Abaqus 6.10

Keywords Reference Manual

Volume I: A–H

SIMULIA

Abaqus Keywords

Reference Manual

Volume I

Legal Notices

Locations

SIMULIA Worldwide Headquarters	Rising Sun Mills, 166 Valley Street, Providence, RI 02909–2499, Tel: +1 401 276 4400, Fax: +1 401 276 4408, simulia.support@3ds.com http://www.simulia.com
SIMULIA European Headquarters	Gaetano Martinolaan 95, P. O. Box 1637, 6201 BP Maastricht, The Netherlands, Tel: +31 43 356 6906, Fax: +31 43 356 6908, simulia.europe.info@3ds.com

Technical Support Centers

United States	Fremont, CA, Tel: +1 510 794 5891, simulia.west.support@3ds.com
	West Lafayette, IN, Tel: +1 765 497 1373, simulia.central.support@3ds.com
	Northville, MI, Tel: +1 248 349 4669, simulia.greatlakes.info@3ds.com
	Woodbury, MN, Tel: +1 612 424 9044, simulia.central.support@3ds.com
	Beachwood, OH, Tel: +1 216 378 1070, simulia.erie.info@3ds.com
	West Chester, OH, Tel: +1 513 275 1430, simulia.central.support@3ds.com
	Warwick, RI, Tel: +1 401 739 3637, simulia.east.support@3ds.com
	Lewisville, TX, Tel: +1 972 221 6500, simulia.south.info@3ds.com
Australia	Richmond VIC, Tel: +61 3 9421 2900, simulia.au.support@3ds.com
Austria	Vienna, Tel: +43 1 22 707 200, simulia.at.info@3ds.com
Benelux	Huizen, The Netherlands, Tel: +31 35 52 58 424, simulia.benelux.support@3ds.com
Canada	Toronto, ON, Tel: +1 416 402 2219, simulia.greatlakes.info@3ds.com
China	Beijing, P. R. China, Tel: +8610 6536 2288, simulia.cn.support@3ds.com
	Shanghai, P. R. China, Tel: +8621 3856 8000, simulia.cn.support@3ds.com
Czech & Slovak Republics	Synerma s. r. o., Psáry, Prague-West, Tel: +420 603 145 769, abaqus@synerma.cz
Finland	Vantaa, Tel: +358 46 712 2247, simulia.nordic.info@3ds.com
France	Velizy Villacoublay Cedex, Tel: +33 1 61 62 72 72, simulia.fr.support@3ds.com
Germany	Aachen, Tel: +49 241 474 01 0, simulia.de.info@3ds.com
	Munich, Tel: +49 89 543 48 77 0, simulia.de.info@3ds.com
Greece	3 Dimensional Data Systems, Crete, Tel: +30 2821040012, support@3dds.gr
India	Chennai, Tamil Nadu, Tel: +91 44 43443000, simulia.in.info@3ds.com
Israel	ADCOM, Givataim, Tel: +972 3 7325311, shmulik.keidar@adcomsim.co.il
Italy	Lainate MI, Tel: +39 02 39211211, simulia.ity.info@3ds.com
Japan	Tokyo, Tel: +81 3 5442 6300, simulia.tokyo.support@3ds.com
	Osaka, Tel: +81 6 4803 5020, simulia.osaka.support@3ds.com
	Yokohama-shi, Kanagawa, Tel: +81 45 470 9381, isight.jp.info@3ds.com
Korea	Mapo-Gu, Seoul, Tel: +82 2 785 6707/8, simulia.kr.info@3ds.com
Latin America	Puerto Madero, Buenos Aires, Tel: +54 11 4312 8700, Horacio.Burbridge@3ds.com
Malaysia	WorleyParsons Advanced Analysis, Kuala Lumpur, Tel: +603 2039 9000, abaqus.my@worleyparsons.com
New Zealand	Matrix Applied Computing Ltd., Auckland, Tel: +64 9 623 1223, abaqus-tech@matrix.co.nz
Poland	BudSoft Sp. z o.o., Poznań, Tel: +48 61 8508 466, info@budsoft.com.pl
Russia, Belarus & Ukraine	TESIS Ltd., Moscow, Tel: +7 495 612 44 22, info@tesis.com.ru
Scandinavia	Västerås, Sweden, Tel: +46 21 150870, simulia.nordic.info@3ds.com
Singapore	WorleyParsons Advanced Analysis, Singapore, Tel: +65 6735 8444, abaqus.sg@worleyparsons.com
South Africa	Finite Element Analysis Services (Pty) Ltd., Parklands, Tel: +27 21 556 6462, feas@feas.co.za
Spain & Portugal	Principia Ingenieros Consultores, S.A., Madrid, Tel: +34 91 209 1482, simulia@principia.es
Taiwan	Simutech Solution Corporation, Taipei, R.O.C., Tel: +886 2 2507 9550, lucille@simutech.com.tw
Thailand	WorleyParsons Advanced Analysis, Singapore, Tel: +65 6735 8444, abaqus.sg@worleyparsons.com
Turkey	A-Ztech Ltd., Istanbul, Tel: +90 216 361 8850, info@a-ztech.com.tr
United Kingdom	Warrington, Tel: +44 1 925 830900, simulia.uk.info@3ds.com
	Sevenoaks, Tel: +44 1 732 834930, simulia.uk.info@3ds.com

Complete contact information is available at http://www.simulia.com/locations/locations.html.

Preface

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

Support

Both technical engineering 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 network of local support offices. Regional contact information is listed in the front of each Abaqus manual and is accessible from the **Locations** page at www.simulia.com.

SIMULIA Online Support System

The SIMULIA Online Support System (SOSS) provides a knowledge database of SIMULIA Answers. The SIMULIA Answers are solutions to questions that we have had to answer or guidelines on how to use Abaqus, SIMULIA SLM, Isight, and other SIMULIA products. You can also submit new requests for support in the SOSS. All support incidents are tracked in the SOSS. If you are contacting us by means outside the SOSS to discuss an existing support problem and you know the incident number, please mention it so that we can consult the database to see what the latest action has been.

To use the SOSS, you need to register with the system. Visit the **My Support** page at www.simulia.com to register.

Many questions about Abaqus can also be answered by visiting the **Products** page and the **Support** page at www.simulia.com.

Anonymous ftp site

To facilitate data transfer with SIMULIA, an anonymous ftp account is available on the computer ftp.simulia.com. Login as user anonymous, and type your e-mail address as your password. Contact support before placing files on the site.

Training

All offices and representatives offer regularly scheduled public training classes. 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 www.simulia.com or call your local office or representative.

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.simulia.com. Complaints should be addressed by contacting your local office or through www.simulia.com by visiting the **Quality Assurance** section of the **Support** page.

Contents

Volume I

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
*AMPLITUDE 1.7
*ANISOTROPIC HYPERELASTIC 1.8
*ANNEAL 1.9
*ANNEAL TEMPERATURE 1.10
*AQUA 1.11
*ASSEMBLY 1.12
*ASYMMETRIC-AXISYMMETRIC 1.13
*AXIAL 1.14

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
*BIAXIAL TEST DATA 2.8
*BLOCKAGE 2.9
*BOND 2.10
*BOUNDARY 2.11
*BRITTLE CRACKING 2.12
*BRITTLE FAILURE 2.13
*BRITTLE SHEAR 2.14
*BUCKLE 2.15
*BUCKLING ENVELOPE 2.16
*BUCKLING LENGTH 2.17
*BUCKLING REDUCTION FACTORS 2.18
*BULK VISCOSITY 2.19

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
*CFILM 3.13
*CFLOW 3.14
*CFLUX 3.15
*CHANGE FRICTION 3.16
*CLAY HARDENING 3.17
*CLAY PLASTICITY 3.18
*CLEARANCE 3.19
*CLOAD 3.20
*COHESIVE BEHAVIOR 3.21
*COHESIVE SECTION 3.22
*COMBINED TEST DATA 3.23
*COMPLEX FREQUENCY 3.24
*CONCRETE 3.25
*CONCRETE COMPRESSION DAMAGE 3.26
*CONCRETE COMPRESSION HARDENING 3.27
*CONCRETE DAMAGED PLASTICITY 3.28
*CONCRETE TENSION DAMAGE 3.29
*CONCRETE TENSION STIFFENING 3.30
*CONDUCTIVITY 3.31
*CONNECTOR BEHAVIOR 3.32
*CONNECTOR CONSTITUTIVE REFERENCE 3.33
*CONNECTOR DAMAGE EVOLUTION 3.34
*CONNECTOR DAMAGE INITIATION 3.35
*CONNECTOR DAMPING 3.36
*CONNECTOR DERIVED COMPONENT 3.37
*CONNECTOR ELASTICITY 3.38
*CONNECTOR FAILURE 3.39
*CONNECTOR FRICTION 3.40

*CONNECTOR HARDENING 3.41
*CONNECTOR LOAD 3.42
*CONNECTOR LOCK 3.43
*CONNECTOR MOTION 3.44
*CONNECTOR PLASTICITY 3.45
*CONNECTOR POTENTIAL 3.46
*CONNECTOR SECTION 3.47
*CONNECTOR STOP 3.48
*CONNECTOR UNIAXIAL BEHAVIOR 3.49
*CONSTRAINT CONTROLS 3.50
*CONTACT 3.51
*CONTACT CLEARANCE 3.52
*CONTACT CLEARANCE ASSIGNMENT 3.53
*CONTACT CONTROLS 3.54
*CONTACT CONTROLS ASSIGNMENT 3.55
*CONTACT DAMPING 3.56
*CONTACT EXCLUSIONS 3.57
*CONTACT FILE 3.58
*CONTACT FORMULATION 3.59
*CONTACT INCLUSIONS 3.60
*CONTACT INITIALIZATION ASSIGNMENT 3.61
*CONTACT INITIALIZATION DATA 3.62
*CONTACT INTERFERENCE 3.63
*CONTACT OUTPUT 3.64
*CONTACT PAIR 3.65
*CONTACT PRINT 3.66
*CONTACT PROPERTY ASSIGNMENT 3.67
*CONTACT RESPONSE 3.68
*CONTACT STABILIZATION 3.69
*CONTOUR INTEGRAL 3.70
*CONTROLS 3.71
*CONWEP CHARGE PROPERTY 3.72
*CORRELATION 3.73
*CO-SIMULATION 3.74
*CO-SIMULATION CONTROLS 3.75
*CO-SIMULATION REGION 3.76
*COUPLED TEMPERATURE-DISPLACEMENT 3.77
*COUPLED THERMAL-ELECTRICAL 3.78
*COUPLING 3.79
*CRADIATE 3.80
*CREEP 3.81
*CREEP STRAIN RATE CONTROL 3.82

*CRUSHABLE FOAM 3.83
*CRUSHABLE FOAM HARDENING 3.84
*CYCLED PLASTIC 3.85
*CYCLIC 3.86
*CYCLIC HARDENING 3.87
*CYCLIC SYMMETRY MODEL 3.88

D

*D ADDED MASS 4.1
*DAMAGE EVOLUTION 4.2
*DAMAGE INITIATION 4.3
*DAMAGE STABILIZATION 4.4
*DAMPING 4.5
*DAMPING CONTROLS 4.6
*DASHPOT 4.7
*DEBOND 4.8
*DECHARGE 4.9
*DECURRENT 4.10
*DEFORMATION PLASTICITY 4.11
*DENSITY 4.12
*DEPVAR 4.13
*DESIGN GRADIENT 4.14
*DESIGN PARAMETER 4.15
*DESIGN RESPONSE 4.16
*DETONATION POINT 4.17
*DFLOW 4.18
*DFLUX 4.19
*DIAGNOSTICS 4.20
*DIELECTRIC 4.21
*DIFFUSIVITY 4.22
*DIRECT CYCLIC 4.23
*DISPLAY BODY 4.24
*DISTRIBUTING 4.25
*DISTRIBUTING COUPLING 4.26
*DISTRIBUTION 4.27
*DISTRIBUTION TABLE 4.28
*DLOAD 4.29
*DRAG CHAIN 4.30
*DRUCKER PRAGER 4.31
*DRUCKER PRAGER CREEP 4.32
*DRUCKER PRAGER HARDENING 4.33
*DSA CONTROLS 4.34
*DSECHARGE 4.35

*DSECURRENT 4.36
*DSFLOW 4.37
*DSFLUX 4.38
*DSLOAD 4.39
*DYNAMIC 4.40
*DYNAMIC TEMPERATURE-DISPLACEMENT 4.41

E

*EL FILE 5.1
*EL PRINT 5.2
*ELASTIC 5.3
*ELCOPY 5.4
*ELECTRICAL CONDUCTIVITY 5.5
*ELEMENT 5.6
*ELEMENT MATRIX OUTPUT 5.7
*ELEMENT OUTPUT 5.8
*ELEMENT RESPONSE 5.9
*ELGEN 5.10
*ELSET 5.11
*EMBEDDED ELEMENT 5.12
*EMISSIVITY 5.13
*END ASSEMBLY 5.14
*END INSTANCE 5.15
*END LOAD CASE 5.16
*END PART 5.17
*END STEP 5.18
*ENERGY FILE 5.19
*ENERGY OUTPUT 5.20
*ENERGY PRINT 5.21
*ENRICHMENT 5.22
*ENRICHMENT ACTIVATION 5.23
*EOS 5.24
*EOS COMPACTION 5.25
*EPJOINT 5.26
*EQUATION 5.27
*EULERIAN BOUNDARY 5.28
*EULERIAN MESH MOTION 5.29
*EULERIAN SECTION 5.30
*EXPANSION 5.31
*EXTREME ELEMENT VALUE 5.32
*EXTREME NODE VALUE 5.33
*EXTREME VALUE 5.34

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
*FLUID BEHAVIOR 6.15
*FLUID BULK MODULUS 6.16
*FLUID CAVITY 6.17
*FLUID DENSITY 6.18
*FLUID EXCHANGE 6.19
*FLUID EXCHANGE ACTIVATION 6.20
*FLUID EXCHANGE PROPERTY 6.21
*FLUID EXPANSION 6.22
*FLUID FLUX 6.23
*FLUID INFLATOR 6.24
*FLUID INFLATOR ACTIVATION 6.25
*FLUID INFLATOR MIXTURE 6.26
*FLUID INFLATOR PROPERTY 6.27
*FLUID LEAKOFF 6.28
*FLUID LINK 6.29
*FLUID PROPERTY 6.30
*FOUNDATION 6.31
*FRACTURE CRITERION 6.32
*FRAME SECTION 6.33
*FREQUENCY 6.34
*FRICTION 6.35

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
*GASKET BEHAVIOR 7.7
*GASKET CONTACT AREA 7.8
*GASKET ELASTICITY 7.9
*GASKET SECTION 7.10
*GASKET THICKNESS BEHAVIOR 7.11
*GAS SPECIFIC HEAT 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

Volume II

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

*MAP SOLUTION 13.1
*MASS 13.2
*MASS DIFFUSION 13.3
*MASS FLOW RATE 13.4
*MATERIAL 13.5
*MATRIX 13.6
*MATRIX ASSEMBLE 13.7
*MATRIX GENERATE 13.8
*MATRIX INPUT 13.9
*MEMBRANE SECTION 13.10
*MODAL DAMPING 13.11
*MODAL DYNAMIC 13.12
*MODAL FILE 13.13
*MODAL OUTPUT 13.14
*MODAL PRINT 13.15
*MODEL CHANGE 13.16
*MOHR COULOMB 13.17
*MOHR COULOMB HARDENING 13.18
*MOISTURE SWELLING 13.19
*MOLECULAR WEIGHT 13.20
*MONITOR 13.21
*MOTION 13.22
*MPC 13.23
*MULLINS EFFECT 13.24
*M1 13.25
*M2 13.26

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
*NONSTRUCTURAL MASS 14.14
*NORMAL 14.15
*NSET 14.16

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
*PERMEABILITY 16.6
*PHYSICAL CONSTANTS 16.7
*PIEZOELECTRIC 16.8
*PIPE-SOIL INTERACTION 16.9
*PIPE-SOIL STIFFNESS 16.10
*PLANAR TEST DATA 16.11
*PLASTIC 16.12
*PLASTIC AXIAL 16.13
*PLASTIC M1 16.14
*PLASTIC M2 16.15
*PLASTIC TORQUE 16.16
*POROUS BULK MODULI 16.17
*POROUS ELASTIC 16.18
*POROUS FAILURE CRITERIA 16.19
*POROUS METAL PLASTICITY 16.20
*POST OUTPUT 16.21
*POTENTIAL 16.22
*PREPRINT 16.23
*PRESSURE PENETRATION 16.24

*PRESSURE STRESS	16.25
*PRESTRESS HOLD	16.26
*PRE-TENSION SECTION	16.27
*PRINT	16.28
*PSD-DEFINITION	16.29

R

*RADIATE	17.1
*RADIATION FILE	17.2
*RADIATION OUTPUT	17.3
*RADIATION PRINT	17.4
*RADIATION SYMMETRY	17.5
*RADIATION VIEWFACTOR	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 PROPERTY 18.51
*SURFACE PROPERTY ASSIGNMENT 18.52
*SURFACE SECTION 18.53
*SURFACE SMOOTHING 18.54
*SWELLING 18.55

*SYMMETRIC MODEL GENERATION 18.56
*SYMMETRIC RESULTS TRANSFER 18.57
*SYSTEM 18.58

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 VELOCITY 19.12
*TRANSVERSE SHEAR STIFFNESS 19.13
*TRIAXIAL TEST DATA 19.14
*TRS 19.15

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

W, X, Y, Z

*WAVE 22.1
*WIND 22.2

1.0.1 BROWSING THE Abaqus Keywords Reference Manual

This manual 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 Manual.

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 2.9.1 of the Abaqus Analysis User’s Manual.

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 Manual, 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 3.2.11 of the Abaqus Analysis User’s Manual) 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 3.2.12 of the Abaqus Analysis User’s Manual.

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 Manual

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 30.1.2 of the Abaqus Analysis User's Manual).

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 Manual, 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 $\mathbf{a}$ to point $\mathbf{b}$, where the coordinates of $\mathbf{a}$ and $\mathbf{b}$ 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 $\mathbf{a}$ on the axis of rotation.
4. Global y-component of point $\mathbf{a}$ on the axis of rotation.

