameter:**

NAME

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

**There are no data lines associated with this option.**

## 12.3 *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.

**Product:** Abaqus/Explicit

**Type:** Model data

**Level:** Model

**References:**

- "Connector uniaxial behavior," Section 31.2.10 of the Abaqus Analysis User's Guide
- "Fabric material behavior," Section 23.4.1 of the Abaqus Analysis User's Guide
- *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. $\mu_0$.
2. $\mu_1$.
3. $\alpha$.

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. $\mu_0$.
2. $\mu_1$.
3. $\alpha$.

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. $\mu_0$.
2. $\mu_1$.
3. $\alpha$.

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.

## 12.4 *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/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "Low-density foams," Section 22.9.1 of the Abaqus Analysis User's Guide
- *UNIAXIAL TEST DATA

**Optional parameters:**

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.

STRAIN RATE

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.

RATE EXTRAPOLATION

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.

TENSION CUTOFF

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.

FAIL

This parameter 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.

**Data line to specify the relaxation coefficients for low-density foams:**

First (and only) line:

1. $\mu_0$. The default is =0.0001 (time units).
2. $\mu_1$. The default is =0.005 (time units).
3. $\alpha$. The default is 2.

# 13. M

## 13.1 *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," Section 26.5.3 of the Abaqus Analysis User's Guide
- "Eddy current analysis," Section 6.7.5 of the Abaqus Analysis User's Guide
- "Magnetostatic analysis," Section 6.7.6 of the Abaqus Analysis User's Guide

**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. $\mu_{11}$. (Units of $FA^{-2}$.)
2. $\mu_{22}$.

3. $\mu_{33}$.
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. $\mu_{11}$. (Units of $FA^{-2}$.)
2. $\mu_{22}$.
3. $\mu_{33}$.
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. $\mu_{11}$. (Units of $FA^{-2}$.)
2. $\mu_{12}$.
3. $\mu_{22}$.
4. $\mu_{13}$.
5. $\mu_{23}$.
6. $\mu_{33}$.

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. $\mu_{11}$. (Units of $FA^{-2}$.)
2. $\mu_{12}$.
3. $\mu_{22}$.
4. $\mu_{13}$.
5. $\mu_{23}$.
6. $\mu_{33}$.
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.**

## 13.2 *MAGNETOSTATIC: Magnetostatic analysis.

This option is used to indicate that the step should be analyzed as a magnetostatic load step.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Electromagnetic analysis procedures," Section 6.7.1 of the Abaqus Analysis User's Guide
- "Magnetostatic analysis," Section 6.7.6 of the Abaqus Analysis User's Guide

**Optional parameter:**

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.

## 13.3 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Model

**Reference:**

- "Mesh-to-mesh solution mapping," Section 12.4.1 of the Abaqus Analysis User's Guide

**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 13.3–1).
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.

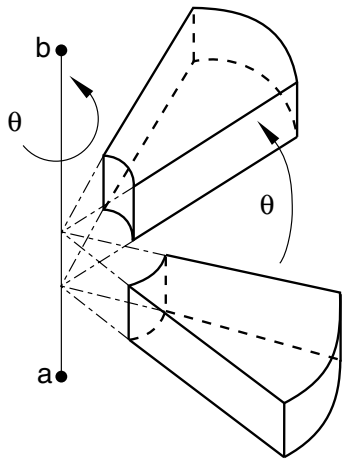



**Figure 13.3–1** Rotation of an old-model mesh.

## 13.4 *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 (for direct-integration dynamic analysis and explicit dynamic analysis) and composite damping (for modal dynamic analysis) associated with MASS elements.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Part, Part instance, Assembly

**Abaqus/CAE:** Property module and Interaction module.

**Reference:**

- "Point masses," Section 30.1.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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 $\alpha_R$ factor to create mass proportional damping for the MASS 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 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," Section 2.2.5 of the Abaqus Analysis User's Guide) 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.

## 13.5 *MASS ADJUST: Adjust and/or redistribute the mass of an element set.

This option is used to either increase or decrease uniformly the mass of the elements in an element set 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. 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.

**Product:** Abaqus/Explicit

**Type:** Model data

**Level:** Assembly

**Reference:**

- "Adjust and/or redistribute mass of an element set," Section 2.6.1 of the Abaqus Analysis User's Guide

**Optional parameter:**

TARGET DT

Set this parameter equal to a default minimum element stable time increment for all of the mass-adjusted element sets. Individualized target time increments can be specified for selected sets, if needed. A nonzero target value causes redistribution of mass within the element sets. If this parameter is omitted, mass redistribution occurs only for element sets for which a minimum element stable time increment value is specified on the data lines.

**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. If mass is to be redistributed to achieve a target time increment without altering the total mass, enter the "word" CURRENT instead of a mass value.
3. Minimum element stable time increment for the element set. The default value is 0.0 or the value of the TARGET DT parameter if it is used. When both the TARGET DT parameter and a value for the element set are specified, the value for the element set is used for that set.

If neither the TARGET DT parameter nor a value for the element set are specified, no mass redistribution occurs for that set.

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.

## 13.6 *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

**Reference:**

- "Mass diffusion analysis," Section 6.9.1 of the Abaqus Analysis User's Guide

**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.

## 13.7 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Uncoupled heat transfer analysis," Section 6.5.2 of the Abaqus Analysis User's Guide
- "UMASFL," Section 1.1.40 of the Abaqus User Subroutines Reference Guide

**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," Section 34.1.2 of the Abaqus Analysis User's Guide). 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," Section 6.1.2 of the Abaqus Analysis User's Guide).

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Guide, 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.

## 13.8 *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/CFD Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**Reference:**

- "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide

**Required parameter:**

NAME

Set this parameter equal to a label that will be used 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," Section 1.2.1 of the Abaqus Analysis User's Guide), 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 be used for regularizing the material data. The default is RTOL=0.03.

SRATE FACTOR

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the factor used for filtering the equivalent plastic strain rate for the evaluation of strain rate-dependent material data. The default value is 0.9.

STRAIN RATE REGULARIZATION

This parameter applies only to Abaqus/Explicit analyses and is used only to regularize strain rate-dependent material data.

Set STRAIN RATE REGULARIZATION=LOGARITHMIC (default) to use a logarithmic regularization for strain rate-dependent material data.

Set STRAIN RATE REGULARIZATION=LINEAR to use a linear regularization for strain rate-dependent material data.

**There are no data lines associated with this option.**

## 13.9 *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 or mass matrix for the user element. It can be used once if only a stiffness or mass is required or twice to give both matrices.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance, Model

**References:**

- "User-defined elements," Section 32.15.1 of the Abaqus Analysis User's Guide
- *USER ELEMENT

**Required parameter:**

TYPE

Set TYPE=MASS to define the mass matrix. Set TYPE=STIFFNESS to define the stiffness matrix.

**Optional parameter:**

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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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.

## 13.10 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Model

**References:**

- "Defining matrices," Section 2.11.1 of the Abaqus Analysis User's Guide
- *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 parameter:**

NSET

Use this parameter to remap nodes of the assembled matrices. Set this parameter equal to the name of the node set that contains the new node labels corresponding to all user-defined nodes of the matrices. 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 that are specified in this option. If necessary, use an unsorted node set to obtain the correct mapping. The matrix nodes are assumed to be sorted in

ascending order of their original labels that were defined at generation or specified in the matrix data.

If this parameter is omitted, all user-defined nodes retain their original labels.

**There are no data lines associated with this option.**

## 13.11 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Generating matrices," Section 10.3.1 of the Abaqus Analysis User's Guide
- "Defining substructures," Section 10.1.2 of the Abaqus Analysis User's Guide
- *MATRIX GENERATE
- *SUBSTRUCTURE GENERATE

**Optional parameter:**

REFERENCE NODE

Set this parameter equal to an existing node label to define the origin of the coordinate frame that will be used for the matrix check.

**There are no data lines associated with this option.**

## 13.12 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**Reference:**

- "Generating matrices," Section 10.3.1 of the Abaqus Analysis User's Guide

**At least one of the following parameters is required:**

STIFFNESS

Include this parameter to generate the stiffness matrix.

MASS

Include this parameter to generate the mass matrix.

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.

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 will evaluate the stiffness associated with frequency-dependent springs and dashpots at zero frequency and will not consider the stiffness contributions from frequency-domain viscoelasticity.

PUBLIC NODES

Use this parameter to specify which nodes will be visible in the matrix usage model. Set this parameter equal to the name of the node set that contains all the nodes that will be presented as user-defined nodes during matrix input; all other nodes are designated as internal nodes and effectively hidden. By default, all user-defined nodes are visible in the matrix usage model.

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.**

## 13.13 *MATRIX INPUT: Read in a matrix for a part of the model.

This option can be used to input a matrix in sparse format.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Model

**References:**

- "Defining matrices," Section 2.11.1 of the Abaqus Analysis User's Guide
- *MATRIX ASSEMBLE

**Required parameter:**

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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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.

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.

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.

## 13.14 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Generating matrices," Section 10.3.1 of the Abaqus Analysis User's Guide
- *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 parameter:**

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.

**There are no data lines associated with this option.**

## 13.15 *MEDIA TRANSPORT: Activate or deactivate a periodic media.

This option is used to activate or deactivate a periodic media.

**Product:** Abaqus/Explicit

**Type:** History data

**Level:** Step

**Reference:**

- “Periodic media analysis,” Section 10.5.1 of the Abaqus Analysis User’s Guide

**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.

## 13.16 *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

**Reference:**

- "Membrane elements," Section 29.1.1 of the Abaqus Analysis User's Guide
- *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," Section 27.1.4 of the Abaqus Analysis User's Guide) 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," Section 27.1.4 of the Abaqus Analysis User's Guide) 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," Section 2.8.1 of the Abaqus Analysis User's Guide) 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.

## 13.17 *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," Section 26.1.1 of the Abaqus Analysis User's Guide
- "Dynamic analysis procedures: overview," Section 6.3.1 of the Abaqus Analysis User's Guide

**Optional, mutually exclusive parameters (if no parameter is specified, Abaqus assumes that the modal damping coefficients are provided on the data lines):**

MODAL

This parameter is superseded by the VISCOUS parameter. It is recommended that you use the VISCOUS parameter.

Set MODAL=DIRECT 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 MODAL=COMPOSITE to select composite modal damping using the damping coefficients that have been calculated in the *FREQUENCY step ("Natural frequency extraction," Section 6.3.5 of the Abaqus Analysis User's Guide). 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," Section 26.1.1 of the Abaqus Analysis User's Guide). Composite modal damping can be used only with DEFINITION=MODE NUMBERS.

RAYLEIGH

This parameter is superseded by the VISCOUS parameter. It is recommended that you use VISCOUS=RAYLEIGH to select Rayleigh damping.

Include this parameter to select Rayleigh damping. The damping term for a particular mode is defined as $\alpha_M m_M + \beta_M k_M$, where $\alpha_M$ and $\beta_M$ 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*.

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," Section 6.3.8 of the Abaqus Analysis User's Guide; "Random response analysis," Section 6.3.11 of the Abaqus Analysis User's Guide; "Transient modal dynamic analysis," Section 6.3.7 of the Abaqus Analysis User's Guide; and "Complex eigenvalue extraction," Section 6.3.6 of the Abaqus Analysis User's Guide). The value of the damping constant, *s*, that multiplies the internal forces is entered on the data line.

VISCOUS

This parameter supersedes the MODAL and RAYLEIGH parameters. It is recommended that you use this parameter.

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," Section 6.3.5 of the Abaqus Analysis User's Guide). 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," Section 26.1.1 of the Abaqus Analysis User's Guide). 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 $\alpha_M$ and $\beta_M$ 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*.

**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 or MODAL=DIRECT and DEFINITION=FREQUENCY RANGE for uncoupled structural and acoustic modes obtained through AMS eigenextraction.

**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 or MODAL=DIRECT 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, $\xi$.

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 or 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, $\alpha_M$.
4. Stiffness proportional damping, $\beta_M$.

Repeat this data line as often as necessary to define modal damping for different modes.

**Data lines to define composite modal damping (VISCOUS=COMPOSITE or MODAL=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 or MODAL=DIRECT and DEFINITION=FREQUENCY RANGE):**

First line:

1. Frequency value (in cycles/time).
2. Fraction of critical damping, $\xi$.

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 or RAYLEIGH and DEFINITION=FREQUENCY RANGE):**

First line:

1. Frequency value (in cycles/time).
2. Mass proportional damping, $\alpha_M$.
3. Stiffness proportional damping, $\beta_M$.

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.

## 13.18 *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

**Reference:**

- "Transient modal dynamic analysis," Section 6.3.7 of the Abaqus Analysis User's Guide

**Optional parameter:**

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.

**Data line for a transient modal dynamic analysis:**

First (and only) line:

1. Time increment to be used.
2. Time period.

## 13.19 *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.

**Reference:**

- "Output to the data and results files," Section 4.1.2 of the Abaqus Analysis User's Guide

**Optional parameter:**

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," Section 4.2.1 of the Abaqus Analysis User's Guide.

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.

## 13.20 *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," Section 4.1.3 of the Abaqus Analysis User's Guide
- *OUTPUT

**Optional parameter:**

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 "Abaqus/Standard output variable identifiers," Section 4.2.1 of the Abaqus Analysis User's Guide.

Repeat this data line as often as necessary to define the modal variables to be written to the output database.

## 13.21 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**Reference:**

- "Output to the data and results files," Section 4.1.2 of the Abaqus Analysis User's Guide

**Optional parameter:**

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," Section 4.2.1 of the Abaqus Analysis User's Guide.

Repeat this data line as often as necessary: each line defines a table.

## 13.22 *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," Section 11.2.1 of the Abaqus Analysis User's Guide
- "Removing and reactivating contact pairs" in "Defining contact pairs in Abaqus/Standard," Section 36.3.1 of the Abaqus Analysis User's Guide

**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 parameter:**

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. Slave surface name used in the contact pair being removed or reactivated.
2. Master surface name used in the contact pair being removed or reactivated. For self-contact the master surface name is omitted or is the same as the slave surface name.

Repeat this data line as often as necessary.

**No data lines are used with this option when the ACTIVATE parameter is included.**

## 13.23 *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," Section 23.3.3 of the Abaqus Analysis User's Guide
- *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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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 $\phi$ 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, $\epsilon$. 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, $\phi$, 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, $\psi$, 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 $\psi$ 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.

## 13.24 *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," Section 23.3.3 of the Abaqus Analysis User's Guide
- *MOHR COULOMB

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 13.25 *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

**Reference:**

- "Moisture swelling," Section 26.6.6 of the Abaqus Analysis User's Guide

**There are no parameters associated with this option.**

**Data lines to define moisture-driven swelling:**

First line:

1. Volumetric moisture swelling strain, $\varepsilon^{ms}$.
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 $\varepsilon^{ms}$–$s$ relationship from $s = 0.0$ to $s = 1.0$ in increasing values of $s$.

## 13.26 *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," Section 11.5.2 of the Abaqus Analysis User's Guide
- "Inflator definition," Section 11.5.4 of the Abaqus Analysis User's Guide
- *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.

## 13.27 *MOMENTUM EQUATION SOLVER: Specify the linear solver and parameters for solving the momentum equations in an Abaqus/CFD analysis.

This option can be used only as a suboption of the *CFD option to enable the linear solver parameters for solving the momentum equations. For a steady-state analysis this option is also used to enable the under-relaxation parameter for the nonlinear momentum solver.

**Products:** Abaqus/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Step module

**References:**

- "Incompressible fluid dynamic analysis," Section 6.6.2 of the Abaqus Analysis User's Guide
- *CFD

**Optional parameters:**

CONVERGENCE

Set CONVERGENCE=ON to write convergence information in the log file.
Set CONVERGENCE=OFF (default) to suppress convergence information.

DIAGNOSTICS

Set DIAGNOSTICS=ON to write diagnostic information about the solver in the log file.
Set DIAGNOSTICS=OFF (default) to suppress diagnostic information.

TYPE

Set TYPE=DSFGMRES (default) to enable the Diagonally Scaled Flexible Generalized Minimum Residual linear solver.

Set TYPE=DSGMRES to enable the Diagonally Scaled Generalized Minimum Residual linear solver.

Set TYPE=ILUFGMRES to enable the Incomplete LU factorization preconditioned Flexible Generalized Minimum Residual linear solver.

**Data line for TYPE=DSFGMRES, TYPE=DSGMRES, and TYPE=ILUFGMRES:**

First (and only) line:

1. Maximum number of iterations (default=50).
2. Interval to check linear convergence (default=2).

3. Linear convergence criterion (default=$10^{-5}$).
4. Number of restart vectors (default=15).
5. Under-relaxation factor (default=0.7). This value is valid only for a steady-state analysis.

## 13.28 *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

**Reference:**

- "Output," Section 4.1.1 of the Abaqus Analysis User's Guide

**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 parameter:**

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.**

## 13.29 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Cavity radiation," Section 41.1.1 of the Abaqus Analysis User's Guide
- "Steady-state transport analysis," Section 6.4.1 of the Abaqus Analysis User's Guide
- "Static stress analysis," Section 6.2.2 of the Abaqus Analysis User's Guide
- "Eddy current analysis," Section 6.7.5 of the Abaqus Analysis User's Guide
- "UMOTION," Section 1.1.44 of the Abaqus User Subroutines Reference Guide

### 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," Section 34.1.2 of the Abaqus Analysis User's Guide).

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.

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," Section 41.1.1 of the Abaqus Analysis User's Guide.

**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," Section 1.2.2 of the Abaqus Analysis User's Guide, 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 (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 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," Section 1.2.2 of the Abaqus Analysis User's Guide, 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 parameter:**

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 parameter:**

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," Section 34.1.2 of the Abaqus Analysis User's Guide).

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 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. 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.

## 13.30 *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," Section 35.2.2 of the Abaqus Analysis User's Guide
- "MPC," Section 1.1.14 of the Abaqus User Subroutines Reference Guide

**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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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," Section 35.2.2 of the Abaqus Analysis User's Guide, 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.

## 13.31 *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," Section 22.5.1 of the Abaqus Analysis User's Guide
- "Hyperelastic behavior in elastomeric foams," Section 22.5.2 of the Abaqus Analysis User's Guide
- "Mullins effect," Section 22.6.1 of the Abaqus Analysis User's Guide
- "Energy dissipation in elastomeric foams," Section 22.6.2 of the Abaqus Analysis User's Guide
- "UMULLINS," Section 1.1.45 of the Abaqus User Subroutines Reference Guide
- "VUMULLINS," Section 1.2.21 of the Abaqus User Subroutines Reference Guide
- *BIAXIAL TEST DATA
- *PLANAR TEST DATA
- *UNIAXIAL TEST DATA

**Optional, mutually exclusive parameters:**

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 can be given directly on the data lines or the damage variable can be defined through user subroutine **UMULLINS** in Abaqus/Standard or **VUMULLINS** in Abaqus/Explicit.

USER

Include this parameter if the damage variable defining the Mullins effect 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 $\beta$ 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, $\beta$ 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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

This parameter is not relevant if the 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 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 $r > 1$.

**To define the material 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, *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 both the TEST DATA INPUT and USER parameters are omitted:**

First line:

1. $r$.
2. $m$.
3. $\beta$ (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 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.

## 13.32 *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," Section 29.3.7 of the Abaqus Analysis User's Guide
- *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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 13.33 *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," Section 29.3.7 of the Abaqus Analysis User's Guide
- *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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

# 14. N

## 14.1 *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/CFD Abaqus/CAE

**Type:** Model data

**Level:** Part, Part instance

**Abaqus/CAE:** Not applicable; copying portions of sketches and instancing of parts serve similar purposes.

**Reference:**

- "Node definition," Section 2.1.1 of the Abaqus Analysis User's Guide

**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," Section 28.3.1 of the Abaqus Analysis User's Guide).

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 14.1–1).
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 14.1–1).
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 14.1–2).
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 14.1–2).
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 14.1–3).
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 14.1–3).
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 14.1–3).
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 14.1–4).
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 14.1–5, 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).

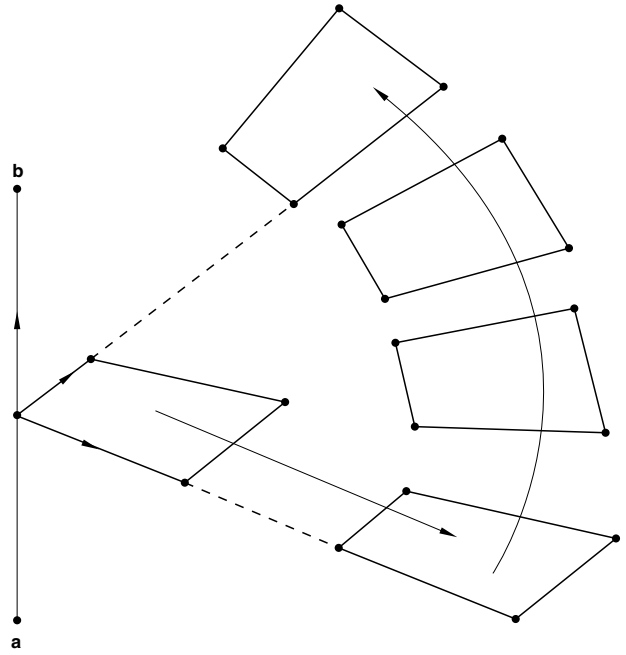



**Figure 14.1–1** *NCOPY, SHIFT option.

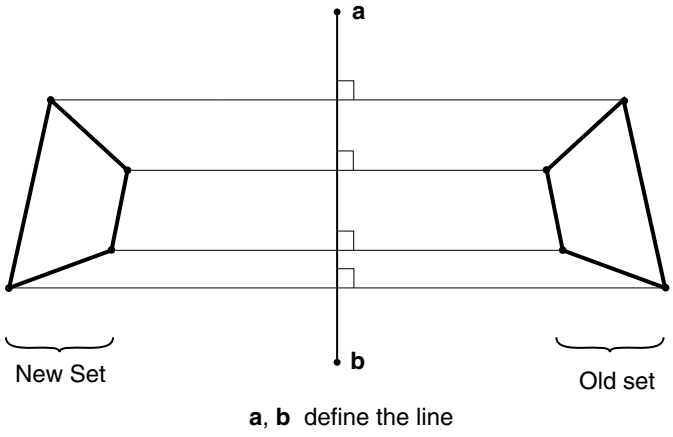



**Figure 14.1–2** *NCOPY, REFLECT=LINE option.

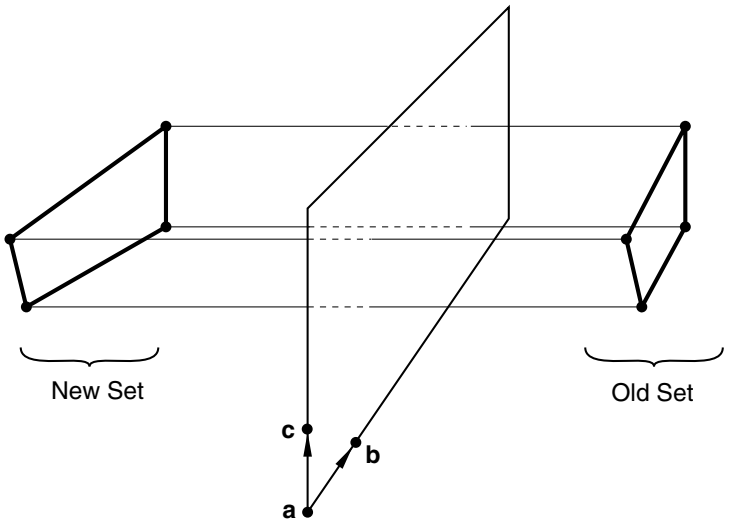



**Figure 14.1–3** *NCOPY, REFLECT=MIRROR option.

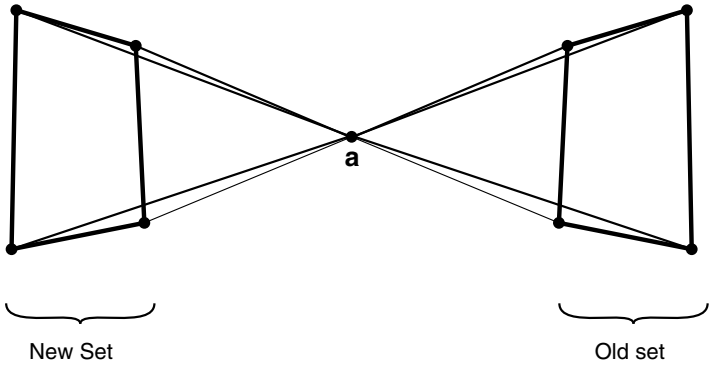



**Figure 14.1–4** *NCOPY, REFLECT=POINT option.

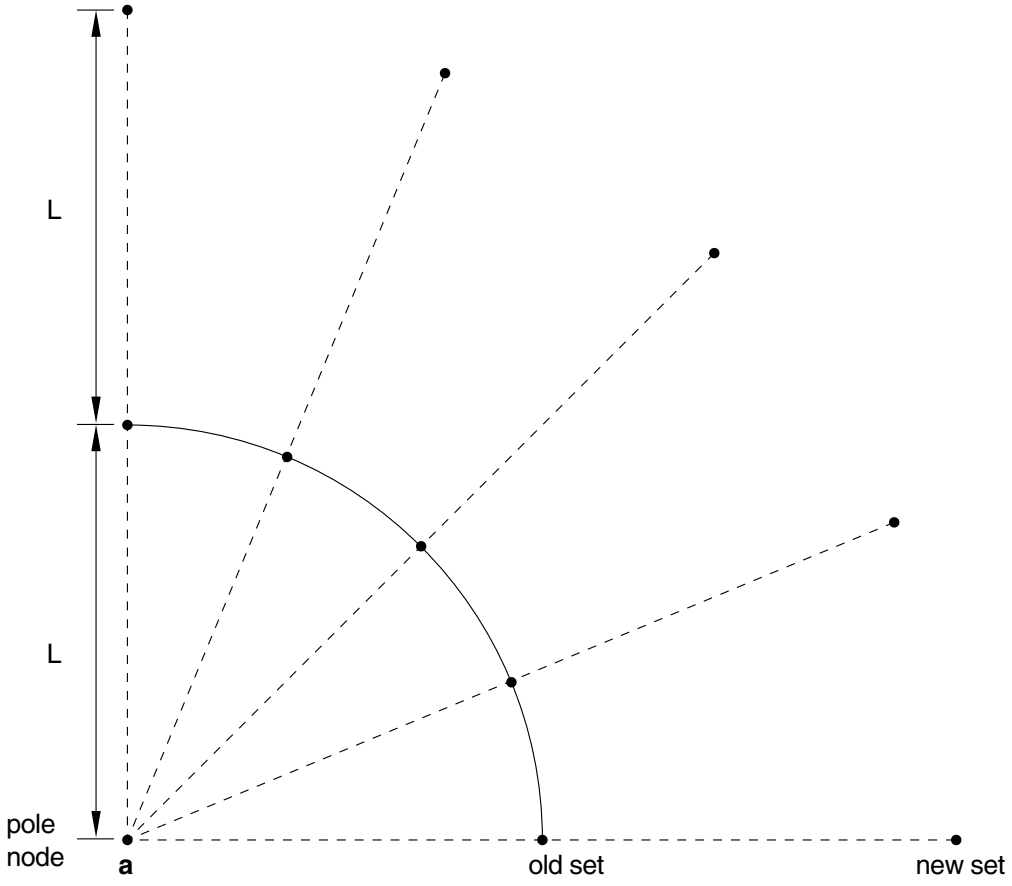



**Figure 14.1–5** *NCOPY, POLE option.

## 14.2 *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/CFD Abaqus/CAE

**Type:** Model data

**Level:** Part, Part instance

**Abaqus/CAE:** Not applicable; nodes are generated when you mesh the model.

**Reference:**

- "Node definition," Section 2.1.1 of the Abaqus Analysis User's Guide

**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.

## 14.3 *NGEN: Generate incremental nodes.

This option is used to generate nodes incrementally.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** Model data

**Level:** Part, Part instance

**Abaqus/CAE:** Not applicable; nodes are generated when you mesh the model.

**Reference:**

- "Node definition," Section 2.1.1 of the Abaqus Analysis User's Guide

**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 14.3–1.

**Data lines to generate nodes incrementally:**

First line:

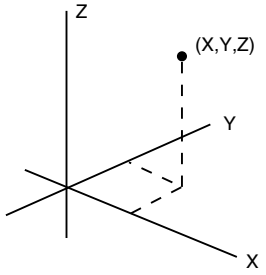
1. Number of the first end node (it must have been previously defined).
2. Number of the second end node (it must have been previously defined).
3. Increment in the numbers between each node along the line. The default is 1.

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).

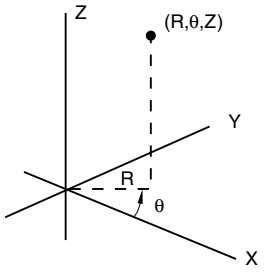
The following entries are used only for a circular arc equal to or larger than 180°:

8. First component of a normal to the circular arc.
9. Second component of a normal to the circular arc.
10. Third component of a normal to the circular arc.

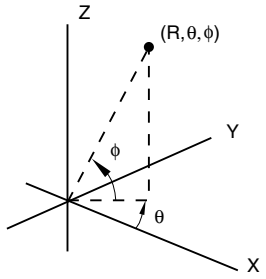
Repeat this data line as often as necessary.




Rectangular Cartesian
(default)




Cylindrical
(θ and ϕ are given in degrees)




Spherical

**Figure 14.3–1** Coordinate systems.

## 14.4 *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/CFD 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.

**Reference:**

- "Node definition," Section 2.1.1 of the Abaqus Analysis User's Guide

**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 14.4–1 defines the origin of the local rectangular coordinate system defining the map. The local $\hat{x}$-axis is defined by the line joining points $a$ and $b$. The local $\hat{x}$–$\hat{y}$ 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 14.4–1 defines the origin of the local cylindrical coordinate system defining the map. The line going through point $a$ and point $b$ defines the $\hat{z}$-axis of the local cylindrical coordinate system. The local $\hat{r}$–$\hat{z}$ plane for $\theta = 0$ 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 14.4–1 defines the origin of the local diamond coordinate system defining the map. The line going through point $a$ and point $b$ defines the $\hat{x}$-axis of the local coordinate system. The line going through point

*a* and point *c* defines the $\hat{y}$-axis of the local coordinate system. The line going through point *a* and point *d* defines the $\hat{z}$-axis of the local coordinate system.

Set TYPE=SPHERICAL to map from spherical coordinates. Point *a* in Figure 14.4–1 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 $\phi = 0$ plane. The plane passing through points *a*, *b*, and *c* defines the local $\theta = 0$ plane.

Set TYPE=TOROIDAL to map from toroidal coordinates. Point *a* in Figure 14.4–1 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 $\phi$ the $\theta$-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 parameter:**

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 14.4–1).
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:

4. *X*-coordinate of point *d*.
5. *Y*-coordinate of point *d*.
6. *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:

2. 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.

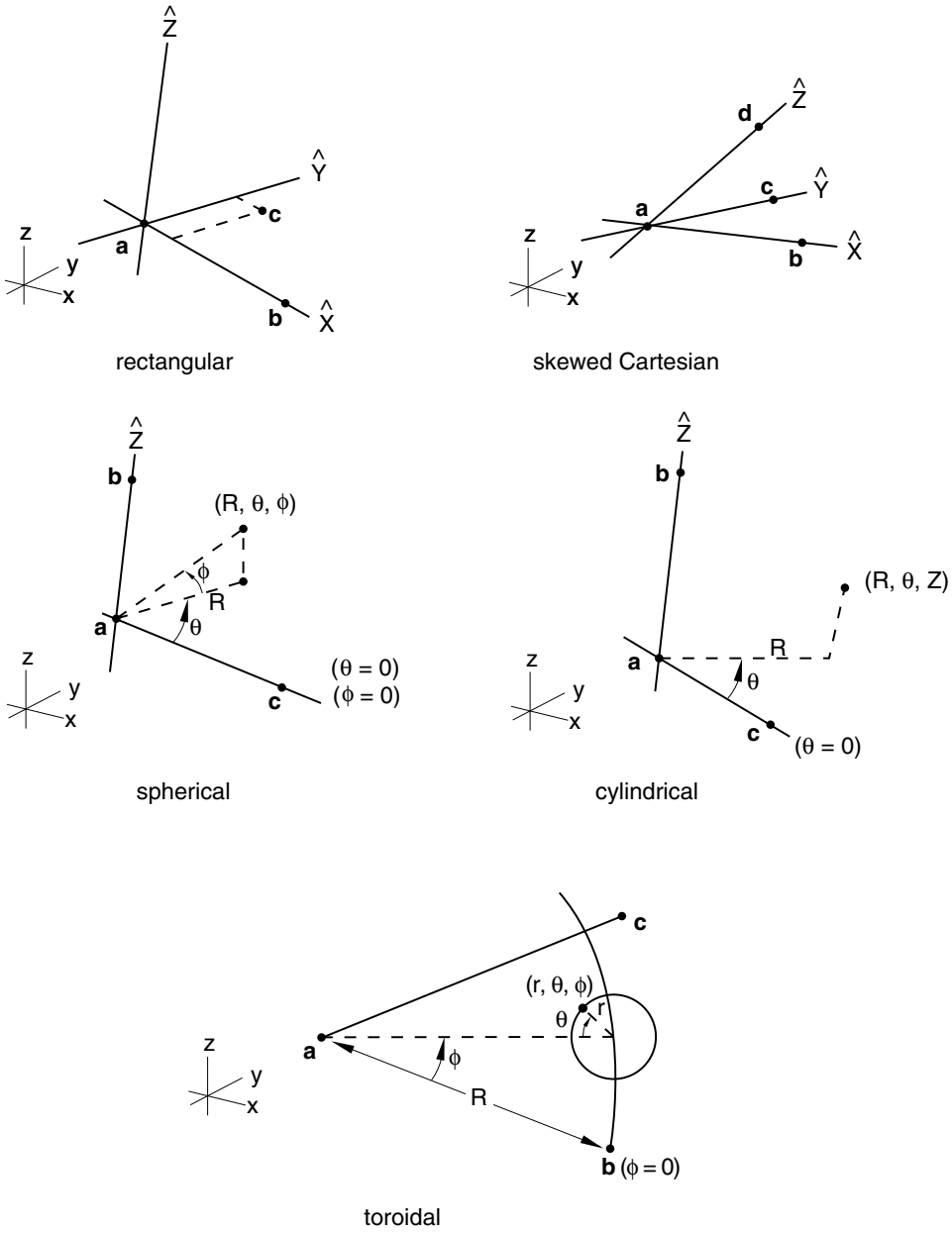



**Figure 14.4–1** Coordinate systems; angles are in degrees.

## 14.5 *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/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "No compression or no tension," Section 22.2.2 of the Abaqus Analysis User's Guide
- *ELASTIC

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

## 14.6 *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/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "No compression or no tension," Section 22.2.2 of the Abaqus Analysis User's Guide
- *ELASTIC

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

## 14.7 *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," Section 11.4.3 of the Abaqus Analysis User's Guide
- *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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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.

## 14.8 *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," Section 2.1.3 of the Abaqus Analysis User's Guide
- "Membrane elements," Section 29.1.1 of the Abaqus Analysis User's Guide
- "Using a shell section integrated during the analysis to define the section behavior," Section 29.6.5 of the Abaqus Analysis User's Guide
- "Using a general shell section to define the section behavior," Section 29.6.6 of the Abaqus Analysis User's Guide
- "Line spring elements for modeling part-through cracks in shells," Section 32.9.1 of the Abaqus Analysis User's Guide
- *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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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.

## 14.9 *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/CFD Abaqus/CAE

**Type:** Model data

**Level:** Part, Part instance, Assembly

**Abaqus/CAE:** Mesh module

**Reference:**

- "Node definition," Section 2.1.1 of the Abaqus Analysis User's Guide

**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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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 14.9–1.

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:**

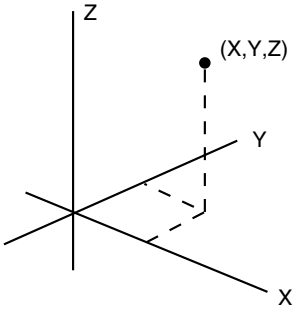
First line:

1. Node number.
2. First coordinate of the node.
3. Second coordinate of the node.

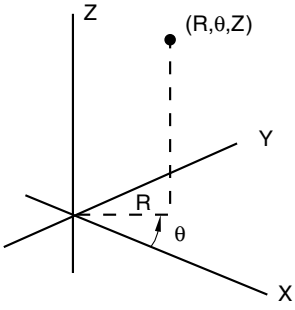
4. Third coordinate of the node.
5. First direction cosine of the normal at the node (optional).
6. Second direction cosine of the normal at the node (optional). For nodes entered in a cylindrical or spherical system, this entry is an angle given in degrees.
7. Third direction cosine of the normal at 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 Part VI, "Elements," of the Abaqus Analysis User's Guide.

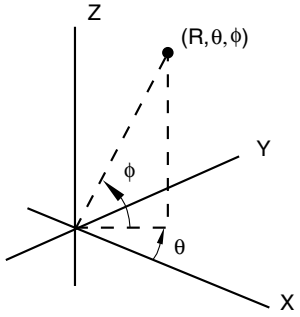
Repeat this data line as often as necessary.




Rectangular Cartesian
(default)




Cylindrical
(θ and ϕ are given in degrees)




Spherical

**Figure 14.9–1** Coordinate systems.

## 14.10 *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," Section 4.1.2 of the Abaqus Analysis User's Guide
- *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," Section 6.3.5 of the Abaqus Analysis User's Guide) and for eigenvalue buckling estimation ("Eigenvalue buckling prediction," Section 6.2.3 of the Abaqus Analysis User's Guide).

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," Section 4.2.1 of the Abaqus Analysis User's Guide, and "Abaqus/Explicit output variable identifiers," Section 4.2.2 of the Abaqus Analysis User's Guide.

Repeat this data line as often as necessary to define the nodal variables to be written to the results or selected results file.

## 14.11 *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/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Step module

**References:**

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Guide
- *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.

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:**

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.

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 parameter:**

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," Section 4.2.1 of the Abaqus Analysis User's Guide, "Abaqus/Explicit output variable identifiers," Section 4.2.2 of the Abaqus Analysis User's Guide, and "Abaqus/CFD output variable identifiers," Section 4.2.3 of the Abaqus Analysis User's Guide.

Repeat this data line as often as necessary to define the nodal variables to be written to the output database.

## 14.12 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**Reference:**

- "Output to the data and results files," Section 4.1.2 of the Abaqus Analysis User's Guide

**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," Section 6.3.5 of the Abaqus Analysis User's Guide), complex eigenvalue extraction ("Complex eigenvalue extraction," Section 6.3.6 of the Abaqus Analysis User's Guide), and eigenvalue buckling estimation ("Eigenvalue buckling prediction," Section 6.2.3 of the Abaqus Analysis User's Guide). 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," Section 4.2.1 of the Abaqus Analysis User's Guide.

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.

## 14.13 *NODE RESPONSE: Define nodal responses for design sensitivity analysis.

This option is used to write nodal response sensitivities to the output database. It must be used in conjunction with the *DESIGN RESPONSE option.

**Product:** Abaqus/Design

**Type:** History data

**Level:** Step

**References:**

- "Design sensitivity analysis," Section 19.1.1 of the Abaqus Analysis User's Guide
- *DESIGN RESPONSE

**Optional parameter:**

NSET

Set this parameter equal to the name of the node set for which this sensitivity output is being made.

**Data lines to request nodal 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 "Design sensitivity analysis," Section 19.1.1 of the Abaqus Analysis User's Guide.

Repeat this data line as often as necessary to define the nodal responses whose sensitivities are to be written to the output database.

## 14.14 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**Reference:**

- "Magnetic permeability," Section 26.5.3 of the Abaqus Analysis User's Guide

**Required parameter:**

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 parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 14.15 *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.

**Reference:**

- "Nonstructural mass definition," Section 2.7.1 of the Abaqus Analysis User's Guide

**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 parameter:**

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.

## 14.16 *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

**Reference:**

- "Normal definitions at nodes," Section 2.1.4 of the Abaqus Analysis User's Guide

**Optional parameter:**

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.

**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. Master surface name.
2. Node number or node set label (on master 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.

## 14.17 *NSET: Assign nodes to a node set.

This option assigns nodes to a node set.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** Model or history data

**Level:** Part, Part instance, Assembly, Model, Step

**Abaqus/CAE:** Set toolset

**Reference:**

- "Node definition," Section 2.1.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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 and GENERATE 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 and GENERATE 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.

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 will be ignored if the ELSET 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 parameter is specified.**

O

## 15. O

## 15.1 *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 (e.g., 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," Section 2.8.1 of the Abaqus Analysis User's Guide).

The *ORIENTATION option can also be used to specify local material directions for anisotropic hyperelastic materials with invariant-based formulation ("Invariant-based formulation" in "Anisotropic hyperelastic behavior," Section 22.5.3 of the Abaqus Analysis User's Guide). The local directions are defined with respect to the local 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.

In Abaqus/Explicit the *ORIENTATION option can be used to 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.

**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," Section 2.2.5 of the Abaqus Analysis User's Guide
- "ORIENT," Section 1.1.15 of the Abaqus User Subroutines Reference Guide

**Required parameter:**

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 15.1–1. A spatially varying local coordinate system can be created for solid continuum elements and shell elements for this parameter value by using a distribution to define spatially varying coordinates for points $a$ and $b$. Using a distribution to define the coordinates of the optional point $c$ is not currently supported. See "Distribution definition," Section 2.8.1 of the Abaqus Analysis User's Guide.

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 15.1–1. 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 $\alpha$. See "Distribution definition," Section 2.8.1 of the Abaqus Analysis User's Guide.

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 15.1–1. 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 necessary, 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 15.1–1). 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 15.1–1). 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 15.1–1. 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.

7. *X*-coordinate of point *c*.
8. *Y*-coordinate of point *c*.
9. *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 $\alpha$ 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:

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.

**Data lines to define a spatially varying orientation for solid continuum and shell elements using a distribution when DEFINITION=COORDINATES:**

First line:

1. Distribution name. Data in the distribution define 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 $\alpha$ 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:

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.

**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.

3. 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 $\alpha$ 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:

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.

**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).

3. 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 $\alpha$ 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:

1. *X*-component of the first local material direction with respect to the orthonormal system at the material point.

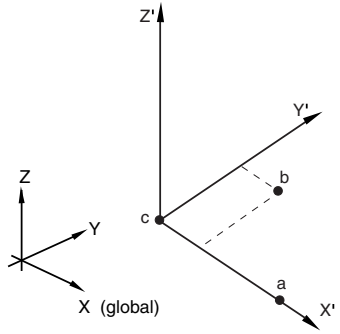
2. *Y*-component of the first local material direction with respect to the orthonormal system at the material point.
3. *Z*-component of the first local material direction with respect to the orthonormal system at the material point.

Repeat the above data line to define additional local directions, as needed, with each direction on a separate line.

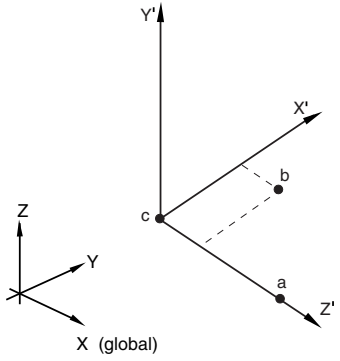
**To define an orientation using a user subroutine (SYSTEM=USER):**

No data lines are used with this option when SYSTEM=USER is specified. Instead, user subroutine **ORIENT** must be used to define the orientation.

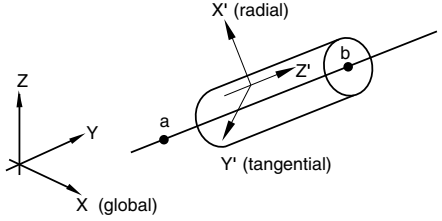
SYSTEM = RECTANGULAR




SYSTEM = Z RECTANGULAR




SYSTEM = CYLINDRICAL




SYSTEM = SPHERICAL

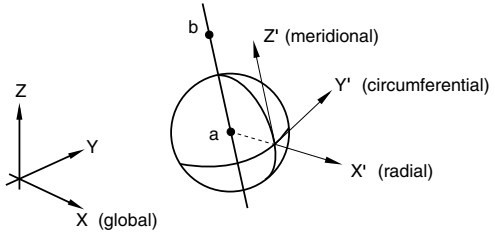



**Figure 15.1–1** Orientation systems.

## 15.2 *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

**Reference:**

- "ORNL – Oak Ridge National Laboratory constitutive model," Section 23.2.12 of the Abaqus Analysis User's Guide

**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 $\alpha$ reset procedure described in Section 4.3.5 of the Nuclear Standard. If this parameter is omitted, the $\alpha$ reset procedure is not used.

**There are no data lines associated with this option.**

## 15.3 *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/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Step module

**References:**

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Guide
- "Abaqus/Standard output variable identifiers," Section 4.2.1 of the Abaqus Analysis User's Guide
- "Abaqus/Explicit output variable identifiers," Section 4.2.2 of the Abaqus Analysis User's Guide
- "Abaqus/CFD output variable identifiers," Section 4.2.3 of the Abaqus Analysis User's Guide
- "Overview of job diagnostics," Section 41.1 of the Abaqus/CAE User's Guide
- *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 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.

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.

**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 parameter:**

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.**

## Defining output requests in an Abaqus/CFD analysis

**One of the following mutually exclusive parameters is required:**

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. Abaqus/CFD will always write field output at the beginning and end of a step.

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. Abaqus/CFD will always write history output at the beginning and end of a step.

**Optional parameters:**

NAME

Set this parameter equal to the name associated with this output definition.

FREQUENCY

Set this parameter equal to the output frequency, in increments. Set FREQUENCY=0 to suppress the output. If this parameter, the NUMBER INTERVAL and the TIME INTERVAL parameters are omitted, field output will be written at 20 equally spaced intervals, and history output at 200 equally spaced intervals.

The FREQUENCY, NUMBER INTERVAL, and TIME INTERVAL parameters are mutually exclusive.

NUMBER INTERVAL

Set this parameter equal to the number of intervals during the step at which the output database states are to be written. Set NUMBER INTERVAL=0 to suppress the output.

The FREQUENCY, NUMBER INTERVAL, and TIME INTERVAL parameters are mutually exclusive.

TIME INTERVAL

Set this parameter equal to the time interval at which the output states are to be written.

The FREQUENCY, NUMBER INTERVAL, and TIME INTERVAL parameters are mutually exclusive.

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 output clock (e.g., FREQUENCY) defined in a previous step. If there is no matching request, this output request will be interpreted as OP=ADD.

**There are no data lines associated with this option in Abaqus/CFD.**

## 16. P, Q

## 16.1 *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 input," Section 1.4.1 of the Abaqus Analysis User's Guide
- "Parametric shape variation," Section 2.1.2 of the Abaqus Analysis User's Guide

**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.

## 16.2 *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 input," Section 1.4.1 of the Abaqus Analysis User's Guide
- "Parametric shape variation," Section 2.1.2 of the Abaqus Analysis User's Guide

**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.

## 16.3 *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," Section 2.1.2 of the Abaqus Analysis User's Guide
- "Design sensitivity analysis," Section 19.1.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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 16.3–1.

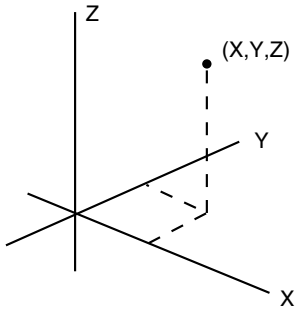
The SYSTEM parameter is entirely local to this option and should not be confused with the *SYSTEM option. As the data lines are read, the shape variation values specified are transformed to the global rectangular Cartesian coordinate system. This transformation requires that the object be centered about 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," Section 2.1.2 of the Abaqus Analysis User's Guide.

**Data lines to define the shape variation if the FILE and INPUT parameters are omitted:**

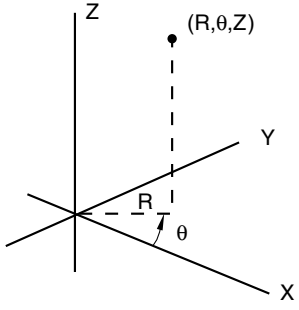
First line:

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.

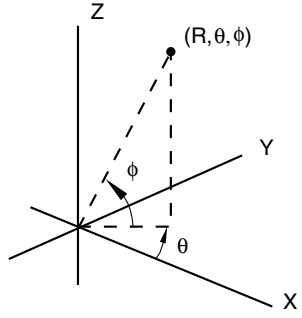
Repeat this data line as often as necessary to define the shape variation. The data given on this data line cannot be parameterized.




Rectangular Cartesian
(SYSTEM=R)
(default)




Cylindrical
(SYSTEM=C)




Spherical
(SYSTEM=S)

(θ and ϕ are given in degrees)

**Figure 16.3–1** Coordinate systems.

## 16.4 *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/CFD Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Part module

**References:**

- "Defining an assembly," Section 2.10.1 of the Abaqus Analysis User's Guide
- *END PART

**Required parameter:**

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.**

## 16.5 *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," Section 41.1.1 of the Abaqus Analysis User's Guide
- *RADIATION SYMMETRY

**Required parameter:**

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 16.5–1). 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 16.5–2). 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 16.5–3). 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 parameter:**

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 16.5–1).
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 16.5–2).
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 16.5–3).
2. $z$-component of periodic distance vector.

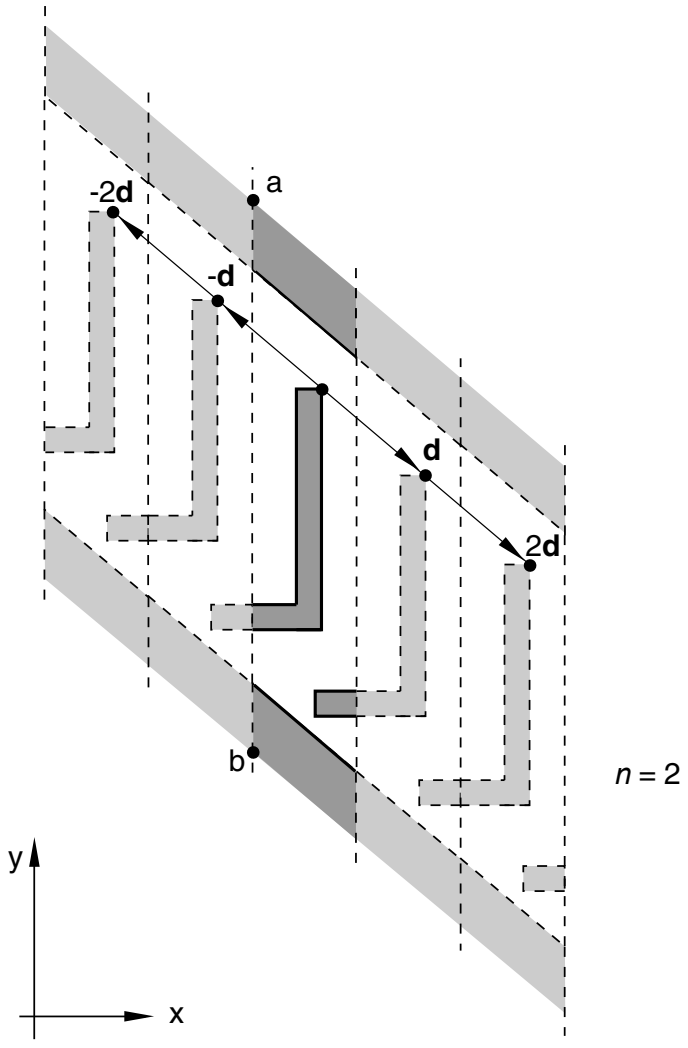



**Figure 16.5–1** *PERIODIC, TYPE=2D option.

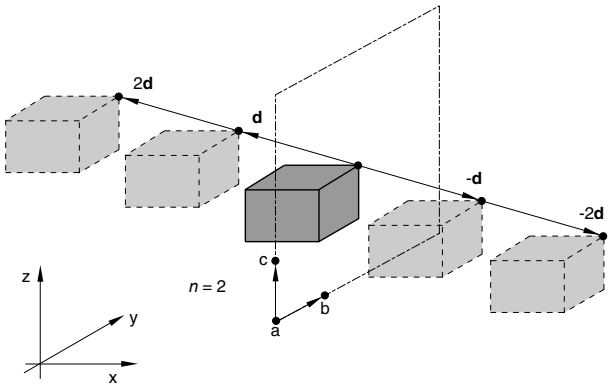



**Figure 16.5–2** *PERIODIC, TYPE=3D option.

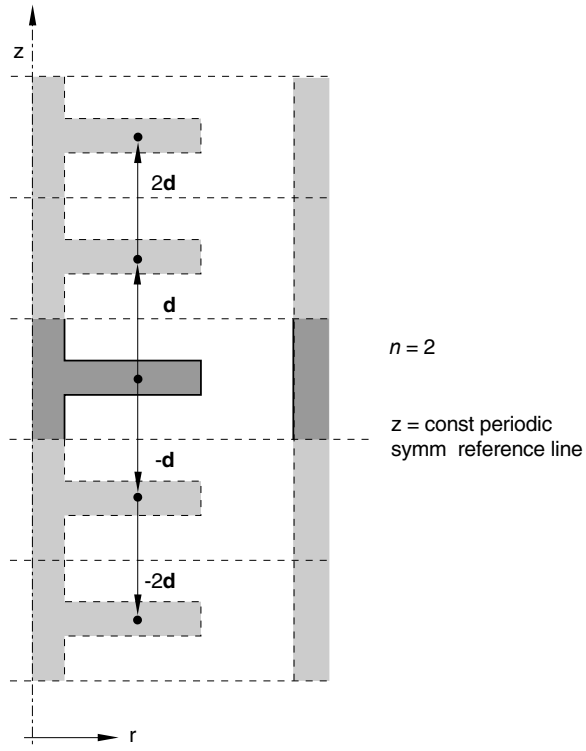



**Figure 16.5–3** *PERIODIC, TYPE=ZDIR option.

## 16.6 *PERIODIC MEDIA: Specify a periodic media.

This option is used to specify a periodic media.

**Product:** Abaqus/Explicit

**Type:** Model data

**Level:** Model

**Reference:**

- "Periodic media analysis," Section 10.5.1 of the Abaqus Analysis User's Guide

**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," Section 2.2.5 of the Abaqus Analysis User's Guide) 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.

## 16.7 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Model

**References:**

- "Magnetic permeability," Section 26.5.3 of the Abaqus Analysis User's Guide
- "Eddy current analysis," Section 6.7.5 of the Abaqus Analysis User's Guide
- "Magnetostatic analysis," Section 6.7.6 of the Abaqus Analysis User's Guide

**Optional parameter:**

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.

## 16.8 *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/CFD Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**Reference:**

- "Permeability," Section 26.6.2 of the Abaqus Analysis User's Guide

### 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.

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, $\gamma_w$ (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," Section 6.8.1 of the Abaqus Analysis User's Guide, 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, $\theta$.
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, $\theta$.
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, $\theta$.

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. $\beta$. (Units of $\mathrm{TL}^{-1}$.) Only $\beta > 0.0$ is allowed.
2. Void ratio, ***e***.

Repeat this data line as often as necessary to define the variation.

## Defining permeability in Abaqus/CFD analyses

**Optional parameters:**

INERTIAL DRAG COEFFICIENT

Set this parameter proportional to the value of the inertial (quadratic or form) drag in the porous medium. The default value is 0.142887.

TYPE

Set TYPE=ISOTROPIC (default) to define fully saturated isotropic permeability.

Set TYPE=CARMAN KOZENY to define permeability as a function of porosity through the Carman-Kozeny relation.

**Data lines to define fully saturated isotropic permeability (TYPE=ISOTROPIC):**

First line:

1. $K$. (Units of $L^2$.)
2. Porosity, $\epsilon$.

Repeat this data line as often as necessary to define the variation.

**Data line to define the Carman-Kozeny permeability relation (TYPE=CARMAN KOZENY):**

First (and only) line:

1. Carman-Kozeny constant. (Dimensionless.)
2. Pore-particle radius (for a fibrous medium this is equal to the characteristic fiber radius). (Units of L.)

## 16.9 *PHYSICAL CONSTANTS: Specify physical constants.

This option is used to define physical constants necessary for an analysis; since Abaqus has no built-in units, no default values are provided. If a physical constant required for the analysis is not given, Abaqus will issue 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," Section 6.5.2 of the Abaqus Analysis User's Guide
- "Mass diffusion analysis," Section 6.9.1 of the Abaqus Analysis User's Guide
- "Fluid cavity definition," Section 11.5.2 of the Abaqus Analysis User's Guide
- "Rate-dependent plasticity: creep and swelling," Section 23.2.4 of the Abaqus Analysis User's Guide
- "Diffusivity," Section 26.4.1 of the Abaqus Analysis User's Guide
- "Solubility," Section 26.4.2 of the Abaqus Analysis User's Guide
- "Thermal contact properties," Section 37.2.1 of the Abaqus Analysis User's Guide
- "Cavity radiation," Section 41.1.1 of the Abaqus Analysis User's Guide

**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.

STEFAN BOLTZMANN

Set this parameter equal to the Stefan Boltzmann constant. For example, STEFAN BOLTZMANN=$5.669 \times 10^{-8}$ 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.

SPL REFERENCE PRESSURE

Set this parameter equal to the reference pressure used for computing sound pressure level. For example, SPL REFERENCE PRESSURE=20. micro pascals for air in SI units.

**There are no data lines associated with this option.**

## 16.10 *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

**Reference:**

- "Piezoelectric behavior," Section 26.5.2 of the Abaqus Analysis User's Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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 $\mathrm{FL}^{-1}\varphi^{-1}$.)
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, $\theta$.
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, $\theta$.
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.

## 16.11 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance

**References:**

- "Pipe-soil interaction elements," Section 32.12.1 of the Abaqus Analysis User's Guide
- "Pipe-soil interaction element library," Section 32.12.2 of the Abaqus Analysis User's Guide
- *PIPE-SOIL STIFFNESS

**Required parameter:**

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 parameter:**

ORIENTATION

Set this parameter equal to the name of the *ORIENTATION definition ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Guide) that gives the orientation of the local system used for material calculations.

**There are no data lines associated with this option.**

## 16.12 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance

**References:**

- "Pipe-soil interaction elements," Section 32.12.1 of the Abaqus Analysis User's Guide
- "Pipe-soil interaction element library," Section 32.12.2 of the Abaqus Analysis User's Guide
- "UMAT," Section 1.1.41 of the Abaqus User Subroutines Reference Guide
- *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," Section 21.1.2 of the Abaqus Analysis User's Guide 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, $\bar{\gamma}$.
2. Coefficient of soil pressure at rest, $K_0$.
3. Interface angle of friction, $\delta$. 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, $\alpha$.
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, $\gamma$.
2. Bearing capacity factor, $N_q$.
3. Bearing capacity factor, $N_\gamma$.
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, $\bar{\varepsilon}_{vp}$.
4. Ultimate relative displacement for upward motion, $\bar{\varepsilon}_{vn}$.
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, $\bar{\varepsilon}_h$.
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, $\bar{\varepsilon}_h$.
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.

## 16.13 *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," Section 22.5.1 of the Abaqus Analysis User's Guide
- "Hyperelastic behavior in elastomeric foams," Section 22.5.2 of the Abaqus Analysis User's Guide
- "Mullins effect," Section 22.6.1 of the Abaqus Analysis User's Guide
- "Energy dissipation in elastomeric foams," Section 22.6.2 of the Abaqus Analysis User's Guide
- *HYPERELASTIC
- *HYPERFOAM
- *MULLINS EFFECT

### Using planar test data to define a hyperelastic material

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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, $\epsilon_S$.

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, $\epsilon_S$.
3. Nominal lateral strain, $\epsilon_3$. Not needed if the POISSON parameter is specified on the *HYPERELASTIC option or if the *VOLUMETRIC TEST DATA option is used.
4. Temperature, $\theta$.
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, $\epsilon_P$.
3. Nominal transverse strain, $\epsilon_3$. 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 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, $\epsilon_P$.

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

## 16.14 *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," Section 23.2.1 of the Abaqus Analysis User's Guide
- "Models for metals subjected to cyclic loading," Section 23.2.2 of the Abaqus Analysis User's Guide
- "Johnson-Cook plasticity," Section 23.2.7 of the Abaqus Analysis User's Guide
- "Permanent set in rubberlike materials," Section 23.7.1 of the Abaqus Analysis User's Guide
- "UHARD," Section 1.1.36 of the Abaqus User Subroutines Reference Guide
- "VUHARD," Section 1.2.17 of the Abaqus User Subroutines Reference Guide

**Optional parameters:**

HARDENING

Set HARDENING=ISOTROPIC (default) to specify isotropic hardening.
Set HARDENING=KINEMATIC to specify linear 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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

**Optional parameter for use with HARDENING=ISOTROPIC:**

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.

**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, $\gamma_1$.
4. Etc., specify kinematic hardening parameters $C_k$ and $\gamma_k$ 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 $\gamma_k$ 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 line for HARDENING=JOHNSON COOK:**

First (and only) line:

1. $A$.
2. $B$.
3. $n$.
4. $m$.
5. Melting temperature, $\theta_{\text{melt}}$.
6. Transition temperature, $\theta_{\text{transition}}$.

**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.

## 16.15 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance

**References:**

- "Frame section behavior," Section 29.4.2 of the Abaqus Analysis User's Guide
- *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.

## 16.16 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance

**References:**

- "Frame section behavior," Section 29.4.2 of the Abaqus Analysis User's Guide
- *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.

## 16.17 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance

**References:**

- "Frame section behavior," Section 29.4.2 of the Abaqus Analysis User's Guide
- *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.

## 16.18 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance

**References:**

- "Frame section behavior," Section 29.4.2 of the Abaqus Analysis User's Guide
- *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.

## 16.19 *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.

**Products:** Abaqus/Standard Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**Reference:**

- "Porous bulk moduli," Section 26.6.3 of the Abaqus Analysis User's Guide

**There are no parameters associated with this option.**

**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.

Repeat this data line as often as necessary to define the dependence of the bulk moduli on temperature.

## 16.20 *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

**Reference:**

- "Elastic behavior of porous materials," Section 22.3.1 of the Abaqus Analysis User's Guide

**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. See "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

SHEAR

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.

**Data lines to define the deviatoric behavior with a constant shear modulus *G*:**

First line:

1. Value of the logarithmic bulk modulus, $\kappa$. (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, $\theta$.
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 $\kappa$, $G$, 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:**

First line:

1. Value of the logarithmic bulk modulus, $\kappa$. (Dimensionless.)
2. Value of Poisson's ratio, $\nu$.
3. Value of the elastic tensile limit, $p_t^{el}$. (This value cannot be negative.)
4. Temperature, $\theta$.
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 $\kappa$, $\nu$, and $p_t^{el}$ on temperature and field variables.

## 16.21 *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," Section 23.2.9 of the Abaqus Analysis User's Guide

**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 $f_F$.

## 16.22 *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," Section 23.2.9 of the Abaqus Analysis User's Guide
- *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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 16.23 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Model

**References:**

- "Output," Section 4.1.1 of the Abaqus Analysis User's Guide
- "Restarting an analysis," Section 9.1.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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," Section 6.2.7 of the Abaqus Analysis User's Guide).

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," Section 6.2.6 of the Abaqus Analysis User's Guide).

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.

## 16.24 *POTENTIAL: Define an anisotropic yield/creep model.

This option is used to define stress ratios for anisotropic yield and creep behavior. It can be used only in conjunction with material models defined by the *CREEP option, the *PLASTIC option (HARDENING=ISOTROPIC, KINEMATIC, or COMBINED; the *POTENTIAL option can be used in conjunction with COMBINED hardening only in Abaqus/Explicit), and/or the *VISCOUS option.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "Classical metal plasticity," Section 23.2.1 of the Abaqus Analysis User's Guide
- "Models for metals subjected to cyclic loading," Section 23.2.2 of the Abaqus Analysis User's Guide
- "Rate-dependent plasticity: creep and swelling," Section 23.2.4 of the Abaqus Analysis User's Guide
- "Anisotropic yield/creep," Section 23.2.6 of the Abaqus Analysis User's Guide
- "Two-layer viscoplasticity," Section 23.2.11 of the Abaqus Analysis User's Guide

**Optional parameter:**

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of $R_{ij}$ in addition to temperature. If this parameter is omitted, it is assumed that the anisotropy ratios are constant or depend only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

**Data lines used to define stress ratios:**

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.

## 16.25 *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

**Reference:**

- "Output," Section 4.1.1 of the Abaqus Analysis User's Guide

**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.**

## 16.26 *PRESSURE EQUATION SOLVER: Specify the linear solver and parameters for solving the pressure equation for incompressible flows.

This option can be used only as a suboption of the *CFD option to enable the linear solver parameters for the pressure equation for incompressible flows. For a steady-state analysis this option is also used to enable the under-relaxation parameter for the pressure correction equation.

**Products:** Abaqus/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Step module

**References:**

- "Incompressible fluid dynamic analysis," Section 6.6.2 of the Abaqus Analysis User's Guide
- *CFD

**Optional parameters:**

CONVERGENCE

Set CONVERGENCE=ON to write convergence information to the log file.
Set CONVERGENCE=OFF (default) to suppress convergence information.

DIAGNOSTICS

Set DIAGNOSTICS=ON to write diagnostic information about the solver to the log file.
Set DIAGNOSTICS=OFF (default) to suppress diagnostic information.

TYPE

Set TYPE=AMG (default) to enable the Algebraic Multigrid linear solver.
Set TYPE=DSCG to enable the Diagonal Scaled Conjugate Gradient linear solver.
Set TYPE=SSORCG to enable the Symmetric Successive Over Relaxation Conjugate Gradient linear solver.

**Data lines for TYPE=AMG:**

First line:

1. Maximum number of iterations (default=250).
2. Interval to check linear convergence (default=2).

3. Linear convergence criterion (default=$10^{-5}$).
4. Under-relaxation factor (default=0.3). This value is valid only for a steady-state analysis.

Second line:

1. Specify the algebraic multigrid smoother as CHEBYCHEV, ILU, ICC, or SSOR (default=ICC).
2. Number of pre-coarsening smoothing steps (default=1).
3. Number of post-coarsening smoothing steps (default=1).
4. Specify the Krylov solver as CG, BCGS, or FGMRES (default=CG).

Third line:

1. Specify the algebraic multigrid cycle as V or W (default=V).

**Data line for TYPE=DSCG and TYPE=SSORCG:**

First (and only) line:

1. Maximum number of iterations (default=250).
2. Interval to check linear convergence (default=2).
3. Linear convergence criterion (default=$10^{-5}$).
4. Under-relaxation factor (default=0.3). This value is valid only for a steady-state analysis.

## 16.27 *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

**Reference:**

- "Pressure penetration loading," Section 37.1.7 of the Abaqus Analysis User's Guide

**Required parameters:**

MASTER

Set this parameter equal to the name of the master surface of the contact pair used in the pressure penetration analysis.

SLAVE

Set this parameter equal to the name of the slave 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," Section 6.1.2 of the Abaqus Analysis User's Guide).

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.

**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 slave 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 master 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 master surface is an analytical rigid surface. This field can be also left blank for a three-dimensional model even if the master 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).

## 16.28 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Mass diffusion analysis," Section 6.9.1 of the Abaqus Analysis User's Guide
- "Predefined fields," Section 34.6.1 of the Abaqus Analysis User's Guide

**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," Section 6.1.2 of the Abaqus Analysis User's Guide).

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Guide, 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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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.

## 16.29 *PRESTRESS HOLD: Keep rebar prestress constant during initial equilibrium solution.

This option is used within a *STATIC step ("Static stress analysis," Section 6.2.2 of the Abaqus Analysis User's Guide) to keep the stress in some or all of the rebar constant during the initial equilibrium solution.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**Reference:**

- "Defining reinforcement," Section 2.2.3 of the Abaqus Analysis User's Guide

**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.

## 16.30 *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

**Reference:**

- "Prescribed assembly loads," Section 34.5.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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).

**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.

## 16.31 *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:**

- "Output," Section 4.1.1 of the Abaqus Analysis User's Guide
- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Guide

**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 will be printed for every increment unless FREQUENCY=0. The default is CONTACT=NO.

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.