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

5. Global z-component of point $\mathbf{a}$ on the axis of rotation.
6. Global x-component of point $\mathbf{b}$ on the axis of rotation.
7. Global y-component of point $\mathbf{b}$ on the axis of rotation.
8. Global z-component of point $\mathbf{b}$ 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 23.3.1 of the Abaqus Analysis User's Manual

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 18.1.2 of the Abaqus Analysis User's Manual, 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 (and only) 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}.)

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

First (and only) 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}.)

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 30.4.5 of the Abaqus Analysis User's Manual

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 Manual
- "ALE adaptive meshing and remapping in Abaqus/Explicit," Section 12.2.3 of the Abaqus Analysis User's Manual
- "Defining ALE adaptive mesh domains in Abaqus/Standard," Section 12.2.6 of the Abaqus Analysis User's Manual
- "ALE adaptive meshing and remapping in Abaqus/Standard," Section 12.2.7 of the Abaqus Analysis User's Manual
- *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 Manual
- "Defining ALE adaptive mesh domains in Abaqus/Standard," Section 12.2.6 of the Abaqus Analysis User's Manual
- "UMESHMOTION," Section 1.1.38 of the Abaqus User Subroutines Reference Manual
- *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 30.1.2 of the Abaqus Analysis User's Manual).

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 Manual
- "ALE adaptive meshing and remapping in Abaqus/Explicit," Section 12.2.3 of the Abaqus Analysis User's Manual
- "Defining ALE adaptive mesh domains in Abaqus/Standard," Section 12.2.6 of the Abaqus Analysis User's Manual
- "ALE adaptive meshing and remapping in Abaqus/Standard," Section 12.2.7 of the Abaqus Analysis User's Manual
- *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, α_C. The default value is $\alpha_C = 1$.

GEOMETRIC ENHANCEMENT

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

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

INITIAL FEATURE ANGLE

Set this parameter equal to the initial geometric feature angle, θ_I, in degrees ($0^\circ \leq \theta_I \leq 180^\circ$). 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, θ_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 θ_C. When adaptive mesh constraints are applied to nodes that are part of a Lagrangian or active geometric edge, the analysis will terminate if the angle between the prescribed constraint and the plane perpendicular to the edge becomes less than θ_C.

MESHING PREDICTOR

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

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

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

MOMENTUM ADVECTION

This parameter applies only to Abaqus/Explicit analyses.

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

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

RESET

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

SMOOTHING OBJECTIVE

This parameter applies only to Abaqus/Explicit analyses.

Set SMOOTHING OBJECTIVE=UNIFORM (default if the adaptive mesh domain has no Eulerian boundary regions in explicit dynamic analysis) to perform adaptive meshing that minimizes element distortion and improves element aspect ratios at the expense of diffusing initial mesh gradation. This objective is recommended for problems with moderate to large overall deformation.

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

TRANSITION FEATURE ANGLE

Set this parameter equal to the transition geometric feature angle, θ_T, in degrees ($0^\circ \le \theta_T \le 180^\circ$). 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 *AMPLITUDE: Define an amplitude curve.

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

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Model, Step

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

Reference:

- "Amplitude curves," Section 30.1.2 of the Abaqus Analysis User's Manual

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 30.1.2 of the Abaqus Analysis User's Manual.

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

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 Manual, 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.

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=USER.

SCALEY

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

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=USER.

SHIFTY

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

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 Manual, for a discussion of these time measures.

VALUE

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

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

Required parameter for DEFINITION=EQUALLY SPACED:

FIXED INTERVAL

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

Optional parameter for DEFINITION=EQUALLY SPACED:

BEGIN

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

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

SMOOTH

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

Optional parameter for DEFINITION=USER:

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 with four data points (eight entries) per each 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 with one data pair (two entries) per each 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 each line:

First line:

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

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

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

First line:

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

Second line:

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

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

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

First (and only) line:

1. A_0.
2. A.
3. t_0.
4. ω_1.
5. ω_2.

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

First (and only) line:

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

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

First (and only) line:

1. Initial amplitude value (default = 1.0).
2. Minimum amplitude value (default = 0.1).
3. Maximum amplitude value (default = 1000.).

Data line to define smooth step data (DEFINITION=SMOOTH STEP):

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 line to define bubble loading (DEFINITION=BUBBLE):

First line:

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

Second line:

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

Third line:

1. Acceleration due to gravity, g.

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

Fourth line:

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

There are no data lines if DEFINITION=USER.

1.8 *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

Type: Model data

Level: Model

References:

- "Anisotropic hyperelastic behavior," Section 19.5.3 of the Abaqus Analysis User's Manual
- "UANISOHYPER_STRAIN," Section 1.1.21 of the Abaqus User Subroutines Reference Manual
- "UANISOHYPER_INV," Section 1.1.20 of the Abaqus User Subroutines Reference Manual
- "VUANISOHYPER_STRAIN," Section 1.2.9 of the Abaqus User Subroutines Reference Manual
- "VUANISOHYPER_INV," Section 1.2.8 of the Abaqus User Subroutines Reference Manual

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 18.1.2 of the Abaqus Analysis User's Manual, 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 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 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 FL^{-2}.)
2. D. (Units of $\mathrm{F}^{-1}\mathrm{L}^{2}$.)
3. k_1. (Units of FL^{-2}.)
4. k_2.
5. Fiber dispersion parameter κ ($0 \leq \kappa \leq 1/3$).
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.

1.9 *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 Manual

Optional parameter:

TEMPERATURE

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

There are no data lines associated with this option.

1.10 *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 20.2.5 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines to define the annealing temperature:

First line:

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

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

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

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

1.11 *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 Manual

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 Manual, 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 Manual, for a description of how to define a property as a function of multiple independent variables.

1.12 *ASSEMBLY: Begin an assembly definition.

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

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Assembly module

References:

- *END ASSEMBLY
- "Defining an assembly," Section 2.9.1 of the Abaqus Analysis User's Manual

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.

1.13 *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 32.1.1 of the Abaqus Analysis User's Manual
- "Contact modeling if asymmetric-axisymmetric elements are present," Section 32.3.10 of the Abaqus Analysis User's Manual
- *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.

1.14 *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 26.3.7 of the Abaqus Analysis User's Manual

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 18.1.2 of the Abaqus Analysis User's Manual, 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).

Product: Abaqus/Standard

Type: History data

Level: Step

References:

- “Natural frequency extraction,” Section 6.3.5 of the Abaqus Analysis User’s Manual
- “Transient modal dynamic analysis,” Section 6.3.7 of the Abaqus Analysis User’s Manual
- “Mode-based steady-state dynamic analysis,” Section 6.3.8 of the Abaqus Analysis User’s Manual
- “Random response analysis,” Section 6.3.11 of the Abaqus Analysis User’s Manual

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.

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 30.1.2 of the Abaqus Analysis User's Manual

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 26.3.3 of the Abaqus Analysis User's Manual
- "Beam section behavior," Section 26.3.5 of the Abaqus Analysis User's Manual

Optional parameters:

ALPHA

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

COMPOSITE

This parameter applies only to Abaqus/Standard analyses.

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

Data line to define additional beam inertia:

First line:

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

6. Rotary inertia around the center of mass about the 2-axis in the local inertia system, I_{22}. Only relevant for beams in space; otherwise, leave blank.
7. Product of inertia, I_{12}. Only relevant for beams in space; otherwise, leave blank.

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

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

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

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 26.3.7 of the Abaqus Analysis User's Manual
- "Beam modeling: overview," Section 26.3.1 of the Abaqus Analysis User's Manual

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.

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 26.3.9 of the Abaqus Analysis User's Manual). 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.

ZERO

This parameter cannot be used when SECTION=MESHED.

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

Data lines for SECTION=GENERAL:

First line:

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

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

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

The entries on this line must be (0, 0, -1) for planar beams. The default for beams in space is (0, 0, -1) if the first beam section axis is not defined by an additional node in the element's connectivity. See "Beam element cross-section orientation," Section 26.3.4 of the Abaqus Analysis User's Manual, 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=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 26.3.4 of the Abaqus Analysis User's Manual, 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 26.3.4 of the Abaqus Analysis User's Manual, 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 10.5.1 of the Abaqus Analysis User's Manual, 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, (ρA).
2. Rotary inertia about the 1-axis of the section, $(\rho I)_{11}$.
3. Rotary product of inertia, $(\rho I)_{12}$.
4. Rotary inertia about the 2-axis of the section, $(\rho I)_{22}$.
5. Local 1-coordinate of the center of mass, x_{1cm}.
6. Local 2-coordinate of the center of mass, x_{2cm}.

Data lines for BOX, 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 26.3.9 of the Abaqus Analysis User's Manual, 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 26.3.4 of the Abaqus Analysis User's Manual, 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=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 26.3.4 of the Abaqus Analysis User's Manual, 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 26.3.6 of the Abaqus Analysis User's Manual
- "Beam modeling: overview," Section 26.3.1 of the Abaqus Analysis User's Manual
- "Pipes and pipebends with deforming cross-sections: elbow elements," Section 26.5.1 of the Abaqus Analysis User's Manual

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 26.3.9 of the Abaqus Analysis User's Manual). 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 hollow circular section.
- RECT, for a solid, rectangular section.

- TRAPEZOID, for a trapezoidal section.

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

Optional parameters:

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 26.3.9 of the Abaqus Analysis User's Manual).

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 26.5.1 of the Abaqus Analysis User's Manual.

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 26.3.9 of the Abaqus Analysis User's Manual, 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 26.3.4 of the Abaqus Analysis User's Manual, 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 26.3.9 of the Abaqus Analysis User's Manual.
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 26.3.9 of the Abaqus Analysis User's Manual. This entry is needed only 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 26.3.9 of the Abaqus Analysis User's Manual. 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 26.3.4 of the Abaqus Analysis User's Manual, 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 10.5.1 of the Abaqus Analysis User's Manual
- "Beam section behavior," Section 26.3.5 of the Abaqus Analysis User's Manual
- "Using a general beam section to define the section behavior," Section 26.3.7 of the Abaqus Analysis User's Manual

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

2.8 *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

Using biaxial test data to define a hyperelastic material

References:

- "Hyperelastic behavior of rubberlike materials," Section 19.5.1 of the Abaqus Analysis User's Manual
- *HYPERELASTIC

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, ϵ_B.

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

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

First line:

1. Nominal stress, T_B.
2. Nominal strain, ϵ_B.
3. Nominal lateral strain, ϵ_3. Not needed if the POISSON parameter is specified on the *HYPERELASTIC option or if the *VOLUMETRIC TEST DATA option is used.
4. Temperature, θ.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

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

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

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

Using biaxial test data to define an elastomeric foam

References:

- "Hyperelastic behavior in elastomeric foams," Section 19.5.2 of the Abaqus Analysis User's Manual
- *HYPERFOAM

There are no parameters associated with this option.

Data lines to specify biaxial test data for a hyperfoam:

First line:

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

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

Using biaxial test data to define the Mullins effect material model

References:

- "Mullins effect in rubberlike materials," Section 19.6.1 of the Abaqus Analysis User's Manual
- "Energy dissipation in elastomeric foams," Section 19.6.2 of the Abaqus Analysis User's Manual
- *MULLINS EFFECT

There are no parameters associated with this option.

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

First line:

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

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

2.9 *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 11.6.3 of the Abaqus Analysis User's Manual
- "Mechanical contact properties: overview," Section 33.1.1 of the Abaqus Analysis User's Manual
- "Contact blockage," Section 33.1.4 of the Abaqus Analysis User's Manual
- *SURFACE INTERACTION
- *FLUID EXCHANGE ACTIVATION

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

2.10 *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 33.1.9 of the Abaqus Analysis User's Manual
- *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.

2.11 *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.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Model, Step

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

Prescribing boundary conditions at nodes

References:

- "Defining a model in Abaqus," Section 1.3.1 of the Abaqus Analysis User's Manual
- "Boundary conditions in Abaqus/Standard and Abaqus/Explicit," Section 30.3.1 of the Abaqus Analysis User's Manual
- "DISP," Section 1.1.4 of the Abaqus User Subroutines Reference Manual
- "VDISP," Section 1.2.1 of the Abaqus User Subroutines Reference Manual
- "Defining ALE adaptive mesh domains in Abaqus/Explicit," Section 12.2.2 of the Abaqus Analysis User's Manual

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 30.1.2 of the Abaqus Analysis User's Manual).

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 ("Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). There are two exceptions. The first is when displacement or rotation components are given with TYPE=DISPLACEMENT, for which the default is always a RAMP function. The second is when displacement or rotation components in a static step or in a dynamic step with APPLICATION=QUASI-STATIC are given with TYPE=VELOCITY, for which the default is always a STEP function.

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

In an Abaqus/Standard dynamic or modal dynamic procedure, amplitude curves specified for TYPE=DISPLACEMENT or TYPE=VELOCITY will be smoothed automatically. In an Abaqus/Explicit analysis, the user must request that such amplitude curves are smoothed. For more information, see "Amplitude curves," Section 30.1.2 of the Abaqus Analysis User's Manual.

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.

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 Manual, 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.

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 30.3.1 of the Abaqus Analysis User’s Manual, for details.

Set TYPE=VELOCITY to give a velocity history. Velocity histories can be specified in static analyses in Abaqus/Standard, as discussed in “Prescribing large rotations” in “Boundary conditions in Abaqus/Standard and Abaqus/Explicit," Section 30.3.1 of the Abaqus Analysis User's Manual. In this case the default variation is STEP.

Set TYPE=ACCELERATION to give an acceleration history. Acceleration histories should not be used in static analysis steps in Abaqus/Standard.

If amplitude functions are specified as piecewise linear functions in Abaqus/Explicit and a displacement history is used, there will be a jump in the velocity and a spike in the acceleration at points on the curve where the curve changes slope. This will result in a "noisy" solution. If possible, use *AMPLITUDE, DEFINITION=SMOOTH STEP; *AMPLITUDE, SMOOTH; or *BOUNDARY, TYPE=VELOCITY or TYPE=ACCELERATION. For TYPE=ACCELERATION the value of the initial velocity (given in *INITIAL CONDITIONS, TYPE=VELOCITY) must be specified to obtain the correct displacement history.

USER

This parameter cannot be used with the FIXED parameter.

For Abaqus/Standard include this parameter to indicate that any nonzero magnitudes associated with variables prescribed through this option can be redefined in user subroutine **DISP**. Any magnitudes defined on the data lines of the option (and possibly modified by the AMPLITUDE parameter) will be passed into user subroutine **DISP** and can be redefined in subroutine **DISP**. The value of the TYPE parameter is ignored when this option is used.

For Abaqus/Explicit include this parameter to indicate that the boundary value associated with variables prescribed through this option are to be defined in user subroutine **VDISP**. Any magnitudes defined on the data lines of the option are ignored and the amplitude, if the AMPLITUDE parameter is included, is passed into the **VDISP** routine for your usage. The type of user prescribed variable in subroutine **VDISP** is determined by the TYPE parameter. The NAME parameter can be used in user subroutine **VDISP** to distinguish multiple boundary conditions. Only translational and rotational degrees of freedom are supported for user-prescribed boundary conditions.

Optional, mutually exclusive parameters for matrix generation and direct-solution, steady-state dynamics analysis (history data only):

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the boundary condition.

REAL

Include this parameter (default) to define the real (in-phase) part of the part of the boundary condition.

Data lines to define zero-valued boundary conditions using the "type" format (model data only):

First line:

1. Node number or node set label.

2. Label specifying the type of boundary condition to be applied (see "Boundary conditions in Abaqus/Standard and Abaqus/Explicit," Section 30.3.1 of the Abaqus Analysis User's Manual). 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:

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 Manual.
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 30.3.1 of the Abaqus Analysis User's Manual). 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.

Defining primary and secondary bases for modal superposition procedures

Reference:

- "Natural frequency extraction," Section 6.3.5 of the Abaqus Analysis User's Manual

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 Manual). 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 Manual). 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 Manual.
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

Reference:

- "Node-based submodeling," Section 10.2.2 of the Abaqus Analysis User's Manual

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 6.1.2 of the Abaqus Analysis User's Manual). 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 Manual.
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.

2.12 *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/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Cracking model for concrete," Section 20.6.2 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

TYPE

Set TYPE=STRAIN (default) to specify the postcracking behavior by entering the postfailure stress-strain relationship directly.

Set TYPE=DISPLACEMENT to define the postcracking behavior by entering the postfailure stress/displacement relationship directly.

Set TYPE=GFI to define the postcracking behavior by entering the failure stress, σ_{tu}^I, and the Mode I fracture energy, G_f^I.

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

First line:

1. Remaining direct stress after cracking, σ_t^I. (Units of FL^{-2}.)

2. Direct cracking 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 cracking strain of 0.0 and gives the failure stress value.

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

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

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

Data lines if the TYPE=DISPLACEMENT parameter is included:

First line:

1. Remaining direct stress after cracking, σ^I_t. (Units of FL^{-2}.)
2. Direct cracking displacement, u^{ck}_n. (Units of L.)
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

The first point at each value of temperature must have a cracking displacement of 0.0 and gives the failure stress value.

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

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

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

Data lines if the TYPE=GFI parameter is included:

First line:

1. Failure stress, σ^I_{tu}. (Units of FL^{-2}.)
2. Mode I fracture energy, G^I_f. (Units of 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.

2.13 *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/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Cracking model for concrete," Section 20.6.2 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, 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.

2.14 *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/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Cracking model for concrete," Section 20.6.2 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, 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, ρ.
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. 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.

2.15 *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 Manual

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.

2.16 *BUCKLING ENVELOPE: Define a nondefault buckling envelope for buckling strut response of frame elements with PIPE sections.

This option is used to define the coefficients characterizing the buckling strut envelope for the buckling strut response of frame elements. It can be used in conjunction with the *FRAME SECTION, SECTION=PIPE, YIELD STRESS=σ^0 option with or without the PINNED parameter.

Product: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- "Frame section behavior," Section 26.4.2 of the Abaqus Analysis User's Manual
- *FRAME SECTION

There are no parameters associated with this option.

Data line to define the buckling strut envelope:

First (and only) data line:

1. ξ, coefficient defining $P_y = \xi\sigma^0 A$ (default value 0.95).
2. γ, coefficient defining the isotropic hardening slope γEA (default value 0.02).
3. α_0, coefficient defining $\alpha = \alpha_0 + \alpha_1 \frac{L}{D}$ (default value 0.03).
4. α_1, coefficient defining $\alpha = \alpha_0 + \alpha_1 \frac{L}{D}$ (default value 0.004).
5. κ, coefficient defining compressive force for discontinuity in buckling envelope (default value 0.28).
6. β, buckling envelope slope coefficient (default value 0.02).
7. ζ, coefficient defining the force axis intercept point (default value $\min(1.0, \frac{5.8}{\xi}(\frac{t}{D})^{0.7})$).