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 and a column listing the value 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" in "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Guide, 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.**

## 16.32 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Model

**References:**

- "Random response analysis," Section 6.3.11 of the Abaqus Analysis User's Guide
- "UPSD," Section 1.1.48 of the Abaqus User Subroutines Reference Guide
- *CORRELATION
- *RANDOM RESPONSE

**Required parameter:**

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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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.

# 17. R

## 17.1 *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 heat transfer, coupled thermal-electrical, and coupled thermal-electrical-structural analyses.

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

**Reference:**

- "Thermal loads," Section 34.4.4 of the Abaqus Analysis User's Guide

**Optional parameters:**

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," Section 6.1.2 of the Abaqus Analysis User's Guide). 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.

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 Part VI, “Elements,” of the Abaqus Analysis User’s Guide).
3. Reference ambient temperature value, $\theta^0$. (Units of $\theta$.)
4. Emissivity, $\epsilon$.

Repeat this data line as often as necessary to define radiation conditions.

## 17.2 *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," Section 41.1.1 of the Abaqus Analysis User's Guide
- "Output," Section 4.1.1 of the Abaqus Analysis User's Guide

**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 parameter:**

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," Section 4.2.1 of the Abaqus Analysis User's Guide.

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.

## 17.3 *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," Section 4.1.3 of the Abaqus Analysis User's Guide
- *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 parameter:**

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," Section 4.2.1 of the Abaqus Analysis User's Guide.

Repeat this data line as often as necessary to define the cavity radiation variables to be written to the output database.

## 17.4 *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).

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Cavity radiation," Section 41.1.1 of the Abaqus Analysis User's Guide
- "Output," Section 4.1.1 of the Abaqus Analysis User's Guide

**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," Section 4.2.1 of the Abaqus Analysis User's Guide.

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.

## 17.5 *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," Section 41.1.1 of the Abaqus Analysis User's Guide
- *CYCLIC
- *PERIODIC
- *REFLECTION

**Required parameter:**

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.**

## 17.6 *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

**Reference:**

- "Cavity radiation," Section 41.1.1 of the Abaqus Analysis User's Guide

**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," Section 2.11.4 of the Abaqus Theory Guide).

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.**

## 17.7 *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," Section 6.3.11 of the Abaqus Analysis User's Guide
- *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," Section 6.3.11 of the Abaqus Analysis User's Guide.
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.

## 17.8 *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=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," Section 23.2.3 of the Abaqus Analysis User's Guide
- "Classical metal plasticity," Section 23.2.1 of the Abaqus Analysis User's Guide
- "Johnson-Cook plasticity," Section 23.2.7 of the Abaqus Analysis User's Guide
- "Extended Drucker-Prager models," Section 23.3.1 of the Abaqus Analysis User's Guide
- "Crushable foam plasticity models," Section 23.3.5 of the Abaqus Analysis User's Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

TYPE

Set TYPE=POWER LAW (default) to specify the Cowper-Symonds overstress power law.

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. $D$.
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 line to define the Johnson-Cook rate parameters (TYPE=JOHNSON COOK):**

First (and only) line:

1. $C$.
2. $\dot{\varepsilon}_0$.

**Data lines to define yield stress ratios (TYPE=YIELD RATIO):**

First line:

1. Yield stress ratio, $R = \bar{\sigma}/\sigma^0$.
2. Equivalent plastic strain rate, $\dot{\bar{\varepsilon}}^{pl}$ (or $\dot{\varepsilon}^{pl}_{axial}|$, 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.

## 17.9 *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," Section 26.6.6 of the Abaqus Analysis User's Guide
- "Rate-dependent plasticity: creep and swelling," Section 23.2.4 of the Abaqus Analysis User's Guide
- *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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 17.10 *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

**Reference:**

- "Equation of state," Section 25.2.1 of the Abaqus Analysis User's Guide

**There are no parameters associated with this option.**

**Data lines to define the reaction rate:**

First line:

1. Initial pressure, $I$. (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.)

## 17.11 *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," Section 2.2.4 of the Abaqus Analysis User's Guide
- "Defining reinforcement," Section 2.2.3 of the Abaqus Analysis User's Guide

**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 $X_1$ (see Figure 17.11–1).
4. Distance $X_2$.

Repeat this data line as often as necessary. Each line defines a single rebar.

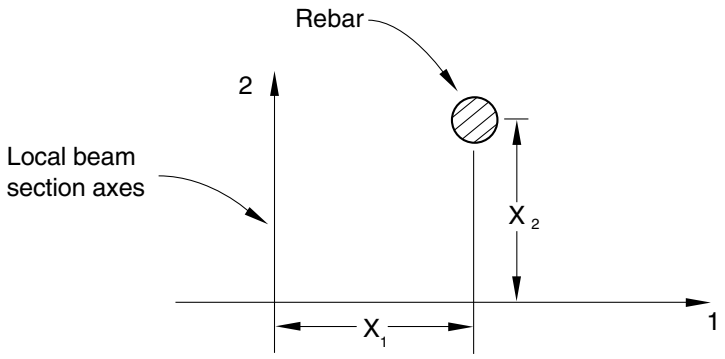



**Figure 17.11–1** Rebar location in a beam section.

**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 17.11–2.

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 17.11–2.

Repeat this data line as often as necessary. Each line defines a layer of rebar.

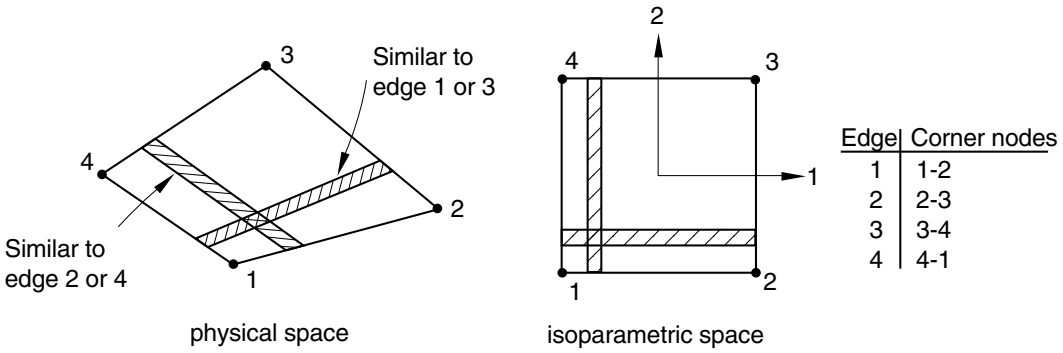



**Figure 17.11–2** "Isoparametric" rebar in a three-dimensional shell or membrane.

**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 17.11–3.
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 17.11–4 or Figure 17.11–7.
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 17.11–3.
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 17.11–5).
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 17.11–6).

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 17.11–7 for the isoparametric direction chosen.

Repeat this data line as often as necessary. Each line defines a single rebar.

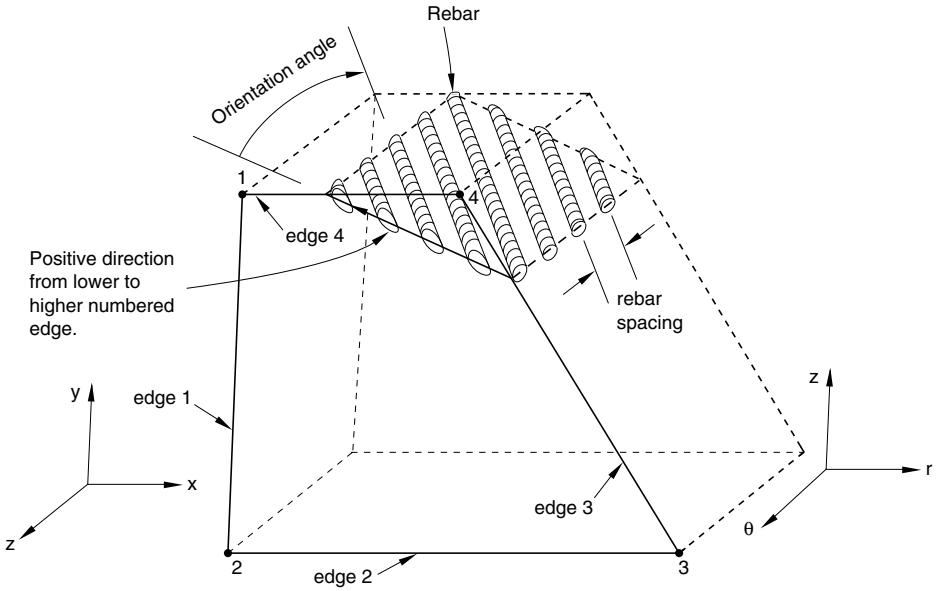



**Figure 17.11–3** Orientation of rebar in plane and axisymmetric solid elements.

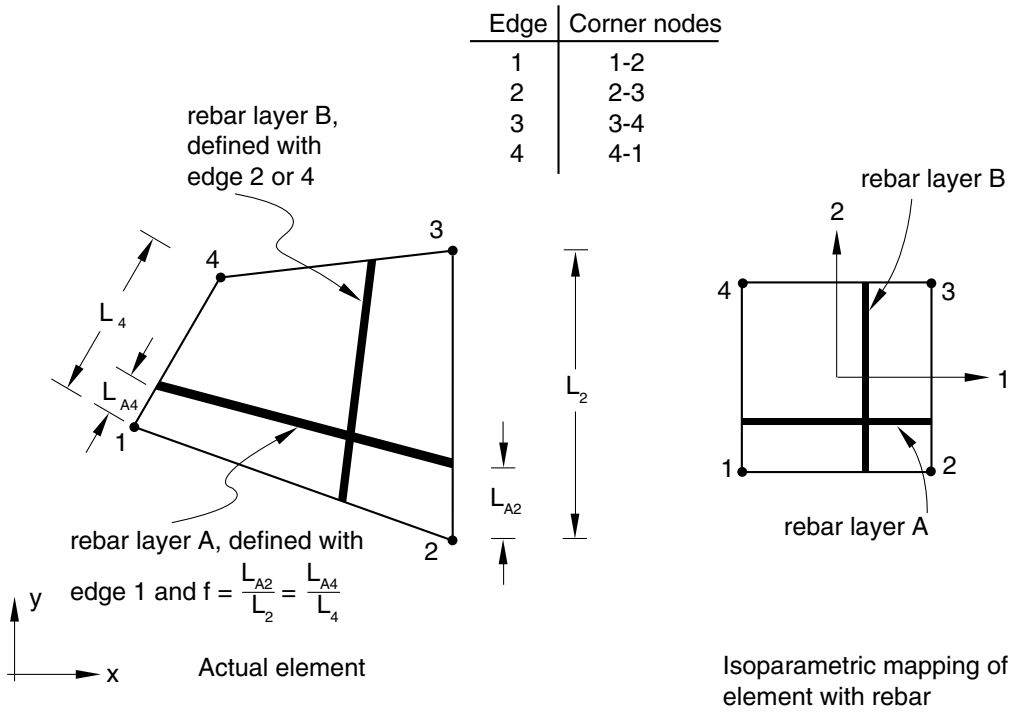



**Figure 17.11–4** Rebar layer definition in solid elements with GEOMETRY=ISOPARAMETRIC.

| Edge | Corner nodes |
|---|---|
| 1 | 1-2 |
| 2 | 2-3 |
| 3 | 3-4 |
| 4 | 4-1 |

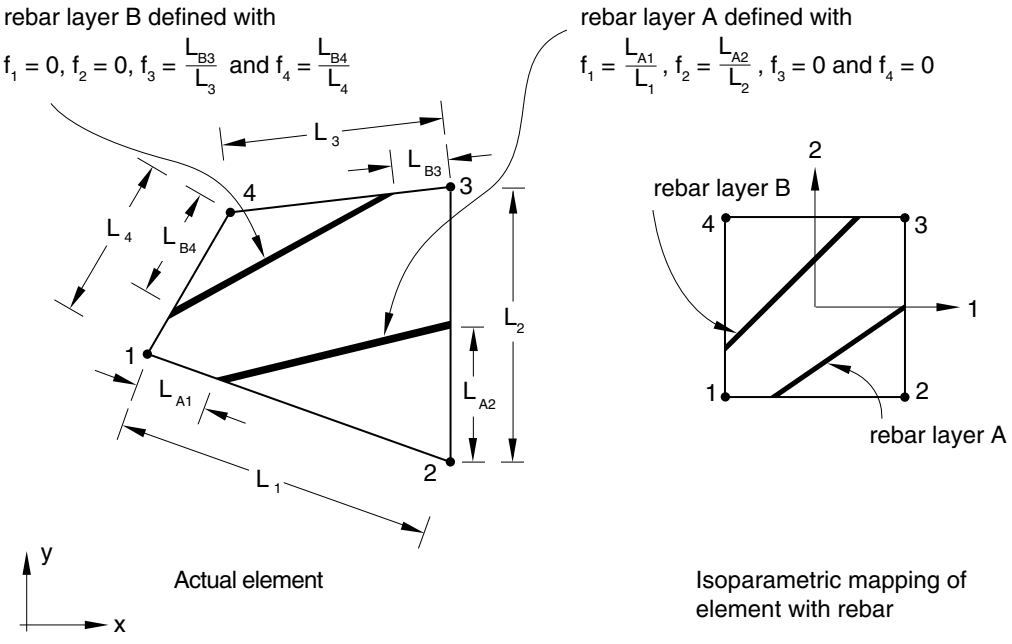



**Figure 17.11–5** Rebar layer definition in solid elements with GEOMETRY=SKEW.

| Edge | Corner nodes |
|---|---|
| 1 | 1-2 |
| 2 | 2-3 |

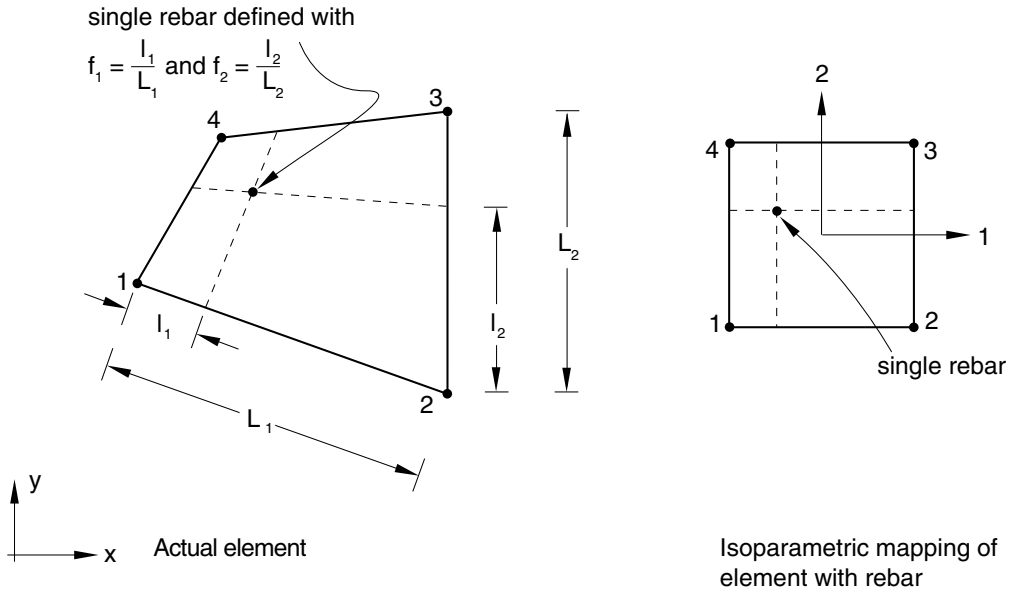
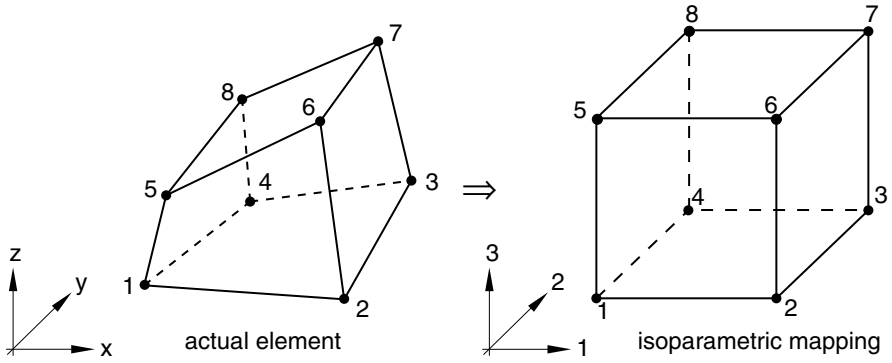



**Figure 17.11–6** SINGLE rebar in a solid element.




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 17.11–7** Isoparametric direction and edge definitions for three-dimensional elements.

## 17.12 *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," Section 2.2.3 of the Abaqus Analysis User's Guide
- "Embedded elements," Section 35.4.1 of the Abaqus Analysis User's Guide
- *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.

## 17.13 *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," Section 41.1.1 of the Abaqus Analysis User's Guide
- *RADIATION SYMMETRY

**Required parameter:**

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 17.13–1. 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 17.13–2. 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 17.13–3. 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 17.13–1).
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 17.13–2).
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 17.13–3).

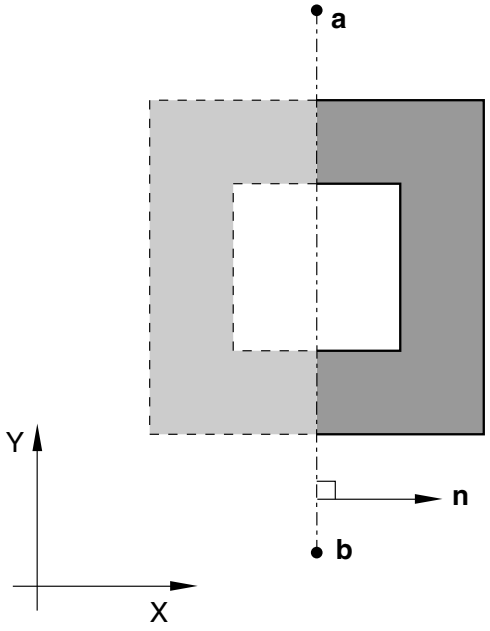



**Figure 17.13–1** *REFLECTION, TYPE=LINE option.

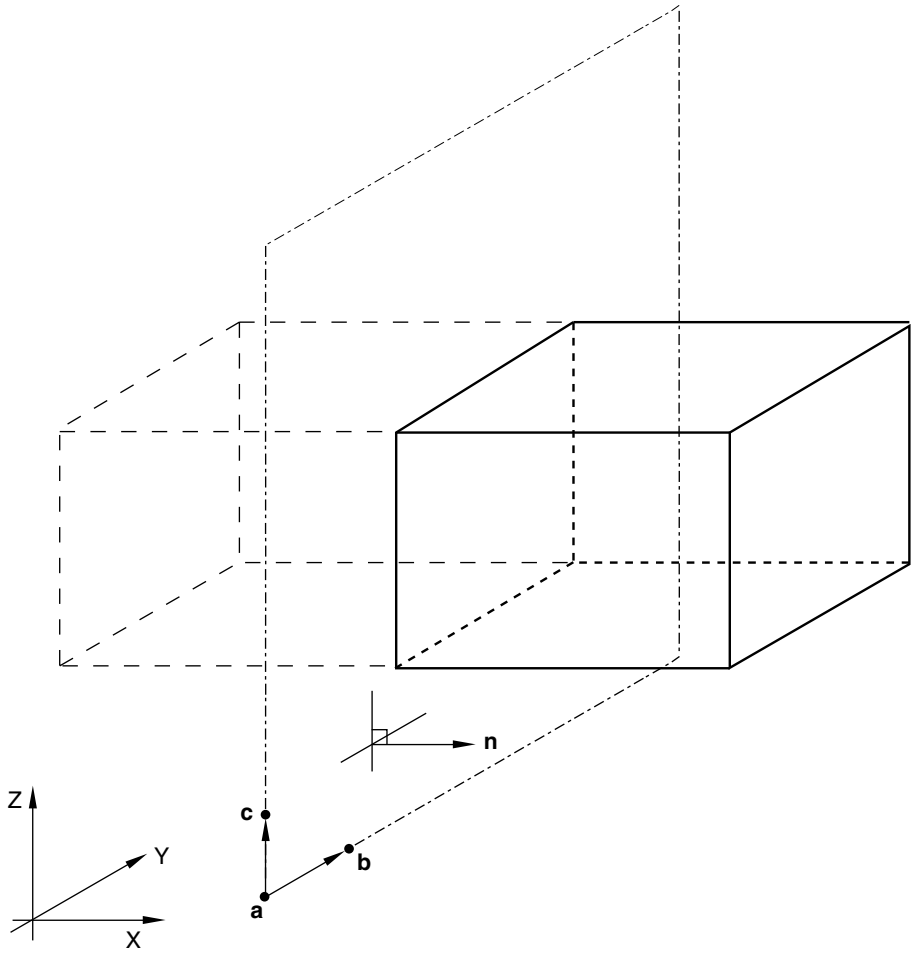



**Figure 17.13–2** *REFLECTION, TYPE=PLANE option.

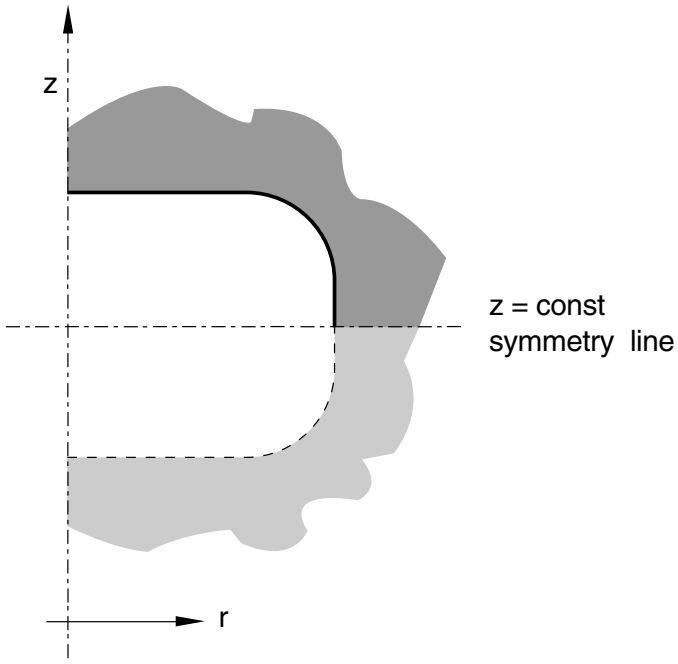



**Figure 17.13–3** *REFLECTION, TYPE=ZCONST option.

## 17.14 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance

**References:**

- "Kinematic coupling constraints," Section 35.2.3 of the Abaqus Analysis User's Guide
- "Element end release," Section 35.5.1 of the Abaqus Analysis User's Guide

**Optional parameter:**

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Guide, 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," Section 35.5.1 of the Abaqus Analysis User's Guide, 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.

## 17.15 *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," Section 6.3.10 of the Abaqus Analysis User's Guide
- *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.

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.

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.

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.

## 17.16 *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/CFD 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

**Reference:**

- "Restarting an analysis," Section 9.1.1 of the Abaqus Analysis User's Guide

### 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. The default is NUMBER INTERVAL=1.

Abaqus/Explicit will always write the restart data at the beginning and 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,” Section 6.3.3 of the Abaqus Analysis User’s Guide).

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.**

## Controlling the writing and reading of restart data in an Abaqus/CFD 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:**

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 boundary condition history, output options, or solver settings, 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/CFD 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.

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.

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/CFD 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.

**There are no data lines associated with this option.**

## 17.17 *RETAINED NODAL DOFS: Specify the degrees of freedom that are to be retained as external to a substructure.

This option is used to list degrees of freedom that are to be retained as external degrees of freedom on the substructure. It can be used only in a *SUBSTRUCTURE GENERATE analysis.

**Products:** Abaqus/Standard Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Load module

**Reference:**

- "Defining substructures," Section 10.1.2 of the Abaqus Analysis User's Guide

**Optional parameter:**

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 will be retained.

Repeat this data line as often as necessary to list all degrees of freedom to be retained.

## 17.18 *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," Section 30.3.1 of the Abaqus Analysis User's Guide
- "Analytical rigid surface definition," Section 2.3.4 of the Abaqus Analysis User's Guide
- "Rigid body definition," Section 2.4.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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:**

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 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.

## 17.19 *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:**

- "Surfaces: overview," Section 2.3.1 of the Abaqus Analysis User's Guide
- "Analytical rigid surface definition," Section 2.3.4 of the Abaqus Analysis User's Guide
- "Drag chains," Section 32.11.1 of the Abaqus Analysis User's Guide
- "RSURFU," Section 1.1.16 of the Abaqus User Subroutines Reference Guide
- *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 parameter:**

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.

Second and subsequent data lines define the various line, circular, and parabolic segments (see below for their format) that form the profile of the rigid surface.

**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 17.19–1).
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.

Fourth and subsequent data lines define the various line, circular, and parabolic segments (see below for their format) that form the profile of the rigid surface.

**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 17.19–2).
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 rigid surface.

**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.

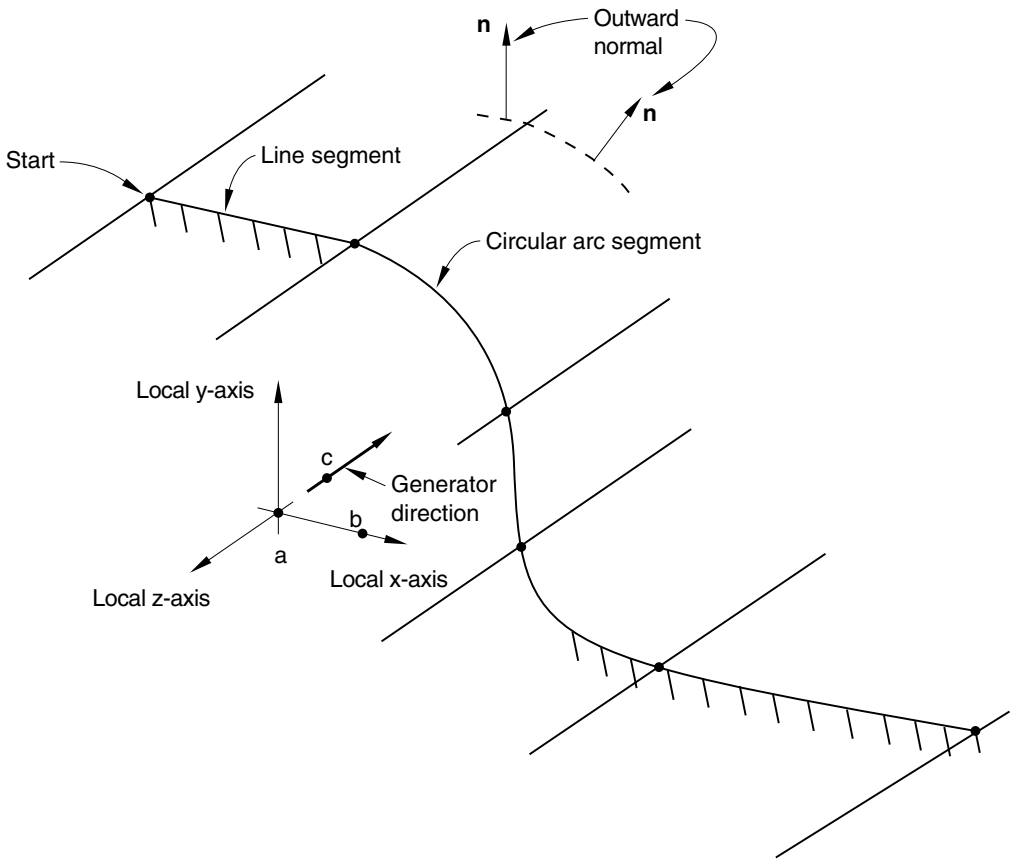



**Figure 17.19–1** *RIGID SURFACE, TYPE=CYLINDER.

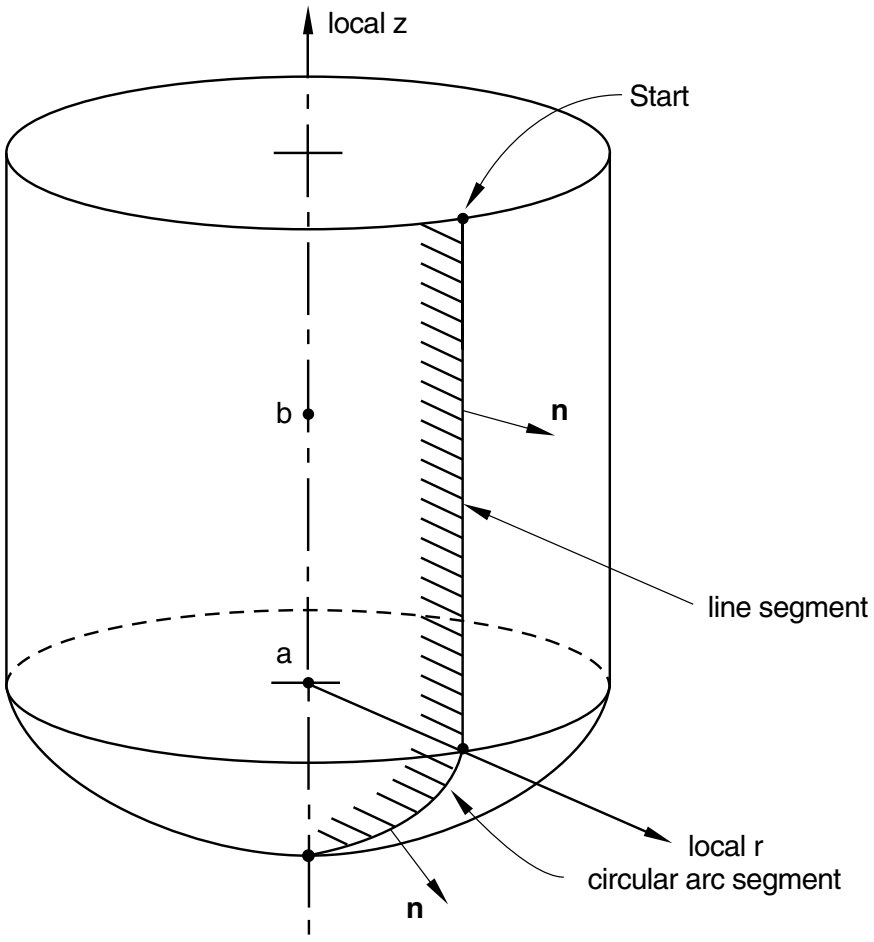



**Figure 17.19–2** *RIGID SURFACE, TYPE=REVOLUTION.

## 17.20 *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.

**Reference:**

- "Rotary inertia," Section 30.2.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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 $\alpha_R$ 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," Section 2.2.5 of the Abaqus Analysis User's Guide) 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.

# 18. S

## 18.1 *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, and
- activate distortion control for solid elements.

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 additional control parameters for smoothed particle hydrodynamic 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, and
- specify control parameters related to smoothed particle hydrodynamic (SPH) analyses.

This option is used in conjunction with the *MEMBRANE SECTION, *SHELL GENERAL SECTION, *SHELL SECTION, and/or *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 the necessary *SECTION CONTROLS settings must be selected 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," Section 27.1.4 of the Abaqus Analysis User's Guide
- *COHESIVE SECTION
- *CONNECTOR SECTION
- *MEMBRANE SECTION
- *SHELL GENERAL SECTION
- *SHELL SECTION
- *SOLID SECTION

**Required parameter:**

NAME

Set this parameter equal to a label that will be used to refer to the section control definition. All section control names in the same input file must be unique.

**Optional parameters:**

CONVERSION CRITERION

This parameter applies only to Abaqus/Explicit analyses involving the conversion of continuum elements to SPH particles.

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.

DISTORTION CONTROL

This parameter applies to Abaqus/Explicit analyses and to solid sections in Abaqus/Standard analyses containing C3D10I 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.

ELEMENT DELETION

This parameter applies to all elements with progressive damage behavior, except connector elements in Abaqus/Explicit.

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 will be 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.

INITIAL GAP OPENING

This parameter applies only to Abaqus/Standard analyses using pore pressure cohesive 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. 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 will be 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 $r = 0.1$.

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 will be assumed to be completely damaged. This parameter also determines the amount of residual stiffness that will be retained by elements for which the ELEMENT DELETION parameter is set to NO. For elements other than cohesive elements, connector elements, 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, and elements with plane stress formulations the default value is always 1.0.

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.

RAMP INITIAL STRESS

This parameter applies to membrane elements 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.

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 $\alpha$ ($0.0 \leq \alpha \leq 1.0$) 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, 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 will use 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 will use 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 in Abaqus/Explicit. If this value is left blank, Abaqus/Explicit will use 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 will use 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 will use 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 will use 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. Not used.
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. Enter a value of 0 (default) to indicate a constant smoothing length throughout the analysis. Enter a value of 1 to indicate a variable smoothing length.

4. Maximum number of nodes associated with a given particle (default is 140).
5. Mean velocity filtering coefficient for modified coordinate updates for particles. Values range from 0.0 (default) to 1.0. If a nonzero value is specified, the method commonly referred to in the literature as the XSPH method is used.
6. Enter a value of 0 (default) to indicate that the classical SPH formulation is to be used. Specify a value of 1 to trigger a modified formulation involving a corrected kernel method referred to in the literature as the normalized SPH (NSPH) method.

Third line (optional, to be used only in conjunction with smoothed particle hydrodynamics 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.

Fourth line (optional, to be used only in conjunction with the conversion of continuum elements to smoothed particle hydrodynamic particles):

1. Number of particles to be generated per isoparametric direction.
2. Threshold value for the conversion criterion.

## 18.2 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Output to the data and results files," Section 4.1.2 of the Abaqus Analysis User's Guide
- "Abaqus/Standard output variable identifiers," Section 4.2.1 of the Abaqus Analysis User's Guide

**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 18.2–1 (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.

4. Third coordinate of point $a$ (ignored if node number given).
5. Node number used to specify point $b$ (blank if coordinates given).
6. First coordinate of point $b$ (ignored if node number given).
7. Second coordinate of point $b$ (ignored if node number given).
8. 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," Section 4.2.1 of the Abaqus Analysis User's Guide.

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," Section 4.1.2 of the Abaqus Analysis User's Guide) will be output.

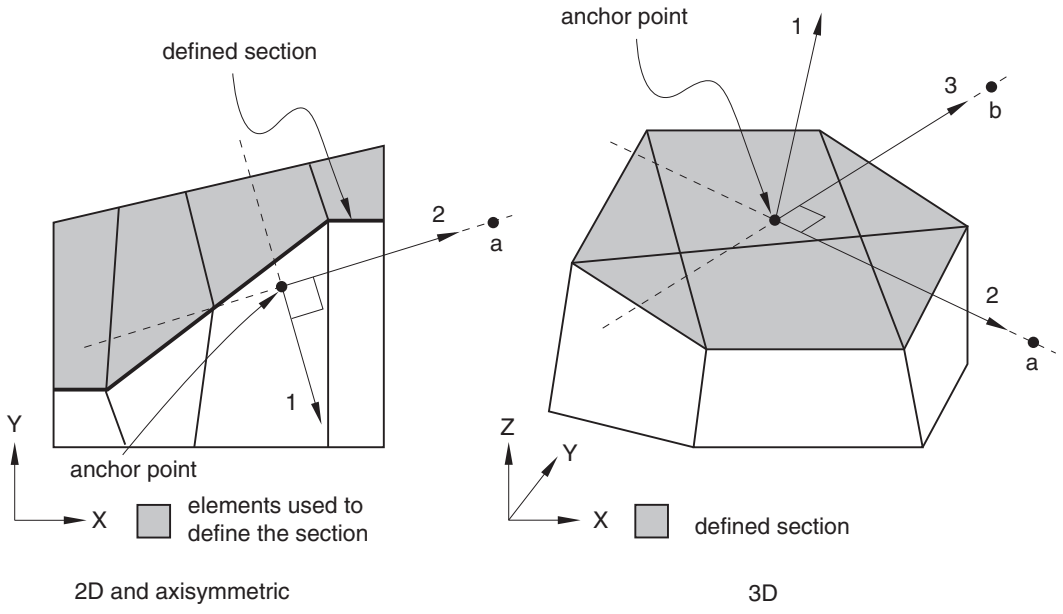



**Figure 18.2–1** User-defined local coordinate system.

## 18.3 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Meshed beam cross-sections," Section 10.6.1 of the Abaqus Analysis User's Guide
- *BEAM SECTION GENERATE

**Optional parameter:**

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.

## 18.4 *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," Section 29.3.7 of the Abaqus Analysis User's Guide
- "Meshed beam cross-sections," Section 10.6.1 of the Abaqus Analysis User's Guide
- *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, $\alpha$, 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 $(\frac{\partial \Psi}{\partial x_1} - (x_2 - x_{2s}))$ with respect to the local 1-direction and the shear center.
8. Derivative of the warping function $(\frac{\partial \Psi}{\partial x_2} + (x_1 - x_{1s}))$ 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.

## 18.5 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Output to the data and results files," Section 4.1.2 of the Abaqus Analysis User's Guide
- "Abaqus/Standard output variable identifiers," Section 4.2.1 of the Abaqus Analysis User's Guide

**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 18.5–1 (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.

4. Third coordinate of point $a$ (ignored if node number given).
5. Node number used to specify point $b$ (blank if coordinates given)
6. First coordinate of point $b$ (ignored if node number given).
7. Second coordinate of point $b$ (ignored if node number given).
8. 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," Section 4.2.1 of the Abaqus Analysis User's Guide.

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," Section 4.1.2 of the Abaqus Analysis User's Guide) will be output.

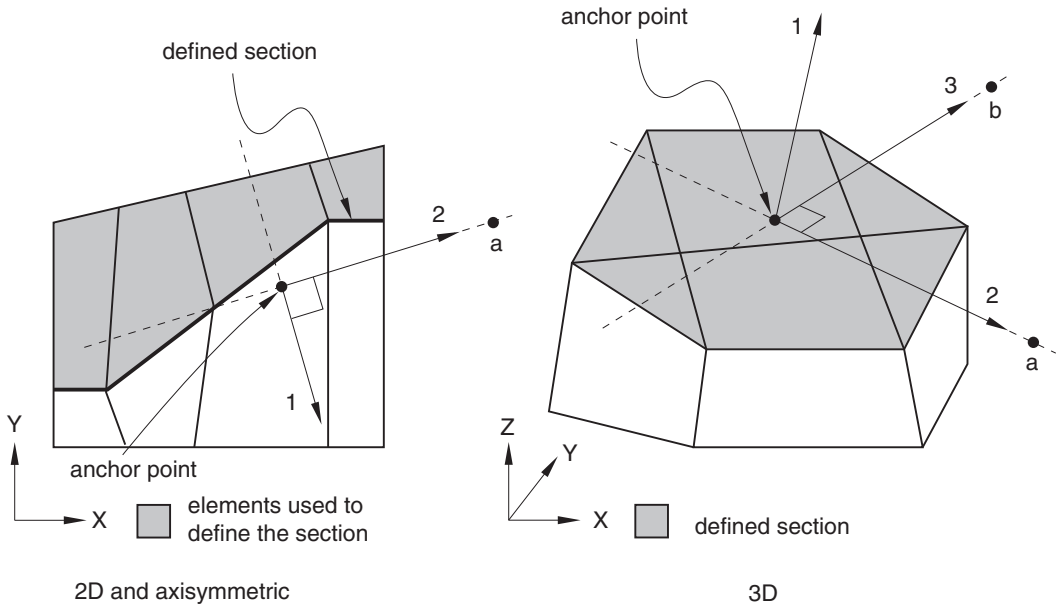



**Figure 18.5–1** User-defined local coordinate system.

## 18.6 *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," Section 10.4.3 of the Abaqus Analysis User's Guide
- *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.**

## 18.7 *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," Section 6.3.2 of the Abaqus Analysis User's Guide
- "Complex eigenvalue extraction," Section 6.3.6 of the Abaqus Analysis User's Guide
- "Transient modal dynamic analysis," Section 6.3.7 of the Abaqus Analysis User's Guide
- "Mode-based steady-state dynamic analysis," Section 6.3.8 of the Abaqus Analysis User's Guide
- "Subspace-based steady-state dynamic analysis," Section 6.3.9 of the Abaqus Analysis User's Guide
- "Defining substructures," Section 10.1.2 of the Abaqus Analysis User's Guide
- "Response spectrum analysis," Section 6.3.10 of the Abaqus Analysis User's Guide
- "Random response analysis," Section 6.3.11 of the Abaqus Analysis User's Guide
- *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.

## 18.8 *SFILM: Define film coefficients and associated sink temperatures over a surface for heat transfer analysis.

This option is used to provide film coefficients and sink temperatures over a surface for fully coupled thermal-stress analysis. In Abaqus/Standard it is also used in heat transfer, coupled thermal-electrical, and coupled thermal-electrical-structural analyses.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Interaction module

**References:**

- "Thermal loads," Section 34.4.4 of the Abaqus Analysis User's Guide
- "FILM," Section 1.1.6 of the Abaqus User Subroutines Reference Guide

**Optional parameters:**

AMPLITUDE

Set this parameter equal to the name of the *AMPLITUDE option that gives the variation of the sink temperature, $\theta^0$, 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," Section 6.1.2 of the Abaqus Analysis User's Guide). 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 (which are available only in Abaqus/Standard), the sink temperature amplitude is defined in user subroutine **FILM**, and AMPLITUDE references are used only to modify the sink temperature passed into the user subroutine.

FILM AMPLITUDE

This parameter applies only to Abaqus/Standard and Abaqus/Explicit analyses.

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 (which are available only in Abaqus/Standard), the film coefficient amplitude is defined in user subroutine **FILM**, 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 *SFILMs applied to the model should be removed.

**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, $\theta^0$. (Units of $\theta$.) For nonuniform film coefficients the sink temperature must be defined in user subroutine **FILM**. If given, this value will be modified by the 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**. 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 for different surfaces.

## 18.9 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Pore fluid flow," Section 34.4.7 of the Abaqus Analysis User's Guide
- "FLOW," Section 1.1.7 of the Abaqus User Subroutines Reference Guide

**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," Section 6.1.2 of the Abaqus Analysis User's Guide). 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 *SFLOWs 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, $u_w^\infty$. (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 $\mathrm{F^{-1}L^3T^{-1}}$.)

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, $u_w^\infty$, 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.

## 18.10 *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 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," Section 29.3.7 of the Abaqus Analysis User's Guide
- *BEAM GENERAL SECTION
- "Meshed beam cross-sections," Section 10.6.1 of the Abaqus Analysis User's Guide

**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.

## 18.11 *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.

**Product:** Abaqus/Explicit

**Type:** Model data

**Level:** Model

**References:**

- "Classical metal plasticity," Section 23.2.1 of the Abaqus Analysis User's Guide
- "Johnson-Cook plasticity," Section 23.2.7 of the Abaqus Analysis User's Guide
- "Dynamic failure models," Section 23.2.8 of the Abaqus Analysis User's Guide
- *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 "Using the DEPENDENCIES parameter to define field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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, $\bar{\varepsilon}_f^{pl}$.
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, $d_5$.

## 18.12 *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, $\varrho$, 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," Section 23.6.1 of the Abaqus Analysis User's Guide
- *CONCRETE
- *FAILURE RATIOS

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

**Data lines to define the shear retention behavior:**

First line:

1. $\varrho^{\text{close}}$. The default is 1.0.
2. $\varepsilon^{\max}$. 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.

## 18.13 *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," Section 22.7.1 of the Abaqus Analysis User's Guide
- *VISCOELASTIC

### Using shear test data to define a viscoelastic material

**Optional parameter:**

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 $1 - \sum_{i=1}^{N} \bar{g}_i^P = g_R(\infty)$.

**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.

## 18.14 *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:**

- "Shell elements: overview," Section 29.6.1 of the Abaqus Analysis User's Guide
- "Using a general shell section to define the section behavior," Section 29.6.6 of the Abaqus Analysis User's Guide
- "UGENS," Section 1.1.35 of the Abaqus User Subroutines Reference Guide

**Required parameter:**

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 $1 \times 10^{-6}$ 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," Section 27.1.4 of the Abaqus Analysis User's Guide) 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," Section 27.1.4 of the Abaqus Analysis User's Guide) 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 Chapter 23, "Composite layups," of the Abaqus/CAE User's Guide). 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 $1 \times 10^{-6}$ 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," Section 2.8.1 of the Abaqus Analysis User's Guide).

Positive values of the offset are in the positive normal direction (see "Shell elements: overview," Section 29.6.1 of the Abaqus Analysis User's Guide). 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. This parameter is ignored for continuum shells.

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," Section 2.2.5 of the Abaqus Analysis User's Guide) 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,” Section 2.8.1 of the Abaqus Analysis User’s Guide), 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,” Section 2.8.1 of the Abaqus Analysis User’s Guide).

THICKNESS MODULUS

This parameter is relevant only for continuum shells.

Set this parameter equal to an effective thickness modulus. The default effective thickness modulus is twice the initial in-plane shear modulus based on the material definition.

ZERO

If the section is defined by its general stiffness, set this parameter equal to $\theta^0$, 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 applies only to Abaqus/Standard analyses and cannot be used with continuum shell elements. See "Using a general shell section to define the section behavior," Section 29.6.6 of the Abaqus Analysis User's Guide, 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**.

**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," Section 2.8.1 of the Abaqus Analysis User's Guide) to define spatially varying thickness. If this parameter is used for a non-composite 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:**

I PROPERTIES

Set this parameter equal to the number of integer property values needed as data in user subroutine **UGENS**. 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**. The default is PROPERTIES=0.

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," Section 2.8.1 of the Abaqus Analysis User's Guide) 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, $\phi$, or the name of a distribution ("Distribution definition," Section 2.8.1 of the Abaqus Analysis User's Guide) 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, $\phi$ is measured relative to the default shell local directions (see "Conventions," Section 1.2.2 of the Abaqus Analysis User's Guide).

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. $\alpha$, scaling modulus for $[\mathbf{F}]$. The default is 0.0.
3. $\theta$, temperature for these values of $Y$ and $\alpha$.
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 $\alpha$ 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. $\alpha$, scaling modulus for $[\mathbf{F}]$. The default is 0.0.
3. $\theta$, temperature for these values of $Y$ and $\alpha$.
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 $\alpha$ 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**. 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 **UGENS**. Enter eight values per line for both real and integer values.

## 18.15 *SHELL SECTION: Specify a shell cross-section.

This option is used to specify a shell cross-section.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Part, Part instance

**Abaqus/CAE:** Property module

**References:**

- "Shell elements: overview," Section 29.6.1 of the Abaqus Analysis User's Guide
- "Using a shell section integrated during the analysis to define the section behavior," Section 29.6.5 of the Abaqus Analysis User's Guide

**Required parameter:**

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," Section 27.1.4 of the Abaqus Analysis User's Guide) 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,"

Section 27.1.4 of the Abaqus Analysis User's Guide) 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 (see Chapter 23, "Composite layups," of the Abaqus/CAE User's Guide). Abaqus/CAE uses this name to identify 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 in an Abaqus/Standard analysis the name of a distribution. See "Distribution definition," Section 2.8.1 of the Abaqus Analysis User's Guide.

Positive values of the offset are in the positive normal direction (see "Shell elements: overview," Section 29.6.1 of the Abaqus Analysis User's Guide). 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. This parameter is ignored for continuum shells.

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 ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Guide) 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," Section 2.8.1 of the Abaqus Analysis User's Guide) to define spatially varying thickness. If this parameter is used for a non-composite 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," Section 2.8.1 of the Abaqus Analysis User's Guide), 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," Section 2.8.1 of the Abaqus Analysis User's Guide).

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* 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 an effective thickness modulus. The default effective thickness modulus is twice the initial in-plane shear modulus based on the material definition.

**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. Positive scalar value defining layer thickness or the name of a distribution ("Distribution definition," Section 2.8.1 of the Abaqus Analysis User's Guide) 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. 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, $\phi$, or the name of a distribution ("Distribution definition," Section 2.8.1 of the Abaqus Analysis User's Guide) 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, $\phi$ is measured relative to the default shell local directions (see "Orientations," Section 2.2.5 of the Abaqus Analysis User's Guide).
5. Name of the ply. Required only for composite layups defined in Abaqus/CAE.

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.

## 18.16 *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," Section 2.3.2 of the Abaqus Analysis User's Guide
- "Node-based surface definition," Section 2.3.3 of the Abaqus Analysis User's Guide
- "Shell-to-solid coupling," Section 35.3.3 of the Abaqus Analysis User's Guide

**Required parameter:**

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.

## 18.17 *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:**

- "Acoustic, shock, and coupled acoustic-structural analysis," Section 6.10.1 of the Abaqus Analysis User's Guide
- "Acoustic and shock loads," Section 34.4.6 of the Abaqus Analysis User's Guide
- *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 parameter:**

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, $\epsilon$, 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.

## 18.18 *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," Section 22.5.2 of the Abaqus Analysis User's Guide
- *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, $\gamma$.
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.

## 18.19 *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," Section 40.3.1 of the Abaqus Analysis User's Guide
- "Slide line contact elements," Section 40.4.1 of the Abaqus Analysis User's Guide

**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.

## 18.20 *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," Section 10.1.1 of the Abaqus Analysis User's Guide
- *SUBSTRUCTURE LOAD CASE

**Optional parameters:**

AMPLITUDE

Set this parameter equal to the name given to an amplitude defined by the *AMPLITUDE option ("Amplitude curves," Section 34.1.2 of the Abaqus Analysis User's Guide). 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," Section 6.1.2 of the Abaqus Analysis User's Guide, for the particular procedure used in the step.

OP

Set OP=MOD (default) to modify or add to the currently active *SLOADs.

Set OP=NEW if all existing *SLOADs applied to the model should be removed and new ones possibly defined.

**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.

## 18.21 *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," Section 6.8.1 of the Abaqus Analysis User's Guide
- "Rate-dependent plasticity: creep and swelling," Section 23.2.4 of the Abaqus Analysis User's Guide

**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," Section 7.1.1 of the Abaqus Analysis User's Guide).

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," Section 23.2.4 of the Abaqus Analysis User's Guide).

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,” Section 7.1.1 of the Abaqus Analysis User’s Guide) 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.

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," Section 7.1.1 of the Abaqus Analysis User's Guide). 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 $2 \times 10^{-4}$ 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 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.

## 18.22 *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/CFD Abaqus/CAE

**Type:** Model data

**Level:** Part, Part instance

**Abaqus/CAE:** Property module

**References:**

- "Solid (continuum) elements," Section 28.1.1 of the Abaqus Analysis User's Guide
- "Infinite elements," Section 28.3.1 of the Abaqus Analysis User's Guide
- "Continuum particle elements," Section 33.2.1 of the Abaqus Analysis User's Guide
- "Truss elements," Section 29.2.1 of the Abaqus Analysis User's Guide

**Required parameters:**

COMPOSITE

This parameter applies only to Abaqus/Standard analyses.

This parameter can be used only with three-dimensional brick solid 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," Section 2.2.5 of the Abaqus Analysis User's Guide) 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," Section 27.1.4 of the Abaqus Analysis User's Guide) 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," Section 27.1.4 of the Abaqus Analysis User's Guide) 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 Chapter 23, "Composite layups," of the Abaqus/CAE User's Guide). 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. Set this parameter equal to 1, 2, or 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 shell are symmetric about a central core. This parameter cannot be used if spatially varying orientation angles are defined on any composite layer using distributions ("Distribution definition," Section 2.8.1 of the Abaqus Analysis User's Guide).

**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 Part VI, "Elements," of the Abaqus Analysis User's Guide 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. 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, $\phi$, or in Abaqus/Standard the name of a distribution ("Distribution definition," Section 2.8.1 of the Abaqus Analysis User's Guide) 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," Section 1.1.15 of the Abaqus User Subroutines Reference Guide), 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.

## 18.23 *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," Section 26.4.2 of the Abaqus Analysis User's Guide
- *DIFFUSIVITY

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

**Data lines to define solubility:**

First line:

1. Solubility. (Units of $PLF^{-1/2}$.)
2. Temperature, $\theta$.
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.

## 18.24 *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 or to specify a separated solution scheme for coupled temperature-displacement and coupled thermal-electrical procedures.

**Products:** Abaqus/Standard Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Step module

**References:**

- "Fully coupled thermal-stress analysis," Section 6.5.3 of the Abaqus Analysis User's Guide
- "Coupled thermal-electrical analysis," Section 6.7.3 of the Abaqus Analysis User's Guide
- "Convergence criteria for nonlinear problems," Section 7.2.3 of the Abaqus Analysis User's Guide

**Required parameter:**

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.

**Optional parameter:**

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.

**There are no data lines associated with this option.**

## 18.25 *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," Section 6.1.5 of the Abaqus Analysis User's Guide
- "Iterative linear equation solver," Section 6.1.6 of the Abaqus Analysis User's Guide

**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 $10^{-3}$ for all general procedures. For linear perturbation procedures, the default is $10^{-6}$.
2. Maximum number of iterations allowed. The default is 300.
3. ILU factorization fill-in level. The default is 0; the maximum is 3. This parameter applies only to porous media (soils) analyses.

## 18.26 *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

**Reference:**

- "Sorption," Section 26.6.4 of the Abaqus Analysis User's Guide

**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 $\mathrm{FL}^{-2}$.)
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, $(du_w/ds)\big|_s$. This slope must be positive and larger than the slope of any segment of the absorption or exsorption behavior definitions.

## 18.27 *SPECIFIC HEAT: Define specific heat.

This option is used to specify a material's specific heat.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CFD Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**Reference:**

- "Specific heat," Section 26.2.3 of the Abaqus Analysis User's Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

TYPE

This parameter applies only to Abaqus/CFD analyses.

Set TYPE=CONSTANT VOLUME (default) to define the specific heat at constant volume.

Set TYPE=CONSTANT PRESSURE to define the specific heat at constant pressure when the energy equation is used for thermal-flow problems.

**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.

## 18.28 *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," Section 6.3.10 of the Abaqus Analysis User's Guide
- *RESPONSE SPECTRUM

### Defining a spectrum using values of $S$ as a function of frequency and damping

**Required parameter:**

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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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.

**Date 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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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.

## 18.29 *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," Section 2.5.5 of the Abaqus Theory Guide).

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," Section 32.1.1 of the Abaqus Analysis User's Guide
- "Flexible joint element," Section 32.3.1 of the Abaqus Analysis User's Guide
- "Tube support elements," Section 32.8.1 of the Abaqus Analysis User's Guide

**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," Section 21.1.2 of the Abaqus Analysis User's Guide, 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," Section 2.2.5 of the Abaqus Analysis User's Guide).

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," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 18.30 *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 heat transfer, coupled thermal-electrical, and coupled thermal-electrical-structural analyses.

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/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Interaction module

**References:**

- "Thermal loads," Section 34.4.4 of the Abaqus Analysis User's Guide
- "Cavity radiation," Section 41.1.1 of the Abaqus Analysis User's Guide

**Optional parameters:**

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," Section 6.1.2 of the Abaqus Analysis User's Guide). 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.

**Data lines to define surface radiation conditions:**

First line:

1. Surface name.
2. Radiation type label R.

3. Reference ambient temperature value, $\theta^0$. (Units of $\theta$.)
4. Emissivity, $\epsilon$.

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, $\epsilon$.

Repeat this data line as often as necessary to define radiation conditions for different surfaces.

## 18.31 *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," Section 6.2.2 of the Abaqus Analysis User's Guide
- "Unstable collapse and postbuckling analysis," Section 6.2.4 of the Abaqus Analysis User's Guide
- "Adiabatic analysis," Section 6.5.4 of the Abaqus Analysis User's Guide
- "Solving nonlinear problems," Section 7.1.1 of the Abaqus Analysis User's Guide
- "Deformation plasticity," Section 23.2.13 of the Abaqus Analysis User's Guide

**No parameters or data lines are used in a linear perturbation 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," Section 7.1.1 of the Abaqus Analysis User's Guide).

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," Section 7.1.1 of the Abaqus Analysis User's Guide) 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.

RIKS

Include this parameter to use the modified Riks method for proportional loading cases ("Unstable collapse and postbuckling analysis," Section 6.2.4 of the Abaqus Analysis User's Guide).

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," Section 7.1.1 of the Abaqus Analysis User's Guide). 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 $2 \times 10^{-4}$ 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 $1.0$ 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, $\Delta l_{in}$. 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 $1.0$ is assumed.

3. Minimum arc length increment, $\Delta l_{min}$. 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, $\Delta l_{max}$. If this value is not specified, no upper limit is imposed.
5. Maximum value of the load proportionality factor, $\lambda_{end}$. 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.

## 18.32 *STEADY STATE CRITERIA: Specify steady-state criteria for terminating a quasi-static uni-directional simulation.

This option is used to specify the norms that must be satisfied to halt a quasi-static uni-directional simulation based on achieving a steady-state condition. It must be used in conjunction with the *STEADY STATE DETECTION option.

**Product:** Abaqus/Explicit

**Type:** History data

**Level:** Step

**References:**

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Guide
- "Steady-state detection," Section 11.8.1 of the Abaqus Analysis User's Guide
- *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.

## 18.33 *STEADY STATE DETECTION: Specify steady-state requirements for terminating a quasi-static uni-directional 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.

**Product:** Abaqus/Explicit

**Type:** History data

**Level:** Step

**References:**

- "Steady-state detection," Section 11.8.1 of the Abaqus Analysis User's Guide
- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Guide
- *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," Section 11.8.1 of the Abaqus Analysis User's Guide, 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.

## 18.34 *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," Section 6.3.4 of the Abaqus Analysis User's Guide
- "Mode-based steady-state dynamic analysis," Section 6.3.8 of the Abaqus Analysis User's Guide
- "Subspace-based steady-state dynamic analysis," Section 6.3.9 of the Abaqus Analysis User's Guide

**Optional and mutually exclusive parameters (used only if the dynamic response is not based on modal superposition):**

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).

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 than 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.

## 18.35 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Steady-state transport analysis," Section 6.4.1 of the Abaqus Analysis User's Guide
- "Symmetric model generation," Section 10.4.1 of the Abaqus Analysis User's Guide

**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," Section 7.1.1 of the Abaqus Analysis User's Guide).

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," Section 7.1.1 of the Abaqus Analysis User's Guide) 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.

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," Section 7.1.1 of the Abaqus Analysis User's Guide). 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 $2 \times 10^{-4}$ 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.

## 18.36 *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/CFD Abaqus/CAE

**Type:** History data

**Level:** Model

**Abaqus/CAE:** Step module

**References:**

- "Defining an analysis," Section 6.1.2 of the Abaqus Analysis User's Guide
- "Convergence criteria for nonlinear problems," Section 7.2.3 of the Abaqus Analysis User's Guide
- "Design sensitivity analysis," Section 19.1.1 of the Abaqus Analysis User's Guide
- *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," Section 6.1.2 of the Abaqus Analysis User's Guide. The default amplitude variation can be overwritten for individual loadings by using the AMPLITUDE parameter on the loading options ("Amplitude curves," Section 34.1.2 of the Abaqus Analysis User's Guide).

This parameter is rarely needed, and changing the defaults may 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 may 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 needed. 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 may 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 linear perturbation procedures," Section 6.1.3 of the Abaqus Analysis User's Guide, "Mesh-to-mesh solution mapping," Section 12.4.1 of the Abaqus Analysis User's Guide, and "Applying loads: overview," Section 34.4.1 of the Abaqus Analysis User's Guide, before using this option.*

SOLVER

Set SOLVER=ITERATIVE to use the iterative linear equation solver. *Please read the discussion in "Iterative linear equation solver," Section 6.1.6 of the Abaqus Analysis User's Guide, before using this option.*

If this parameter is omitted, the default direct sparse solver 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," Section 6.1.2 of the Abaqus Analysis User's Guide, 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.

## Beginning a step in an Abaqus/CFD analysis

**Optional parameter:**

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.

**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.

## 18.37 *SUBCYCLING: Define a subcycling zone.

This option is used to define a subcycling zone.

**Product:** Abaqus/Explicit

**Type:** Model data

**Level:** Model

**References:**

- "Explicit dynamic analysis," Section 6.3.3 of the Abaqus Analysis User's Guide
- "Fully coupled thermal-stress analysis," Section 6.5.3 of the Abaqus Analysis User's Guide
- "Selective subcycling," Section 11.7.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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.**

## 18.38 *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

**Reference:**

- "Submodeling: overview," Section 10.2.1 of the Abaqus Analysis User's Guide

**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.

## 18.39 *SUBSTRUCTURE COPY: Copy a substructure definition.

This option is used to copy a substructure definition from one library to another or from one substructure identifier to another within one library.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** This option is not supported in a model defined in terms of an assembly of part instances.

**Reference:**

- "Using substructures," Section 10.1.1 of the Abaqus Analysis User's Guide

**Required parameters:**

NEW TYPE

Set this parameter equal to the TYPE identifier of the substructure being created.

OLD TYPE

Set this parameter equal to the TYPE identifier of the substructure being copied.

**Optional parameters:**

NEW LIBRARY

Set this parameter equal to the name of the substructure library in which the substructure is stored. The default library name is *jobname*.

OLD LIBRARY

Set this parameter equal to the name of the substructure library from which the substructure is being copied. The default library name is *jobname*.

**There are no data lines associated with this option.**

## 18.40 *SUBSTRUCTURE DELETE: Remove a substructure from the substructure library.

This option is used to delete a substructure from a substructure library.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** This option is not supported in a model defined in terms of an assembly of part instances.

**Reference:**

- "Using substructures," Section 10.1.1 of the Abaqus Analysis User's Guide

**Required parameter:**

TYPE

Set this parameter equal to the TYPE identifier of the substructure being removed.

**Optional parameter:**

LIBRARY

Set this parameter equal to the name of the substructure library from which the substructure is being deleted. The default library name is *jobname*.

**There are no data lines associated with this option.**

## 18.41 *SUBSTRUCTURE DIRECTORY: List information about the substructures on a substructure library.

This option is used to provide a summary of information about the substructures stored on a substructure library.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** This option is not supported in a model defined in terms of an assembly of part instances.

**Reference:**

- "Using substructures," Section 10.1.1 of the Abaqus Analysis User's Guide

**Required parameter:**

LIBRARY

Set this parameter equal to the name of the substructure library.

**There are no data lines associated with this option.**

## 18.42 *SUBSTRUCTURE GENERATE: Substructure generation analysis.

This option is used to indicate that the step should be 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

**Reference:**

- "Defining substructures," Section 10.1.2 of the Abaqus Analysis User's Guide

**Required parameter:**

TYPE

Set this parameter equal to the identifier to be assigned to this substructure in a substructure library. The identifier must be *Z* followed by a number that cannot exceed 9999.

Substructure identifiers must be unique within a library. If a substructure already exists in the library with this same identifier, the analysis will terminate with an error message unless the OVERWRITE parameter is specified.

**Optional parameters:**

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.

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 name of the substructure library on which the substructure data will be written. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Guide, for the syntax of such library names. The default library name is *jobname*.

MASS MATRIX

Set MASS MATRIX=YES to calculate the substructure's reduced mass matrix. The default is MASS MATRIX=NO.

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 an existing substructure with the same TYPE identifier in the library. 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 will evaluate the stiffness associated with frequency-dependent springs and dashpots at zero frequency and will 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.

STRUCTURAL DAMPING MATRIX

Set STRUCTURAL DAMPING MATRIX=YES to calculate the substructure's reduced structural damping matrix. The default is STRUCTURAL DAMPING MATRIX=NO.

VISCOUS DAMPING MATRIX

Set VISCOUS DAMPING MATRIX=YES to calculate the substructure's reduced viscous damping matrix. The default is VISCOUS DAMPING MATRIX=NO.

**There are no data lines associated with this option.**

## 18.43 *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:**

- "Defining substructures," Section 10.1.2 of the Abaqus Analysis User's Guide
- *SLOAD

**Required parameter:**

NAME

Set this parameter equal to a label that will be used to refer to the load case in *SLOAD option specifications when applying loads to the substructure during an analysis.

**To define the loads:**

Enter any mechanical loading options ("Concentrated loads," Section 34.4.2 of the Abaqus Analysis User's Guide, and "Distributed loads," Section 34.4.3 of the Abaqus Analysis User's Guide) or thermal loading options ("Thermal loads," Section 34.4.4 of the Abaqus Analysis User's Guide) to define the loads forming the load case. Specify a magnitude for each load. This magnitude will be 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.

## 18.44 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** This option is not supported in a model defined in terms of an assembly of part instances.

**Reference:**

- "Defining substructures," Section 10.1.2 of the Abaqus Analysis User's Guide

**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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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 output format," Section 5.1.2 of the Abaqus Analysis User's Guide.

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," Section 32.15.1 of the Abaqus Analysis User's Guide). 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.**

## 18.45 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** This option is not supported in a model defined in terms of an assembly of part instances.

**Reference:**

- "Using substructures," Section 10.1.1 of the Abaqus Analysis User's Guide

**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.**

## 18.46 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** This option is not supported in a model defined in terms of an assembly of part instances.

**Reference:**

- "Using substructures," Section 10.1.1 of the Abaqus Analysis User's Guide

**Required parameter:**

ELSET

Set this parameter equal to the name of the element set containing the substructures for which properties are being defined.

**Optional parameter:**

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 18.46–1).
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 18.46–2).
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 18.46–1).
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 18.46–2).
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.

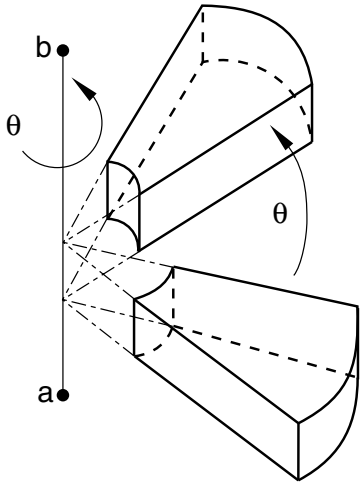



**Figure 18.46–1** Substructure rotation.

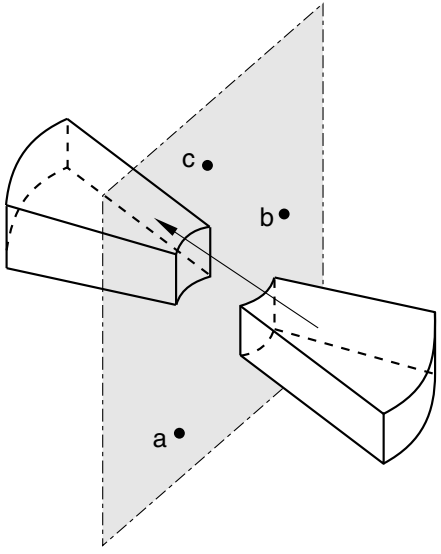



**Figure 18.46–2** Substructure reflection. Points *a*, *b*, and *c* cannot be colinear.

## 18.47 *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/CFD 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:**

- "Surfaces: overview," Section 2.3.1 of the Abaqus Analysis User's Guide
- "Element-based surface definition," Section 2.3.2 of the Abaqus Analysis User's Guide
- "Node-based surface definition," Section 2.3.3 of the Abaqus Analysis User's Guide
- "Analytical rigid surface definition," Section 2.3.4 of the Abaqus Analysis User's Guide
- "Eulerian surface definition," Section 2.3.5 of the Abaqus Analysis User's Guide
- "Operating on surfaces," Section 2.3.6 of the Abaqus Analysis User's Guide
- "Eulerian analysis," Section 14.1.1 of the Abaqus Analysis User's Guide
- "Contact interaction analysis: overview," Section 36.1.1 of the Abaqus Analysis User's Guide
- "RSURFU," Section 1.1.16 of the Abaqus User Subroutines Reference Guide
- "Modeling discontinuities as an enriched feature using the extended finite element method," Section 10.7.1 of the Abaqus Analysis User's Guide

**Required parameter:**

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," Section 41.1.1 of the Abaqus Analysis User's Guide.

**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," Section 12.2.2 of the Abaqus Analysis User's Guide.

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 master 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. The generated surface is supported only for the application of distributed pressure loads. This option applies only to Abaqus/Standard.

**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," Section 2.3.2 of the Abaqus Analysis User's Guide, 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 18.47–1).
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 18.47–2).
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.