In the above data line A is the cross-section area, σ^0 is a yield stress value, E is Young's modulus, L is the element length, D is the outer pipe diameter, and t is the pipe wall thickness.

2.17 *BUCKLING LENGTH: Define buckling length data for buckling strut response of frame elements with PIPE sections.

This option is used to define two sets of coefficients used in the ISO equation that predicts P_{cr} for frame elements with buckling strut response. For a user-defined buckling envelope it can be used only in conjunction with both the *FRAME SECTION, SECTION=PIPE, YIELD STRESS=σ^0 option and the *BUCKLING ENVELOPE option. For the default buckling envelope it can be used only in conjunction with the *FRAME SECTION, BUCKLING, SECTION=PIPE, YIELD STRESS=σ^0 option, with or without the PINNED parameter.

Product: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- "Frame section behavior," Section 26.4.2 of the Abaqus Analysis User's Manual
- *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.

2.18 *BUCKLING REDUCTION FACTORS: Define buckling reduction factors for buckling strut response of frame elements with PIPE sections.

This option is used to define two coefficients used in the ISO equation, which predicts P_{cr}, the axial load at which the response switches to buckling only, for frame elements with buckling strut response. For a nondefault buckling envelope the *BUCKLING REDUCTION FACTORS option can be used only in conjunction with both the *FRAME SECTION, SECTION=PIPE, YIELD STRESS=σ^0 option and the *BUCKLING ENVELOPE option. For the default buckling envelope it can be used only in conjunction with the *FRAME SECTION, BUCKLING, SECTION=PIPE, YIELD STRESS=σ^0 option.

Product: Abaqus/Standard

Type: Model data

Level: Part, Part instance, Model

References:

- "Frame elements," Section 26.4.1 of the Abaqus Analysis User's Manual
- "Frame section behavior," Section 26.4.2 of the Abaqus Analysis User's Manual
- "Buckling strut response for frame elements," Section 3.9.3 of the Abaqus Theory Manual

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$(1.0 - 0.4 f_c/F_{e2}, 0.85)$, 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.

2.19 *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 Manual

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 Manual

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 20.3.2 of the Abaqus Analysis User's Manual
- *CAP PLASTICITY
- *CAP HARDENING
- "CREEP," Section 1.1.1 of the Abaqus User Subroutines Reference Manual

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 18.1.2 of the Abaqus Analysis User's Manual, 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.

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. α. (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 20.3.2 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, 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 20.3.2 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines to define Drucker-Prager/Cap plasticity yield surface parameters:

First line:

1. Material cohesion, d, in the p–t plane (Abaqus/Standard) or in the p–q plane (Abaqus/Explicit). (Units of FL^{-2}.)
2. Material angle of friction, β, in the p–t plane (Abaqus/Standard) or in the p–q plane (Abaqus/Explicit). Give the value in degrees.
3. Cap eccentricity parameter, R. Its value must be greater than zero (typically $0.0001 \le R \le 1000.0$).
4. Initial cap yield surface position on the volumetric inelastic strain axis, $\varepsilon^{in}_{vol}|_0$.

5. Transition surface radius parameter, α. Its value should be a small number compared to unity. If this field is left blank, the default of 0.0 is used (i.e., no transition surface). If creep properties are included in the material model, α must be set to zero.
6. (Not used in Abaqus/Explicit) K, the ratio of the flow stress in triaxial tension to the flow stress in triaxial compression. The value of K should be such that $0.778 \leq K \leq 1.0$. 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.

Product: Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- "Fluid cavity definition," Section 11.6.2 of the Abaqus Analysis User's Manual
- "Inflator definition," Section 11.6.4 of the Abaqus Analysis User's Manual
- *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 $\text{JMOLE}^{-1}\text{K}^{-4}$.)
5. $\tilde{e}$, the fifth molar heat capacity coefficient. (Units of $\text{JMOLE}^{-1}\text{K}$.)

Data lines for TYPE=TABULAR:

First line:

1. Molar heat capacity, $\tilde{c}_p$, at constant pressure. (Units of $\mathrm{JMOLE^{-1}K^{-1}}$.)
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 20.2.10 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines to define compression hardening:

First line:

1. Yield stress in compression, σ_c.
2. Absolute value of the corresponding plastic strain. (The first tabular value entered must always be zero.)
3. Not used.
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

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

1. Fifth field variable.

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

Repeat this set of data lines as often as necessary to define the dependence of the yield stress on plastic strain and, if needed, on temperature and other predefined field variables.

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 20.2.10 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

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

First line:

1. Value of the plastic "Poisson's ratio," ν_{pl}, where $-1.0 < \nu_{pl} \leq 0.5$. (Dimensionless.) If no value is provided, a default value of 0.04 is assumed.
2. Temperature, θ.
3. First field variable.
4. Etc., up to six field variables.

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

1. Seventh field variable.

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

Repeat this set of data lines as often as necessary to define the dependence of the material parameter ν_{pl} on temperature and field variables.

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 20.2.10 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines to define tension hardening:

First line:

1. Yield stress in uniaxial tension, σ_t.
2. Corresponding plastic strain. (The first tabular value entered must always be zero.)
3. Not used.
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

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

1. Fifth field variable.

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

Repeat this set of data lines as often as necessary to define the dependence of the yield stress on plastic strain and, if needed, on temperature and other predefined field variables.

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 37.1.1 of the Abaqus Analysis User's Manual
- *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.

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.

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.

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 6.7.3 of the Abaqus Analysis User's Manual

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 ("Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual).

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 a coupled thermal-electrical analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

Reference:

- "Coupled thermal-electrical analysis," Section 6.7.2 of the Abaqus Analysis User's Manual

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 30.1.2 of the Abaqus Analysis User's Manual). 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 ("Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual).

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 26.3.7 of the Abaqus Analysis User's Manual
- "Meshed beam cross-sections," Section 10.5.1 of the Abaqus Analysis User's Manual

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 *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 and coupled thermal-electrical analyses.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- "Thermal loads," Section 30.4.4 of the Abaqus Analysis User's Manual
- "FILM," Section 1.1.6 of the Abaqus User Subroutines Reference Manual

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the *AMPLITUDE option that gives the variation of the sink temperature, θ^0, with time.

If this parameter is omitted in an Abaqus/Standard analysis, the reference sink temperature is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the *STEP option ("Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). 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 ignored.

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 nonuniform film coefficient is defined in user subroutine **FILM** or if a film coefficient is defined to be a function of temperature and field variables via the *FILM PROPERTY option.

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, θ^0. (Units of θ.) For nonuniform film coefficients the sink temperature must be defined in user subroutine **FILM**. If given, this value will be 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 passed into the user subroutine.

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

3.14 *CFLOW: Specify concentrated fluid flow.

This option is used to apply concentrated fluid flow to any node 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 Manual
- "Geostatic stress state," Section 6.8.2 of the Abaqus Analysis User's Manual
- "Pore fluid flow," Section 30.4.6 of the Abaqus Analysis User's Manual

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 "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual).

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.

Data lines to define concentrated flow:

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.

3.15 *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, and mass diffusion analyses.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

Reference:

- “Thermal loads,” Section 30.4.4 of the Abaqus Analysis User’s Manual

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 30.1.2 of the Abaqus Analysis User’s Manual).

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 “Procedures: overview,” Section 6.1.1 of the Abaqus Analysis User’s Manual). If this parameter is omitted in an Abaqus/Explicit analysis, the reference magnitude is applied immediately at the beginnning of the step.

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.

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.

3.16 *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 33.1.1 of the Abaqus Analysis User's Manual
- "Frictional behavior," Section 33.1.5 of the Abaqus Analysis User's Manual
- "Connector behavior," Section 28.2.1 of the Abaqus Analysis User's Manual
- *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 30.1.2 of the Abaqus Analysis User's Manual).

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 "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). 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.

3.17 *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 20.3.4 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, 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. Absolute value of the corresponding volumetric 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 the yield surface size on volumetric plastic strain and, if needed, on temperature and other predefined field variables.

3.18 *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 20.3.4 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

HARDENING

Set HARDENING=EXPONENTIAL (default for Abaqus/Standard) to specify an exponential hardening/softening law. This hardening law is not supported in Abaqus/Explicit.

Set HARDENING=TABULAR (default and only option for Abaqus/Explicit) to specify a piecewise linear hardening/softening relationship. The *CLAY HARDENING option must be used in this case. HARDENING=TABULAR and the use of the INTERCEPT parameter are mutually exclusive.

INTERCEPT

This parameter applies only to Abaqus/Standard analyses.

It is used as an alternative to the direct specification of the initial yield surface size, a_0, when the exponential hardening law is specified. Set this parameter equal to e_1, the intercept of the virgin consolidation line with the void ratio axis in a plot of void ratio versus the logarithm of pressure stress. If this parameter is included, the value given for a_0 on the data line is ignored. This parameter cannot be used when the HARDENING=TABULAR parameter is used.

Data lines to define Cam-clay plasticity:

First line:

1. Logarithmic plastic bulk modulus, λ (dimensionless). This data item is ignored if HARDENING=TABULAR.
2. Stress ratio at critical state, M.
3. Enter the initial yield surface size, a_0 (units of FL^{-2}), if HARDENING=EXPONENTIAL. Enter the initial volumetric plastic strain, $\varepsilon_{\rm vol}^{pl}|_0$, 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. β, the parameter defining the size of the yield surface on the "wet" side of critical state. If this value is omitted or set to zero, a value of 1.0 is assumed.
5. K, the ratio of the flow stress in triaxial tension to the flow stress in triaxial compression. $0.778 \leq K \leq 1.0$. If this value is left blank or set to zero, a value of 1.0 is assumed.
6. Temperature.
7. First field variable.
8. Second field variable.

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

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

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

3.19 *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 35.1.2 of the Abaqus Analysis User's Manual
- "Common difficulties associated with contact modeling using contact pairs in Abaqus/Explicit," Section 35.2.2 of the Abaqus Analysis User's Manual
- "Adjusting initial surface positions and specifying initial clearances in Abaqus/Standard contact pairs," Section 32.3.5 of the Abaqus Analysis User's Manual
- "Adjusting initial surface positions and specifying initial clearances for contact pairs in Abaqus/Explicit," Section 32.5.4 of the Abaqus Analysis User's Manual

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 Manual, 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 35.1.2 of the Abaqus Analysis User's Manual, and "Common difficulties associated with contact modeling using contact pairs in Abaqus/Explicit," Section 35.2.2 of the Abaqus Analysis User's Manual).

Data lines if both the TABULAR parameter and the BOLT parameter are included without the INPUT parameter (see Figure 3.19–1 and Figure 3.19–2):

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 $d\text{-}0.649519p$.

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 32.3.5 of the Abaqus Analysis User's Manual, and "Specifying initial clearance values precisely" in "Adjusting initial surface positions and specifying initial clearances for contact pairs in Abaqus/Explicit," Section 32.5.4 of the Abaqus Analysis User's Manual).

To define a clearance value by using the VALUE parameter:

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

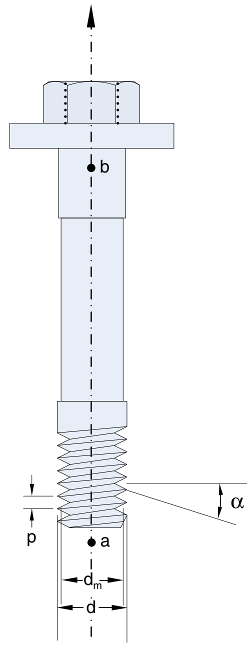

Figure 3.19–1 Thread geometry.

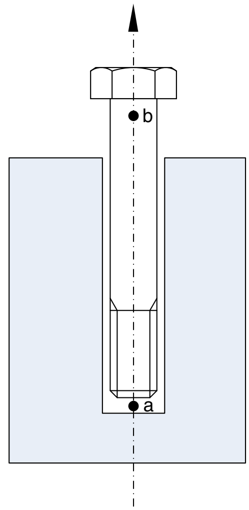

Figure 3.19–2 Points a and b on the center line of the bolt and bolt-hole assembly.

3.20 *CLOAD: Specify concentrated forces and moments.

This option is used to apply concentrated forces and moments at any node in the model. The *CLOAD option can also be used to specify concentrated buoyancy, drag, and inertia loads in an Abaqus/Aqua analysis.

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

Type: History data

Level: Step

Abaqus/CAE: Load module

Applying concentrated loads

References:

- "Concentrated loads," Section 30.4.2 of the Abaqus Analysis User's Manual
- "Defining ALE adaptive mesh domains in Abaqus/Explicit," Section 12.2.2 of the Abaqus Analysis User's Manual
- "Analysis of models that exhibit cyclic symmetry," Section 10.4.3 of the Abaqus Analysis User's Manual

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 "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). 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.

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 Manual), 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.

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

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the loading.

REAL

Include this parameter (default) to define the real (in-phase) part of the loading.

Data lines to define concentrated loads for specific degrees of freedom:

First line:

1. Node number or node set label.
2. Degree of freedom.
3. Reference magnitude for load.

Repeat this data line as often as necessary to define concentrated loads.

Applying Abaqus/Aqua loads

Reference:

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

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 "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual).

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:

8. Density of the fluid outside the element. This value will override the fluid density given on the data line of the *AQUA option.
9. Free surface elevation of the fluid outside the element. This value will override the fluid surface elevation given on the data line of the *AQUA option.
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, ΔA.
5. Drag coefficient, C_n.
6. Structural velocity factor, α_R. The default value is 1.0 if this entry is left blank or set equal to 0.0.
7. For load type TFD, name of the *AMPLITUDE curve used for scaling steady current velocities (A_c). For load type TWD, name of the *AMPLITUDE curve used for scaling the local x-direction wind velocity (A_x). If this data item is blank, the velocities are not scaled ($A_c = 1$ or $A_x = 1$).
8. For load type TFD, name of the *AMPLITUDE curve used for scaling wave velocities (A_w). For load type TWD, name of the *AMPLITUDE curve used for scaling the local y-direction wind velocity (A_y). If this data item is blank, the velocities are not scaled ($A_w = 1$ or $A_y = 1$).

Second line:

Give the following direction cosines in the local coordinate system if the *TRANSFORM option was used at this node:

1. *X*-direction cosine of the outward normal to the exposed transition section area, pointing into the fluid, in the initial configuration.
2. *Y*-direction cosine of the outward normal to the exposed transition section area, pointing into the fluid, in the initial configuration.
3. *Z*-direction cosine of the outward normal to the exposed transition section area, pointing into the fluid, in the initial configuration.

Repeat this pair of data lines as often as necessary to define concentrated fluid or wind drag loading at various nodes or node sets.

Data lines to define concentrated fluid inertia loading:

First line:

1. Node number or node set label.
2. Load type label, TSI.
3. Magnitude factor, *M*. The default value is 1.0. This factor will be scaled by any *AMPLITUDE specification associated with this *CLOAD option.
4. Tangential inertia coefficient, K_{ts}.
5. Fluid acceleration shape factor for the tangential inertia term, F_{1s}.
6. Tangential added-mass coefficient, L_{ts}.
7. Structural acceleration shape factor for the tangential inertia term, F_{2s}.
8. Name of the *AMPLITUDE curve to be used for scaling fluid particle accelerations (A_w). If this data item is blank, the fluid particle accelerations are not scaled ($A_w = 1$).

Second line:

Give the following direction cosines in the local coordinate system if the *TRANSFORM option was used at this node:

1. *X*-direction cosine of the outward normal to the exposed transition section area, pointing into the fluid, in the initial configuration.
2. *Y*-direction cosine of the outward normal to the exposed transition section area, pointing into the fluid, in the initial configuration.
3. *Z*-direction cosine of the outward normal to the exposed transition section area, pointing into the fluid, in the initial configuration.

Repeat this pair of data lines as often as necessary to define concentrated fluid inertia loading for various nodes or node sets.

3.21 *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 33.1.10 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, 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.

3.22 *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 29.5.1 of the Abaqus Analysis User's Manual
- "Defining the constitutive response of cohesive elements using a continuum approach," Section 29.5.5 of the Abaqus Analysis User's Manual

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 24.1.4 of the Abaqus Analysis User’s Manual). The *SECTION CONTROLS option can be used to specify whether the cohesive elements should be deleted once they completely fail. This option may also be used to specify a maximum value of the scalar degradation (damage) parameter, D, and/or the viscosity coefficient, μ, for viscous regularization.

ORIENTATION

Set this parameter equal to the name given for the *ORIENTATION option (“Orientations,” Section 2.2.5 of the Abaqus Analysis User’s Manual) 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.

3.23 *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 19.7.1 of the Abaqus Analysis User's Manual
- *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.

3.24 *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 Manual

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.

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 cut-off value for complex modes. Only complex modes with the real part of the eigenvalue higher than the cut-off 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.

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.

3.25 *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 20.6.1 of the Abaqus Analysis User's Manual; "Defining rebar as an element property," Section 2.2.4 of the Abaqus Analysis User's Manual; and the appropriate sections in the Theory Manual and the Example Problems Manual.

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 Manual).

The *BRITTLE CRACKING option is used to analyze concrete in an Abaqus/Explicit analysis (see "Cracking model for concrete," Section 20.6.2 of the Abaqus Analysis User's Manual).

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Concrete smeared cracking," Section 20.6.1 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, 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.

3.26 *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 20.6.3 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, 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.

3.27 *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 20.6.3 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines to define compressive hardening:

First line:

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

5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

The first point at each value of temperature must have a crushing strain of 0.0 and gives the initial yield stress value, σ_{c0}.

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

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

Repeat this set of data lines as often as necessary to define the dependence of the compressive yield stress on crushing strain, crushing strain rate, and other predefined field variables.

3.28 *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 20.6.3 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

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

First line:

1. Dilation angle, ψ, in the p–q plane. Give the value in degrees.

2. Flow potential eccentricity, ϵ. The eccentricity is a small positive number that defines the rate at which the hyperbolic flow potential approaches its asymptote. If this field is left blank or a value of 0.0 is entered, the default of $\epsilon = 0.1$ is used.
3. σ_{b0}/σ_{c0}, the ratio of initial equibiaxial compressive yield stress to initial uniaxial compressive yield stress. If this field is left blank or a value of 0.0 is entered, the default of 1.16 is used.
4. K_c, the ratio of the second stress invariant on the tensile meridian, $q_{(\mathrm{TM})}$, to that on the compressive meridian, $q_{(\mathrm{CM})}$, at initial yield for any given value of the pressure invariant p such that the maximum principal stress is negative, $\hat{\sigma}_{\max} < 0$. It must satisfy the condition $0.5 < K_c \leq 1.0$. If this field is left blank or a value of 0.0 is entered, the default of $2/3$ is used.
5. Viscosity parameter, μ, used for the visco-plastic regularization of the concrete constitutive equations in Abaqus/Standard analyses. This parameter is ignored in Abaqus/Explicit. The default value is 0.0. (Units of T.)
6. Temperature.
7. First field variable.
8. Second field variable.

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

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

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

3.29 *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 20.6.3 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, 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.

3.30 *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 20.6.3 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

TYPE

Set TYPE=STRAIN (default) to specify the postcracking behavior by entering the postfailure stress/cracking-strain relationship.

Set TYPE=DISPLACEMENT to define the postcracking behavior by entering the postfailure stress/cracking-displacement relationship.

Set TYPE=GFI to define the postcracking behavior by entering the failure stress, σ_{t0}, and the fracture energy, G_f.

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

First line:

1. Remaining direct stress after cracking, σ_t. (Units of FL^{-2}.)
2. Direct cracking strain, $\tilde{\varepsilon}_t^{ck}$.
3. Direct cracking strain rate, $\dot{\tilde{\varepsilon}}_t^{ck}$. (Units of T^{-1}.)
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

The first point at each value of temperature must have a cracking strain of 0.0 and gives the failure stress value, σ_{t0}.

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

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

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

Data lines if the TYPE=DISPLACEMENT parameter is included:

First line:

1. Remaining direct stress after cracking, σ_t. (Units of FL^{-2}.)
2. Direct cracking displacement, u_t^{ck}. (Units of L.)
3. Direct cracking displacement rate, $\dot{u}_t^{ck}$. (Units of LT^{-1}.)
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

The first point at each value of temperature must have a cracking displacement of 0.0 and gives the failure stress value.

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

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

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

Data lines if the TYPE=GFI parameter is included:

First line:

1. Failure stress, σ_{t0}. (Units of FL^{-2}.)
2. Fracture energy, G_f. (Units of FL^{-1}.)
3. Direct cracking displacement rate, $\dot{u}_t^{ck}$. (Units of 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.

3.31 *CONDUCTIVITY: Specify thermal conductivity.

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

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Reference:

- "Conductivity," Section 23.2.2 of the Abaqus Analysis User's Manual

Optional parameters:

DEPENDENCIES

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 18.1.2 of the Abaqus Analysis User's Manual, 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.

Data lines to define isotropic thermal conductivity (TYPE=ISO):

First line:

1. Conductivity, k. (Units of $\mathrm{JT}^{-1}\mathrm{L}^{-1}\,\theta^{-1}$.)
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 $\mathrm{JT}^{-1}\mathrm{L}^{-1}\ \theta^{-1}$.)
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 $\mathrm{JT}^{-1}\mathrm{L}^{-1}\theta^{-1}$.)
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.

3.32 *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 28.2.1 of the Abaqus Analysis User's Manual
- *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.

3.33 *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 28.2.1 of the Abaqus Analysis User's Manual
- *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.

3.34 ∗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 28.1.5 of the Abaqus Analysis User's Manual
- "Connector behavior," Section 28.2.1 of the Abaqus Analysis User's Manual
- "Connector damage behavior," Section 28.2.7 of the Abaqus Analysis User's Manual
- ∗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 18.1.2 of the Abaqus Analysis User's Manual, 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 28.2.1 of the Abaqus Analysis User's Manual, 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 28.2.1 of the Abaqus Analysis User’s Manual, for a description of the connector relative motions.

2. Exponential law parameter, α (see “Connector damage behavior,” Section 28.2.7 of the Abaqus Analysis User’s Manual).
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 28.2.1 of the Abaqus Analysis User’s Manual, 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.

3.35 *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 28.1.5 of the Abaqus Analysis User's Manual
- "Connector behavior," Section 28.2.1 of the Abaqus Analysis User's Manual
- "Connector damage behavior," Section 28.2.7 of the Abaqus Analysis User's Manual
- *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 28.1.5 of the Abaqus Analysis User's Manual, 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 18.1.2 of the Abaqus Analysis User's Manual, 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 28.2.6 of the Abaqus Analysis User’s Manual, 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.

3.36 *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 28.2.1 of the Abaqus Analysis User's Manual
- "Connector damping behavior," Section 28.2.3 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, 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

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.

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 both the COMPONENT and NONLINEAR parameters omitted; all 21 damping constants must be specified, regardless of whether temperature or field variable dependencies are included):

First line:

1. C_{11}. (Units of FTL^{-1}.)
2. C_{12}.
3. C_{22}.
4. C_{13}.
5. C_{23}.
6. C_{33}.
7. C_{14}. (Units of FT.)
8. C_{24}.

Second line:

1. C_{34}.
2. C_{44}. (Units of FTL.)
3. C_{15}. (Units of FT.)
4. C_{25}.
5. C_{35}.
6. C_{45}. (Units of FTL.)
7. C_{55}.
8. C_{16}. (Units of FT.)

Third line:

1. C_{26}.
2. C_{36}.
3. C_{46}. (Units of FTL.)
4. C_{56}.

5. C_{66}.
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 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}.

3.37 *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 28.2.1 of the Abaqus Analysis User's Manual
- "Connector functions for coupled behavior," Section 28.2.4 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

EXTRAPOLATION

Set EXTRAPOLATION=CONSTANT (default unless *CONNECTOR BEHAVIOR, EXTRAPOLATION=LINEAR is used) to use constant extrapolation of the dependent variables outside the specified range of the independent variables.

Set EXTRAPOLATION=LINEAR to use linear extrapolation of the dependent variables outside the specified range of the independent variables.

INDEPENDENT COMPONENTS

Set INDEPENDENT COMPONENTS=POSITION (default) to specify dependencies on components of relative position included in the derived component definition.

Set INDEPENDENT COMPONENTS=CONSTITUTIVE MOTION to specify dependencies on components of constitutive relative motion included in the derived component definition.

OPERATOR

Set OPERATOR=NORM (default) to use a square root of a sum of the squares function of the contributing components.

Set OPERATOR=MACAULEY SUM to sum the contributing components with a Macauley bracket function applied to each contribution.

Set OPERATOR=SUM to sum the contributing components directly.

REGULARIZE

This parameter applies only to Abaqus/Explicit analyses.

Set REGULARIZE=ON (default unless *CONNECTOR BEHAVIOR, REGULARIZE=OFF is used) to regularize the user-defined tabular connector derived component data.

Set REGULARIZE=OFF to use the user-defined tabular connector derived component data directly without regularization.

RTOL

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the tolerance to be used to regularize the connector derived component data.

If this parameter is omitted, the default is RTOL=0.03 unless the tolerance is specified on the *CONNECTOR BEHAVIOR option.

SIGN

Set SIGN=POSITIVE (default) to provide an overall positive sign to the derived component definition.

Set SIGN=NEGATIVE to provide an overall negative sign to the derived component definition.

Data lines to define the derived component if the INDEPENDENT COMPONENTS parameter is omitted:

First line:

1. First component number (1–6) to be used in the definition of the derived component.
2. Second component number (1–6) to be used in the definition of the derived component.
3. Etc., up to N_c entries (maximum six).

Subsequent lines:

1. Scaling constant (α_1) that multiplies the first component identified on the first data line.
2. Scaling constant (α_2) that multiplies the second component identified on the first data line.
3. Etc., up to N_c entries as identified on the first data line.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to eight entries per line.

If the number of data entries exceeds the limit of eight entries per line, continue the input on the next data line.

Do not repeat the first data line. Repeat the subsequent data lines as often as necessary to define the contributions to the derived component as a function of temperature and field variables.

Data lines to define the derived component if the INDEPENDENT COMPONENTS parameter is included:

First line:

1. First independent component number (1–6).
2. Second independent component number (1–6).
3. Etc., up to N_i entries (maximum six).

Second line:

1. First component number (1–6) to be used in the definition of the derived component.
2. Second component number (1–6) to be used in the definition of the derived component.
3. Etc., up to N_c entries (maximum six).

Third line:

1. Scaling constant (α_1) that multiplies the first component identified on the second data line.
2. Scaling constant (α_2) that multiplies the second component identified on the second data line.
3. Etc., up to N_c entries as identified on the second data line.
4. Connector relative position or constitutive relative motion in the first independent component identified on the first data line.
5. Connector relative position or constitutive relative motion in the second independent component identified on the first data line.
6. Etc., up to N_i entries as identified on the first data line.
7. Temperature.
8. First field variable.

If the number of data entries exceeds the limit of eight entries per line, continue the input on the next data line.

Continuation line (if needed):

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

Do not repeat the first two data lines. Repeat the subsequent data lines as often as necessary to define the contributions to the derived component as a function of connector relative position or constitutive relative motion, temperature, and field variables.

3.38 *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 28.2.1 of the Abaqus Analysis User's Manual
- "Connector elastic behavior," Section 28.2.2 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, 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

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.

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 and NONLINEAR 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 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.

3.39 *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 28.2.1 of the Abaqus Analysis User's Manual
- "Connection-type library," Section 28.1.5 of the Abaqus Analysis User's Manual
- *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 28.1.5 of the Abaqus Analysis User's Manual, 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.

3.40 *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 28.1.5 of the Abaqus Analysis User's Manual
- "Connector behavior," Section 28.2.1 of the Abaqus Analysis User's Manual
- "Connector friction behavior," Section 28.2.5 of the Abaqus Analysis User's Manual
- *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 28.1.5 of the Abaqus Analysis User's Manual.

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 18.1.2 of the Abaqus Analysis User’s Manual, 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 28.1.5 of the Abaqus Analysis User’s Manual.

2. Second friction parameter.
3. Etc., up to as many friction parameters discussed in "Connection-type library," Section 28.1.5 of the Abaqus Analysis User's Manual.

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.

3.41 *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 28.2.1 of the Abaqus Analysis User's Manual
- "Connector plastic behavior," Section 28.2.6 of the Abaqus Analysis User's Manual
- "Models for metals subjected to cyclic loading," Section 20.2.2 of the Abaqus Analysis User's Manual
- *CONNECTOR BEHAVIOR
- *CONNECTOR ELASTICITY
- *CONNECTOR HARDENING
- *CONNECTOR PLASTICITY
- *CONNECTOR POTENTIAL

Optional parameters:

DEFINITION

Set DEFINITION=EXPONENTIAL LAW to specify the isotropic hardening parameters Q_{inf} and b directly. This parameter is valid only for TYPE=ISOTROPIC.

Set DEFINITION=HALF CYCLE (default for TYPE=KINEMATIC) to provide force/moment versus plastic motion data of a first half-cycle. This parameter is valid only for TYPE=KINEMATIC.

Set DEFINITION=PARAMETERS to specify the kinematic hardening parameters C and γ directly. This parameter is valid only for TYPE=KINEMATIC.

Set DEFINITION=STABILIZED to provide force/moment versus plastic motion data of a stabilized cycle. This parameter is valid only for TYPE=KINEMATIC.

Set DEFINITION=TABULAR (default for TYPE=ISOTROPIC) to provide force/moment versus plastic motion values. Either uniaxial test data or processed data (as explained in "Connector behavior," Section 28.2.1 of the Abaqus Analysis User's Manual) 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 18.1.2 of the Abaqus Analysis User's Manual, 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 γ. Set γ=0 to specify linear Ziegler kinematic hardening.
4. Temperature.

5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

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

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

Repeat this set of data lines as often as necessary to define the yield force/moment at zero relative plastic motion and the kinematic hardening parameters as functions of temperature and field variables.

3.42 *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 28.1.3 of the Abaqus Analysis User's Manual

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 "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). 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 Manual), 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.

3.43 *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 28.2.1 of the Abaqus Analysis User's Manual
- "Connection-type library," Section 28.1.5 of the Abaqus Analysis User's Manual
- *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 28.1.5 of the Abaqus Analysis User's Manual, 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 18.1.2 of the Abaqus Analysis User's Manual, 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.

3.44 *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 28.1.5 of the Abaqus Analysis User's Manual
- "DISP," Section 1.1.4 of the Abaqus User Subroutines Reference Manual

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 30.1.2 of the Abaqus Analysis User's Manual).

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 ("Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). 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 30.1.2 of the Abaqus Analysis User's Manual.

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 28.1.5 of the Abaqus Analysis User's Manual, 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.

3.45 *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 28.2.1 of the Abaqus Analysis User's Manual
- "Connector plastic behavior," Section 28.2.6 of the Abaqus Analysis User's Manual
- *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.

3.46 *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 28.1.5 of the Abaqus Analysis User's Manual
- "Connector behavior," Section 28.2.1 of the Abaqus Analysis User's Manual
- "Connector functions for coupled behavior," Section 28.2.4 of the Abaqus Analysis User's Manual
- *CONNECTOR BEHAVIOR
- *CONNECTOR DAMAGE EVOLUTION
- *CONNECTOR DAMAGE INITIATION
- *CONNECTOR DERIVED COMPONENT
- *CONNECTOR FRICTION
- *CONNECTOR PLASTICITY

Optional parameters:

EXPONENT

This parameter can be used only if OPERATOR=SUM.

Set this parameter equal to the inverse of the overall exponent in the potential definition, β. β must be a positive number. The default value is $\beta = 2.0$.

OPERATOR

Set OPERATOR=SUM (default) to define the potential as the sum of the contributions defined on each data line.

Set OPERATOR=MAX to define the potential as the contribution coming from the data line that yields the maximum value. The EXPONENT parameter is ignored in this case.

Data lines to define the potential:

First line:

1. Connector component number (1–6) or connector derived component name that is used in the contribution.
2. Nonzero scaling factor R. The default value is $R = 1.0$.
3. Positive exponent α. The default value is that of the EXPONENT parameter, $\alpha = \beta$. The exponent is ignored if OPERATOR=MAX.
4. The function H to be used to generate the contribution. H can be ABS (absolute value), MACAULEY (Macauley bracket), or NONE (the identity function). 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.

3.47 *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 28.1.2 of the Abaqus Analysis User's Manual
- "Connection-type library," Section 28.1.5 of the Abaqus Analysis User's Manual
- *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 24.1.4 of the Abaqus Analysis User's Manual) to be used for the connector elements.

Section controls can be used to specify whether the connector elements should be deleted once they fail completely. If this parameter is omitted, the failed elements will not be deleted. Section controls can also be used to specify a maximum value of the scalar degradation (damage) parameter, $D_{\max}$, and to specify the viscosity coefficient, μ, for viscous damping or regularization.

ELIMINATION

This parameter applies only to Abaqus/Explicit analyses.

Set ELIMINATION=NO (default) if the constraint or kinetic forces/moments of the associated connector elements are to be solved for directly in the implicit constraint solver in Abaqus/Explicit.

Set ELIMINATION=YES if the constraint or kinetic forces/moments of the associated connector elements are to be solved for using a condensation technique.

Data lines to define the connection attributes:

First line:

1. Basic translational connection type, basic rotational connection type, assembled connection type, or complex connection type from "Connection-type library," Section 28.1.5 of the Abaqus Analysis User's Manual. 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, β_s (default value is 1.0).

Omit the third line if no data are specified.

3.48 *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 28.2.1 of the Abaqus Analysis User's Manual
- *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.

3.49 *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 28.2.1 of the Abaqus Analysis User's Manual
- "Connector uniaxial behavior," Section 28.2.10 of the Abaqus Analysis User's Manual
- *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.

3.50 *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 31.6.1 of the Abaqus Analysis User's Manual
- "Connectors: overview," Section 28.1.1 of the Abaqus Analysis User's Manual
- "Mesh tie constraints," Section 31.3.1 of the Abaqus Analysis User's Manual
- "Common difficulties associated with contact modeling in Abaqus/Standard," Section 35.1.2 of the Abaqus Analysis User's Manual

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.

3.51 *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 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 32.2.1 of the Abaqus Analysis User's Manual
- "Defining general contact interactions in Abaqus/Explicit," Section 32.4.1 of the Abaqus Analysis User's Manual

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.

There are no data lines associated with this option.

3.52 *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 32.4.4 of the Abaqus Analysis User's Manual
- *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 2.7.1 of the Abaqus Analysis User's Manual). 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.

3.53 *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 32.4.4 of the Abaqus Analysis User's Manual
- *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.

3.54 *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

Specifying additional controls for contact in an Abaqus/Standard analysis

WARNING: The parameters LAGRANGE MULTIPLIER, MAXCHP, PERRMX, and UERRMX are intended for experienced analysts and should be used with care.

Reference:

- "Adjusting contact controls in Abaqus/Standard," Section 32.3.6 of the Abaqus Analysis User's Manual

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:

APPROACH

Include this parameter to automatically address situations where an initial rigid body mode exists normal to the contact direction. This option activates viscous damping in the normal direction to prevent numerical difficulties associated with rigid body motion that occurs when surfaces not initially in contact are brought into contact. The bodies should be moved into contact in a single step, but there should not be significant further deformation of the parts during the step due to the loads bringing them into contact. This parameter must be used in conjunction with the SLAVE and MASTER parameters to specify a contact pair. For more general situations in which rigid body modes need to be controlled, use the STABILIZE parameter instead.

AUTOMATIC TOLERANCES

Include this parameter to have Abaqus/Standard automatically compute an overclosure tolerance and a separation pressure tolerance to prevent chattering in contact. This parameter cannot be used with the MAXCHP, PERRMX, and UERRMX parameters.

FRICTION ONSET

Set FRICTION ONSET=IMMEDIATE (default) to include friction in the increment when contact occurs. Set FRICTION ONSET=DELAYED to delay the application of friction to the increment after contact occurs.

LAGRANGE MULTIPLIER

Set LAGRANGE MULTIPLIER=YES to enforce the contact constraints with Lagrange multipliers.