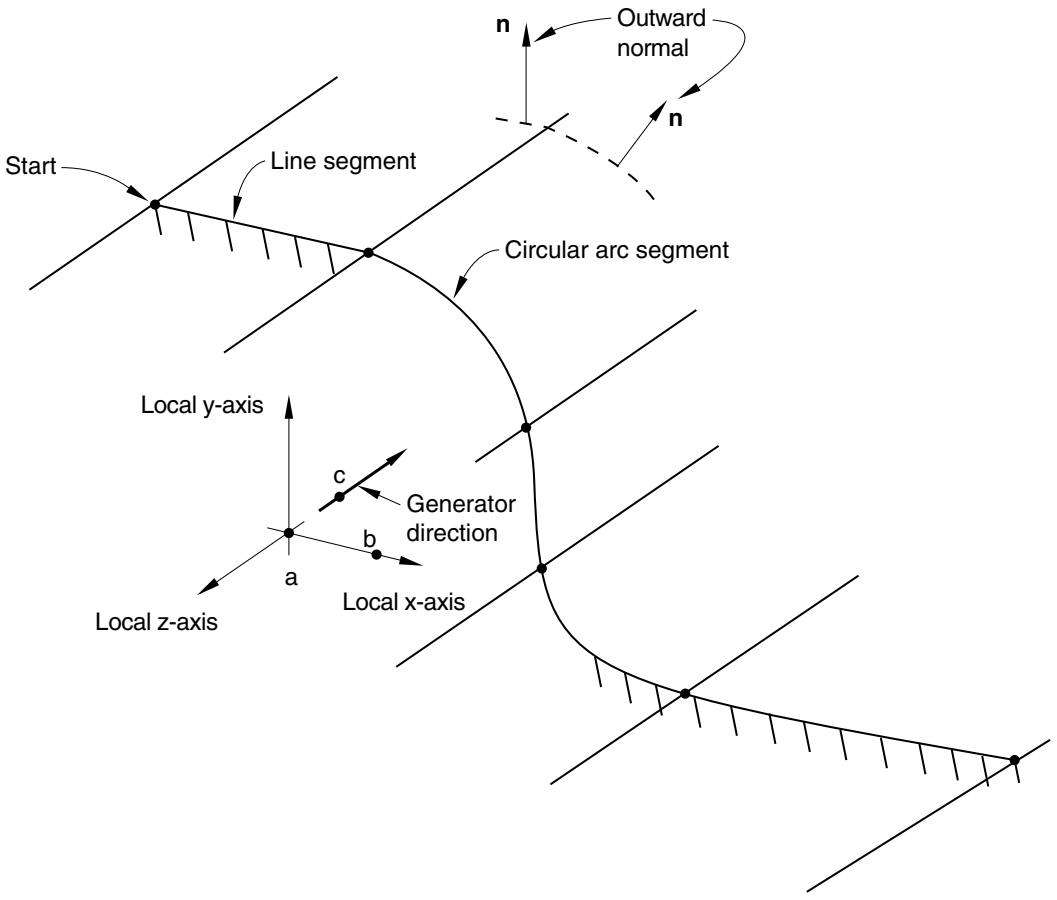



**Figure 18.47–1** *SURFACE, TYPE=CYLINDER.

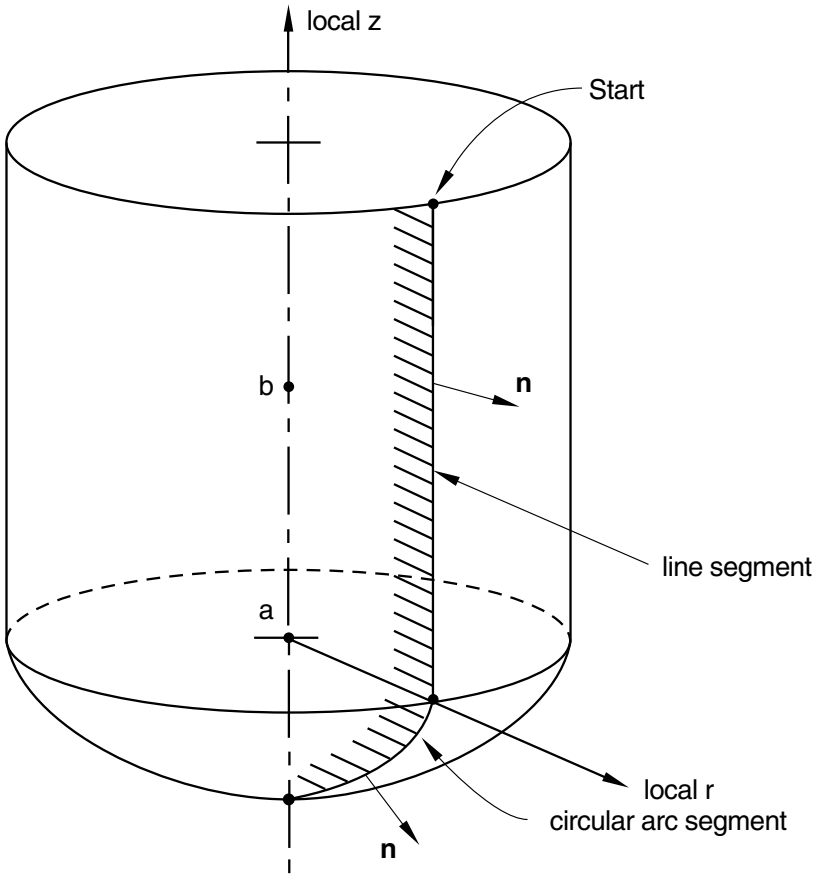



**Figure 18.47–2** *SURFACE, TYPE=REVOLUTION.

## 18.48 *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:**

- "Mechanical contact properties: overview," Section 37.1.1 of the Abaqus Analysis User's Guide
- "Contact pressure-overclosure relationships," Section 37.1.2 of the Abaqus Analysis User's Guide
- *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," Section 38.1.2 of the Abaqus Analysis User's Guide, 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," Section 38.1.2 of the Abaqus Analysis User's Guide, 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," Section 38.1.2 of the Abaqus Analysis User's Guide, 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. Note that 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.

If a contact area is not defined, such as may 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.

**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 18.48–1). 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 18.48–1), 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 18.48–1). 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 18.48–2).
2. Pressure at zero clearance, $p_0$.

The following data item is available only in Abaqus/Explicit analyses:

3. Value of the maximum stiffness, $k_{max}$. When using penalty contact, large stiffness values obtained from the exponential law may significantly lower the stable time increment size. Choose a suitable maximum stiffness to mitigate this effect (see Figure 18.48–2). 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 18.48–3), 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 18.48–3), 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 18.48–3), which defines the geometric scaling of the "base" stiffness and must be greater than one.
4. Initial stiffness scale factor, $s_0$ (see Figure 18.48–3), 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 18.48–4).

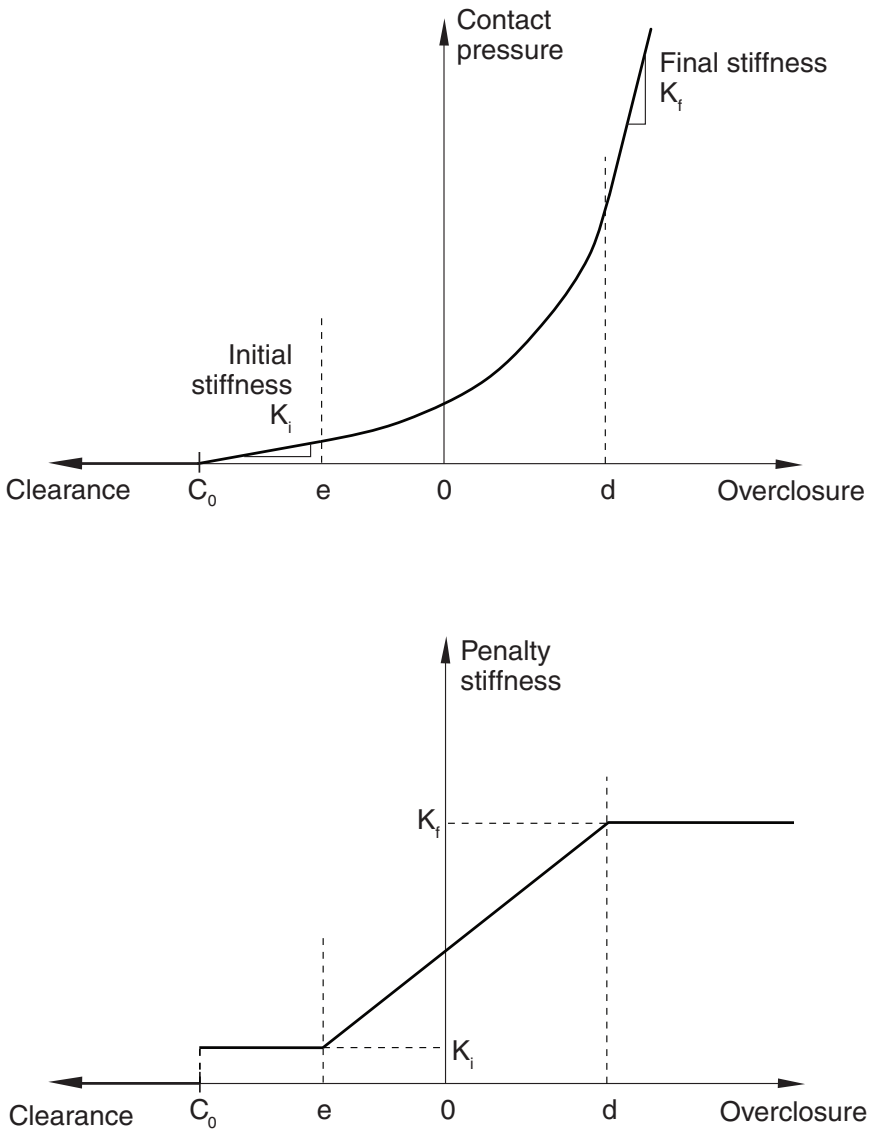



**Figure 18.48–1** Nonlinear penalty pressure-overclosure relationship.

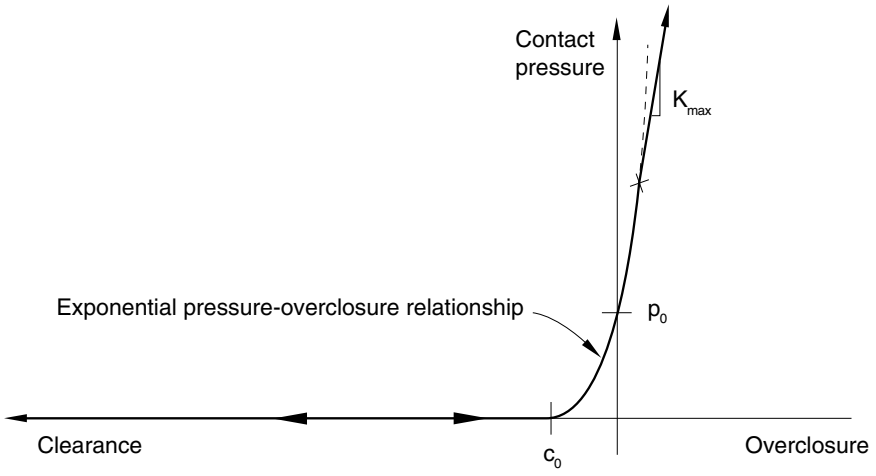



**Figure 18.48–2** Exponential pressure-overclosure relationship.

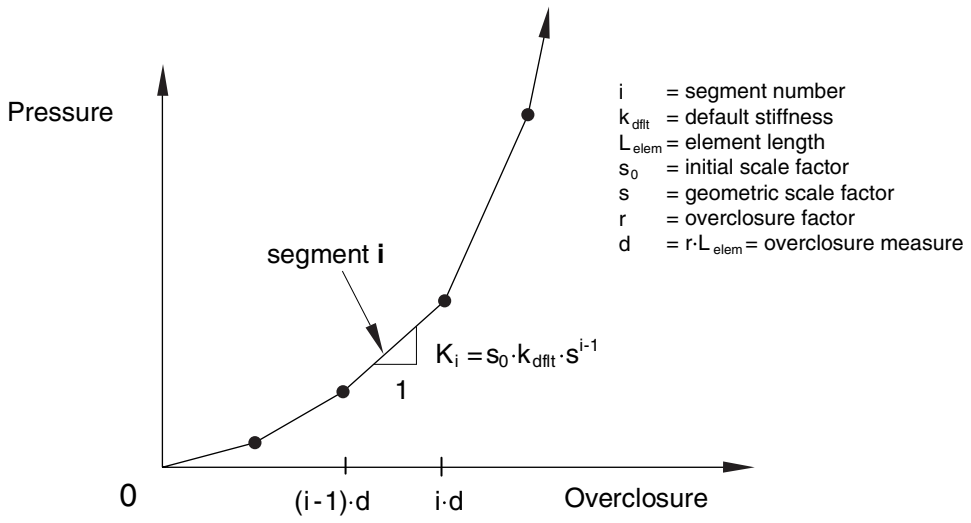



**Figure 18.48–3** Scale factor pressure-overclosure relationship.

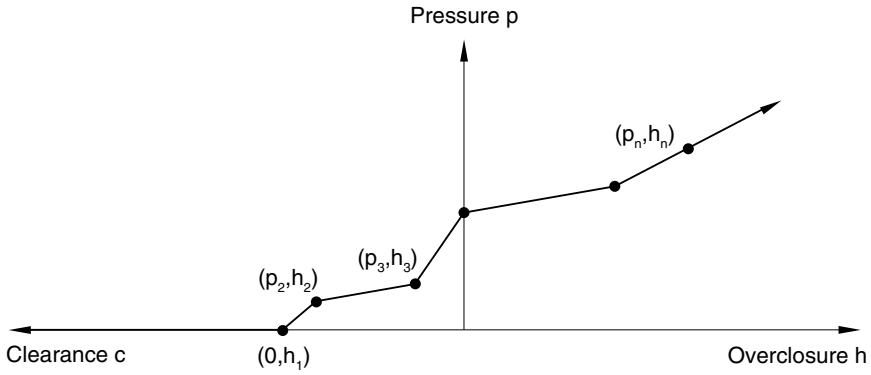



**Figure 18.48–4** Pressure-overclosure relationship defined in tabular form.

## 18.49 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** Part, Part instance

**Reference:**

- "Line spring elements for modeling part-through cracks in shells," Section 32.9.1 of the Abaqus Analysis User's Guide

**Required parameter:**

SIDE

Set SIDE=POSITIVE or SIDE=NEGATIVE to indicate which surface is cracked.

**Optional parameter:**

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Guide, 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.

## 18.50 *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," Section 36.5.2 of the Abaqus Analysis User's Guide
- "Mechanical contact properties: overview," Section 37.1.1 of the Abaqus Analysis User's Guide
- "Contact pressure-overclosure relationships," Section 37.1.2 of the Abaqus Analysis User's Guide
- "Contact damping," Section 37.1.3 of the Abaqus Analysis User's Guide
- "Frictional behavior," Section 37.1.5 of the Abaqus Analysis User's Guide
- "User-defined interfacial constitutive behavior," Section 37.1.6 of the Abaqus Analysis User's Guide
- "Breakable bonds," Section 37.1.9 of the Abaqus Analysis User's Guide
- "Thermal contact properties," Section 37.2.1 of the Abaqus Analysis User's Guide
- "Electrical contact properties," Section 37.3.1 of the Abaqus Analysis User's Guide
- "Pore fluid contact properties," Section 37.4.1 of the Abaqus Analysis User's Guide
- "UINTER," Section 1.1.39 of the Abaqus User Subroutines Reference Guide
- "VUINTER," Section 1.2.18 of the Abaqus User Subroutines Reference Guide
- "VUINTERACTION," Section 1.2.19 of the Abaqus User Subroutines Reference Guide

**Required parameter:**

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.

TRACKING THICKNESS

Set this parameter equal to the thickness that determines the contacting surfaces to be tracked.

In Abaqus/Standard 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. In Abaqus/Standard this parameter extends the range over which contact opening distance (COPEN) output is provided for separated surfaces.

In Abaqus/Explicit this parameter applies only if user subroutine **VUINTER** or **VUINTERACTION** is also 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.

**Optional data line for two-dimensional models in Abaqus/Standard or for contact pairs involving node-based surfaces in Abaqus/Standard, if the USER parameter is omitted:**

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 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.

## 18.51 *SURFACE OUTPUT: Specify surface variables to be written to the output database.

This option is used to write surface variables to the output database. It must be used in conjunction with the *OUTPUT option.

**Products:** Abaqus/CFD

**Type:** History data

**Level:** Step

**References:**

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Guide
- *OUTPUT

**Required parameter:**

SURFACE

Set this parameter equal to the name of the surface for which this output request is being made.

**Data lines to request surface 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/CFD output variable identifiers," Section 4.2.3 of the Abaqus Analysis User's Guide. Depending on the model and the variables requested, not all variables may be available for a given surface.

Repeat this data line as often as necessary to define the list of variables to be written to the output database.

## 18.52 *SURFACE PROPERTY: Define surface properties for cavity radiation.

This option is used to define surface properties for cavity radiation analysis. It must immediately precede the *EMISSIVITY option.

**Products:** Abaqus/Standard Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Interaction module

**References:**

- "Cavity radiation," Section 41.1.1 of the Abaqus Analysis User's Guide
- *EMISSIVITY

**Required parameter:**

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. Only element-based surfaces can be used.

**There are no data lines associated with this option.**

## 18.53 *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," Section 36.2.2 of the Abaqus Analysis User's Guide
- "Assigning surface properties for general contact in Abaqus/Explicit," Section 36.4.2 of the Abaqus Analysis User's Guide
- *CONTACT

**Required parameter:**

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=FEATURE EDGE CRITERIA to control which primary feature edges and secondary feature edges should be activated in the general contact domain.

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=THICKNESS to assign the surface thickness.

**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=FEATURE EDGE CRITERIA in Abaqus/Standard:**

First line:

1. Surface name. If the surface name 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), the "words" PERIMETER EDGES, or the "words" 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), the "words" PERIMETER EDGES, or the "words" NO FEATURE EDGES. The entry on this field activates edges to participate in edge-to-edge contact. The default is NO FEATURE EDGES.

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. A non-blank surface name is required if the ALL EDGES or PICKED EDGES options are specified. If the surface name is omitted when using the PERIMETER EDGES, NO FEATURE EDGES, or cutoff feature angle options, a default surface that encompasses the entire general contact domain is assumed.
2. The criterion for primary feature edges. The "words" PERIMETER EDGES (default), the "words" ALL EDGES, the "words" PICKED EDGES, the "words" NO FEATURE EDGES, or a scalar value representing the cutoff feature angle (in degrees).
3. The criterion for secondary feature edges. The "words" ALL REMAINING EDGES, the "words" PERIMETER EDGES, the "words" PICKED EDGES, or a scalar value representing the cutoff feature angle (in degrees).

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 to define smoothing on regions of two-dimensional surfaces that correspond (or nearly correspond) to a circular arc:**

First line:

1. Surface name.
2. The "word" CIRCUMFERENTIAL.
3. Global $X$-coordinate of point $a$, the approximate center (origin) of the circular arc (see Figure 18.53–1).

4. Global *Y*-coordinate of point *a*, the approximate center (origin) of the circular arc.

Repeat this data line as often as necessary. If the geometry correction assignments overlap, the last assignment applies in the overlap region.

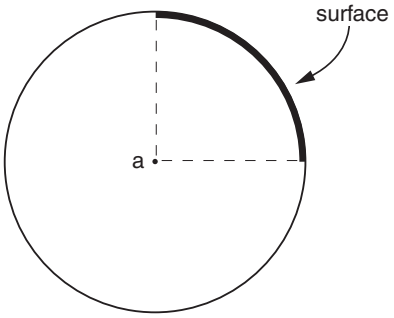



**Figure 18.53–1** Two-dimensional circumferential smoothing.

**Data lines for PROPERTY=GEOMETRIC CORRECTION to define smoothing on regions of surfaces that correspond (or nearly correspond) to a surface of revolution:**

First line:

1. Surface name.
2. The "word" CIRCUMFERENTIAL.
3. Global *X*-coordinate of point *a* on the approximate axis of revolution for the surface (see Figure 18.53–2).
4. Global *Y*-coordinate of point *a* on the approximate axis of revolution for the surface.
5. Global *Z*-coordinate of point *a* on the approximate axis of revolution for the surface.
6. Global *X*-coordinate of point *b* on the approximate axis of revolution for the surface (see Figure 18.53–2).
7. Global *Y*-coordinate of point *b* on the approximate axis of revolution for the surface.
8. Global *Z*-coordinate of point *b* on the approximate axis of revolution for the surface.

Repeat this data line as often as necessary. If the geometry correction assignments overlap, the last assignment applies in the overlap region.

**Data lines for PROPERTY=GEOMETRIC CORRECTION to define smoothing on regions of surfaces that correspond (or nearly correspond) to a spherical section:**

First line:

1. Surface name.
2. The "word" SPHERICAL.

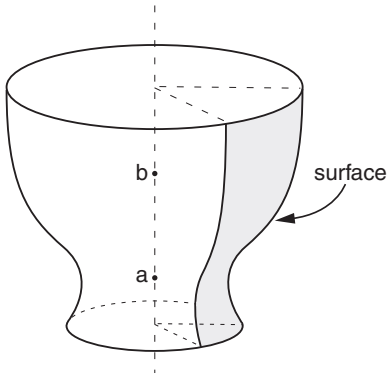



**Figure 18.53–2** Three-dimensional circumferential smoothing.

3. Global *X*-coordinate of point *a*, the approximate center (origin) of the sphere (see Figure 18.53–3).
4. Global *Y*-coordinate of point *a*, the approximate center (origin) of the sphere.
5. Global *Z*-coordinate of point *a*, the approximate center (origin) of the sphere.

Repeat this data line as often as necessary. If the geometry correction assignments overlap, the last assignment applies in the overlap region.

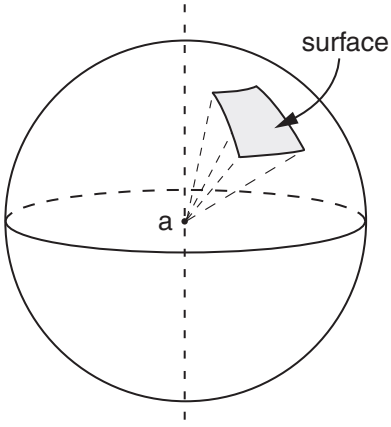



**Figure 18.53–3** Spherical smoothing.

**Data lines for PROPERTY=GEOMETRIC CORRECTION to define smoothing on regions of surfaces that correspond (or nearly correspond) to a circular arc revolved about an axis:**

First line:

1. Surface name.
2. The "word" TOROIDAL.
3. Global *X*-coordinate of point *a* on the approximate axis of revolution for the surface (see Figure 18.53–4).
4. Global *Y*-coordinate of point *a* on the approximate axis of revolution for the surface.
5. Global *Z*-coordinate of point *a* on the approximate axis of revolution for the surface.
6. Global *X*-coordinate of point *b* on the approximate axis of revolution for the surface (see Figure 18.53–4).
7. Global *Y*-coordinate of point *b* on the approximate axis of revolution for the surface.
8. Global *Z*-coordinate of point *b* on the approximate axis of revolution for the surface.
9. Distance *R* of the center of the circular arc from the axis of revolution (see Figure 18.53–4).

Repeat this data line as often as necessary. If the geometry correction assignments overlap, the last assignment applies in the overlap region. The line joining point *a* and the center of the circular arc should be perpendicular to the axis of revolution.

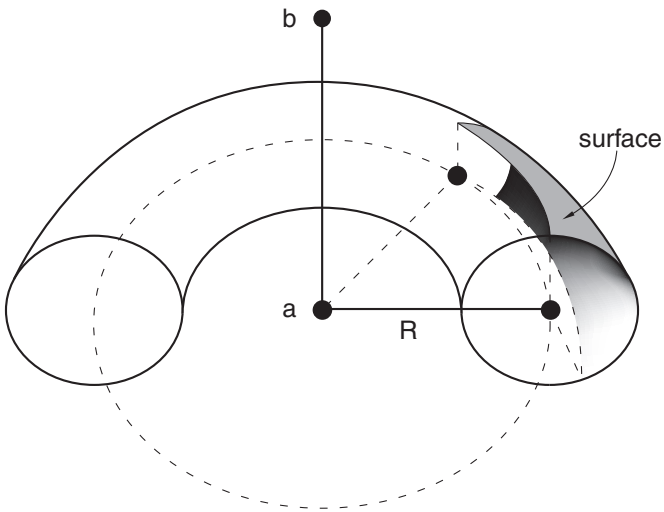



**Figure 18.53–4** Three-dimensional toroidal smoothing.

**Data lines for PROPERTY=GEOMETRIC CORRECTION to define surface regions that should not be smoothed (corresponds to default):**

First line:

1. Surface name.
2. Leave blank or specify the "word" NONE.

Repeat this data line as often as necessary. If the geometry correction assignments overlap, the last assignment applies in the overlap region.

**Data lines for PROPERTY=OFFSET FRACTION:**

First line:

1. Surface name. If the surface name 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. The "word" ORIGINAL (default), the "word" SPOS, the "word" 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.

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=THICKNESS:**

First line:

1. Surface name. If the surface name is omitted, a default surface that encompasses the entire general contact domain is assumed.
2. The "word" ORIGINAL (default), the "word" THINNING (available only in Abaqus/Explicit), or a scalar value representing a nominal thickness.
3. A constant scaling factor (default is 1.0).

Repeat this data line as often as necessary. If the thickness assignments overlap, the last assignment applies in the overlap region.

## 18.54 *SURFACE SECTION: Specify section properties for surface elements.

This option is used to specify a surface element cross-section. It must be used in conjunction with the *REBAR LAYER option. 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," Section 32.7.1 of the Abaqus Analysis User's Guide
- "Defining reinforcement," Section 2.2.3 of the Abaqus Analysis User's Guide
- *REBAR LAYER

**Required parameter:**

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.

**There are no data lines associated with this option.**

## 18.55 *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

**Reference:**

- "Smoothing contact surfaces in Abaqus/Standard," Section 38.1.3 of the Abaqus Analysis User's Guide

**Required parameter:**

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 18.55–1):

1. Name of slave surface or surface corresponding to a subregion of the slave surface. The name can be omitted when smoothing is applied only to a subregion of the master surface.
2. Name of master surface or surface corresponding to a subregion of the master surface. The name can be omitted when smoothing is applied only to a subregion of the slave 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.

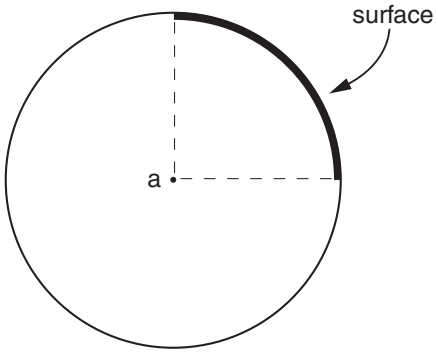



**Figure 18.55–1** Two-dimensional circumferential smoothing.

Data line to define smoothing on regions of surfaces that correspond (or nearly correspond) to a surface of revolution (see Figure 18.55–2):

1. Name of slave surface or surface corresponding to a subregion of the slave surface. The name can be omitted when smoothing is applied only to a subregion of the master surface.
2. Name of master surface or surface corresponding to a subregion of the master surface. The name can be omitted when smoothing is applied only to a subregion of the slave surface.
3. The "word" CIRCUMFERENTIAL.
4. Global *X*-coordinate of point *a*.
5. Global *Y*-coordinate of point *a*.
6. Global *Z*-coordinate of point *a*.
7. Global *X*-coordinate of point *b* on the approximate axis of revolution for the surface (see Figure 18.55–2).
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.

Data line to define smoothing on regions of surfaces that correspond (or nearly correspond) to a sphere section (see Figure 18.55–3):

1. Name of slave surface or surface corresponding to a subregion of the slave surface. The name can be omitted when smoothing is applied only to a subregion of the master surface.
2. Name of master surface or surface corresponding to a subregion of the master surface. The name can be omitted when smoothing is applied only to a subregion of the slave 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.

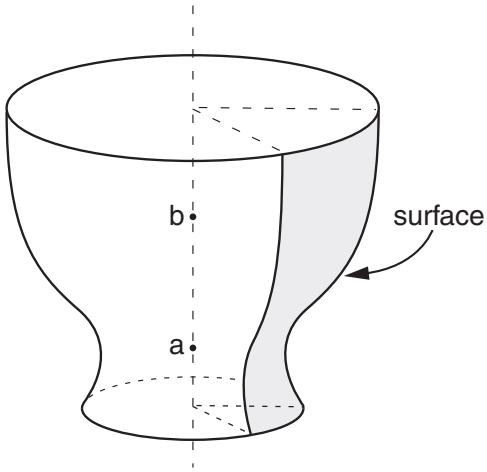



**Figure 18.55–2** Three-dimensional circumferential smoothing.

6. Global *Z*-coordinate of point *a*, the approximate center (origin) of the sphere.

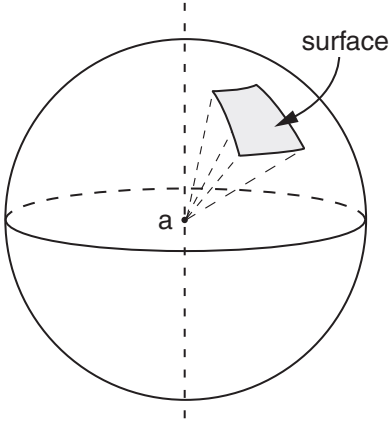



**Figure 18.55–3** Spherical smoothing.

Data line to define smoothing on regions of surfaces that correspond (or nearly correspond) to a toroidal surface (see Figure 18.55–4):

1. Name of slave surface or surface corresponding to a subregion of the slave surface. The name can be omitted when smoothing is applied only to a subregion of the master surface.

2. Name of master surface or surface corresponding to a subregion of the master surface. The name can be omitted when smoothing is applied only to a subregion of the slave surface.
3. The "word" TOROIDAL.
4. Global *X*-coordinate of point *a* on the approximate axis of revolution for the surface (see Figure 18.55–4).
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 18.55–4).
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 18.55–4).

Repeat the above data lines as often as necessary to define all surface regions that require smoothing.

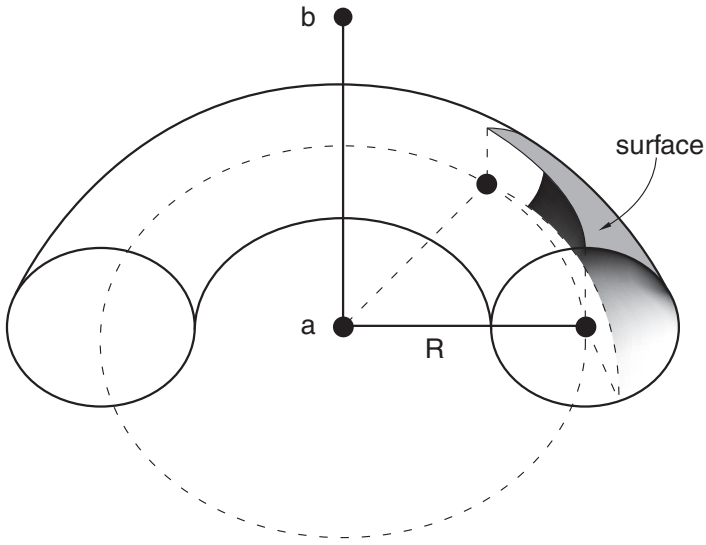



**Figure 18.55–4** Three-dimensional toroidal smoothing.

## 18.56 *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," Section 23.2.4 of the Abaqus Analysis User's Guide
- "CREEP," Section 1.1.1 of the Abaqus User Subroutines Reference Guide

**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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 18.57 *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.

**Product:** Abaqus/Standard

**Type:** Model data

**Level:** This option is not supported in a model defined in terms of an assembly of part instances.

**Reference:**

- "Symmetric model generation," Section 10.4.1 of the Abaqus Analysis User's Guide

**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 18.57–1).

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 18.57–3).

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 18.57–4).

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 18.57–2.

**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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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, $\theta$ (in degrees), of the original three-dimensional sector. $0.0^\circ < \theta \leq 360.0^\circ$.

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, $\theta$, 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, $\theta$ (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, $\theta$, 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, $\theta$, 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.

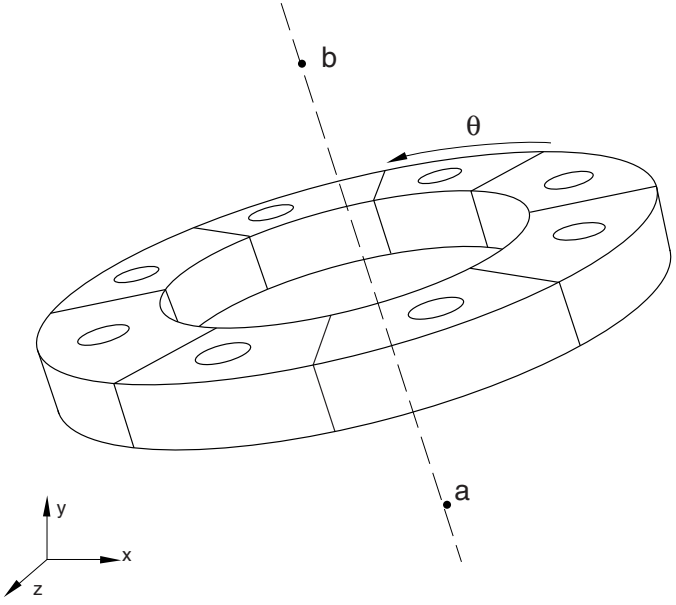



**Figure 18.57–1** Revolving a single three-dimensional repetitive sector to create a periodic structure.

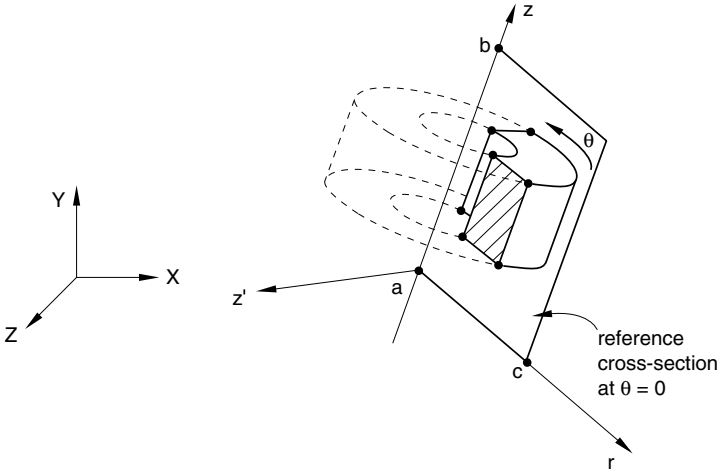



**Figure 18.57–2** Revolving an axisymmetric cross-section.

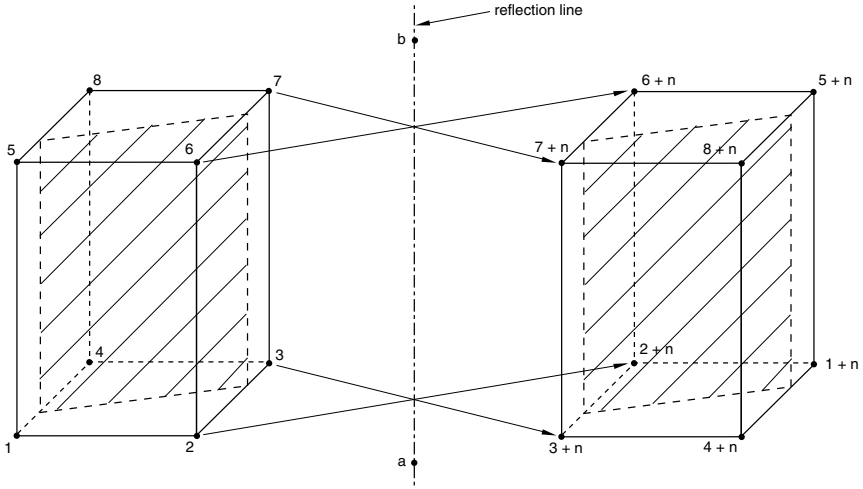



**Figure 18.57–3** Reflecting a three-dimensional model through line *ab* with node offset *n*.

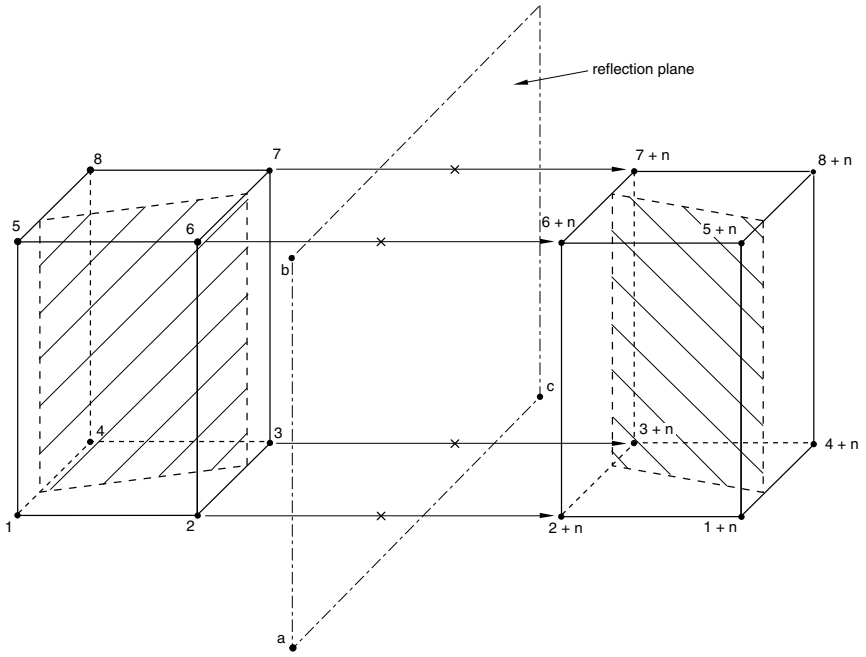



**Figure 18.57–4** Reflecting a three-dimensional model through a plane *abc* with node offset *n*.

## 18.58 *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.

**Product:** 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," Section 10.4.2 of the Abaqus Analysis User's Guide
- "Symmetric model generation," Section 10.4.1 of the Abaqus Analysis User's Guide
- *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.

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.**

## 18.59 *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/CFD 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.

**Reference:**

- "Node definition," Section 2.1.1 of the Abaqus Analysis User's Guide

**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 18.59–1).
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:

4. $X_b$, global ***X***-coordinate of a point on the $X^1$-axis of the local coordinate system (point ***b*** in Figure 18.59–1).
5. $Y_b$, global ***Y***-coordinate of a point on the $X^1$-axis of the local coordinate system.
6. $Z_b$, 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 18.59–1).
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.

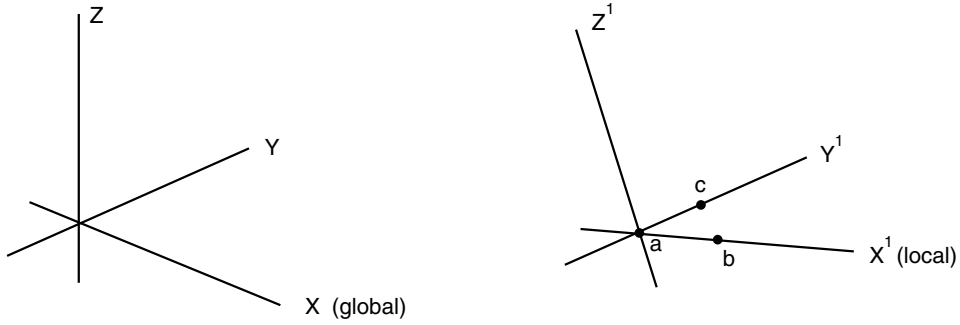



**Figure 18.59–1** Local coordinate system.

# 19. T

## 19.1 *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," Section 34.6.1 of the Abaqus Analysis User's Guide
- "UTEMP," Section 1.1.51 of the Abaqus User Subroutines Reference Guide

**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," Section 34.1.2 of the Abaqus Analysis User's Guide).

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," Section 6.1.2 of the Abaqus Analysis User's Guide). 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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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.

**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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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`:**

USER

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to indicate that user subroutine **`UTEMP`** will be used to define temperature values. **`UTEMP`** 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 or an output database file has been specified in addition to user subroutine **UTEMP**, values read from this file will be passed into **UTEMP** for possible modification.

**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.

**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 a user subroutine is not used (FILE parameter included, USER parameter omitted).**

**Data lines to define temperatures using user subroutine `UTEMP` using the data line format (FILE parameter omitted, 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.

**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.

## 19.2 *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.

**Product:** Abaqus/Explicit

**Type:** Model data

**Level:** Model

**References:**

- "Equation of state," Section 25.2.1 of the Abaqus Analysis User's Guide
- "Classical metal plasticity," Section 23.2.1 of the Abaqus Analysis User's Guide
- "Johnson-Cook plasticity," Section 23.2.7 of the Abaqus Analysis User's Guide
- "Dynamic failure models," Section 23.2.8 of the Abaqus Analysis User's Guide
- *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 "Using the DEPENDENCIES parameter to define field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 19.3 *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," Section 23.3.3 of the Abaqus Analysis User's Guide
- *MOHR COULOMB
- *MOHR COULOMB HARDENING

**Optional parameter:**

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, $\sigma_t$.
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.

## 19.4 *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," Section 23.6.1 of the Abaqus Analysis User's Guide
- *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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 19.5 *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:**

- "Using a general beam section to define the section behavior," Section 29.3.7 of the Abaqus Analysis User's Guide
- *BEAM GENERAL SECTION

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 19.6 *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," Section 35.3.1 of the Abaqus Analysis User's Guide
- "Element-based surface definition," Section 2.3.2 of the Abaqus Analysis User's Guide
- "Node-based surface definition," Section 2.3.3 of the Abaqus Analysis User's Guide
- "Analysis of models that exhibit cyclic symmetry," Section 10.4.3 of the Abaqus Analysis User's Guide
- "Acoustic, shock, and coupled acoustic-structural analysis," Section 6.10.1 of the Abaqus Analysis User's Guide

**Required parameter:**

NAME

Set this parameter equal to a label that will be used 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 slave surface are tied to the master surface. The calculation of the distance between the slave and master surface for a particular slave node depends on factors such as shell element thickness, the setting of the TYPE parameter, and the types of surfaces involved. Slave nodes that do not satisfy the position tolerance are not tied to the master 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 slave surface that will be tied to the master 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 slave surface onto the master surface in the initial configuration, without any strain.

Set ADJUST=NO if the slave nodes will not be moved. This is the default if the slave 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 master reference surface and the slave node at which the translational constraint should act. By default, Abaqus will attempt to choose this distance such that the translational constraint acts precisely at the interface.

CYCLIC SYMMETRY

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to invoke a constraint between the faces bounding a repetitive sector of a cyclic symmetric structure. This parameter can be used only in conjunction with the *CYCLIC SYMMETRY MODEL option.

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 will be tied if applicable, in addition to the translation degrees of freedom.

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 slave node projects onto the master surface.

**Data lines to define the surfaces forming the constraint pairs:**

First line:

1. The slave surface name.
2. The master 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 that will be tied together.

## 19.7 *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 parameter:**

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, $\Delta t$.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Guide, 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.

## 19.8 *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," Section 29.3.7 of the Abaqus Analysis User's Guide
- *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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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.

## 19.9 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**Reference:**

- "Slide line contact elements," Section 40.4.1 of the Abaqus Analysis User's Guide

**Optional parameter:**

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.**

## 19.10 *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.

**Product:** Abaqus/Explicit

**Type:** History data

**Level:** Step

**References:**

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Guide
- *ELEMENT OUTPUT
- *NODE OUTPUT

**Required parameter:**

TRACER SET

Set this parameter equal to the name of the tracer set to which these tracer particles will be assigned.

**Optional parameter:**

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.

## 19.11 *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/CFD Abaqus/CAE

**Type:** Model data

**Level:** Part, Part instance, Assembly

**Abaqus/CAE:** Define nodal coordinate systems for prescribed conditions in the Load module.

**Reference:**

- "Transformed coordinate systems," Section 2.1.5 of the Abaqus Analysis User's Guide

**Required parameter:**

NSET

Set this parameter equal to the name of the node set for which the local transformed system is being given.

**Optional parameter:**

TYPE

Set TYPE=R (default) to indicate a rectangular Cartesian system (Figure 19.11–1). Set TYPE=C to indicate a cylindrical system (Figure 19.11–2). Set TYPE=S to indicate a spherical system (Figure 19.11–3).

**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.

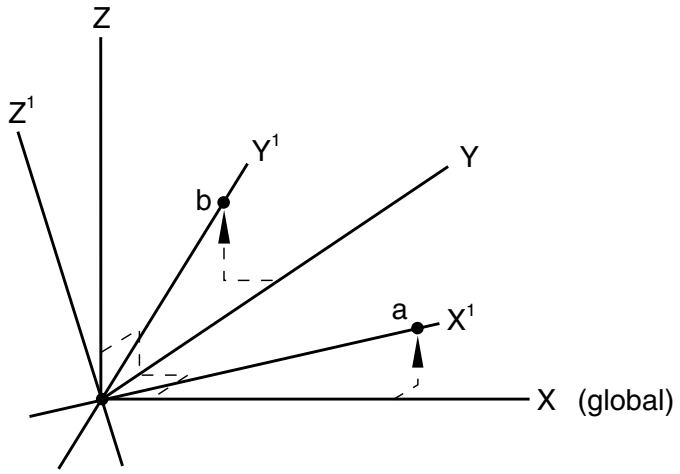



**Figure 19.11–1** Cartesian transformation option.

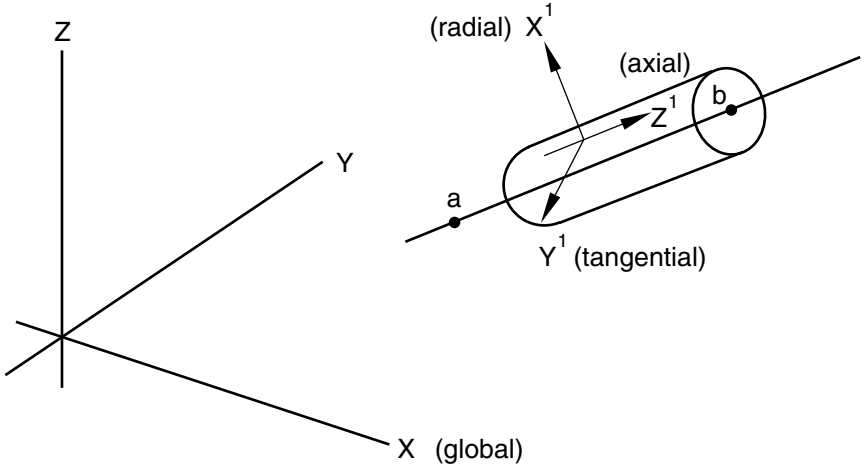



**Figure 19.11–2** Cylindrical transformation option.

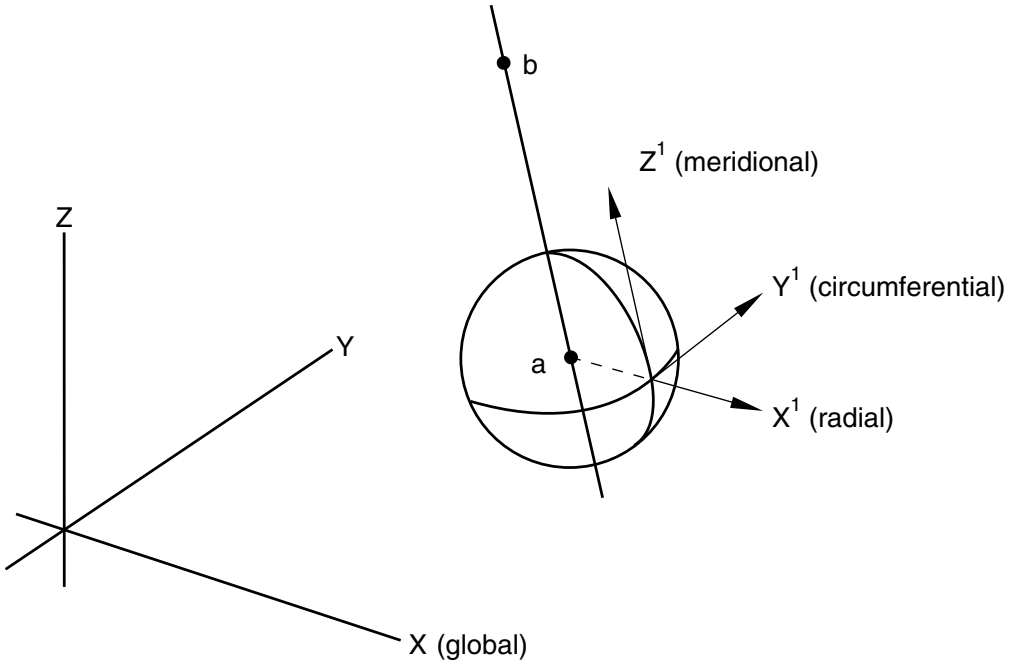



**Figure 19.11–3** Spherical transformation option.

## 19.12 *TRANSPORT EQUATION SOLVER: Specify the linear solver and parameters for solving the transport equations in an Abaqus/CFD analysis.

This option can be used only as a suboption of the *CFD option to enable the linear solver parameters for solving the transport equations of scalar variables such as temperature, species concentration, turbulent eddy viscosity, turbulent kinetic energy, etc. For a steady-state analysis this option is also used to enable the under-relaxation parameter for the transport of the scalar variables.

**Products:** Abaqus/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Step module

**References:**

- "Incompressible fluid dynamic analysis," Section 6.6.2 of the Abaqus Analysis User's Guide
- *CFD

**Optional parameters:**

CONVERGENCE

Set CONVERGENCE=ON to write convergence information to the log file.

Set CONVERGENCE=OFF (default) to suppress convergence information.

DIAGNOSTICS

Set DIAGNOSTICS=ON to write diagnostic information about the solver to the log file.

Set DIAGNOSTICS=OFF (default) to suppress diagnostic information.

TYPE

Set TYPE=DSFGMRES (default) to enable the Diagonally Scaled Flexible Generalized Minimum Residual linear solver.

Set TYPE=DSGMRES to enable the Diagonally Scaled Generalized Minimum Residual linear solver.

Set TYPE=ILUFGMRES to enable the Incomplete LU factorization preconditioned Flexible Generalized Minimum Residual linear solver.

**Data line for TYPE=DSFGMRES, TYPE=DSGMRES, and TYPE=ILUFGMRES:**

First (and only) line:

1. Maximum number of iterations (default=50).

2. Interval to check linear convergence (default=2).
3. Linear convergence criterion (default=$10^{-5}$).
4. Number of restart vectors (default=15).
5. Under-relaxation factor (default=0.7). This value is valid only for a steady-state analysis.

## 19.13 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Steady-state transport analysis," Section 6.4.1 of the Abaqus Analysis User's Guide
- "Symmetric model generation," Section 10.4.1 of the Abaqus Analysis User's Guide
- "UMOTION," Section 1.1.44 of the Abaqus User Subroutines Reference Guide

**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," Section 34.1.2 of the Abaqus Analysis User's Guide).

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," Section 6.1.2 of the Abaqus Analysis User's Guide).

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.

## 19.14 *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. 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," Section 29.6.4 of the Abaqus Analysis User's Guide
- "Choosing a beam element," Section 29.3.3 of the Abaqus Analysis User's Guide
- "Defining the constitutive response of cohesive elements using a continuum approach," Section 32.5.5 of the Abaqus Analysis User's Guide
- "Defining the constitutive response of cohesive elements using a traction-separation description," Section 32.5.6 of the Abaqus Analysis User's Guide
- *SHELL GENERAL SECTION
- *SHELL SECTION
- *BEAM GENERAL SECTION
- *BEAM SECTION
- *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, $K_{12}^{ts}$.

If either value $K_{11}^{ts}$ or $K_{22}^{ts}$ is omitted or given as zero, the nonzero value will be 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.

## 19.15 *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

**Reference:**

- "Extended Drucker-Prager models," Section 23.3.1 of the Abaqus Analysis User's Guide

**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, $\sigma_3$.

Repeat this data line as often as necessary to give the yield stress at different levels of confining stress.

## 19.16 *TRS: Used to define temperature-time shift for time history viscoelastic analysis.

This option can be used only in conjunction with the *VISCOELASTIC option and, in Abaqus/Explicit, with the *VISCOSITY option.

In an Abaqus/Standard analysis, either the nonlinear viscoelasticity must be defined by using the *VISCOELASTIC, NONLINEAR option or viscoelasticity must be defined in the time domain by using the *VISCOELASTIC, TIME option.

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "Time domain viscoelasticity," Section 22.7.1 of the Abaqus Analysis User's Guide
- "Parallel rheological framework," Section 22.8.2 of the Abaqus Analysis User's Guide
- "UTRS," Section 1.1.53 of the Abaqus User Subroutines Reference Guide
- "UTRSNETWORK," Section 1.1.54 of the Abaqus User Subroutines Reference Guide
- "VUTRS," Section 1.2.23 of the Abaqus User Subroutines Reference Guide
- *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," Section 22.7.1 of the Abaqus Analysis User's Guide) or in user subroutine **UTRSNETWORK** for the nonlinear viscoelastic model defined using the parallel rheological framework (see "Parallel rheological framework," Section 22.8.2 of the Abaqus Analysis User's Guide). In Abaqus/Explicit analyses define the shift function in user subroutine **VUTRS** for the linear viscoelastic model.

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, $\theta_0$.
2. Calibration constant, $C_1$.
3. Calibration constant, $C_2$.

**Data line to define the shift function by the Arrhenius approximation:**

First (and only) line:

1. Reference temperature, $\theta_0$.
2. Activation energy, $E_0$.

In addition, you need to specify the universal gas constant and absolute zero using the *PHYSICAL CONSTANTS option.

**Data 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.

## 19.17 *TURBULENCE MODEL: Specify turbulence models for fluid analyses.

This option can be used only as a suboption of the *CFD option to enable turbulence modeling for fluid flow simulations.

**Products:** Abaqus/CFD Abaqus/CAE

**Type:** History data

**Level:** Step

**Abaqus/CAE:** Step module

**References:**

- "Incompressible fluid dynamic analysis," Section 6.6.2 of the Abaqus Analysis User's Guide
- *CFD

**Required parameter:**

TYPE

Set TYPE=SPALART ALLMARAS to specify the Spalart-Allmaras turbulence model.
Set TYPE=RNG KEPSILON to specify the $k$-$\varepsilon$ renormalization group turbulence model.
Set TYPE=KEPSILON REALIZABLE to specify the $k$-$\varepsilon$ realizable turbulence model.
Set TYPE=KOMEGA SST to specify the $k$-$\omega$ shear-stress transport turbulence model.

**Data lines for TYPE=SPALART ALLMARAS:**

First line:

1. $c_{b1}$, production coefficient (default=0.1355).
2. $c_{b2}$, cross diffusion coefficient (default=0.622).
3. $c_{v1}$, production damping (default=7.1).
4. $c_{v2}$, Spalart correction for production damping (default=5).
5. $c_{w1}$, for near-wall equilibrium conditions (default=3.2391).
6. $c_{w2}$, for near-wall friction coefficient calibration (default=0.3).
7. $c_{w3}$, near-wall damping coefficient (default=2).
8. $\sigma$, diffusion coefficient for the diffusive transport of the turbulent kinematic viscosity ($\widetilde{\nu}$). The default value for $\sigma$ is 0.6667.

Second line:

1. $\kappa$, Karman constant (default=0.41).

Third line (enter a blank line unless ENERGY EQUATION=TEMPERATURE is used in the associated *CFD option):

1. $Pr_t$, turbulent Prandtl number (default=0.8889).

**Data lines for TYPE=RNG KEPSILON:**

First line:

1. $C_\mu$, turbulent viscosity coefficient (default=0.085).
2. $C_{\varepsilon_1}$, production coefficient for the $\varepsilon$ equation (default=1.42).
3. $\widetilde{C}_{\varepsilon_2}$, unadjusted dissipation coefficient for the $\varepsilon$ equation (default=1.68).
4. $\sigma_k$, diffusion coefficient for the turbulent kinetic energy diffusive transport (default=0.72).
5. $\sigma_\varepsilon$, diffusion coefficient for the energy dissipation rate diffusive transport (default=0.72).
6. $\beta$, second coefficient for the dissipation coefficient in the $\varepsilon$ equation (default=0.012).
7. $\eta_0$, fixed point ratio of the turbulent to mean strain time scale (default=4.38).

Second line (enter a blank line unless ENERGY EQUATION=TEMPERATURE is used in the associated *CFD option):

1. $Pr_t$, turbulent Prandtl number (default=0.8889).

**Data lines for TYPE=KEPSILON REALIZABLE:**

First line:

1. $C_\mu$, turbulent viscosity coefficient (default=0.09).
2. $A_0$, realizable viscosity coefficient (default=4.0).
3. $C_{\varepsilon 1 min}$, minimum value for the production coefficient for the $\varepsilon$ equation (default=0.43).
4. $C_{\varepsilon 2}$, dissipation coefficient for the $\varepsilon$ equation (default=1.9).
5. $\sigma_k$, diffusion coefficient for the turbulent kinetic energy equation (default=1.0).
6. $\sigma_\varepsilon$, diffusion coefficient for the energy dissipation rate equation (default=1.2).

Second line:

1. $\kappa$, Karman constant (default=0.41).
2. $E$, law of the wall constant (default=8.43171).
3. $Re^*$, two-layer blending threshold (default=60.0).
4. $\Delta Re^*$, two-layer thickness parameter (default=6.0).
5. $A_\mu$, two-layer turbulent viscosity coefficient (default=70.0).
6. $A_\varepsilon$, two-layer $\varepsilon$ prescribed coefficient (default=2.0).
7. $C_{\varepsilon 2}$, standard dissipation coefficient for the $\varepsilon$ equation (default=1.9).

Third line (enter a blank line unless ENERGY EQUATION=TEMPERATURE is used in the associated *CFD option):

1. $Pr_t$, turbulent Prandtl number (default=0.8889).
2. $C_{\varepsilon 1}$, standard production coefficient for the $\varepsilon$ equation (default=1.44).

**Data lines for TYPE=KOMEGA SST:**

First line:

1. $a_1$shear stress transport model coefficient (default=0.31).
2. $\beta^*$ dissipation coefficient of the $k$ equation (default=0.09).
3. $\kappa$ Karman constant (default=0.41).
4. $\sigma_{k1}$ turbulent Prandtl number for the $k$ equation used in the near wall region (default=0.85).
5. $\sigma_{k2}$ turbulent Prandtl number for the $k$ equation used in the outer wall region (default=1.0).
6. $\sigma_{\omega 1}$ turbulent Prandtl number for the $\omega$ equation used in the near wall region (default=0.5).
7. $\sigma_{\omega 2}$ turbulent Prandtl number for the $\omega$ equation used in the outer wall region (default=0.856).
8. $\beta_1$ dissipation coefficient for the $\omega$ equation used in the near wall region (default=0.075).
9. $\beta_2$ dissipation coefficient for the $\omega$ equation used in the outer wall region (default=0.0828).
10. $\gamma_1$ production coefficient for the $\omega$ equation used in the near wall region (default=0.5532).
11. $\gamma_2$ production coefficient for the $\omega$ equation used in the outer wall region (default=0.4404).

Second line (enter a blank line unless ENERGY EQUATION=TEMPERATURE is used on the associated *CFD option):

1. $Pr_t$ turbulent Prandtl number (default=0.8889).

## 20. U

## 20.1 *UEL PROPERTY: Define property values to be used with a user element type.

This option is used to define the properties of a user element.

**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," Section 32.15.1 of the Abaqus Analysis User's Guide
- *USER ELEMENT

**Required parameter:**

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," Section 2.2.5 of the Abaqus Analysis User's Guide) to be used to define a local coordinate system for material calculations in the elements in this set.

**Optional parameters (relevant only for direct-integration dynamic analysis with linear user elements):**

ALPHA

Set this parameter equal to the Rayleigh mass damping factor, $\alpha$.

BETA

Set this parameter equal to the Rayleigh stiffness damping factor, $\beta$.

**To define the properties of linear user elements:**

There are no data lines required.

**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.

## 20.2 *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," Section 34.4.6 of the Abaqus Analysis User's Guide
- *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, $\gamma$.

Second line:

1. Density of charge material, $\rho_c$.
2. Mass of charge material, $m_c$.
3. Acceleration due to gravity, $g$.
4. Atmospheric pressure, $p_{atm}$.
5. Wave effect parameter, $\eta$. 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, $T_{\text{final}}$.
2. Maximum number of time steps for the bubble simulation, $N_{\text{steps}}$. The bubble amplitude simulation ceases when the number of steps reaches $N_{\text{steps}}$ or the time duration, $T_{\text{final}}$, is reached. The default is 1500.
3. Relative step size control parameter, $\Omega_{\text{rel}}$. The default is $1 \times 10^{-11}$.
4. Absolute step size control parameter, $X_{\text{abs}}$. The default is $1 \times 10^{-11}$.
5. Step size control exponent, $\beta$. The step size, $\Delta t$, 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.

## 20.3 *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.

**Product:** Abaqus/Explicit

**Type:** Model data

**Level:** Model

**References:**

- "Fabric material behavior," Section 23.4.1 of the Abaqus Analysis User's Guide
- *FABRIC
- *LOADING DATA
- *UNLOADING DATA

**Required parameter:**

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.**

## 20.4 *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," Section 22.5.1 of the Abaqus Analysis User's Guide
- "Hyperelastic behavior in elastomeric foams," Section 22.5.2 of the Abaqus Analysis User's Guide
- "Low-density foams," Section 22.9.1 of the Abaqus Analysis User's Guide
- "Mullins effect," Section 22.6.1 of the Abaqus Analysis User's Guide
- "Energy dissipation in elastomeric foams," Section 22.6.2 of the Abaqus Analysis User's Guide
- *HYPERELASTIC
- *HYPERFOAM
- *LOW DENSITY FOAM
- *MULLINS EFFECT

### Using uniaxial test data to define a hyperelastic material

**Optional parameter:**

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:**

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 (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, $\epsilon_U$.

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 (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, $\epsilon_U$.
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, $\theta$.
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, $\epsilon_U$.
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 parameter:**

DIRECTION

Set DIRECTION=TENSION to define tensile behavior.
Set DIRECTION=COMPRESSION to define compressive behavior.

**Optional parameter:**

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, $\epsilon_U$. (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, $\theta$.
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, $\epsilon_U$. (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, $\theta$.
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, $\epsilon_U$.

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

## 20.5 *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.

**Product:** Abaqus/Explicit

**Type:** Model data

**Level:** Model

**References:**

- "Connector uniaxial behavior," Section 31.2.10 of the Abaqus Analysis User's Guide
- "Fabric material behavior," Section 23.4.1 of the Abaqus Analysis User's Guide
- *CONNECTOR BEHAVIOR
- *CONNECTOR UNIAXIAL BEHAVIOR
- *FABRIC
- *LOADING DATA
- *UNIAXIAL

### Defining the unloading response for uniaxial behavior in connectors

**Required parameter:**

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 parameter:**

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 parameter:**

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 parameter:**

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.

## 20.6 *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 via user subroutine **USDFLD** in Abaqus/Standard analyses or user subroutine **VUSDFLD** in Abaqus/Explicit analyses. If the *USER DEFINED FIELD option is used, it must appear within a *MATERIAL definition ("Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide).

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "USDFLD," Section 1.1.50 of the Abaqus User Subroutines Reference Guide
- "VUSDFLD," Section 1.2.22 of the Abaqus User Subroutines Reference Guide

**Optional parameter:**

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**.

**Data lines to define material properties when PROPERTIES 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.

## 20.7 *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," Section 32.15.1 of the Abaqus Analysis User's Guide
- "UEL," Section 1.1.28 of the Abaqus User Subroutines Reference Guide
- "VUEL," Section 1.2.12 of the Abaqus User Subroutines Reference Guide
- *ELEMENT
- *MATRIX
- *UEL PROPERTY

### Introducing a linear user-defined element (Abaqus/Standard only)

**Required parameter:**

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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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," Section 1.2.2 of the Abaqus Analysis User's Guide, 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," Section 1.2.2 of the Abaqus Analysis User's Guide, 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.

## 20.8 *USER MATERIAL: Define material constants for use in subroutine UMAT, UMATHT, or VUMAT.

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 Abaqus/Standard it is also used to input material constants for use in a user-defined thermal material model (user subroutine **UMATHT**).

**Products:** Abaqus/Standard Abaqus/Explicit Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "User-defined mechanical material behavior," Section 26.7.1 of the Abaqus Analysis User's Guide
- "User-defined thermal material behavior," Section 26.7.2 of the Abaqus Analysis User's Guide
- "UMAT," Section 1.1.41 of the Abaqus User Subroutines Reference Guide
- "UMATHT," Section 1.1.42 of the Abaqus User Subroutines Reference Guide
- "VUMAT," Section 1.2.20 of the Abaqus User Subroutines Reference Guide

**Required parameter:**

CONSTANTS

Set this parameter equal to the number of constants being entered.

**Optional parameters:**

HYBRID FORMULATION

This parameter applies only to Abaqus/Standard analyses.

Set HYBRID FORMULATION=INCREMENTAL (default) to use an incremental Lagrange multiplier–based formulation with hybrid elements.

Set HYBRID FORMULATION=TOTAL to use a total Lagrange multiplier–based formulation with hybrid elements.

Set HYBRID FORMULATION=INCOMPRESSIBLE to define an incompressible material response with hybrid elements.

TYPE

This parameter applies only to Abaqus/Standard analyses.

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.

## 20.9 *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

**Reference:**

- "UVARM," Section 1.1.55 of the Abaqus User Subroutines Reference Guide

**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.

# 21. V

## 21.1 *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," Section 11.6.1 of the Abaqus Analysis User's Guide
- "Output," Section 4.1.1 of the Abaqus Analysis User's Guide

**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.**

## 21.2 *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.

**Product:** Abaqus/Standard

**Type:** History data

**Level:** Step

**References:**

- "Cavity radiation," Section 41.1.1 of the Abaqus Analysis User's Guide
- "Output," Section 4.1.1 of the Abaqus Analysis User's Guide

**Required parameter:**

CAVITY

Set this parameter equal to the name of the cavity for which this output request is being made.

**Optional parameter:**

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.**

## 21.3 *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," Section 6.2.5 of the Abaqus Analysis User's Guide
- "Rate-dependent plasticity: creep and swelling," Section 23.2.4 of the Abaqus Analysis User's Guide

**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," Section 7.1.1 of the Abaqus Analysis User's Guide).

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," Section 23.2.4 of the Abaqus Analysis User's Guide, 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," Section 7.1.1 of the Abaqus Analysis User's Guide) 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," Section 7.1.1 of the Abaqus Analysis User's Guide). 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 $2 \times 10^{-4}$ 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 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.

## 21.4 *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," Section 22.8.2 of the Abaqus Analysis User's Guide
- "Time domain viscoelasticity," Section 22.7.1 of the Abaqus Analysis User's Guide
- "Frequency domain viscoelasticity," Section 22.7.2 of the Abaqus Analysis User's Guide
- "UCREEPNETWORK," Section 1.1.23 of the Abaqus User Subroutines Reference Guide
- *COMBINED TEST DATA
- *SHEAR TEST DATA
- *TRS
- *VOLUMETRIC TEST DATA

**Required, mutually exclusive parameters:**

FREQUENCY

This parameter applies only to 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.

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

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.

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, isotropic, 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=BERGSTROM-BOYCE to choose a Bergstrom-Boyce law.

Set LAW=HYPERB to choose a hyperbolic-sine law.

Set LAW=POWER LAW to choose a power law model.

Set LAW=STRAIN to choose a strain hardening power law.

Set LAW=USER to input the creep law using user subroutine **UCREEPNETWORK**. This value applies only to Abaqus/Standard analyses.

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 applies only to Abaqus/Standard analyses and 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" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, 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**. 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 viscoelasic 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" in "Defining the constitutive response of cohesive elements using a continuum approach," Section 32.5.5 of the Abaqus Analysis User's Guide, or gasket elements with a material response—see "Defining the gasket behavior using a material model," Section 32.6.5 of the Abaqus Analysis User's Guide). In Abaqus/Explicit this parameter should also be used for the definition of viscoelastic properties for cohesive elements with elastic traction-separation behavior ("Modeling rate-dependent traction-separation behavior in Abaqus/Explicit" in "Defining the constitutive response of cohesive elements using a traction-separation description," Section 32.5.6 of the Abaqus Analysis User's Guide).

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," Section 32.6.5 of the Abaqus Analysis User's Guide).