Set LAGRANGE MULTIPLIER=NO to enforce the constraints without Lagrange multipliers. Setting LAGRANGE MULTIPLIER=NO is not recommended for problems with a high stiffness since it may lead to numerical problems during equation solution (such as singularities). For the default direct enforcement of hard contact LAGRANGE MULTIPLIER=NO is not allowed.

The value of the contact stiffness determines whether or not Lagrange multipliers are used by default. When the default penalty stiffness settings are used for penalty or augmented Lagrange contact, Lagrange multipliers are not used by default. If the penalty stiffness used in penalty or augmented Lagrange contact is set to greater than 1000 times the representative underlying element stiffness, Lagrange multipliers are used by default. For softened contact using the direct enforcement method (see "Contact pressure-overclosure relationships," Section 33.1.2 of the Abaqus Analysis User's Manual), Lagrange multipliers are used by default only if the maximum slope of the pressure-overclosure relationship exceeds 1000 times the representative underlying element stiffness.

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.

MAXCHP

Set this parameter equal to the maximum number of points that are permitted to violate contact conditions in any increment. The amount by which the condition can be violated is limited by the

PERRMX and UERRMX parameters, at least one of which must be used in conjunction with this parameter. If more than this number of points violate the contact conditions, the solution will not be accepted.

PERRMX

Set this parameter equal to the maximum value of tensile stress (tensile force in GAP- or ITT-type contact elements) allowed to be transmitted at a contact point. If any point in contact has a tensile stress/force across the contact interface greater than PERRMX, iteration will occur regardless of the value of MAXCHP (which must be used in conjunction with this parameter). By default, no tensile stress can be transmitted.

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.

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 will calculate the damping coefficient automatically. If a value is assigned to this parameter, Abaqus will multiply the automatically calculated damping coefficient by this value. If the damping coefficient is defined directly on the data line, any value assigned to this parameter will be ignored.

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. Only contact constraints enforced with the augmented Lagrangian and penalty methods will be affected by this parameter. 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.

UERRMX

Set this parameter equal to the maximum overclosure distance allowed at a slave node that is considered to be open. If any contact point violates the contact constraint by more than UERRMX, iteration will occur regardless of the value of MAXCHP (which must be used in conjunction with this parameter). By default, no overclosure is allowed.

Optional data line if the STABILIZE parameter is included:

First (and only) line:

1. Damping coefficient to be used in the contact interface. The value entered overrides the damping coefficient calculated by Abaqus. When a nonzero value is entered, the value assigned to the STABILIZE parameter is ignored.
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.

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.

References:

- “Defining contact pairs in Abaqus/Explicit,” Section 32.5.1 of the Abaqus Analysis User’s Manual
- “Contact controls for contact pairs in Abaqus/Explicit,” Section 32.5.5 of the Abaqus Analysis User’s Manual
- *CONTACT PAIR

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.

3.55 *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 32.4.5 of the Abaqus Analysis User's Manual
- *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.

TYPE

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

Set TYPE=ENHANCED EDGE TRACKING to activate an enhanced 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.

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.

3.56 *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 33.1.1 of the Abaqus Analysis User's Manual
- "Contact damping," Section 33.1.3 of the Abaqus Analysis User's Manual

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, $\eta \quad (0 \leq \eta \leq 1)$. The default is $\eta = 0.0$.

3.57 *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 32.2.1 of the Abaqus Analysis User's Manual
- "Defining general contact interactions in Abaqus/Explicit," Section 32.4.1 of the Abaqus Analysis User's Manual
- *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.

3.58 *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 Manual

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 Manual.

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.

3.59 *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 32.2.6 of the Abaqus Analysis User's Manual
- "Contact formulation for general contact in Abaqus/Explicit," Section 34.2.1 of the Abaqus Analysis User's Manual
- *CONTACT

Required parameter:

TYPE

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.

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.

3.60 *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 32.2.1 of the Abaqus Analysis User's Manual
- "Defining general contact interactions in Abaqus/Explicit," Section 32.4.1 of the Abaqus Analysis User's Manual
- *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; any data lines specified will be ignored.

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.

3.61 *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 32.2.4 of the Abaqus Analysis User's Manual
- *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.

3.62 *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 32.2.4 of the Abaqus Analysis User's Manual
- "Common difficulties associated with contact modeling in Abaqus/Standard," Section 35.1.2 of the Abaqus Analysis User's Manual
- *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.

3.63 *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 32.3.4 of the Abaqus Analysis User's Manual
- "Adjusting initial surface positions and specifying initial clearances in Abaqus/Standard contact pairs," Section 32.3.5 of the Abaqus Analysis User's Manual

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.

3.64 *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 Manual
- *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 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).

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 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.

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 Manual, and "Abaqus/Explicit output variable identifiers," Section 4.2.2 of the Abaqus Analysis User's Manual.

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

3.65 *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

Defining contacting surfaces in an Abaqus/Standard analysis

References:

- "Defining contact pairs in Abaqus/Standard," Section 32.3.1 of the Abaqus Analysis User's Manual
- "Adjusting initial surface positions and specifying initial clearances in Abaqus/Standard contact pairs," Section 32.3.5 of the Abaqus Analysis User's Manual
- "Defining tied contact in Abaqus/Standard," Section 32.3.7 of the Abaqus Analysis User's Manual
- "Adjusting contact controls in Abaqus/Standard," Section 32.3.6 of the Abaqus Analysis User's Manual
- "Contact formulations in Abaqus/Standard," Section 34.1.1 of the Abaqus Analysis User's Manual
- "Smoothing contact surfaces in Abaqus/Standard," Section 34.1.3 of the Abaqus Analysis User's Manual
- "Common difficulties associated with contact modeling in Abaqus/Standard," Section 35.1.2 of the Abaqus Analysis User's Manual

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. This parameter is not allowed with self-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 tangential slip directions on the slave surface.
4. Optional orientation name to specify the tangential slip 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

References:

- "Defining contact pairs in Abaqus/Explicit," Section 32.5.1 of the Abaqus Analysis User's Manual
- "Contact formulations for contact pairs in Abaqus/Explicit," Section 34.2.2 of the Abaqus Analysis User's Manual
- "Adjusting initial surface positions and specifying initial clearances for contact pairs in Abaqus/Explicit," Section 32.5.4 of the Abaqus Analysis User's Manual

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.

3.66 *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 Manual

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 Manual.

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

3.67 *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 32.2.3 of the Abaqus Analysis User's Manual
- "Assigning contact properties for general contact in Abaqus/Explicit," Section 32.4.3 of the Abaqus Analysis User's Manual
- *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.

3.68 *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 16.1.1 of the Abaqus Analysis User's Manual
- *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 16.1.1 of the Abaqus Analysis User's Manual.

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

3.69 *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 32.2.5 of the Abaqus Analysis User's Manual
- *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.

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.

3.70 *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 Manual, 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 Manual
- "Modeling discontinuities as an enriched feature using the extended finite element method," Section 10.6.1 of the Abaqus Analysis User's Manual

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.

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.

3.71 *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 Manual, contains a discussion of the types of problems that may occur and the use of the *CONTROLS option to overcome these problems. This option can also be used in some cases to obtain a solution in a more efficient manner. Use of the option for this latter purpose is intended for experienced users only.*

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

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

Required, mutually exclusive parameters:

ANALYSIS

Set ANALYSIS=DISCONTINUOUS to set parameters that will usually improve efficiency for severely discontinuous behavior, such as frictional sliding or concrete cracking, by allowing relatively many iterations prior to beginning any checks on the convergence rate. This parameter overrides any values that may be set for the variables I_0 and I_R on the data lines associated with PARAMETERS=TIME INCREMENTATION. A less efficient solution may result if this parameter is set in problems that do not exhibit severely discontinuous behavior.

PARAMETERS

Set PARAMETERS=FIELD to set parameters for satisfying a field equation. In this case the FIELD parameter can be used to define the field for which the parameters are being given. If the FIELD parameter is omitted, the parameters are being set for all fields that are active in the problem.

Set PARAMETERS=CONSTRAINTS to set tolerances on constraint equations.

Set PARAMETERS=LINE SEARCH to set line search control parameters.

Set PARAMETERS=TIME INCREMENTATION to set time incrementation control parameters.

RESET

Include this parameter to reset all values to their defaults. The option should have no data lines when this parameter is used.

TYPE

Set TYPE=DIRECT CYCLIC to set parameters that will be used to control the stabilized state and plastic ratcheting detections and to specify when to impose the periodicity condition for direct cyclic analysis.

Set TYPE=VCCT LINEAR SCALING to set the β parameter that will be used with linear scaling for a VCCT debonding analysis.

Optional parameter:

FIELD

This parameter can be used only with PARAMETERS=FIELD.

Set FIELD=CONCENTRATION to set parameters for the mass concentration field equilibrium equations.

Set FIELD=DISPLACEMENT to set parameters for the displacement field and warping degree of freedom equilibrium equations.

Set FIELD=ELECTRICAL POTENTIAL to set parameters for the electrical potential field equilibrium equations.

Set FIELD=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.

Data lines for PARAMETERS=FIELD:

First line:

1. R_n^α, 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^α, convergence criterion for the ratio of the largest solution correction to the largest corresponding incremental solution value. Default $C_n^\alpha = 10^{-2}$.
3. $\tilde{q}_0^\alpha$, initial value of the time average flux for this step. The default is the time average flux from previous steps or 10^{-2} if this is Step 1.
4. $\tilde{q}_u^\alpha$, user-defined average flux. When this value is defined, $\tilde{q}^\alpha(t) = \tilde{q}_u^\alpha$ for all t.

The remaining items rarely need to be reset from their default values.

5. R_P^α, alternative residual convergence criterion to be used after I_P^α iterations. Default $R_P^\alpha = 2 \times 10^{-2}$.
6. ϵ^α, criterion for zero flux compared to $\tilde{q}^\alpha$. Default $\epsilon^\alpha = 10^{-5}$.
7. C_ϵ^α, convergence criterion for the ratio of the largest solution correction to the largest corresponding incremental solution value when there is zero flux in the model. Default $C_\epsilon^\alpha = 10^{-3}$.
8. R_l^α, 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. ϵ_l^α, criterion for zero flux compared to the time averaged value of the largest flux $\tilde{q}_{\max}^\alpha$ in the model during the current step. Default $\epsilon_l^\alpha = 10^{-5}$.
3. ϵ_d^α, criterion for zero displacement increment (and/or zero penetration if CONVERT SDI=YES) compared to the characteristic element length in the model. This item is used only when FIELD=DISPLACEMENT. Default $\epsilon_d^\alpha = 10^{-8}$.

Data line for PARAMETERS=CONSTRAINTS:

First (and only) line:

These items rarely need to be reset from their default values. The relevance of certain parameters depends on the value of the CONVERT SDI parameter on the *STEP option.

1. T^{vol}, volumetric strain compatibility tolerance for hybrid solid elements. Default $T^{vol} = 10^{-5}$.
2. T^{axial}, axial strain compatibility tolerance for hybrid beam elements. Default $T^{axial} = 10^{-5}$.

3. T^{tshear}, transverse shear strain compatibility tolerance for hybrid beam elements. Default $T^{tshear} = 10^{-5}$.

4. T^{cont}, contact and slip compatibility tolerance. For CONVERT SDI=YES, the ratio of the maximum error in the contact or slip constraints to the maximum displacement increment must be less than this tolerance.
For CONVERT SDI=NO, this is used only with softened contact specified with the *SURFACE BEHAVIOR, PRESSURE-OVERCLOSURE option. The ratio of the error in the soft contact constraint clearance to the user-specified clearance at which the contact pressure is zero must lie below this tolerance for $p > p^0$, where p^0 is the pressure value at zero clearance. Default $T^{cont} = 5 \times 10^{-3}$.

5. T^{soft}, soft contact compatibility tolerance for low pressure. This tolerance, which is used only if CONVERT SDI=NO, is similar to T^{cont} for softened contact, except that it represents the tolerance when $p = 0.0$. The actual tolerance is interpolated linearly between T^{cont} and T^{soft} for $0 \leq p \leq p^0$. 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. η^{ls}, ratio of new to old correction scale factors below which line searching terminates. Default $\eta^{ls} = 0.10$.

Data lines for PARAMETERS=TIME INCREMENTATION:

First line:

The relevance of certain parameters depends on the value of the CONVERT SDI parameter on the *STEP option.

1. I_0, number of equilibrium iterations (without severe discontinuities) after which the check is made whether the residuals are increasing in two consecutive iterations. Minimum value is $I_0 = 3$. Default $I_0 = 4$. If ANALYSIS=DISCONTINUOUS, $I_0 = 8$.
2. I_R, number of consecutive equilibrium iterations (without severe 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 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 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^α, 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^α, stabilized state detection criterion for the ratio of the largest correction to the displacement coefficient on any terms in the Fourier series to the largest displacement coefficient. Default $CU_n^\alpha = 5 \times 10^{-3}$.
4. CR_0^α, plastic 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^α, 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. β parameter. Default $\beta = 0.9$.

3.72 *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 30.4.5 of the Abaqus Analysis User's Manual
- *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).

3.73 *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 Manual
- *PSD-DEFINITION
- "UCORR," Section 1.1.22 of the Abaqus User Subroutines Reference Manual

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 Manual, 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.

3.74 *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 to define parameters that control the co-simulation.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- "Co-simulation: overview," Section 14.1.1 of the Abaqus Analysis User's Manual
- "Preparing an Abaqus/Standard or Abaqus/Explicit analysis for co-simulation," Section 14.1.2 of the Abaqus Analysis User's Manual
- "Abaqus/Standard to Abaqus/Explicit co-simulation," Section 14.1.4 of the Abaqus Analysis User's Manual
- *CO-SIMULATION CONTROLS
- *CO-SIMULATION REGION

Required parameters:

CONTROLS

Set this parameter equal to the name of the co-simulation controls to be used to define the rendezvousing scheme. This parameter is not valid for PROGRAM=MADYMO.

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 Manual), 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.

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

Set PROGRAM=DCI for exchange of data between Abaqus and certain third-party analysis programs. Consult the User's Guide for the third-party analysis program to determine when this option is applicable.

Set PROGRAM=MADYMO for exchange of data between Abaqus/Explicit and the occupant simulation program MADYMO.

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.

There are no data lines associated with this option if PROGRAM=MULTIPHYSICS, PROGRAM=ABAQUS, PROGRAM=DCI, or PROGRAM=MPCCI is specified.

Data line to define the conversion factors for the physical units of mass, length, and time if PROGRAM=MADYMO and the unit system in the Abaqus/Explicit model is different than the unit system in the MADYMO model:

First (and only) line:

1. Conversion factor for the mass unit used in the Abaqus model to that in the MADYMO model.
2. Conversion factor for the length unit used in the Abaqus model to that in the MADYMO model.
3. Conversion factor for the time unit used in the Abaqus model to that in the MADYMO model.

For example, Abaqus/Explicit will multiply the coordinate values by the above length conversion factor prior to exporting these values to MADYMO. Appropriate scale factors based on the above conversion factors are used for the various quantities that are exported to MADYMO. Similarly, Abaqus will divide the imported values from MADYMO with appropriate scale factors based on the above conversion factors.

3.75 *CO-SIMULATION CONTROLS: Specify the rendezvousing scheme for co-simulation.

This option is used to specify the 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 not required for co-simulation with MADYMO.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- "Co-simulation: overview," Section 14.1.1 of the Abaqus Analysis User's Manual
- "Preparing an Abaqus/Standard or Abaqus/Explicit analysis for co-simulation," Section 14.1.2 of the Abaqus Analysis User's Manual
- *CO-SIMULATION
- *CO-SIMULATION REGION

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.

Required parameter for use with the *CO-SIMULATION, PROGRAM=MULTIPHYSICS or MPCCI option; optional parameter for use with the *CO-SIMULATION, PROGRAM=ABAQUS option:

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. This setting is valid only when used with the *CO-SIMULATION, PROGRAM=MULTIPHYSICS or MPCCI option.

Set STEP SIZE=EXPORT for Abaqus to export a coupling step size to the external program for the next coupling step. This setting is valid only when used with the *CO-SIMULATION, PROGRAM=MULTIPHYSICS or MPCCI option.

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. This setting is valid only when used with the *CO-SIMULATION, PROGRAM=MULTIPHYSICS option.

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. This setting is valid only when used with the *CO-SIMULATION, PROGRAM=MULTIPHYSICS option.

When you specify PROGRAM=ABAQUS with the *CO-SIMULATION option, you can specify the STEP SIZE parameter only in the Abaqus/Explicit analysis. If you do not specify the STEP SIZE parameter, the step size is computed automatically.

Required parameter for use with the *CO-SIMULATION, PROGRAM=MULTIPHYSICS option:

COUPLING SCHEME

Set COUPLING SCHEME=GAUSS-SEIDEL to select a Gauss-Seidel coupling algorithm (also referred to as a serial coupling scheme) where the simulations are executed in sequential order.

Set COUPLING SCHEME=JACOBI to select the Jacobi coupling algorithm (also referred to as a parallel coupling scheme) where both simulations are executed concurrently, exchanging fields to update the respective solutions at the next target time.

Required parameter for COUPLING SCHEME=GAUSS-SEIDEL:

SCHEME MODIFIER

Set SCHEME MODIFIER=LEAD if Abaqus leads the co-simulation. In this case the third-party analysis program needs to lag the co-simulation.

Set SCHEME MODIFIER=LAG if Abaqus lags the co-simulation. In this case the third-party analysis program needs to lead the co-simulation.

Set SCHEME MODIFIER=SEND PREDICTOR if Abaqus lags the co-simulation and can compute predictor fields for the next coupling step. In this case the third-party analysis program needs to lead the co-simulation and be able to receive predictor fields.

Set SCHEME MODIFIER=RECEIVE PREDICTOR if Abaqus leads the co-simulation and can receive predictor fields to advance the co-simulation. In this case the third-party analysis program needs to lag the co-simulation and be able to send predictor fields for the upcoming coupling step.

Optional parameters:

FACTORIZATION FREQUENCY

This parameter is valid only for an Abaqus/Standard analysis used with the parameter TIME INCREMENTATION=SUBCYCLE and *CO-SIMULATION, PROGRAM=ABAQUS option.

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.

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

TIME MARKS

Set TIME MARKS=YES (default) to enforce the target time in an exact manner; that is, Abaqus will temporarily cut back the increment such that the exchange occurs at the specified target time.

Set TIME MARKS=NO to enforce the target time in a loose manner. This setting is applicable only when TIME INCREMENTATION=SUBCYCLE is used.

This parameter does not apply when using the *CO-SIMULATION, PROGRAM=ABAQUS option; the target time will always be enforced in an exact manner.

There are no data lines associated with this option.

3.76 *CO-SIMULATION REGION: Identify the interface regions in the Abaqus model and specify the fields to be exchanged during co-simulation.

This option is used to identify the regions across which data will be exchanged and to specify the fields to be passed across those regions. It must be used in conjunction with the *CO-SIMULATION option to identify the analysis program for co-simulation with Abaqus.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

Defining a co-simulation region for exchange with a third-party analysis program

This section defines the use of this option when the PROGRAM parameter for the *CO-SIMULATION option is set to any value other than ABAQUS.

References:

- "Co-simulation: overview," Section 14.1.1 of the Abaqus Analysis User's Manual
- "Preparing an Abaqus/Standard or Abaqus/Explicit analysis for co-simulation," Section 14.1.2 of the Abaqus Analysis User's Manual
- *CO-SIMULATION

Optional parameter:

REGION ID

Integer identifier of the co-simulation region. The default value is 1. The third-party analysis program uses this integer identifier to distinguish one region from another when multiple *CO-SIMULATION REGION options are associated with a single *CO-SIMULATION option.

Optional parameters (mutually exclusive—if neither parameter is specified, Abaqus selects the coordinates for export and the concentrated forces for import):

EXPORT

Include this parameter to specify fields and accompanying regions for export to the third-party analysis program.

IMPORT

Include this parameter to specify fields and accompanying regions for import from the third-party analysis program.

Data lines to define the co-simulation regions and the fields passed across them if the EXPORT or IMPORT parameter is included:

First line:

1. The name of the element-based surface.
2. Field identifier for the field to be passed across this surface. The field identifiers are defined in "Preparing an Abaqus/Standard or Abaqus/Explicit analysis for co-simulation," Section 14.1.2 of the Abaqus Analysis User's Manual.
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 surface or to define additional co-simulation regions and their import or export fields.

Defining a co-simulation region for exchange with another Abaqus analysis

This section defines the use of this option for Abaqus/Standard to Abaqus/Explicit co-simulation (*CO-SIMULATION, PROGRAM=ABAQUS).

References:

- "Co-simulation: overview," Section 14.1.1 of the Abaqus Analysis User's Manual
- "Preparing an Abaqus/Standard or Abaqus/Explicit analysis for co-simulation," Section 14.1.2 of the Abaqus Analysis User's Manual
- *CO-SIMULATION

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.

3.77 *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 6.5.4 of the Abaqus Analysis User's Manual
- "Rate-dependent plasticity: creep and swelling," Section 20.2.4 of the Abaqus Analysis User's Manual

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 Manual).

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 20.2.4 of the Abaqus Analysis User's Manual 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.

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 Manual) 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 Manual). If this parameter is omitted, the stabilization algorithm is not activated. If this parameter is included without a specified value, the default value of the dissipated energy fraction is 2×10^{-4} and the adaptive automatic damping algorithm will be activated by default with ALLSDTOL =0.05 in this step; set ALLSDTOL=0 to deactivate the adaptive automatic damping algorithm. If the FACTOR parameter is used, any value of the dissipated energy fraction will be 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.

3.78 *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 6.7.2 of the Abaqus Analysis User's Manual

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.

3.79 *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 31.3.2 of the Abaqus Analysis User's Manual
- "Element-based surface definition," Section 2.3.2 of the Abaqus Analysis User's Manual
- "Node-based surface definition," Section 2.3.3 of the Abaqus Analysis User's Manual

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 node. If the name of a node set is chosen, the node set must contain exactly one node.

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 Manual) 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.

3.80 *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 and coupled thermal-electrical analyses.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

Reference:

- "Thermal loads," Section 30.4.4 of the Abaqus Analysis User's Manual

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 "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). 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, θ^0. (Units of θ.)
4. Emissivity, ϵ.

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

3.81 *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 20.2.4 of the Abaqus Analysis User's Manual
- "Anisotropic yield/creep," Section 20.2.6 of the Abaqus Analysis User's Manual
- "CREEP," Section 1.1.1 of the Abaqus User Subroutines Reference Manual

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 18.1.2 of the Abaqus Analysis User's Manual, 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**.

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. 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.

3.82 *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 20.2.4 of the Abaqus Analysis User's Manual

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 30.1.2 of the Abaqus Analysis User's Manual).

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 18.1.2 of the Abaqus Analysis User's Manual) 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 18.1.2 of the Abaqus Analysis User's Manual, 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.

3.83 *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 20.3.5 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, 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. ν_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.

3.84 *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 20.3.5 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines to define crushable foam hardening:

First line:

1. σ_c, yield stress in uniaxial compression, given as a positive value.
2. Absolute value of the corresponding plastic strain. (The first tabular value entered must always be zero.)
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

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

1. Sixth field variable.

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

Repeat this set of data lines as often as necessary to define the dependence of the yield stress in uniaxial compression on the corresponding axial plastic strain and, if needed, on temperature and other predefined field variables.

3.85 *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 20.2.12 of the Abaqus Analysis User's Manual

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.

3.86 *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 37.1.1 of the Abaqus Analysis User's Manual
- *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 3.86–1. 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 3.86–2. 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 3.86–1).
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 3.86–2).
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.

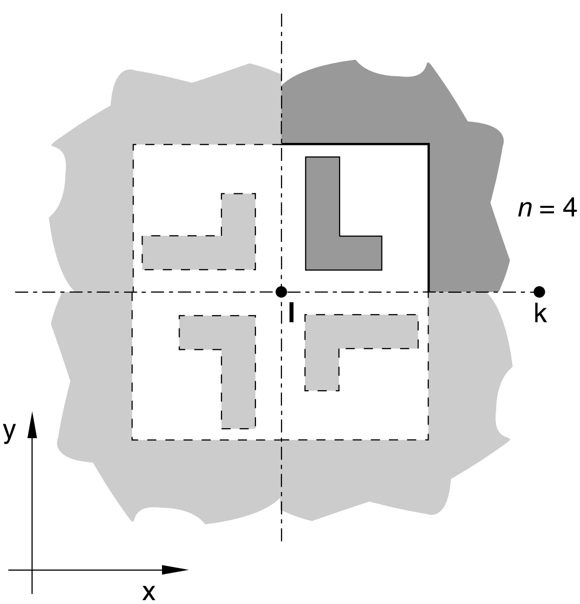

Figure 3.86–1 *CYCLIC, TYPE=POINT option.

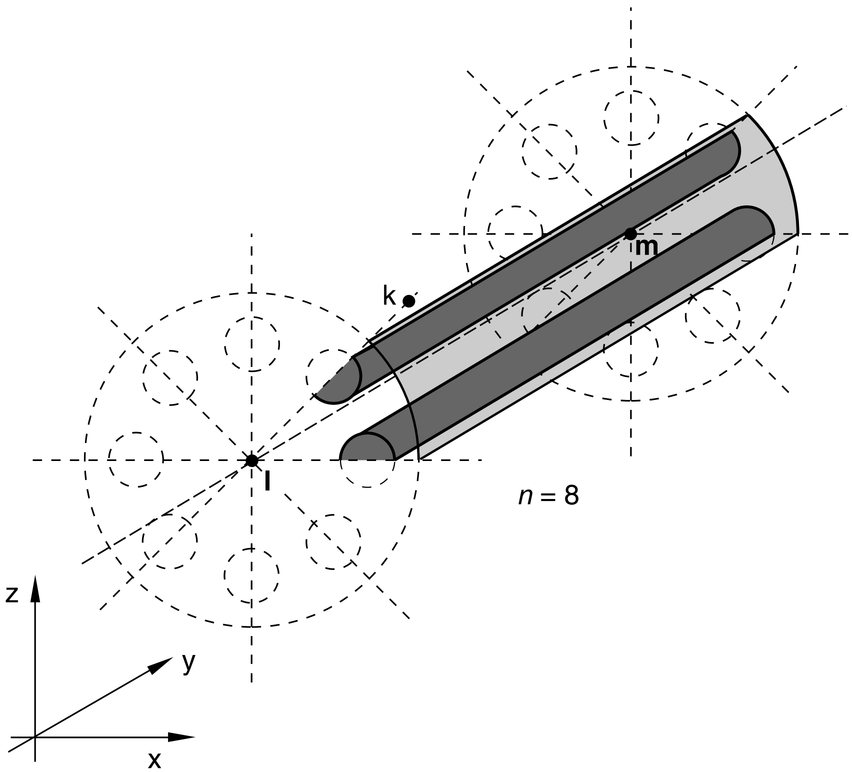

Figure 3.86–2 *CYCLIC, TYPE=AXIS option.

3.87 *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 20.2.2 of the Abaqus Analysis User's Manual
- *PLASTIC
- "UHARD," Section 1.1.31 of the Abaqus User Subroutines Reference Manual
- "VUHARD," Section 1.2.14 of the Abaqus User Subroutines Reference Manual

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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

PARAMETERS

Include this parameter to provide the material parameters Q_∞ 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_∞.
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.

3.88 *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 Manual
- *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 Manual

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 *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 21.2.3 of the Abaqus Analysis User's Manual
- "Damage evolution and element removal for fiber-reinforced composites," Section 21.3.3 of the Abaqus Analysis User's Manual
- "Damage evolution for ductile materials in low-cycle fatigue," Section 21.4.3 of the Abaqus Analysis User's Manual
- "Defining the constitutive response of cohesive elements using a traction-separation description," Section 29.5.6 of the Abaqus Analysis User's Manual
- "Surface-based cohesive behavior," Section 33.1.10 of the Abaqus Analysis User's Manual
- "Modeling discontinuities as an enriched feature using the extended finite element method," Section 10.6.1 of the Abaqus Analysis User's Manual

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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

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 29.5.6 of the Abaqus Analysis User's Manual, or "Defining damage evolution data as a tabular function of mode mix" in "Surface-based cohesive behavior," Section 33.1.10 of the Abaqus Analysis User's Manual, for further details.

Set MODE MIX RATIO=ENERGY (default) to define the mode mix in terms of a ratio of fracture energy in the different modes. This definition must 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.

4.3 *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

Defining damage initiation as a material property

References:

- "Damage initiation for ductile metals," Section 21.2.2 of the Abaqus Analysis User's Manual
- "Damage initiation for fiber-reinforced composites," Section 21.3.2 of the Abaqus Analysis User's Manual
- "Damage initiation for ductile materials in low-cycle fatigue," Section 21.4.2 of the Abaqus Analysis User's Manual
- "Defining the constitutive response of cohesive elements using a traction-separation description," Section 29.5.6 of the Abaqus Analysis User's Manual
- "Modeling discontinuities as an enriched feature using the extended finite element method," Section 10.6.1 of the Abaqus Analysis User's Manual

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.

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 α.

Set DEFINITION=FLD to specify the MSFLD damage initiation criterion by providing the limit major strain as a tabular function of minor strain.

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the damage initiation properties. If this parameter is omitted, it is assumed that the damage initiation properties are constant or depend only on temperature. This parameter cannot be used with CRITERION=JOHNSON COOK.

FEQ

This parameter can be used only in conjunction with CRITERION=MK.

Set this parameter equal to the critical value of the deformation severity index for equivalent plastic strains, f_{eq}^{crit}. The default value is $f_{eq}^{\mathrm{crit}} = 10$.

Set this parameter equal to zero if the deformation severity factor for equivalent plastic strains should not be considered for the evaluation of the Marciniak-Kuczynski criterion.

FNN

This parameter can be used only in conjunction with CRITERION=MK.

Set this parameter equal to the critical value of the deformation severity index for strains normal to the groove direction, f_{nn}^{crit}. The default value is $f_{nn}^{\mathrm{crit}} = 10$.

Set this parameter equal to zero if the deformation severity factor for strains normal to the groove should not be considered for the evaluation of the Marciniak-Kuczynski criterion.

FNT

This parameter can be used only in conjunction with CRITERION=MK.

Set this parameter equal to the critical value of the deformation severity index for shear strains, f_{nt}^{crit}. The default value is $f_{nt}^{\mathrm{crit}} = 10$.

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 $k_s = 0$.

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 ω used for filtering the ratio of principal strain rates used for the evaluation of the MSFLD damage initiation criterion. The default value is $\omega = 1.0$.

PEINC

This parameter can be used only in conjunction with CRITERION=MSFLD in Abaqus/Explicit.

Set this parameter equal to the accumulated increment in equivalent plastic strain used to trigger the evaluation of the MSFLD damage initiation criterion. The default value is 0.002 (0.2 %).

TOLERANCE

This parameter can be used only in conjunction with CRITERION=MAXPE, CRITERION=MAXPS, CRITERION=MAXE, CRITERION=MAXS, CRITERION=QUADE, or CRITERION=QUADS 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:

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=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, θ_{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, α.
3. Equivalent plastic strain rate.
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

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

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

Repeat this set of data lines as often as necessary to define the equivalent plastic strain at damage initiation as a function of α, equivalent plastic strain rate, temperature, and other predefined field variables.

Data lines to specify damage initiation for CRITERION=MSFLD, DEFINITION=FLD:

First line:

1. Major principal strain at initiation of localized necking.
2. Minor principal strain.
3. Equivalent plastic strain rate.
4. Temperature, if temperature dependent.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

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

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

Repeat this set of data lines as often as necessary to define the major principal strain at damage initiation as a function of minor principal strain, equivalent plastic strain rate, temperature, and other predefined field variables.

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.

Defining damage initiation as part of a contact property model

Reference:

- "Surface-based cohesive behavior," Section 33.1.10 of the Abaqus Analysis User's Manual

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.

4.4 *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

Reference:

- "Viscous regularization" in "Damage evolution and element removal for fiber-reinforced composites," Section 21.3.3 of the Abaqus Analysis User's Manual
- "Surface-based cohesive behavior," Section 33.1.10 of the Abaqus Analysis User's Manual
- "Modeling discontinuities as an enriched feature using the extended finite element method," Section 10.6.1 of the Abaqus Analysis User's Manual

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.

4.5 *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 23.1.1 of the Abaqus Analysis User's Manual.

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 23.1.1 of the Abaqus Analysis User's Manual
- "Dynamic analysis procedures: overview," Section 6.3.1 of the Abaqus Analysis User's Manual
- "Explicit dynamic analysis," Section 6.3.3 of the Abaqus Analysis User's Manual

Optional parameters:

ALPHA

Set this parameter equal to the α_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 follow Lanczos or subspace iteration eigenvalue extraction.

The default is ALPHA=0. (Units of T^{-1}.)

BETA

Set this parameter equal to the β_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 follow Lanczos or subspace iteration eigenvalue extraction.

The default is BETA=0. (Units of T.)

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 Lanczos or subspace iteration eigenvalue extraction, except for *STEADY STATE DYNAMICS, SUBSPACE PROJECTION. Use *MODAL DAMPING, COMPOSITE option to activate it.

The default is COMPOSITE=0.

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

This parameter is ignored in mode-based procedures that follow a Lanczos or subspace iteration eigenvalue extraction.

The default is STRUCTURAL=0.

There are no data lines associated with this option.

4.6 *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 Manual, for details).

Product: Abaqus/Standard

Type: Model or history data

Level: Part, Part instance

References:

- "Material damping," Section 23.1.1 of the Abaqus Analysis User's Manual
- "Damping in dynamic analysis" in "Dynamic analysis procedures: overview," Section 6.3.1 of the Abaqus Analysis User's Manual
- *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.

4.7 *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 29.2.1 of the Abaqus Analysis User's Manual
- "Flexible joint element," Section 29.3.1 of the Abaqus Analysis User's Manual
- "Tube support elements," Section 29.9.1 of the Abaqus Analysis User's Manual

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 18.1.2 of the Abaqus Analysis User's Manual, 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 Manual).

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 18.1.2 of the Abaqus Analysis User’s Manual, 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.

4.8 *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.

Product: Abaqus/Standard

Type: History data

Level: Step

References:

- "Crack propagation analysis," Section 11.4.3 of the Abaqus Analysis User's Manual
- *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:

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.

4.9 *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 6.7.3 of the Abaqus Analysis User’s Manual

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 (“Procedures: overview,” Section 6.1.1 of the Abaqus Analysis User’s Manual).

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 Manual).
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.

4.10 *DECURRENT: Specify distributed current densities in an electric conduction analysis.

This option is used to input distributed current densities in a coupled thermal-electrical analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

Reference:

- "Coupled thermal-electrical analysis," Section 6.7.2 of the Abaqus Analysis User's Manual

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 30.1.2 of the Abaqus Analysis User's Manual). 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 ("Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual).

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.

Data lines to define distributed electrical current densities:

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 Manual).
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.

4.11 *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 20.2.13 of the Abaqus Analysis User's Manual

There are no parameters associated with this option.

Data lines to define deformation plasticity:

First line:

1. Young's modulus, E.
2. Poisson's ratio, ν.
3. Yield stress, σ^0.
4. Exponent, n.
5. Yield offset, α.
6. Temperature.

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

4.12 *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 2.7.1 of the Abaqus Analysis User's Manual).

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Reference:

- "Density," Section 18.2.1 of the Abaqus Analysis User's Manual

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 18.1.2 of the Abaqus Analysis User's Manual, 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 2.7.1 of the Abaqus Analysis User's Manual.

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}.

4.13 *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 15.1.1 of the Abaqus Analysis User's Manual
- "User-defined mechanical material behavior," Section 23.8.1 of the Abaqus Analysis User's Manual

Optional parameter:

DELETE

This parameter applies only to Abaqus/Explicit analyses.

Set this Abaqus/Explicit parameter equal to the state variable number controlling the element deletion flag (see "User-defined mechanical material behavior," Section 23.8.1 of the Abaqus Analysis User's Manual).

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 Manual), 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 Manual), 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.

4.14 *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 16.1.1 of the Abaqus Analysis User's Manual
- *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.

4.15 *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 16.1.1 of the Abaqus Analysis User's Manual
- *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.

4.16 *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 16.1.1 of the Abaqus Analysis User's Manual
- *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.

4.17 *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 22.2.1 of the Abaqus Analysis User's Manual

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 Manual). The default is 0.

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

4.18 *DFLOW: Specify distributed seepage flows for consolidation analysis.