**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 $\omega g^*$. $(\omega \Re(g^*) = G_\ell / G_\infty)$
2. Imaginary part of $\omega g^*$. $(\omega \Im(g*) = 1 - G_s / G_\infty)$
3. Real part of $\omega k^*$. $(\omega \Re(k^*) = \mathrm{K}_\ell / \mathrm{K}_\infty)$ If the material is incompressible, this value is ignored.
4. Imaginary part of $\omega k^*$. $(\omega \Im(k^*) = 1 - \mathrm{K}_s / \mathrm{K}_\infty)$ 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:**

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. $\tau_1$, 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=PRONY:**

First line:

1. $\bar{g}_1^P$, the modulus ratio in the first term in the Prony series expansion of the shear traction relaxation modulus.

2. $\bar{k}_1^P$, the modulus ratio in the first term in the Prony series expansion of the normal traction relaxation modulus.
3. $\tau_1$, 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:**

First line:

1. Real part of $\omega g^*$. ($\omega\Re(g^*) = G_\ell/G_\infty$)
2. Imaginary part of $\omega g^*$. ($\omega\Im(g*) = 1 - G_s/G_\infty$)
3. Real part of $\omega k^*$. ($\omega\Re(k^*) = \mathrm{K}_\ell/\mathrm{K}_\infty$)
4. Imaginary part of $\omega k^*$. ($\omega\Im(k^*) = 1 - \mathrm{K}_s/\mathrm{K}_\infty$)
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 $\omega k^*$. ($\omega\Re(k^*) = k_\ell/k_\infty$), where $k^*$ represents the complex effective thickness direction dynamic stiffness.

2. Imaginary part of $\omega k^*$. ($\omega \Im(k*) = 1 - k_s/k_\infty$), 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=BERGSTROM-BOYCE:**

First line:

1. Creep parameter, $A$.
2. Effective stress exponent, $m$.
3. Creep strain exponent, $C$.
4. 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.

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. First field variable.
5. Second field variable.
6. Third field variable.
7. Fourth field variable.
8. Fifth field variable.

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 creep constants on predefined field variables.

**Data lines for LAW=POWER LAW:**

First line:

1. $q_0$. (Units of $\mathrm{FL}^{-2}$.)
2. $n$.
3. $m$.
4. $a$. The default is 0.0.
5. $\dot{\varepsilon}_0$. The default is 1.0. (Units of $\mathrm{T}^{-1}$.)
6. Temperature.
7. First field variable.
8. Second field variable.
9. 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=STRAIN:**

First line:

1. $A$. (Units of $\mathrm{F}^{-n}\mathrm{L}^{2n}\mathrm{T}^{-1-m}$.)
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.

## 21.5 *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/Explicit Abaqus/CFD Abaqus/CAE

**Type:** Model data

**Level:** Model

**Abaqus/CAE:** Property module

**References:**

- "Viscosity," Section 26.1.4 of the Abaqus Analysis User's Guide
- "Equation of state," Section 25.2.1 of the Abaqus Analysis User's Guide
- "VUVISCOSITY," Section 1.2.24 of the Abaqus User Subroutines Reference Guide
- *EOS
- *TRS

**Optional parameters:**

DEFINITION

Use this parameter to choose the shear viscosity of the material.

Set DEFINITION=CARREAU-YASUDA to define the Carreau-Yasuda viscous shear behavior.

Set DEFINITION=CROSS to define the Cross viscous shear behavior.

Set DEFINITION=ELLIS-METER to define the Ellis-Meter viscous shear behavior.

Set DEFINITION=HERSCHEL-BULKLEY to define the Herschel-Bulkley viscous shear behavior.

Set DEFINITION=NEWTONIAN (default) to define Newtonian (linear) viscous shear behavior.

Set DEFINITION=POWELL-EYRING to define the Powell-Eyring viscous shear behavior.

Set DEFINITION=POWER LAW to define the power law viscous shear behavior.

Set DEFINITION=TABULAR 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.

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.

**Data lines to define the Carreau-Yasuda viscous shear behavior (DEFINITION=CARREAU-YASUDA):**

First line:

1. Shear viscosity at low shear rates (Newtonian limit), $\eta_0$. (Units of $FL^{-2}T$.)
2. Shear viscosity at large shear rates, $\eta_\infty$. (Units of $FL^{-2}T$.)
3. Time constant, $\lambda$. (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), $\eta_0$. (Units of $FL^{-2}T$.)
2. Shear viscosity at large shear rates, $\eta_\infty$. (Units of $FL^{-2}T$.)
3. Time constant, $\lambda$. (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), $\eta_0$. (Units of $\mathrm{FL^{-2}T}$.)
2. Shear viscosity at large shear rates, $\eta_\infty$. (Units of $\mathrm{FL^{-2}T}$.)
3. Effective shear stress, $\tau_{1/2}$, at which the viscosity is 50% between the Newtonian limit, $\eta_0$, and the infinite shear viscosity, $\eta_\infty$. (Units of $\mathrm{FL^{-2}}$.)
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):**

First line:

1. Shear viscosity at low shear rates, $\eta_0$. (Units of $\mathrm{FL^{-2}T}$.)
2. Yield shear stress, $\tau_0$. (Units of $\mathrm{FL^{-2}}$.)
3. Consistency index, $k$. (Units of $\mathrm{FL^{-2}T^{n}}$.)
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 Herschel-Bulkley viscosity model as a function of temperature and other predefined field variables.

**Data lines to define Newtonian viscous shear behavior (DEFINITION=NEWTONIAN):**

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 the Powell-Eyring viscous shear behavior (DEFINITION=POWELL-EYRING):**

First line:

1. Shear viscosity at low shear rates (Newtonian limit), $\eta_0$. (Units of $FL^{-2}T$.)
2. Shear viscosity at large shear rates, $\eta_\infty$. (Units of $FL^{-2}T$.)
3. Time constant, $\lambda$. (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):**

First line:

1. Consistency index, $k$. (Units of $\mathrm{FL^{-2}T}^n$.)
2. Flow behavior index, $n$.
3. Maximum viscosity, $\eta_{max}$. (Units of $\mathrm{FL^{-2}T}$.)
4. Minimum viscosity, $\eta_{min}$. (Units of $\mathrm{FL^{-2}T}$.)
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 viscous shear behavior in tabular form (DEFINITION=TABULAR):**

First line:

1. Viscosity, $\eta$. (Units of $\mathrm{FL^{-2}T}$.)
2. Effective shear strain rate, $\dot{\gamma}$. (Units of $\mathrm{T^{-1}}$.)
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 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.

## 21.6 *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," Section 23.2.11 of the Abaqus Analysis User's Guide
- *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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

LAW

Set LAW=STRAIN (default) to choose a strain-hardening power law.
Set LAW=TIME to choose a time-hardening 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.

TIME

This parameter is relevant only when LAW=TIME.
Set TIME=CREEP to use creep time in the time-hardening relation.
Set TIME=TOTAL (default) to use total time in the time-hardening relation.

**Data lines for LAW=TIME or LAW=STRAIN:**

First line:

1. $A$. (Units of $\mathrm{F}^{-n}\mathrm{L}^{2n}\mathrm{T}^{-1-m}$.)
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 $\mathrm{FL}^{-2}$.)
2. $Q/R$. (Units of $\theta$.)
3. $A$. (Units of $\mathrm{T}^{-1}$.)
4. $\xi$.

5. $m$.
6. $A_0$. (Units of $\mathrm{FL}^{-2}$.)
7. $\hat{s}$. (Units of $\mathrm{FL}^{-2}$.)
8. $n$.

Second line:

1. $a$.
2. $S_2$. (Units of $\mathrm{FL}^{-2}\theta^{-1}$.)
3. $S_3$. (Units of $\mathrm{FL}^{-2}\theta^{-2}$.)
4. $A_1$. (Units of $\mathrm{FL}^{-2}\theta^{-1}$.)
5. $A_2$. (Units of $\mathrm{FL}^{-2}\theta^{-2}$.)
6. $A_3$. (Units of $\mathrm{FL}^{-2}\mathrm{T}$.)
7. $A_4$. (Units of $\mathrm{FL}^{-2}\mathrm{T}^2$.)
8. $f$.

**Data lines for LAW=DARVEAUX:**

First line:

1. $C_{\mathrm{ss}}$. (Units of $\mathrm{T}^{-1}$.)
2. $Q/R$. (Units of $\theta$.)
3. $\alpha$. (Units of $\mathrm{F}^{-1}\mathrm{L}^2$.)
4. $n$.
5. $\epsilon_T$.
6. $B$.
7. $f$.

**Data lines for LAW=DOUBLE POWER:**

First line:

1. $A_1$. (Units of $\mathrm{T}^{-1}$.)
2. $B_1$. (Units of $\theta$.)
3. $C_1$.
4. $A_2$. (Units of $\mathrm{T}^{-1}$.)
5. $B_2$. (Units of $\theta$.)
6. $C_2$.
7. $\sigma_0$. (Units of $\mathrm{FL}^{-2}$.)
8. $f$.

## 21.7 *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," Section 23.2.9 of the Abaqus Analysis User's Guide
- *POROUS METAL PLASTICITY

**Optional parameter:**

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 "Specifying field variable dependence" in "Material data definition," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

**Data lines to define void nucleation:**

First line:

1. $\varepsilon_N$, 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 $\varepsilon_N$, $s_N$, and $f_N$ on temperature and other predefined field variables.

## 21.8 *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," Section 22.5.1 of the Abaqus Analysis User's Guide
- "Hyperelastic behavior in elastomeric foams," Section 22.5.2 of the Abaqus Analysis User's Guide
- "Time domain viscoelasticity," Section 22.7.1 of the Abaqus Analysis User's Guide
- *HYPERELASTIC
- *HYPERFOAM
- *VISCOELASTIC

### Hyperelastic material model

Volumetric loading test data can be provided by this option to include user-defined material compressibility. Compressibility can alternatively be included by using the POISSON parameter on the *HYPERELASTIC option or, in the case of the Marlow model, 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 will select a default value for the compressibility. See the discussion under "Compressibility" in "Hyperelastic behavior of rubberlike materials," Section 22.5.1 of the Abaqus Analysis User's Guide, before using this option in Abaqus/Explicit.

**Optional parameter:**

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:**

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," Section 21.1.2 of the Abaqus Analysis User's Guide, for more information.

**Data lines to specify volumetric test data for hyperelasticity other than the Marlow model (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 model (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, $\theta$.
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 parameter:**

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 $j_K(t),\ (j_K(t) \geq 1)$.
2. Time $t,\ (t > 0)$.

Repeat this data line as often as necessary to give the compliance-time data.

**Data lines to specify volumetric relaxation test data for a viscoelastic material:**

First line:

1. Normalized volumetric (bulk) modulus $k_R(t),\ (0 \leq k_R(t) \leq 1)$.
2. Time $t,\ (t > 0)$.

Repeat this data line as often as necessary to give the modulus-time data.

# 22. W, X, Y, Z

## 22.1 *WAVE: Define gravity waves for use in immersed structure calculations.

This option is used to define gravity waves for use in applying loads.

**Product:** Abaqus/Aqua

**Type:** Model data

**Level:** Model

**References:**

- "Abaqus/Aqua analysis," Section 6.11.1 of the Abaqus Analysis User's Guide
- "VWAVE," Section 1.2.25 of the Abaqus User Subroutines Reference Guide
- "UWAVE," Section 1.1.56 of the Abaqus User Subroutines Reference Guide

**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," Section 1.2.1 of the Abaqus Analysis User's Guide, 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, $\tau_N$. If this parameter is omitted, the second field in the data line specifies the wavelength, $\lambda_N$.

**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," Section 6.11.1 of the Abaqus Analysis User's Guide.

**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, $\tau$.
3. Phase angle, $\phi$, 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, $\lambda_N$ (default), or wave period of this component, $\tau_N$ (if the parameter WAVE PERIOD is used).
3. Phase angle of this component, $\phi_N$, 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 $d_N$ (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. $x$-coordinate of the origin of the wave data grid.
2. $y$-coordinate of the origin of the wave data grid.
3. $z$-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.

## 22.2 *WIND: Define wind velocity profile for wind loading.

This option is used to define a wind velocity profile for use in applying loads.

**Product:** Abaqus/Aqua

**Type:** Model data

**Level:** Model

**Reference:**

- "Abaqus/Aqua analysis," Section 6.11.1 of the Abaqus Analysis User's Guide

**There are no parameters associated with this option.**

**Data line to define the wind velocity profile:**

First (and only) line:

1. Air density, $\rho_a$.
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. $\alpha$ (default value 1/7).

## I.1. Product Index

# Abaqus/Standard

Section 1.1 *ACOUSTIC FLOW VELOCITY
Section 1.2 *ACOUSTIC MEDIUM
Section 1.3 *ACOUSTIC WAVE FORMULATION
Section 1.4 *ADAPTIVE MESH
Section 1.5 *ADAPTIVE MESH CONSTRAINT
Section 1.6 *ADAPTIVE MESH CONTROLS
Section 1.8 *ADJUST
Section 1.9 *AMPLITUDE
Section 1.10 *ANISOTROPIC HYPERELASTIC
Section 1.12 *ANNEAL TEMPERATURE
Section 1.14 *ASSEMBLY
Section 1.15 *ASYMMETRIC-AXISYMMETRIC
Section 1.16 *AXIAL
Section 2.1 *BASE MOTION
Section 2.2 *BASELINE CORRECTION
Section 2.3 *BEAM ADDED INERTIA
Section 2.4 *BEAM FLUID INERTIA
Section 2.5 *BEAM GENERAL SECTION
Section 2.6 *BEAM SECTION
Section 2.7 *BEAM SECTION GENERATE
Section 2.8 *BEAM SECTION OFFSET
Section 2.9 *BIAXIAL TEST DATA
Section 2.12 *BOUNDARY
Section 2.16 *BUCKLE
Section 2.17 *BUCKLING ENVELOPE
Section 2.18 *BUCKLING LENGTH
Section 2.19 *BUCKLING REDUCTION FACTORS
Section 3.2 *CAP CREEP
Section 3.3 *CAP HARDENING
Section 3.4 *CAP PLASTICITY
Section 3.6 *CAST IRON COMPRESSION HARDENING
Section 3.7 *CAST IRON PLASTICITY
Section 3.8 *CAST IRON TENSION HARDENING
Section 3.9 *CAVITY DEFINITION
Section 3.10 *CECHARGE
Section 3.11 *CECURRENT

Section 3.12 *CENTROID
Section 3.14 *CFILM
Section 3.15 *CFLOW
Section 3.16 *CFLUX
Section 3.17 *CHANGE FRICTION
Section 3.19 *CLAY HARDENING
Section 3.20 *CLAY PLASTICITY
Section 3.21 *CLEARANCE
Section 3.22 *CLOAD
Section 3.23 *COHESIVE BEHAVIOR
Section 3.24 *COHESIVE SECTION
Section 3.25 *COMBINED TEST DATA
Section 3.26 *COMPLEX FREQUENCY
Section 3.27 *COMPOSITE MODAL DAMPING
Section 3.28 *CONCRETE
Section 3.29 *CONCRETE COMPRESSION DAMAGE
Section 3.30 *CONCRETE COMPRESSION HARDENING
Section 3.31 *CONCRETE DAMAGED PLASTICITY
Section 3.32 *CONCRETE TENSION DAMAGE
Section 3.33 *CONCRETE TENSION STIFFENING
Section 3.34 *CONDUCTIVITY
Section 3.35 *CONNECTOR BEHAVIOR
Section 3.36 *CONNECTOR CONSTITUTIVE REFERENCE
Section 3.37 *CONNECTOR DAMAGE EVOLUTION
Section 3.38 *CONNECTOR DAMAGE INITIATION
Section 3.39 *CONNECTOR DAMPING
Section 3.40 *CONNECTOR DERIVED COMPONENT
Section 3.41 *CONNECTOR ELASTICITY
Section 3.42 *CONNECTOR FAILURE
Section 3.43 *CONNECTOR FRICTION
Section 3.44 *CONNECTOR HARDENING
Section 3.45 *CONNECTOR LOAD
Section 3.46 *CONNECTOR LOCK
Section 3.47 *CONNECTOR MOTION
Section 3.48 *CONNECTOR PLASTICITY
Section 3.49 *CONNECTOR POTENTIAL
Section 3.50 *CONNECTOR SECTION
Section 3.51 *CONNECTOR STOP
Section 3.53 *CONSTRAINT CONTROLS
Section 3.54 *CONTACT
Section 3.57 *CONTACT CONTROLS
Section 3.59 *CONTACT DAMPING

Section 3.60 *CONTACT EXCLUSIONS
Section 3.61 *CONTACT FILE
Section 3.62 *CONTACT FORMULATION
Section 3.63 *CONTACT INCLUSIONS
Section 3.64 *CONTACT INITIALIZATION ASSIGNMENT
Section 3.65 *CONTACT INITIALIZATION DATA
Section 3.66 *CONTACT INTERFERENCE
Section 3.67 *CONTACT OUTPUT
Section 3.68 *CONTACT PAIR
Section 3.69 *CONTACT PERMEABILITY
Section 3.70 *CONTACT PRINT
Section 3.71 *CONTACT PROPERTY ASSIGNMENT
Section 3.73 *CONTACT STABILIZATION
Section 3.74 *CONTOUR INTEGRAL
Section 3.75 *CONTROLS
Section 3.77 *CORRELATION
Section 3.78 *CO-SIMULATION
Section 3.79 *CO-SIMULATION CONTROLS
Section 3.80 *CO-SIMULATION REGION
Section 3.81 *COUPLED TEMPERATURE-DISPLACEMENT
Section 3.82 *COUPLED THERMAL-ELECTRICAL
Section 3.83 *COUPLING
Section 3.84 *CRADIATE
Section 3.85 *CREEP
Section 3.86 *CREEP STRAIN RATE CONTROL
Section 3.87 *CRUSHABLE FOAM
Section 3.88 *CRUSHABLE FOAM HARDENING
Section 3.89 *CYCLED PLASTIC
Section 3.90 *CYCLIC
Section 3.91 *CYCLIC HARDENING
Section 3.92 *CYCLIC SYMMETRY MODEL
Section 4.2 *D EM POTENTIAL
Section 4.3 *DAMAGE EVOLUTION
Section 4.4 *DAMAGE INITIATION
Section 4.5 *DAMAGE STABILIZATION
Section 4.6 *DAMPING
Section 4.7 *DAMPING CONTROLS
Section 4.8 *DASHPOT
Section 4.9 *DEBOND
Section 4.10 *DECHARGE
Section 4.11 *DECURRENT
Section 4.12 *DEFORMATION PLASTICITY

Section 4.13 *DENSITY
Section 4.14 *DEPVAR
Section 4.18 *DETONATION POINT
Section 4.19 *DFLOW
Section 4.21 *DFLUX
Section 4.22 *DIAGNOSTICS
Section 4.23 *DIELECTRIC
Section 4.24 *DIFFUSIVITY
Section 4.25 *DIRECT CYCLIC
Section 4.27 *DISPLAY BODY
Section 4.28 *DISTRIBUTING
Section 4.29 *DISTRIBUTING COUPLING
Section 4.30 *DISTRIBUTION
Section 4.31 *DISTRIBUTION TABLE
Section 4.32 *DLOAD
Section 4.34 *DRAG CHAIN
Section 4.35 *DRUCKER PRAGER
Section 4.36 *DRUCKER PRAGER CREEP
Section 4.37 *DRUCKER PRAGER HARDENING
Section 4.39 *DSECHARGE
Section 4.40 *DSECURRENT
Section 4.41 *DSFLOW
Section 4.42 *DSFLUX
Section 4.43 *DSLOAD
Section 4.44 *DYNAMIC
Section 5.1 *EL FILE
Section 5.2 *EL PRINT
Section 5.3 *ELASTIC
Section 5.4 *ELCOPY
Section 5.5 *ELECTRICAL CONDUCTIVITY
Section 5.6 *ELECTROMAGNETIC
Section 5.7 *ELEMENT
Section 5.8 *ELEMENT MATRIX OUTPUT
Section 5.9 *ELEMENT OPERATOR OUTPUT
Section 5.10 *ELEMENT OUTPUT
Section 5.12 *ELGEN
Section 5.13 *ELSET
Section 5.14 *EMBEDDED ELEMENT
Section 5.15 *EMISSIVITY
Section 5.16 *END ASSEMBLY
Section 5.17 *END INSTANCE
Section 5.18 *END LOAD CASE

Section 5.19 *END PART
Section 5.20 *END STEP
Section 5.22 *ENERGY FILE
Section 5.23 *ENERGY OUTPUT
Section 5.24 *ENERGY PRINT
Section 5.25 *ENRICHMENT
Section 5.26 *ENRICHMENT ACTIVATION
Section 5.28 *EOS COMPACTION
Section 5.29 *EPJOINT
Section 5.30 *EQUATION
Section 5.34 *EXPANSION
Section 6.2 *FAIL STRAIN
Section 6.3 *FAIL STRESS
Section 6.4 *FAILURE RATIOS
Section 6.5 *FASTENER
Section 6.6 *FASTENER PROPERTY
Section 6.7 *FIELD
Section 6.8 *FILE FORMAT
Section 6.10 *FILM
Section 6.11 *FILM PROPERTY
Section 6.14 *FLOW
Section 6.15 *FLEXIBLE BODY
Section 6.16 *FLUID BEHAVIOR
Section 6.18 *FLUID BULK MODULUS
Section 6.19 *FLUID CAVITY
Section 6.20 *FLUID DENSITY
Section 6.21 *FLUID EXCHANGE
Section 6.23 *FLUID EXCHANGE PROPERTY
Section 6.24 *FLUID EXPANSION
Section 6.25 *FLUID FLUX
Section 6.30 *FLUID LEAKOFF
Section 6.32 *FOUNDATION
Section 6.33 *FRACTURE CRITERION
Section 6.34 *FRAME SECTION
Section 6.35 *FREQUENCY
Section 6.36 *FRICTION
Section 7.1 *GAP
Section 7.2 *GAP CONDUCTANCE
Section 7.3 *GAP ELECTRICAL CONDUCTANCE
Section 7.4 *GAP FLOW
Section 7.5 *GAP HEAT GENERATION
Section 7.6 *GAP RADIATION

Section 7.8 *GASKET BEHAVIOR
Section 7.9 *GASKET CONTACT AREA
Section 7.10 *GASKET ELASTICITY
Section 7.11 *GASKET SECTION
Section 7.12 *GASKET THICKNESS BEHAVIOR
Section 7.13 *GEL
Section 7.14 *GEOSTATIC
Section 7.15 *GLOBAL DAMPING
Section 8.1 *HEADING
Section 8.2 *HEAT GENERATION
Section 8.3 *HEAT TRANSFER
Section 8.4 *HEATCAP
Section 8.5 *HOURGLASS STIFFNESS
Section 8.6 *HYPERELASTIC
Section 8.7 *HYPERFOAM
Section 8.8 *HYPOELASTIC
Section 8.9 *HYSTERESIS
Section 9.1 *IMPEDANCE
Section 9.2 *IMPEDANCE PROPERTY
Section 9.3 *IMPERFECTION
Section 9.4 *IMPORT
Section 9.5 *IMPORT CONTROLS
Section 9.6 *IMPORT ELSET
Section 9.7 *IMPORT NSET
Section 9.8 *INCIDENT WAVE
Section 9.9 *INCIDENT WAVE FLUID PROPERTY
Section 9.10 *INCIDENT WAVE INTERACTION
Section 9.11 *INCIDENT WAVE INTERACTION PROPERTY
Section 9.12 *INCIDENT WAVE PROPERTY
Section 9.13 *INCIDENT WAVE REFLECTION
Section 9.14 *INCLUDE
Section 9.15 *INCREMENTATION OUTPUT
Section 9.16 *INELASTIC HEAT FRACTION
Section 9.17 *INERTIA RELIEF
Section 9.18 *INITIAL CONDITIONS
Section 9.19 *INSTANCE
Section 9.22 *INTERFACE
Section 9.23 *ITS
Section 10.1 *JOINT
Section 10.2 *JOINT ELASTICITY
Section 10.3 *JOINT PLASTICITY
Section 10.4 *JOINTED MATERIAL

Section 10.5 *JOULE HEAT FRACTION
Section 11.1 *KAPPA
Section 11.2 *KINEMATIC
Section 11.3 *KINEMATIC COUPLING
Section 12.1 *LATENT HEAT
Section 12.2 *LOAD CASE
Section 13.1 *MAGNETIC PERMEABILITY
Section 13.2 *MAGNETOSTATIC
Section 13.3 *MAP SOLUTION
Section 13.4 *MASS
Section 13.6 *MASS DIFFUSION
Section 13.7 *MASS FLOW RATE
Section 13.8 *MATERIAL
Section 13.9 *MATRIX
Section 13.10 *MATRIX ASSEMBLE
Section 13.11 *MATRIX CHECK
Section 13.12 *MATRIX GENERATE
Section 13.13 *MATRIX INPUT
Section 13.14 *MATRIX OUTPUT
Section 13.16 *MEMBRANE SECTION
Section 13.17 *MODAL DAMPING
Section 13.18 *MODAL DYNAMIC
Section 13.19 *MODAL FILE
Section 13.20 *MODAL OUTPUT
Section 13.21 *MODAL PRINT
Section 13.22 *MODEL CHANGE
Section 13.23 *MOHR COULOMB
Section 13.24 *MOHR COULOMB HARDENING
Section 13.25 *MOISTURE SWELLING
Section 13.26 *MOLECULAR WEIGHT
Section 13.28 *MONITOR
Section 13.29 *MOTION
Section 13.30 *MPC
Section 13.31 *MULLINS EFFECT
Section 13.32 *M1
Section 13.33 *M2
Section 14.1 *NCOPY
Section 14.2 *NFILL
Section 14.3 *NGEN
Section 14.4 *NMAP
Section 14.5 *NO COMPRESSION
Section 14.6 *NO TENSION

Section 14.7 *NODAL ENERGY RATE
Section 14.8 *NODAL THICKNESS
Section 14.9 *NODE
Section 14.10 *NODE FILE
Section 14.11 *NODE OUTPUT
Section 14.12 *NODE PRINT
Section 14.14 *NONLINEAR BH
Section 14.15 *NONSTRUCTURAL MASS
Section 14.16 *NORMAL
Section 14.17 *NSET
Section 15.1 *ORIENTATION
Section 15.2 *ORNL
Section 15.3 *OUTPUT
Section 16.1 *PARAMETER
Section 16.2 *PARAMETER DEPENDENCE
Section 16.3 *PARAMETER SHAPE VARIATION
Section 16.4 *PART
Section 16.5 *PERIODIC
Section 16.7 *PERMANENT MAGNETIZATION
Section 16.8 *PERMEABILITY
Section 16.9 *PHYSICAL CONSTANTS
Section 16.10 *PIEZOELECTRIC
Section 16.11 *PIPE-SOIL INTERACTION
Section 16.12 *PIPE-SOIL STIFFNESS
Section 16.13 *PLANAR TEST DATA
Section 16.14 *PLASTIC
Section 16.15 *PLASTIC AXIAL
Section 16.16 *PLASTIC M1
Section 16.17 *PLASTIC M2
Section 16.18 *PLASTIC TORQUE
Section 16.19 *POROUS BULK MODULI
Section 16.20 *POROUS ELASTIC
Section 16.22 *POROUS METAL PLASTICITY
Section 16.23 *POST OUTPUT
Section 16.24 *POTENTIAL
Section 16.25 *PREPRINT
Section 16.27 *PRESSURE PENETRATION
Section 16.28 *PRESSURE STRESS
Section 16.29 *PRESTRESS HOLD
Section 16.30 *PRE-TENSION SECTION
Section 16.31 *PRINT
Section 16.32 *PSD-DEFINITION

Section 17.1 *RADIATE
Section 17.2 *RADIATION FILE
Section 17.3 *RADIATION OUTPUT
Section 17.4 *RADIATION PRINT
Section 17.5 *RADIATION SYMMETRY
Section 17.6 *RADIATION VIEW FACTOR
Section 17.7 *RANDOM RESPONSE
Section 17.8 *RATE DEPENDENT
Section 17.9 *RATIOS
Section 17.11 *REBAR
Section 17.12 *REBAR LAYER
Section 17.13 *REFLECTION
Section 17.14 *RELEASE
Section 17.15 *RESPONSE SPECTRUM
Section 17.16 *RESTART
Section 17.17 *RETAINED NODAL DOFS
Section 17.18 *RIGID BODY
Section 17.19 *RIGID SURFACE
Section 17.20 *ROTARY INERTIA
Section 18.1 *SECTION CONTROLS
Section 18.2 *SECTION FILE
Section 18.3 *SECTION ORIGIN
Section 18.4 *SECTION POINTS
Section 18.5 *SECTION PRINT
Section 18.6 *SELECT CYCLIC SYMMETRY MODES
Section 18.7 *SELECT EIGENMODES
Section 18.8 *SFILM
Section 18.9 *SFLOW
Section 18.10 *SHEAR CENTER
Section 18.12 *SHEAR RETENTION
Section 18.13 *SHEAR TEST DATA
Section 18.14 *SHELL GENERAL SECTION
Section 18.15 *SHELL SECTION
Section 18.16 *SHELL TO SOLID COUPLING
Section 18.17 *SIMPEDANCE
Section 18.18 *SIMPLE SHEAR TEST DATA
Section 18.19 *SLIDE LINE
Section 18.20 *SLOAD
Section 18.21 *SOILS
Section 18.22 *SOLID SECTION
Section 18.23 *SOLUBILITY
Section 18.24 *SOLUTION TECHNIQUE

Section 18.25 *SOLVER CONTROLS
Section 18.26 *SORPTION
Section 18.27 *SPECIFIC HEAT
Section 18.28 *SPECTRUM
Section 18.29 *SPRING
Section 18.30 *SRADIATE
Section 18.31 *STATIC
Section 18.34 *STEADY STATE DYNAMICS
Section 18.35 *STEADY STATE TRANSPORT
Section 18.36 *STEP
Section 18.38 *SUBMODEL
Section 18.39 *SUBSTRUCTURE COPY
Section 18.40 *SUBSTRUCTURE DELETE
Section 18.41 *SUBSTRUCTURE DIRECTORY
Section 18.42 *SUBSTRUCTURE GENERATE
Section 18.43 *SUBSTRUCTURE LOAD CASE
Section 18.44 *SUBSTRUCTURE MATRIX OUTPUT
Section 18.45 *SUBSTRUCTURE PATH
Section 18.46 *SUBSTRUCTURE PROPERTY
Section 18.47 *SURFACE
Section 18.48 *SURFACE BEHAVIOR
Section 18.49 *SURFACE FLAW
Section 18.50 *SURFACE INTERACTION
Section 18.52 *SURFACE PROPERTY
Section 18.53 *SURFACE PROPERTY ASSIGNMENT
Section 18.54 *SURFACE SECTION
Section 18.55 *SURFACE SMOOTHING
Section 18.56 *SWELLING
Section 18.57 *SYMMETRIC MODEL GENERATION
Section 18.58 *SYMMETRIC RESULTS TRANSFER
Section 18.59 *SYSTEM
Section 19.1 *TEMPERATURE
Section 19.3 *TENSION CUTOFF
Section 19.4 *TENSION STIFFENING
Section 19.5 *THERMAL EXPANSION
Section 19.6 *TIE
Section 19.7 *TIME POINTS
Section 19.8 *TORQUE
Section 19.9 *TORQUE PRINT
Section 19.11 *TRANSFORM
Section 19.13 *TRANSPORT VELOCITY
Section 19.14 *TRANSVERSE SHEAR STIFFNESS

Section 19.15 *TRIAXIAL TEST DATA
Section 19.16 *TRS
Section 20.1 *UEL PROPERTY
Section 20.2 *UNDEX CHARGE PROPERTY
Section 20.4 *UNIAXIAL TEST DATA
Section 20.6 *USER DEFINED FIELD
Section 20.7 *USER ELEMENT
Section 20.8 *USER MATERIAL
Section 20.9 *USER OUTPUT VARIABLES
Section 21.2 *VIEW FACTOR OUTPUT
Section 21.3 *VISCO
Section 21.4 *VISCOELASTIC
Section 21.6 *VISCOUS
Section 21.7 *VOID NUCLEATION
Section 21.8 *VOLUMETRIC TEST DATA

## Abaqus/Explicit

Section 1.2 *ACOUSTIC MEDIUM
Section 1.3 *ACOUSTIC WAVE FORMULATION
Section 1.4 *ADAPTIVE MESH
Section 1.5 *ADAPTIVE MESH CONSTRAINT
Section 1.6 *ADAPTIVE MESH CONTROLS
Section 1.7 *ADAPTIVE MESH REFINEMENT
Section 1.8 *ADJUST
Section 1.9 *AMPLITUDE
Section 1.10 *ANISOTROPIC HYPERELASTIC
Section 1.11 *ANNEAL
Section 1.12 *ANNEAL TEMPERATURE
Section 1.14 *ASSEMBLY
Section 1.16 *AXIAL
Section 2.3 *BEAM ADDED INERTIA
Section 2.4 *BEAM FLUID INERTIA
Section 2.5 *BEAM GENERAL SECTION
Section 2.6 *BEAM SECTION
Section 2.9 *BIAXIAL TEST DATA
Section 2.10 *BLOCKAGE
Section 2.11 *BOND
Section 2.12 *BOUNDARY
Section 2.13 *BRITTLE CRACKING
Section 2.14 *BRITTLE FAILURE
Section 2.15 *BRITTLE SHEAR
Section 2.20 *BULK VISCOSITY