This option is used to input seepage flows (pore fluid velocities normal to surfaces of the model) in consolidation problems.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- "Pore fluid flow," Section 30.4.6 of the Abaqus Analysis User's Manual
- "DFLOW," Section 1.1.2 of the Abaqus User Subroutines Reference Manual

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 "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). 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 Manual).

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 Manual).
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.

4.19 *DFLUX: Specify distributed fluxes in heat transfer 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.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- "Thermal loads," Section 30.4.4 of the Abaqus Analysis User's Manual
- "DFLUX," Section 1.1.3 of the Abaqus User Subroutines Reference Manual

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 30.1.2 of the Abaqus Analysis User's Manual).

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 ("Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). If this parameter is omitted in an Abaqus/Explicit 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 Manual).
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.

4.20 *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 written to the status (`.sta`) file or to the message (`.msg`) file if problems are detected during an analysis.

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.

Product: Abaqus/Explicit

Type: History data

Level: Step

References:

- "Explicit dynamic analysis," Section 6.3.3 of the Abaqus Analysis User's Manual
- "Output and diagnostics for ALE adaptive meshing in Abaqus/Explicit," Section 12.2.5 of the Abaqus Analysis User's Manual
- "Contact diagnositcs in an Abaqus/Explicit analysis," Section 35.2.1 of the Abaqus Analysis User's Manual
- *CONTACT CONTROLS

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 cut-off 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.

4.21 *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 23.6.2 of the Abaqus Analysis User's Manual

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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

TYPE

Set TYPE=ISO (default) for isotropic behavior. Set TYPE=ORTHO for orthotropic behavior. Set TYPE=ANISO for fully anisotropic behavior.

Data lines to define isotropic behavior (TYPE=ISO):

First line:

1. Dielectric constant. (Units of $C\varphi^{-1}L^{-1}$.)
2. Temperature, θ.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

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

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

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

Data lines to define orthotropic behavior (TYPE=ORTHO):

First line:

1. $D_{11}^{\varphi(\varepsilon)}$. (Units of C$\varphi^{-1}$L^{-1}.)
2. $D_{22}^{\varphi(\varepsilon)}$.
3. $D_{33}^{\varphi(\varepsilon)}$.
4. Temperature, θ.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

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

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

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

Data lines to define anisotropic behavior (TYPE=ANISO):

First line:

1. $D_{11}^{\varphi(\varepsilon)}$. (Units of C$\varphi^{-1}$L^{-1}.)
2. $D_{12}^{\varphi(\varepsilon)}$.
3. $D_{22}^{\varphi(\varepsilon)}$.
4. $D_{13}^{\varphi(\varepsilon)}$.
5. $D_{23}^{\varphi(\varepsilon)}$.
6. $D_{33}^{\varphi(\varepsilon)}$.
7. Temperature, θ.
8. First field variable.

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

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

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

4.22 *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 23.5.1 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, 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, θ.
4. First field variable.

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

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

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

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

Data lines to define orthotropic diffusivity (TYPE=ORTHO):

First line:

1. D_{11}. (Units of L^2T^{-1}.)
2. D_{22}.
3. D_{33}.
4. Concentration, c.
5. Temperature, θ.
6. First field variable.
7. Second field variable.
8. Third field variable

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

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

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

Data lines to define anisotropic diffusivity (TYPE=ANISO):

First line:

1. D_{11}. (Units of L^2T^{-1}.)
2. D_{12}.
3. D_{22}.
4. D_{13}.
5. D_{23}.
6. D_{33}.
7. Concentration, c.
8. Temperature, θ.

Subsequent lines (only needed if the DEPENDENCIES parameter is used):

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

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

4.23 *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 Manual
- "Low-cycle fatigue analysis using the direct cyclic approach," Section 6.2.7 of the Abaqus Analysis User's Manual
- *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 100. 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 100. 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 100. 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 100. 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. The default is 100.
2. Maximum increment in number of cycles over which the damage is extrapolated forward. It must be greater than 0. The default is 1000.

3. Total number of cycles allowed in a step. If this entry is zero or not specified, the default value is equal to one plus half of the maximum increment in number of cycles over which the damage is extrapolated.
4. Damage extrapolation tolerance. The maximum extrapolated damage increment will be limited by this value. The default is 1.0.

4.24 *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 2.8.1 of the Abaqus Analysis User's Manual

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).

4.25 *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 31.3.2 of the Abaqus Analysis User's Manual
- *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 Manual, 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).

4.26 *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 29.4.1 of the Abaqus Analysis User's Manual

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.

4.27 *DISTRIBUTION: Define spatial distributions.

This option is used to define a spatial distribution.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly, Model

Abaqus/CAE: Property module

References:

- "Distribution definition," Section 2.7.1 of the Abaqus Analysis User's Manual
- *CONTACT CLEARANCE
- *DENSITY
- *DISTRIBUTION TABLE
- *ELASTIC
- *EXPANSION
- *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.

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 Manual, 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 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 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. ν.

Repeat this data line as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of orthotropic elastic moduli:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent uses of this data line.
2. D_{1111}.
3. D_{1122}.
4. D_{2222}.
5. D_{1133}.
6. D_{2233}.
7. D_{3333}.
8. D_{1212}.

Second line:

1. D_{1313}.
2. D_{2323}.

Repeat this set of data lines as often as necessary to define data for element numbers or element sets.

Data lines to define a distribution of orthotropic elastic moduli using engineering constants:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent uses of this data line.
2. E_1.
3. E_2.
4. E_3.
5. ν_{12}.
6. ν_{13}.
7. ν_{23}.
8. G_{12}.

Second line:

1. G_{13}.
2. G_{23}.

Repeat this set of data lines as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of orthotropic elastic moduli in plane stress:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. E_1.
3. E_2.
4. ν_{12}.
5. G_{12}.
6. G_{13}. This shear modulus is needed to define transverse shear behavior in shells.
7. G_{23}. This shear modulus is needed to define transverse shear behavior in shells.

Repeat this data line as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of anisotropic elastic moduli:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent uses of this data line.
2. D_{1111}.
3. D_{1122}.
4. D_{2222}.

5. D_{1133}.
6. D_{2233}.
7. D_{3333}.
8. D_{1112}.

Second line:

1. D_{2212}.
2. D_{3312}.
3. D_{1212}.
4. D_{1113}.
5. D_{2213}.
6. D_{3313}.
7. D_{1213}.
8. D_{1313}.

Third line:

1. D_{1123}.
2. D_{2223}.
3. D_{3323}.
4. D_{1223}.
5. D_{1323}.
6. D_{2323}.

Repeat this set of data lines as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of mass density:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. Density.

Repeat this data line as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of isotropic thermal expansion:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. α.

Repeat this data line as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of orthotropic thermal expansion:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. α_{11}.
3. α_{22}.
4. α_{33}. (Not used for plane stress case.)

Repeat this data line as often as necessary to define the data for element numbers or element sets.

Data lines to define a distribution of anisotropic thermal expansion:

First line:

1. Blank space to define default data for the first use of this data line. Element number or element set for subsequent data lines.
2. α_{11}.
3. α_{22}.
4. α_{33}. (Not used for plane stress case.)
5. α_{12}.
6. α_{13}.
7. α_{23}.

Repeat this data line as often as necessary to define the data for element numbers or element sets.

4.28 *DISTRIBUTION TABLE: Define a distribution table.

This option is used to define a distribution table that defines the format of the data given on the data lines for a spatial distribution.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Distribution definition," Section 2.7.1 of the Abaqus Analysis User's Manual
- *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.

4.29 *DLOAD: Specify distributed loads.

This option is used to prescribe distributed loading. It is also used to apply concentrated or distributed wind, wave, or buoyancy loading in an Abaqus/Aqua analysis.

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

Type: History data

Level: Step

Abaqus/CAE: Load module

Applying distributed loads

References:

- "Distributed loads," Section 30.4.3 of the Abaqus Analysis User's Manual
- "DLOAD," Section 1.1.5 of the Abaqus User Subroutines Reference Manual
- "Defining ALE adaptive mesh domains in Abaqus/Explicit," Section 12.2.2 of the Abaqus Analysis User's Manual
- "Analysis of models that exhibit cyclic symmetry," Section 10.4.3 of the Abaqus Analysis User's Manual

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 "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). If this parameter is omitted in an Abaqus/Explicit analysis, the reference magnitude is applied immediately at the beginning of the step.

Amplitude references are ignored for nonuniform loads given by user subroutine **DLOAD** in an Abaqus/Standard analysis. Amplitude references are passed into user subroutine **VDLOAD** in an Abaqus/Explicit analysis.

Only the load magnitude is changed with time. Quantities such as the direction of an applied gravity load and the fluid surface level in hydrostatic pressure loading are not changed.

CONSTANT RESULTANT

Set CONSTANT RESULTANT=NO (default) if surface traction vectors, edge traction vectors, or edge moments are to be integrated over the surface in the current configuration.

Set CONSTANT RESULTANT=YES if surface traction vectors, edge traction vectors, or edge moments are to be integrated over the surface in the reference configuration.

The CONSTANT RESULTANT parameter is valid only for uniform and nonuniform surface tractions and edge loads (including edge moments); it is ignored for all other load types.

FOLLOWER

Set FOLLOWER=YES (default) if a prescribed traction or shell-edge load is to rotate with the surface or shell edge in a large-displacement analysis (live load).

Set FOLLOWER=NO if a prescribed traction or edge load is to remain fixed in a large-displacement analysis (dead load).

The FOLLOWER parameter is valid only for traction and edge load labels TRVEC*n*, TRVEC, TRVEC*n*NU, TRVECNU, EDLD*n*, and EDLD*n*NU. It is ignored for all other load labels.

OP

Set OP=MOD (default) for existing *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 Manual) 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 Manual).
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 gravity loading:

First line:

1. The element number or element set label is optional for gravity loads. If this field is left blank, Abaqus automatically includes all elements in the model that have mass contributions (including point mass elements) in an element set called **`_Whole_Model_Gravity_Elset`** and applies the gravity load to all elements in this element set.
2. Distributed load type label GRAV.
3. Actual magnitude of the load, which can be modified by the use of the *AMPLITUDE option.
4. 1-component of the gravity vector.
5. 2-component of the gravity vector.
6. 3-component of the gravity vector.

For axisymmetric elements the gravity load must be in the z-direction; therefore, only component 2 should be nonzero.

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

Data lines to define external and internal pressure in pipe or elbow elements:

First line:

1. Element number or element set label.
2. Distributed load type label PE, PI, PENU, or PINU.
3. Actual magnitude of the load, which can be modified by the use of the *AMPLITUDE option. For nonuniform loads the magnitude must be defined in user subroutine **DLOAD**.
4. Effective inner or outer diameter.

Repeat this data line as often as necessary to define internal or external pressure loading for different pipe or elbow elements or element sets.

Data lines to define hydrostatic pressure:

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

Reference:

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

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 "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). 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:

8. Density of the fluid outside the element. This value will override the fluid density given on the data line of the *AQUA option.
9. Free surface elevation of the fluid outside the element. This value will override the fluid surface elevation given on the data line of the *AQUA option.
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, α_R. The default value is 1.0 if this entry is left blank or set equal to 0.0.
7. For load type FDD, name of the *AMPLITUDE curve used for scaling steady current velocities (A_c). For load type WDD, name of the *AMPLITUDE curve used for scaling the local x-direction wind velocity (A_x). If this entry is blank, the velocities are not scaled ($A_c = 1$ or $A_x = 1$).
8. For load type FDD, name of the *AMPLITUDE curve used for scaling wave velocities (A_w). For load type WDD, name of the *AMPLITUDE curve used for scaling the local y-direction wind velocity (A_y). If this is blank, the velocities are not scaled ($A_w = 1$ or $A_y = 1$).

Repeat this data line as often as necessary to define distributed transverse fluid or wind drag on various elements or element sets.

Data lines to define distributed tangential fluid drag:

First line:

1. Element number or element set label.
2. Distributed load type label FDT.
3. Magnitude factor, *M* (default value is 1.0). This factor will be scaled by any *AMPLITUDE specification associated with this *DLOAD option.
4. Effective outer diameter of the member, *D*.
5. Drag coefficient, C_t.
6. Structural velocity factor, α_R. The default value is 1.0 if this entry is left blank or set equal to 0.0.
7. Exponent *h*. The default value is 2.0 if this entry is left blank or set equal to 0.0.
8. Name of the *AMPLITUDE curve (A_c) used for scaling steady current velocities. If this entry is blank, the current velocities are not scaled ($A_c = 1$).
9. Name of the *AMPLITUDE curve (A_w) used for scaling wave velocities. If this entry is blank, the wave velocities are not scaled ($A_w = 1$).

Repeat this data line as often as necessary to define distributed tangential fluid drag on various elements or element sets.

Data lines to define distributed fluid inertia loading:

First line:

1. Element number or element set label.
2. Distributed load type label FI.
3. Magnitude factor, *M* (default value is 1.0). This factor will be scaled by any *AMPLITUDE specification associated with this *DLOAD option.
4. Effective outer diameter of the member, *D*.
5. Transverse fluid inertia coefficient, C_M.
6. Transverse added-mass coefficient, C_A.
7. Name of the *AMPLITUDE curve used for scaling fluid particle accelerations (A_w). If this entry is blank, the fluid particle accelerations are not scaled ($A_w = 1$).

Repeat this data line as often as necessary to define fluid inertia loading for various elements or element sets.

Data lines to define concentrated fluid and wind drag loading on the ends of elements:

First line:

1. Element number or element set label.
2. Distributed load type label FD1, FD2, WD1, or WD2.

3. Magnitude factor, M (default value is 1.0). This factor will be scaled by any AMPLITUDE specification associated with this *DLOAD option.
4. Exposed area, ΔA.
5. Drag coefficient, C.
6. Structural velocity factor, α_R. The default value is 1.0 if this entry is left blank or set equal to 0.0.
7. For load types FD1 or FD2, name of the *AMPLITUDE curve used for scaling steady current velocities (A_c). For load types WD1 or WD2, name of the *AMPLITUDE curve used for scaling the local x-direction wind velocity (A_x). If this entry is blank, the velocities are not scaled ($A_c = 1$ or $A_x = 1$).
8. For load types FD1 or FD2, name of the *AMPLITUDE curve used for scaling wave velocities (A_w). For load types WD1 or WD2, name of the *AMPLITUDE curve used for scaling the local y-direction wind velocity (A_y). If this entry is blank, the velocities are not scaled ($A_w = 1$ or $A_y = 1$).

Repeat this data line as often as necessary to define concentrated fluid or wind drag loading on the ends of elements.

Data lines to define concentrated fluid inertia loading on the ends of elements:

First line:

1. Element number or element set label.
2. Distributed load type label FI1 or FI2.
3. Magnitude factor, M (default value is 1.0). This factor will be scaled by any AMPLITUDE specification associated with this *DLOAD option.
4. Fluid inertia coefficient, K_{ts}.
5. Fluid acceleration shape factor, F_{1s}.
6. Added-mass coefficient, L_{ts}.
7. Structural acceleration shape factor, F_{2s}.
8. Name of the *AMPLITUDE curve used for scaling fluid particle accelerations. If this entry is blank, the fluid particle accelerations are not scaled.

Repeat this data line as often as necessary to define concentrated fluid inertia loading on the ends of elements.

4.30 *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 29.12.1 of the Abaqus Analysis User's Manual

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, ℓ.
2. Friction limit between the chain and the seabed.

Data line for DRAG3D elements:

First (and only) line:

1. Total length of chain.
2. Friction coefficient.
3. Weight of chain (per unit length).

4.31 *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 20.3.1 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

ECCENTRICITY

This parameter is only for use with SHEAR CRITERION=HYPERBOLIC or SHEAR CRITERION=EXPONENT FORM or if creep material properties are included with SHEAR CRITERION=LINEAR.

It is used to define the flow potential eccentricity, ϵ. The eccentricity is a small positive number that defines the rate at which the hyperbolic flow potential approaches its asymptote. The default is $\epsilon = 0.1$ for the exponent model; and if $\psi = \beta$, it is set to $\epsilon = (d'|_0 - p_t|_0 \tan\beta)/(\bar{\sigma}|_0 \tan\beta)$ for the hyperbolic model to ensure associated flow (the terms are defined in "Extended Drucker-Prager models," Section 20.3.1 of the Abaqus Analysis User's Manual).

SHEAR CRITERION

Set SHEAR CRITERION=LINEAR (default) to define the linear yield criterion. This is required if creep material behavior is included for an Abaqus/Standard analysis.

Set SHEAR CRITERION=HYPERBOLIC to define the hyperbolic yield criterion.

Set SHEAR CRITERION=EXPONENT FORM to define the exponent form as a yield criterion.

TEST DATA

This parameter is only for use with SHEAR CRITERION=EXPONENT FORM.

Include this parameter if the material constants for the exponent model are to be computed by Abaqus from triaxial test data at different levels of confining pressure. The *TRIAXIAL TEST DATA option must be used for this purpose.

Data lines to define a linear Drucker-Prager plasticity model (SHEAR CRITERION=LINEAR):

First line:

1. Material angle of friction, β, in the p–t plane. Give the value in degrees.
2. K, the ratio of the flow stress in triaxial tension to the flow stress in triaxial compression. $0.778 \leq K \leq 1.0$. If this field is left blank or a value of 0.0 is entered, the default of 1.0 is used. If creep material behavior is included, K should be set to 1.0.
3. Dilation angle, ψ, in the p–t plane. Give the value in degrees.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

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

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

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

Data lines to define a hyperbolic Drucker-Prager plasticity model (SHEAR CRITERION=HYPERBOLIC):

First line:

1. Material angle of friction, β, at high confining pressure in the p–q plane. Give the value in degrees.
2. Initial hydrostatic tension strength, $p_t|_0$. (Units of FL^{-2}.)
3. Not used.
4. Dilation angle, ψ, at high confining pressure in the p–q plane. Give the value in degrees.

5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

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

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

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

Data lines to define a Drucker-Prager plasticity model with the exponent law (SHEAR CRITERION=EXPONENT FORM) and without test data (TEST DATA):

First line:

1. Material constant a.
2. Exponent b. To ensure a convex yield surface, $b \geq 1$.
3. Not used.
4. Dilation angle, ψ, at high confining pressure in the p–q plane. Give the value in degrees.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

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

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

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

Data lines to define a Drucker-Prager plasticity model with the exponent law (SHEAR CRITERION=EXPONENT FORM) and with test data (TEST DATA):

First line:

1. Not used.
2. Not used.
3. Not used.
4. Dilation angle, ψ, at high confining pressure in the p–q plane. Give the value in degrees.
5. Temperature.
6. First field variable.