Section 3.3 *CAP HARDENING
Section 3.4 *CAP PLASTICITY
Section 3.5 *CAPACITY
Section 3.6 *CAST IRON COMPRESSION HARDENING
Section 3.7 *CAST IRON PLASTICITY
Section 3.8 *CAST IRON TENSION HARDENING
Section 3.12 *CENTROID
Section 3.14 *CFILM
Section 3.16 *CFLUX
Section 3.18 *CHARACTERISTIC LENGTH
Section 3.19 *CLAY HARDENING
Section 3.20 *CLAY PLASTICITY
Section 3.21 *CLEARANCE
Section 3.22 *CLOAD
Section 3.23 *COHESIVE BEHAVIOR
Section 3.24 *COHESIVE SECTION
Section 3.25 *COMBINED TEST DATA
Section 3.29 *CONCRETE COMPRESSION DAMAGE
Section 3.30 *CONCRETE COMPRESSION HARDENING
Section 3.31 *CONCRETE DAMAGED PLASTICITY
Section 3.32 *CONCRETE TENSION DAMAGE
Section 3.33 *CONCRETE TENSION STIFFENING
Section 3.34 *CONDUCTIVITY
Section 3.35 *CONNECTOR BEHAVIOR
Section 3.36 *CONNECTOR CONSTITUTIVE REFERENCE
Section 3.37 *CONNECTOR DAMAGE EVOLUTION
Section 3.38 *CONNECTOR DAMAGE INITIATION
Section 3.39 *CONNECTOR DAMPING
Section 3.40 *CONNECTOR DERIVED COMPONENT
Section 3.41 *CONNECTOR ELASTICITY
Section 3.42 *CONNECTOR FAILURE
Section 3.43 *CONNECTOR FRICTION
Section 3.44 *CONNECTOR HARDENING
Section 3.45 *CONNECTOR LOAD
Section 3.46 *CONNECTOR LOCK
Section 3.47 *CONNECTOR MOTION
Section 3.48 *CONNECTOR PLASTICITY
Section 3.49 *CONNECTOR POTENTIAL
Section 3.50 *CONNECTOR SECTION
Section 3.51 *CONNECTOR STOP
Section 3.52 *CONNECTOR UNIAXIAL BEHAVIOR
Section 3.54 *CONTACT

Section 3.55 *CONTACT CLEARANCE
Section 3.56 *CONTACT CLEARANCE ASSIGNMENT
Section 3.57 *CONTACT CONTROLS
Section 3.58 *CONTACT CONTROLS ASSIGNMENT
Section 3.59 *CONTACT DAMPING
Section 3.60 *CONTACT EXCLUSIONS
Section 3.62 *CONTACT FORMULATION
Section 3.63 *CONTACT INCLUSIONS
Section 3.67 *CONTACT OUTPUT
Section 3.68 *CONTACT PAIR
Section 3.71 *CONTACT PROPERTY ASSIGNMENT
Section 3.76 *CONWEP CHARGE PROPERTY
Section 3.78 *CO-SIMULATION
Section 3.79 *CO-SIMULATION CONTROLS
Section 3.80 *CO-SIMULATION REGION
Section 3.83 *COUPLING
Section 3.84 *CRADIATE
Section 3.87 *CRUSHABLE FOAM
Section 3.88 *CRUSHABLE FOAM HARDENING
Section 3.91 *CYCLIC HARDENING
Section 4.3 *DAMAGE EVOLUTION
Section 4.4 *DAMAGE INITIATION
Section 4.5 *DAMAGE STABILIZATION
Section 4.6 *DAMPING
Section 4.8 *DASHPOT
Section 4.13 *DENSITY
Section 4.14 *DEPVAR
Section 4.18 *DETONATION POINT
Section 4.21 *DFLUX
Section 4.22 *DIAGNOSTICS
Section 4.26 *DISCRETE SECTION
Section 4.27 *DISPLAY BODY
Section 4.28 *DISTRIBUTING
Section 4.30 *DISTRIBUTION
Section 4.31 *DISTRIBUTION TABLE
Section 4.32 *DLOAD
Section 4.33 *DOMAIN DECOMPOSITION
Section 4.35 *DRUCKER PRAGER
Section 4.37 *DRUCKER PRAGER HARDENING
Section 4.42 *DSFLUX
Section 4.43 *DSLOAD
Section 4.44 *DYNAMIC

Section 4.45 *DYNAMIC TEMPERATURE-DISPLACEMENT
Section 5.1 *EL FILE
Section 5.3 *ELASTIC
Section 5.4 *ELCOPY
Section 5.7 *ELEMENT
Section 5.10 *ELEMENT OUTPUT
Section 5.12 *ELGEN
Section 5.13 *ELSET
Section 5.14 *EMBEDDED ELEMENT
Section 5.16 *END ASSEMBLY
Section 5.17 *END INSTANCE
Section 5.19 *END PART
Section 5.20 *END STEP
Section 5.22 *ENERGY FILE
Section 5.23 *ENERGY OUTPUT
Section 5.27 *EOS
Section 5.28 *EOS COMPACTION
Section 5.30 *EQUATION
Section 5.31 *EULERIAN BOUNDARY
Section 5.32 *EULERIAN MESH MOTION
Section 5.33 *EULERIAN SECTION
Section 5.34 *EXPANSION
Section 5.35 *EXTREME ELEMENT VALUE
Section 5.36 *EXTREME NODE VALUE
Section 5.37 *EXTREME VALUE
Section 6.1 *FABRIC
Section 6.2 *FAIL STRAIN
Section 6.3 *FAIL STRESS
Section 6.5 *FASTENER
Section 6.6 *FASTENER PROPERTY
Section 6.7 *FIELD
Section 6.9 *FILE OUTPUT
Section 6.10 *FILM
Section 6.11 *FILM PROPERTY
Section 6.12 *FILTER
Section 6.13 *FIXED MASS SCALING
Section 6.16 *FLUID BEHAVIOR
Section 6.18 *FLUID BULK MODULUS
Section 6.19 *FLUID CAVITY
Section 6.20 *FLUID DENSITY
Section 6.21 *FLUID EXCHANGE
Section 6.22 *FLUID EXCHANGE ACTIVATION

Section 6.23 *FLUID EXCHANGE PROPERTY
Section 6.24 *FLUID EXPANSION
Section 6.25 *FLUID FLUX
Section 6.26 *FLUID INFLATOR
Section 6.27 *FLUID INFLATOR ACTIVATION
Section 6.28 *FLUID INFLATOR MIXTURE
Section 6.29 *FLUID INFLATOR PROPERTY
Section 6.33 *FRACTURE CRITERION
Section 6.36 *FRICTION
Section 7.2 *GAP CONDUCTANCE
Section 7.5 *GAP HEAT GENERATION
Section 7.6 *GAP RADIATION
Section 7.7 *GAS SPECIFIC HEAT
Section 8.1 *HEADING
Section 8.4 *HEATCAP
Section 8.6 *HYPERELASTIC
Section 8.7 *HYPERFOAM
Section 9.1 *IMPEDANCE
Section 9.2 *IMPEDANCE PROPERTY
Section 9.3 *IMPERFECTION
Section 9.4 *IMPORT
Section 9.5 *IMPORT CONTROLS
Section 9.6 *IMPORT ELSET
Section 9.7 *IMPORT NSET
Section 9.8 *INCIDENT WAVE
Section 9.9 *INCIDENT WAVE FLUID PROPERTY
Section 9.10 *INCIDENT WAVE INTERACTION
Section 9.11 *INCIDENT WAVE INTERACTION PROPERTY
Section 9.12 *INCIDENT WAVE PROPERTY
Section 9.13 *INCIDENT WAVE REFLECTION
Section 9.14 *INCLUDE
Section 9.15 *INCREMENTATION OUTPUT
Section 9.16 *INELASTIC HEAT FRACTION
Section 9.18 *INITIAL CONDITIONS
Section 9.19 *INSTANCE
Section 9.20 *INTEGRATED OUTPUT
Section 9.21 *INTEGRATED OUTPUT SECTION
Section 11.2 *KINEMATIC
Section 12.1 *LATENT HEAT
Section 12.3 *LOADING DATA
Section 12.4 *LOW DENSITY FOAM
Section 13.4 *MASS

Section 13.5 *MASS ADJUST
Section 13.8 *MATERIAL
Section 13.15 *MEDIA TRANSPORT
Section 13.16 *MEMBRANE SECTION
Section 13.23 *MOHR COULOMB
Section 13.24 *MOHR COULOMB HARDENING
Section 13.26 *MOLECULAR WEIGHT
Section 13.28 *MONITOR
Section 13.30 *MPC
Section 13.31 *MULLINS EFFECT
Section 13.32 *M1
Section 13.33 *M2
Section 14.1 *NCOPY
Section 14.2 *NFILL
Section 14.3 *NGEN
Section 14.4 *NMAP
Section 14.7 *NODAL ENERGY RATE
Section 14.8 *NODAL THICKNESS
Section 14.9 *NODE
Section 14.10 *NODE FILE
Section 14.11 *NODE OUTPUT
Section 14.15 *NONSTRUCTURAL MASS
Section 14.16 *NORMAL
Section 14.17 *NSET
Section 15.1 *ORIENTATION
Section 15.3 *OUTPUT
Section 16.1 *PARAMETER
Section 16.2 *PARAMETER DEPENDENCE
Section 16.3 *PARAMETER SHAPE VARIATION
Section 16.4 *PART
Section 16.6 *PERIODIC MEDIA
Section 16.9 *PHYSICAL CONSTANTS
Section 16.13 *PLANAR TEST DATA
Section 16.14 *PLASTIC
Section 16.21 *POROUS FAILURE CRITERIA
Section 16.22 *POROUS METAL PLASTICITY
Section 16.24 *POTENTIAL
Section 16.25 *PREPRINT
Section 16.31 *PRINT
Section 17.1 *RADIATE
Section 17.8 *RATE DEPENDENT
Section 17.10 *REACTION RATE

Section 17.11 *REBAR
Section 17.12 *REBAR LAYER
Section 17.16 *RESTART
Section 17.18 *RIGID BODY
Section 17.20 *ROTARY INERTIA
Section 18.1 *SECTION CONTROLS
Section 18.4 *SECTION POINTS
Section 18.8 *SFILM
Section 18.10 *SHEAR CENTER
Section 18.11 *SHEAR FAILURE
Section 18.13 *SHEAR TEST DATA
Section 18.14 *SHELL GENERAL SECTION
Section 18.15 *SHELL SECTION
Section 18.16 *SHELL TO SOLID COUPLING
Section 18.17 *SIMPEDANCE
Section 18.18 *SIMPLE SHEAR TEST DATA
Section 18.22 *SOLID SECTION
Section 18.27 *SPECIFIC HEAT
Section 18.29 *SPRING
Section 18.30 *SRADIATE
Section 18.32 *STEADY STATE CRITERIA
Section 18.33 *STEADY STATE DETECTION
Section 18.36 *STEP
Section 18.37 *SUBCYCLING
Section 18.38 *SUBMODEL
Section 18.47 *SURFACE
Section 18.48 *SURFACE BEHAVIOR
Section 18.50 *SURFACE INTERACTION
Section 18.53 *SURFACE PROPERTY ASSIGNMENT
Section 18.54 *SURFACE SECTION
Section 18.59 *SYSTEM
Section 19.1 *TEMPERATURE
Section 19.2 *TENSILE FAILURE
Section 19.3 *TENSION CUTOFF
Section 19.5 *THERMAL EXPANSION
Section 19.6 *TIE
Section 19.7 *TIME POINTS
Section 19.8 *TORQUE
Section 19.10 *TRACER PARTICLE
Section 19.11 *TRANSFORM
Section 19.14 *TRANSVERSE SHEAR STIFFNESS
Section 19.15 *TRIAXIAL TEST DATA

Section 19.16 *TRS
Section 20.1 *UEL PROPERTY
Section 20.2 *UNDEX CHARGE PROPERTY
Section 20.3 *UNIAXIAL
Section 20.4 *UNIAXIAL TEST DATA
Section 20.5 *UNLOADING DATA
Section 20.6 *USER DEFINED FIELD
Section 20.7 *USER ELEMENT
Section 20.8 *USER MATERIAL
Section 21.1 *VARIABLE MASS SCALING
Section 21.4 *VISCOELASTIC
Section 21.5 *VISCOSITY
Section 21.7 *VOID NUCLEATION
Section 21.8 *VOLUMETRIC TEST DATA

## Abaqus/CFD

Section 1.9 *AMPLITUDE
Section 1.14 *ASSEMBLY
Section 2.12 *BOUNDARY
Section 3.13 *CFD
Section 3.22 *CLOAD
Section 3.34 *CONDUCTIVITY
Section 3.75 *CONTROLS
Section 3.78 *CO-SIMULATION
Section 3.79 *CO-SIMULATION CONTROLS
Section 3.80 *CO-SIMULATION REGION
Section 4.13 *DENSITY
Section 4.19 *DFLOW
Section 4.21 *DFLUX
Section 4.30 *DISTRIBUTION
Section 4.31 *DISTRIBUTION TABLE
Section 4.32 *DLOAD
Section 4.42 *DSFLUX
Section 4.43 *DSLOAD
Section 5.4 *ELCOPY
Section 5.7 *ELEMENT
Section 5.10 *ELEMENT OUTPUT
Section 5.12 *ELGEN
Section 5.13 *ELSET
Section 5.16 *END ASSEMBLY
Section 5.17 *END INSTANCE
Section 5.19 *END PART

Section 5.20 *END STEP
Section 5.21 *ENERGY EQUATION SOLVER
Section 5.23 *ENERGY OUTPUT
Section 5.27 *EOS
Section 5.34 *EXPANSION
Section 6.17 *FLUID BOUNDARY
Section 6.31 *FLUID SECTION
Section 8.1 *HEADING
Section 8.3 *HEAT TRANSFER
Section 9.14 *INCLUDE
Section 9.18 *INITIAL CONDITIONS
Section 9.19 *INSTANCE
Section 13.8 *MATERIAL
Section 13.27 *MOMENTUM EQUATION SOLVER
Section 14.1 *NCOPY
Section 14.2 *NFILL
Section 14.3 *NGEN
Section 14.4 *NMAP
Section 14.9 *NODE
Section 14.11 *NODE OUTPUT
Section 14.17 *NSET
Section 15.3 *OUTPUT
Section 16.4 *PART
Section 16.8 *PERMEABILITY
Section 16.26 *PRESSURE EQUATION SOLVER
Section 17.16 *RESTART
Section 18.8 *SFILM
Section 18.22 *SOLID SECTION
Section 18.27 *SPECIFIC HEAT
Section 18.30 *SRADIATE
Section 18.36 *STEP
Section 18.47 *SURFACE
Section 18.51 *SURFACE OUTPUT
Section 18.59 *SYSTEM
Section 19.11 *TRANSFORM
Section 19.12 *TRANSPORT EQUATION SOLVER
Section 19.17 *TURBULENCE MODEL
Section 21.5 *VISCOSITY

## Abaqus/CAE

Section 1.2 *ACOUSTIC MEDIUM
Section 1.3 *ACOUSTIC WAVE FORMULATION

Section 1.4 *ADAPTIVE MESH
Section 1.5 *ADAPTIVE MESH CONSTRAINT
Section 1.6 *ADAPTIVE MESH CONTROLS
Section 1.8 *ADJUST
Section 1.9 *AMPLITUDE
Section 1.10 *ANISOTROPIC HYPERELASTIC
Section 1.11 *ANNEAL
Section 1.12 *ANNEAL TEMPERATURE
Section 1.14 *ASSEMBLY
Section 2.2 *BASELINE CORRECTION
Section 2.4 *BEAM FLUID INERTIA
Section 2.5 *BEAM GENERAL SECTION
Section 2.6 *BEAM SECTION
Section 2.9 *BIAXIAL TEST DATA
Section 2.12 *BOUNDARY
Section 2.13 *BRITTLE CRACKING
Section 2.14 *BRITTLE FAILURE
Section 2.15 *BRITTLE SHEAR
Section 2.16 *BUCKLE
Section 2.20 *BULK VISCOSITY
Section 3.2 *CAP CREEP
Section 3.3 *CAP HARDENING
Section 3.4 *CAP PLASTICITY
Section 3.5 *CAPACITY
Section 3.6 *CAST IRON COMPRESSION HARDENING
Section 3.7 *CAST IRON PLASTICITY
Section 3.8 *CAST IRON TENSION HARDENING
Section 3.9 *CAVITY DEFINITION
Section 3.10 *CECHARGE
Section 3.11 *CECURRENT
Section 3.12 *CENTROID
Section 3.13 *CFD
Section 3.14 *CFILM
Section 3.15 *CFLOW
Section 3.16 *CFLUX
Section 3.17 *CHANGE FRICTION
Section 3.19 *CLAY HARDENING
Section 3.20 *CLAY PLASTICITY
Section 3.21 *CLEARANCE
Section 3.22 *CLOAD
Section 3.23 *COHESIVE BEHAVIOR
Section 3.24 *COHESIVE SECTION

Section 3.25 *COMBINED TEST DATA
Section 3.26 *COMPLEX FREQUENCY
Section 3.28 *CONCRETE
Section 3.29 *CONCRETE COMPRESSION DAMAGE
Section 3.30 *CONCRETE COMPRESSION HARDENING
Section 3.31 *CONCRETE DAMAGED PLASTICITY
Section 3.32 *CONCRETE TENSION DAMAGE
Section 3.33 *CONCRETE TENSION STIFFENING
Section 3.34 *CONDUCTIVITY
Section 3.35 *CONNECTOR BEHAVIOR
Section 3.36 *CONNECTOR CONSTITUTIVE REFERENCE
Section 3.37 *CONNECTOR DAMAGE EVOLUTION
Section 3.38 *CONNECTOR DAMAGE INITIATION
Section 3.39 *CONNECTOR DAMPING
Section 3.40 *CONNECTOR DERIVED COMPONENT
Section 3.41 *CONNECTOR ELASTICITY
Section 3.42 *CONNECTOR FAILURE
Section 3.43 *CONNECTOR FRICTION
Section 3.44 *CONNECTOR HARDENING
Section 3.45 *CONNECTOR LOAD
Section 3.46 *CONNECTOR LOCK
Section 3.47 *CONNECTOR MOTION
Section 3.48 *CONNECTOR PLASTICITY
Section 3.49 *CONNECTOR POTENTIAL
Section 3.50 *CONNECTOR SECTION
Section 3.51 *CONNECTOR STOP
Section 3.52 *CONNECTOR UNIAXIAL BEHAVIOR
Section 3.54 *CONTACT
Section 3.57 *CONTACT CONTROLS
Section 3.59 *CONTACT DAMPING
Section 3.60 *CONTACT EXCLUSIONS
Section 3.61 *CONTACT FILE
Section 3.62 *CONTACT FORMULATION
Section 3.63 *CONTACT INCLUSIONS
Section 3.64 *CONTACT INITIALIZATION ASSIGNMENT
Section 3.65 *CONTACT INITIALIZATION DATA
Section 3.66 *CONTACT INTERFERENCE
Section 3.67 *CONTACT OUTPUT
Section 3.68 *CONTACT PAIR
Section 3.71 *CONTACT PROPERTY ASSIGNMENT
Section 3.73 *CONTACT STABILIZATION
Section 3.74 *CONTOUR INTEGRAL

Section 3.75 *CONTROLS
Section 3.78 *CO-SIMULATION
Section 3.79 *CO-SIMULATION CONTROLS
Section 3.80 *CO-SIMULATION REGION
Section 3.81 *COUPLED TEMPERATURE-DISPLACEMENT
Section 3.82 *COUPLED THERMAL-ELECTRICAL
Section 3.83 *COUPLING
Section 3.84 *CRADIATE
Section 3.85 *CREEP
Section 3.87 *CRUSHABLE FOAM
Section 3.88 *CRUSHABLE FOAM HARDENING
Section 3.89 *CYCLED PLASTIC
Section 3.90 *CYCLIC
Section 3.91 *CYCLIC HARDENING
Section 3.92 *CYCLIC SYMMETRY MODEL
Section 4.2 *D EM POTENTIAL
Section 4.3 *DAMAGE EVOLUTION
Section 4.4 *DAMAGE INITIATION
Section 4.5 *DAMAGE STABILIZATION
Section 4.6 *DAMPING
Section 4.7 *DAMPING CONTROLS
Section 4.8 *DASHPOT
Section 4.9 *DEBOND
Section 4.10 *DECHARGE
Section 4.11 *DECURRENT
Section 4.12 *DEFORMATION PLASTICITY
Section 4.13 *DENSITY
Section 4.14 *DEPVAR
Section 4.18 *DETONATION POINT
Section 4.19 *DFLOW
Section 4.21 *DFLUX
Section 4.23 *DIELECTRIC
Section 4.24 *DIFFUSIVITY
Section 4.25 *DIRECT CYCLIC
Section 4.27 *DISPLAY BODY
Section 4.28 *DISTRIBUTING
Section 4.29 *DISTRIBUTING COUPLING
Section 4.30 *DISTRIBUTION
Section 4.31 *DISTRIBUTION TABLE
Section 4.32 *DLOAD
Section 4.35 *DRUCKER PRAGER
Section 4.36 *DRUCKER PRAGER CREEP

Section 4.37 *DRUCKER PRAGER HARDENING
Section 4.39 *DSECHARGE
Section 4.40 *DSECURRENT
Section 4.41 *DSFLOW
Section 4.42 *DSFLUX
Section 4.43 *DSLOAD
Section 4.44 *DYNAMIC
Section 4.45 *DYNAMIC TEMPERATURE-DISPLACEMENT
Section 5.1 *EL FILE
Section 5.3 *ELASTIC
Section 5.4 *ELCOPY
Section 5.5 *ELECTRICAL CONDUCTIVITY
Section 5.6 *ELECTROMAGNETIC
Section 5.7 *ELEMENT
Section 5.10 *ELEMENT OUTPUT
Section 5.12 *ELGEN
Section 5.13 *ELSET
Section 5.14 *EMBEDDED ELEMENT
Section 5.15 *EMISSIVITY
Section 5.16 *END ASSEMBLY
Section 5.17 *END INSTANCE
Section 5.18 *END LOAD CASE
Section 5.19 *END PART
Section 5.20 *END STEP
Section 5.22 *ENERGY FILE
Section 5.23 *ENERGY OUTPUT
Section 5.25 *ENRICHMENT
Section 5.26 *ENRICHMENT ACTIVATION
Section 5.27 *EOS
Section 5.28 *EOS COMPACTION
Section 5.30 *EQUATION
Section 5.31 *EULERIAN BOUNDARY
Section 5.32 *EULERIAN MESH MOTION
Section 5.33 *EULERIAN SECTION
Section 5.34 *EXPANSION
Section 6.2 *FAIL STRAIN
Section 6.3 *FAIL STRESS
Section 6.4 *FAILURE RATIOS
Section 6.5 *FASTENER
Section 6.6 *FASTENER PROPERTY
Section 6.8 *FILE FORMAT
Section 6.9 *FILE OUTPUT

Section 6.10 *FILM
Section 6.11 *FILM PROPERTY
Section 6.12 *FILTER
Section 6.16 *FLUID BEHAVIOR
Section 6.17 *FLUID BOUNDARY
Section 6.18 *FLUID BULK MODULUS
Section 6.19 *FLUID CAVITY
Section 6.20 *FLUID DENSITY
Section 6.21 *FLUID EXCHANGE
Section 6.23 *FLUID EXCHANGE PROPERTY
Section 6.24 *FLUID EXPANSION
Section 6.30 *FLUID LEAKOFF
Section 6.31 *FLUID SECTION
Section 6.32 *FOUNDATION
Section 6.33 *FRACTURE CRITERION
Section 6.35 *FREQUENCY
Section 6.36 *FRICTION
Section 7.1 *GAP
Section 7.2 *GAP CONDUCTANCE
Section 7.4 *GAP FLOW
Section 7.5 *GAP HEAT GENERATION
Section 7.6 *GAP RADIATION
Section 7.7 *GAS SPECIFIC HEAT
Section 7.8 *GASKET BEHAVIOR
Section 7.9 *GASKET CONTACT AREA
Section 7.10 *GASKET ELASTICITY
Section 7.11 *GASKET SECTION
Section 7.12 *GASKET THICKNESS BEHAVIOR
Section 7.13 *GEL
Section 7.14 *GEOSTATIC
Section 7.15 *GLOBAL DAMPING
Section 8.1 *HEADING
Section 8.2 *HEAT GENERATION
Section 8.3 *HEAT TRANSFER
Section 8.4 *HEATCAP
Section 8.5 *HOURGLASS STIFFNESS
Section 8.6 *HYPERELASTIC
Section 8.7 *HYPERFOAM
Section 8.8 *HYPOELASTIC
Section 8.9 *HYSTERESIS
Section 9.2 *IMPEDANCE PROPERTY
Section 9.4 *IMPORT

Section 9.8 *INCIDENT WAVE
Section 9.9 *INCIDENT WAVE FLUID PROPERTY
Section 9.10 *INCIDENT WAVE INTERACTION
Section 9.11 *INCIDENT WAVE INTERACTION PROPERTY
Section 9.12 *INCIDENT WAVE PROPERTY
Section 9.14 *INCLUDE
Section 9.15 *INCREMENTATION OUTPUT
Section 9.16 *INELASTIC HEAT FRACTION
Section 9.17 *INERTIA RELIEF
Section 9.18 *INITIAL CONDITIONS
Section 9.19 *INSTANCE
Section 9.20 *INTEGRATED OUTPUT
Section 9.21 *INTEGRATED OUTPUT SECTION
Section 9.22 *INTERFACE
Section 9.23 *ITS
Section 10.1 *JOINT
Section 10.5 *JOULE HEAT FRACTION
Section 11.1 *KAPPA
Section 11.2 *KINEMATIC
Section 11.3 *KINEMATIC COUPLING
Section 12.1 *LATENT HEAT
Section 12.2 *LOAD CASE
Section 12.4 *LOW DENSITY FOAM
Section 13.1 *MAGNETIC PERMEABILITY
Section 13.4 *MASS
Section 13.6 *MASS DIFFUSION
Section 13.8 *MATERIAL
Section 13.16 *MEMBRANE SECTION
Section 13.17 *MODAL DAMPING
Section 13.18 *MODAL DYNAMIC
Section 13.19 *MODAL FILE
Section 13.20 *MODAL OUTPUT
Section 13.22 *MODEL CHANGE
Section 13.23 *MOHR COULOMB
Section 13.24 *MOHR COULOMB HARDENING
Section 13.25 *MOISTURE SWELLING
Section 13.26 *MOLECULAR WEIGHT
Section 13.27 *MOMENTUM EQUATION SOLVER
Section 13.28 *MONITOR
Section 13.30 *MPC
Section 13.31 *MULLINS EFFECT
Section 14.1 *NCOPY

Section 14.2 *NFILL
Section 14.3 *NGEN
Section 14.4 *NMAP
Section 14.5 *NO COMPRESSION
Section 14.6 *NO TENSION
Section 14.8 *NODAL THICKNESS
Section 14.9 *NODE
Section 14.10 *NODE FILE
Section 14.11 *NODE OUTPUT
Section 14.15 *NONSTRUCTURAL MASS
Section 14.17 *NSET
Section 15.1 *ORIENTATION
Section 15.2 *ORNL
Section 15.3 *OUTPUT
Section 16.4 *PART
Section 16.5 *PERIODIC
Section 16.8 *PERMEABILITY
Section 16.9 *PHYSICAL CONSTANTS
Section 16.10 *PIEZOELECTRIC
Section 16.13 *PLANAR TEST DATA
Section 16.14 *PLASTIC
Section 16.19 *POROUS BULK MODULI
Section 16.20 *POROUS ELASTIC
Section 16.21 *POROUS FAILURE CRITERIA
Section 16.22 *POROUS METAL PLASTICITY
Section 16.24 *POTENTIAL
Section 16.25 *PREPRINT
Section 16.26 *PRESSURE EQUATION SOLVER
Section 16.27 *PRESSURE PENETRATION
Section 16.30 *PRE-TENSION SECTION
Section 16.31 *PRINT
Section 17.1 *RADIATE
Section 17.2 *RADIATION FILE
Section 17.3 *RADIATION OUTPUT
Section 17.5 *RADIATION SYMMETRY
Section 17.6 *RADIATION VIEW FACTOR
Section 17.7 *RANDOM RESPONSE
Section 17.8 *RATE DEPENDENT
Section 17.9 *RATIOS
Section 17.10 *REACTION RATE
Section 17.12 *REBAR LAYER
Section 17.13 *REFLECTION

Section 17.15 *RESPONSE SPECTRUM
Section 17.16 *RESTART
Section 17.17 *RETAINED NODAL DOFS
Section 17.18 *RIGID BODY
Section 17.19 *RIGID SURFACE
Section 17.20 *ROTARY INERTIA
Section 18.1 *SECTION CONTROLS
Section 18.4 *SECTION POINTS
Section 18.6 *SELECT CYCLIC SYMMETRY MODES
Section 18.7 *SELECT EIGENMODES
Section 18.8 *SFILM
Section 18.10 *SHEAR CENTER
Section 18.12 *SHEAR RETENTION
Section 18.13 *SHEAR TEST DATA
Section 18.14 *SHELL GENERAL SECTION
Section 18.15 *SHELL SECTION
Section 18.16 *SHELL TO SOLID COUPLING
Section 18.17 *SIMPEDANCE
Section 18.18 *SIMPLE SHEAR TEST DATA
Section 18.19 *SLIDE LINE
Section 18.20 *SLOAD
Section 18.21 *SOILS
Section 18.22 *SOLID SECTION
Section 18.23 *SOLUBILITY
Section 18.24 *SOLUTION TECHNIQUE
Section 18.25 *SOLVER CONTROLS
Section 18.26 *SORPTION
Section 18.27 *SPECIFIC HEAT
Section 18.28 *SPECTRUM
Section 18.29 *SPRING
Section 18.30 *SRADIATE
Section 18.31 *STATIC
Section 18.34 *STEADY STATE DYNAMICS
Section 18.36 *STEP
Section 18.38 *SUBMODEL
Section 18.42 *SUBSTRUCTURE GENERATE
Section 18.43 *SUBSTRUCTURE LOAD CASE
Section 18.47 *SURFACE
Section 18.48 *SURFACE BEHAVIOR
Section 18.50 *SURFACE INTERACTION
Section 18.52 *SURFACE PROPERTY
Section 18.53 *SURFACE PROPERTY ASSIGNMENT

Section 18.54 *SURFACE SECTION
Section 18.55 *SURFACE SMOOTHING
Section 18.56 *SWELLING
Section 18.59 *SYSTEM
Section 19.1 *TEMPERATURE
Section 19.3 *TENSION CUTOFF
Section 19.4 *TENSION STIFFENING
Section 19.6 *TIE
Section 19.7 *TIME POINTS
Section 19.11 *TRANSFORM
Section 19.12 *TRANSPORT EQUATION SOLVER
Section 19.14 *TRANSVERSE SHEAR STIFFNESS
Section 19.15 *TRIAXIAL TEST DATA
Section 19.16 *TRS
Section 19.17 *TURBULENCE MODEL
Section 20.1 *UEL PROPERTY
Section 20.2 *UNDEX CHARGE PROPERTY
Section 20.4 *UNIAXIAL TEST DATA
Section 20.6 *USER DEFINED FIELD
Section 20.7 *USER ELEMENT
Section 20.8 *USER MATERIAL
Section 20.9 *USER OUTPUT VARIABLES
Section 21.3 *VISCO
Section 21.4 *VISCOELASTIC
Section 21.5 *VISCOSITY
Section 21.6 *VISCOUS
Section 21.7 *VOID NUCLEATION
Section 21.8 *VOLUMETRIC TEST DATA

## Abaqus/AMS

Section 6.35 *FREQUENCY

## Abaqus/Aqua

Section 1.13 *AQUA
Section 3.1 *C ADDED MASS
Section 3.22 *CLOAD
Section 4.1 *D ADDED MASS
Section 4.32 *DLOAD
Section 9.18 *INITIAL CONDITIONS
Section 18.54 *SURFACE SECTION
Section 22.1 *WAVE

Section 22.2 *WIND

## Abaqus/Design

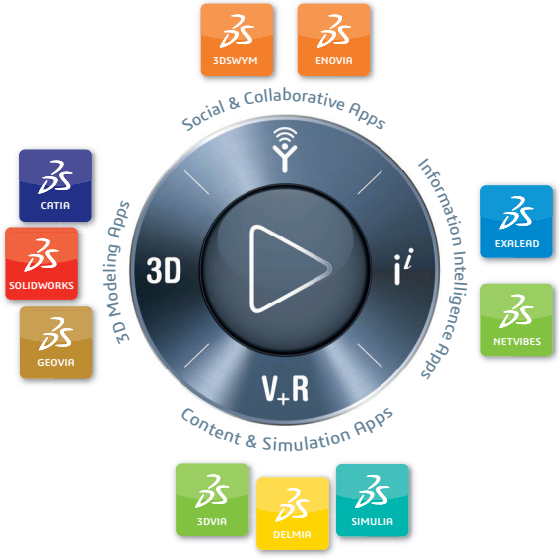
Section 3.72 *CONTACT RESPONSE
Section 4.15 *DESIGN GRADIENT
Section 4.16 *DESIGN PARAMETER
Section 4.17 *DESIGN RESPONSE
Section 4.38 *DSA CONTROLS
Section 5.11 *ELEMENT RESPONSE
Section 14.13 *NODE RESPONSE

## About SIMULIA

Dassault Systèmes SIMULIA applications, including Abaqus, Isight, Tosca, and Simulation Lifecycle Management, enable users to leverage physics-based simulation and high-performance computing to explore real-world behavior of products, nature, and life. As an integral part of Dassault Systèmes' **3D**EXPERIENCE platform, SIMULIA applications accelerate the process of making highly informed, mission-critical design and engineering decisions before committing to costly and time-consuming physical prototypes. **www.3ds.com/simulia**

## Our **3D**EXPERIENCE® Platform powers our brand applications, serving 12 industries, and provides a rich portfolio of industry solution experiences.

Dassault Systèmes, the **3D**EXPERIENCE Company, provides business and people with virtual universes to imagine sustainable innovations. Its world-leading solutions transform the way products are designed, produced, and supported. Dassault Systèmes' collaborative solutions foster social innovation, expanding possibilities for the virtual world to improve the real world. The group brings value to over 170,000 customers of all sizes in all industries in more than 140 countries. For more information, visit **www.3ds.com**.




© Dassault Systèmes 2014, all rights reserved. 3DEXPERIENCE®, CATIA, SolidWorks, ENOVIA, SIMULIA, DELMIA, 3DVIA, 3DSwym, EXALEAD, and Netvibes are registered trademarks of Dassault Systèmes or its subsidiaries in the US and/or other countries.

DASSAULT SYSTEMES | The **3D**EXPERIENCE Company

**Europe/Middle East/Africa**
Dassault Systèmes
10, rue Marcel Dassault
CS 40501
78946 Vélizy-Villacoublay Cedex
France

**Asia-Pacific**
Dassault Systèmes
Pier City Shibaura Bldg 10F
3-18-1 Kaigan, Minato-Ku
Tokyo 108-002
Japan

**Americas**
Dassault Systèmes
175 Wyman Street
Waltham, Massachusetts
02451-1223
USA