7. Second field variable.
8. Third field variable.

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

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

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

4.32 *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 20.3.1 of the Abaqus Analysis User's Manual
- *DRUCKER PRAGER
- *DRUCKER PRAGER HARDENING
- "CREEP," Section 1.1.1 of the Abaqus User Subroutines Reference Manual

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 18.1.2 of the Abaqus Analysis User's Manual, 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**.

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 T^{-1}.)
2. α. (Units of $F^{-1}L^{2}$.)
3. m.
4. t_1. (Units of T.)
5. Temperature.
6. First field variable.
7. 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.

4.33 *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 20.3.1 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

RATE

Set this parameter equal to the equivalent plastic strain rate, $\dot{\bar{\varepsilon}}^{pl}$, for which this hardening curve applies. This parameter should be omitted if the *RATE DEPENDENT option or the *DRUCKER PRAGER CREEP option is used. Rate-independent behavior is assumed if the RATE parameter, the *RATE DEPENDENT option, and the *DRUCKER PRAGER CREEP option are not used.

TYPE

Set TYPE=COMPRESSION (default) to define the hardening behavior by giving the uniaxial compression yield stress, σ_c, as a function of uniaxial compression plastic strain, $\bar{\varepsilon}^{pl} = |\varepsilon_{11}^{pl}|$.

Set TYPE=TENSION to define the hardening behavior by giving the uniaxial tension yield stress, σ_t, as a function of uniaxial tension plastic strain, $\bar{\varepsilon}^{pl} = \varepsilon_{11}^{pl}$.

Set TYPE=SHEAR to define the hardening behavior by giving the cohesion, $d = \frac{\sqrt{3}}{2}\tau(1+\frac{1}{K})$, as a function of equivalent shear plastic strain, $\bar{\varepsilon}^{pl} = \gamma^{pl}/\sqrt{3}$, where τ is the yield stress in shear, K is the ratio of flow stress in triaxial tension to the flow stress in triaxial compression, and γ^{pl} is the engineering shear plastic strain.

Data lines to define Drucker-Prager hardening:

First line:

1. Yield stress.
2. Absolute value of the corresponding plastic strain. (The first tabular value entered must always be zero.)
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

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

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

Repeat this set of data lines as often as necessary to define the dependence of yield stress on plastic strain and, if needed, on temperature and other predefined field variables.

4.34 *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 16.1.1 of the Abaqus Analysis User's Manual

Optional parameters:

FORMULATION

Use this parameter to select the design sensitivity analysis formulation type in a multi-increment analysis. This parameter will be ignored if used as history data.

Set FORMULATION=INCREMENTAL (default) to select incremental design sensitivity analysis.

Set FORMULATION=TOTAL to select total design sensitivity analysis.

RESET

Include this parameter to reset the values to those specified on the model data options or to the original default values if no model data options exist. This action takes effect before applying any additional changes to the values.

SIZING FREQUENCY

Set this parameter equal to the frequency in increments (static steps) or modes (frequency steps) at which the default perturbation sizing algorithm is to be executed. The algorithm will always be executed for the first increment or first eigenmode in each step for which DSA calculations are done, even if SIZING FREQUENCY is set to 0. The default is SIZING FREQUENCY=0.

TOLERANCE

Set this parameter equal to the tolerance to be used with the default perturbation sizing algorithm. The default is TOLERANCE=1.0×10^{-4}.

Data lines to override the default perturbation sizing algorithm for selected design parameters (The SIZING FREQUENCY and TOLERANCE parameters will be ignored for these design parameters.):

First line:

1. Design parameter.
2. Set this entry to FD to use forward difference. Set this entry to CD to use central difference.
3. Absolute value of perturbation.

Repeat this data line for each design parameter for which the default algorithm is to be overridden.

4.35 *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 6.7.3 of the Abaqus Analysis User's Manual

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 ("Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual).

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.

4.36 *DSECURRENT: Specify distributed current densities over a surface in an electric conduction analysis.

This option is used to input distributed current densities over a surface in a coupled thermal-electrical analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

Reference:

- "Coupled thermal-electrical analysis," Section 6.7.2 of the Abaqus Analysis User's Manual

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 30.1.2 of the Abaqus Analysis User's Manual). 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 ("Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual).

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.

Data lines to define distributed electrical current densities:

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.

4.37 *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 30.4.6 of the Abaqus Analysis User's Manual
- "DFLOW," Section 1.1.2 of the Abaqus User Subroutines Reference Manual

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 "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). 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.

4.38 *DSFLUX: Specify distributed surface fluxes for heat transfer analysis.

This option is used to apply distributed surface fluxes in fully coupled thermal-stress analysis. In Abaqus/Standard it is also used for heat transfer and coupled thermal-electrical analyses.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- "Thermal loads," Section 30.4.4 of the Abaqus Analysis User's Manual
- "DFLUX," Section 1.1.3 of the Abaqus User Subroutines Reference Manual

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 30.1.2 of the Abaqus Analysis User's Manual).

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 ("Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). If this parameter is omitted in an Abaqus/Explicit analysis, the reference magnitude is applied immediately at the beginning of the step.

For nonuniform flux type SNU (which is available only in Abaqus/Standard), the flux magnitude is defined in user subroutine **DFLUX**, and AMPLITUDE references are ignored.

OP

Set OP=MOD (default) for existing *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.

4.39 *DSLOAD: Specify distributed surface loads.

This option is used to prescribe distributed surface loading.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

Applying distributed loads

References:

- "Distributed loads," Section 30.4.3 of the Abaqus Analysis User's Manual
- "DLOAD," Section 1.1.5 of the Abaqus User Subroutines Reference Manual
- "Analysis of models that exhibit cyclic symmetry," Section 10.4.3 of the Abaqus Analysis User's Manual

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 "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). 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 Manual) 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)

References:

- "Submodeling: overview," Section 10.2.1 of the Abaqus Analysis User's Manual
- "Surface-based submodeling," Section 10.2.3 of the Abaqus Analysis User's Manual

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 6.1.2 of the Abaqus Analysis User's Manual).

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.

4.40 *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

Defining a dynamic analysis in Abaqus/Standard

References:

- "Implicit dynamic analysis using direct integration," Section 6.3.2 of the Abaqus Analysis User's Manual
- "Adiabatic analysis," Section 6.5.5 of the Abaqus Analysis User's Manual

Optional parameter for the subspace projection method:

SUBSPACE

Include this parameter to choose the subspace projection method (explicit integration of the model projected onto the eigenvectors obtained in the last *FREQUENCY step preceding this step).

If this parameter is omitted, implicit time integration of the dynamic equations for all global level degrees of freedom is used.

Optional parameters for the general implicit integration method:

ADIABATIC

Include this parameter if an adiabatic stress analysis is to be performed. This parameter is relevant only for isotropic metal plasticity materials with a Mises yield surface and when the *INELASTIC HEAT FRACTION option has been specified.

ALPHA

Set this parameter equal to a nondefault value of the numerical (artificial) damping control parameter, α, in the implicit operator for TIME INTEGRATOR=HHT-TF, HHT-MD, or HYBRID. Allowable values are 0 (no damping) to –0.5. The value of −0.333 provides maximum damping. The default for TIME INTEGRATOR=HHT-TF is ALPHA=−0.05, which provides slight numerical damping.

APPLICATION

Use this parameter to choose a time integration method. Other parameter values are determined by the time integration method selected. You can override the defaults by specifying these parameter values directly.

Set APPLICATION=TRANSIENT FIDELITY (default for problems without contact in the model) to choose a method for an accurate solution with slight numerical damping. The TIME INTEGRATOR=HHT-TF, IMPACT=AVERAGE TIME, and INCREMENTATION=CONSERVATIVE are set.

Set APPLICATION=MODERATE DISSIPATION (default for problems with contact in the model) to choose a method with larger than default numerical damping and a more aggressive time incrementation scheme at the expense of some solution accuracy. The TIME INTEGRATOR=HHT-MD, IMPACT=NO, and INCREMENTATION=AGGRESSIVE are set.

Set APPLICATION=QUASI-STATIC to choose a method with very significant numerical damping that is primarily intended to obtain quasi-static solutions. The TIME INTEGRATOR=BWE, IMPACT=NO, and INCREMENTATION=AGGRESSIVE values are set. In addition, the default step amplitude is set to RAMP instead of STEP.

BETA

Set this parameter equal to a nondefault value, β, in the implicit operator for TIME INTEGRATOR=HHT-TF, HHT-MD, or HYBRID. Allowable values are positive.

DIRECT

Include this parameter to choose direct user control of the incrementation through the step. If this parameter is included and no contact impacts or releases occur, constant increments of the size defined on the data line are used. If this parameter is omitted, Abaqus/Standard uses the automatic time incrementation scheme after trying the user's initial time increment for the first attempt at the first increment. The DIRECT parameter and the HAFTOL and HALFINC SCALE FACTOR parameters are mutually exclusive.

The DIRECT parameter may have the value NO STOP. If this value is included, the solution to an increment is accepted after the maximum number of iterations allowed (as defined in the *CONTROLS option) have been done, even if the equilibrium tolerances are not satisfied. Small increments and a minimum of two iterations are usually necessary if this value is used. *This approach is not generally recommended; it should be used only in special cases when the analyst has a thorough understanding of how to interpret results obtained in this way.*

GAMMA

Set this parameter equal to a nondefault value, γ, in the implicit operator for TIME INTEGRATOR=HHT-TF, HHT-MD, or HYBRID. 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.

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.

Set TIME INTEGRATOR=HYBRID to choose a hybrid time integrator that closely resembles the Hilber-Hughes-Taylor time integrator with slight numerical damping except that it has fully implicit treatment of contact.

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

References:

- "Explicit dynamic analysis," Section 6.3.3 of the Abaqus Analysis User's Manual
- "Adiabatic analysis," Section 6.5.5 of the Abaqus Analysis User's Manual

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 20.1.1 of the Abaqus Analysis User's Manual). 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 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 elements with plane stress formulations.

SCALE FACTOR

Set this parameter equal to the factor that is used to scale the time increment computed by Abaqus/Explicit. The default scaling factor is 1.0. This parameter can be used to scale the default global time estimate, and it can be used in conjunction with the ELEMENT BY ELEMENT and FIXED TIME INCREMENTATION parameters. It cannot be used in conjunction with the DIRECT USER CONTROL parameter.

Data line for automatic time incrementation (global or ELEMENT BY ELEMENT estimation):

First (and only) line:

1. Enter a blank field.
2. T, time period of the step.
3. Enter a blank field.
4. Δt_{max}, maximum time increment allowed. If this value is not specified, no upper limit is imposed.

Data line for fixed time incrementation using DIRECT USER CONTROL:

First (and only) line:

1. Δt, time increment to be used throughout the step.
2. T, time period of the step.

Data line for fixed time incrementation using FIXED TIME INCREMENTATION:

First (and only) line:

1. Enter a blank field.
2. T, time period of the step.

4.41 *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 6.5.4 of the Abaqus Analysis User's Manual

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 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 elements with plane stress formulations.

SCALE FACTOR

Set this parameter equal to the factor that is used to scale the time increment computed by Abaqus/Explicit. The default scaling factor is 1.0. This parameter can be used to scale the default global time estimate, and it can be used in conjunction with the ELEMENT BY ELEMENT and FIXED TIME INCREMENTATION parameters. It cannot be used in conjunction with the DIRECT USER CONTROL parameter.

Data line for automatic time incrementation (global or ELEMENT BY ELEMENT estimation):

First (and only) line:

1. Enter a blank field.
2. T, time period of the step.
3. Enter a blank field.
4. Δt_{max}, maximum time increment allowed. If this value is not specified, no upper limit is imposed.

Data line for fixed time incrementation using DIRECT USER CONTROL:

First (and only) line:

1. Δt, time increment to be used throughout the step.
2. T, time period of the step.

Data line for fixed time incrementation using FIXED TIME INCREMENTATION:

First (and only) line:

1. Enter a blank field.
2. T, time period of the step.

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 Manual
- *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 Manual, 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 Manual) and for eigenvalue buckling estimation (“Eigenvalue buckling prediction,” Section 6.2.3 of the Abaqus Analysis User’s Manual). 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 Manual) and for eigenvalue buckling estimation (“Eigenvalue buckling prediction,” Section 6.2.3 of the Abaqus Analysis User’s Manual). 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 Manual. 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 Manual.

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 Manual.

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 Manual

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 Manual), complex eigenvalue extraction ("Complex eigenvalue extraction," Section 6.3.6 of the Abaqus Analysis User's Manual), and for eigenvalue buckling estimation ("Eigenvalue buckling prediction," Section 6.2.3 of the Abaqus Analysis User's Manual). 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 *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

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 Manual. 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 Manual. 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 2.7.1 of the Abaqus Analysis User’s Manual).

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Reference:

- “Linear elastic behavior,” Section 19.2.1 of the Abaqus Analysis User’s Manual

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 18.1.2 of the Abaqus Analysis User’s Manual, 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 2.7.1 of the Abaqus Analysis User’s Manual.

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 Interface for Moldflow. Any data lines given will be ignored. Material properties will be read from the ASCII neutral file identified as *jobid*`.shf`. See the Abaqus Interface for Moldflow User's Manual 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 2.7.1 of the Abaqus Analysis User's Manual.

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. 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.

Data lines to define orthotropic elasticity with moduli (TYPE=ENGINEERING CONSTANTS):

First line:

1. E_1.
2. E_2.
3. E_3.
4. ν_{12}.
5. ν_{13}.
6. ν_{23}.
7. G_{12}.
8. G_{13}.

Second line:

1. G_{23}.
2. Temperature, θ.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

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

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

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

Data lines to define isotropic elasticity (TYPE=ISOTROPIC):

First line:

1. Young's modulus, E.
2. Poisson's ratio, ν.
3. Temperature, θ.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

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

1. Sixth field variable.

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

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

Data lines to define orthotropic elasticity in plane stress (TYPE=LAMINA):

First line:

1. E_1.
2. E_2.
3. ν_{12}.
4. G_{12}.
5. G_{13}. This shear modulus is needed to define transverse shear behavior in shells.
6. G_{23}. This shear modulus is needed to define transverse shear behavior in shells.
7. Temperature.
8. First field variable.

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

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

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

Data lines to define orthotropic elasticity directly (TYPE=ORTHOTROPIC):

First line:

1. D_{1111}. (Units of FL^{-2}.)
2. D_{1122}.
3. D_{2222}.
4. D_{1133}.
5. D_{2233}.
6. D_{3333}.
7. D_{1212}.
8. D_{1313}.

Second line:

1. D_{2323}.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

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

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

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

Data lines to define isotropic elastic shear behavior (TYPE=SHEAR):

First line:

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

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

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

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

Data lines to define orthotropic shear behavior for 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; K_{nn} for cohesive elements.
2. G_1 for warping elements; K_{ss} for cohesive elements.
3. G_2 for warping elements; K_{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/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 Manual

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 elements in coupled thermal-electrical analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Electrical conductivity," Section 23.6.1 of the Abaqus Analysis User's Manual
- "Coupled thermal-electrical analysis," Section 6.7.2 of the Abaqus Analysis User's Manual

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. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

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):

First line:

1. Electrical conductivity. (Units of $\mathrm{CT}^{-1}\mathrm{L}^{-1}\varphi^{-1}$.)
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 orthotropic electrical conductivity (TYPE=ORTHO):

First line:

1. σ^E_{11}. (Units of $\mathrm{CT}^{-1}\mathrm{L}^{-1}\varphi^{-1}$.)
2. σ^E_{22}.
3. σ^E_{33}.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

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

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

Repeat this set of data lines as often as necessary to define orthotropic electrical conductivity as a function of temperature and field variables.

Data lines to define anisotropic electrical conductivity (TYPE=ANISO):

First line:

1. σ^E_{11}. (Units of $\mathrm{CT}^{-1}\mathrm{L}^{-1}\varphi^{-1}$.)
2. σ^E_{12}.
3. σ^E_{22}.
4. σ^E_{13}.
5. σ^E_{23}.
6. σ^E_{33}.
7. Temperature.
8. First field variable.

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

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

Repeat this set of data lines as often as necessary to define anisotropic electrical conductivity as a function of temperature and field variables.

5.6 *ELEMENT: Define elements by giving their nodes.

This option is used to define an element directly by specifying its nodes.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Mesh module

Reference:

- "Element definition," Section 2.2.1 of the Abaqus Analysis User's Manual

Required parameter:

TYPE

Set this parameter equal to the element type, as defined in Part VI, "Elements," of the Abaqus Analysis User's Manual.

For user elements specify the U*n* type identification (see "User-defined elements," Section 29.16.1 of the Abaqus Analysis User's Manual). 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 Manual).

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 Manual, 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 Manual).

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 Manual, 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 Manual, 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 29.6.4 of the Abaqus Analysis User's Manual, and "Defining the cohesive element's initial geometry," Section 29.5.4 of the Abaqus Analysis User's Manual).

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 Manual.

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).

5.7 *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 Manual

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 Manual, 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 Manual.

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 29.16.1 of the Abaqus Analysis User's Manual). 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.

5.8 *ELEMENT OUTPUT: Define output database requests for element variables.

This option is used to write element variables to the output database. It must be used in conjunction with the *OUTPUT option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Manual
- *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

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 is omitted, the 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).

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 Manual, and "Abaqus/Explicit output variable identifiers," Section 4.2.2 of the Abaqus Analysis User's Manual.

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

5.9 *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 16.1.1 of the Abaqus Analysis User's Manual
- *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 16.1.1 of the Abaqus Analysis User's Manual.

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

5.10 *ELGEN: Incremental element generation.

This option is used to generate elements incrementally.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Not applicable; elements are generated when you mesh the model.

Reference:

- "Element definition," Section 2.2.1 of the Abaqus Analysis User's Manual

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 $N1 \times N2 \times N3$ elements, where $N1$ is the number of elements in a row, $N2$ is the number of rows in a layer, and $N3$ is the number of layers.

5.11 *ELSET: Assign elements to an element set.

This option is used to assign elements to an element set.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

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

Abaqus/CAE: Set toolset

Reference:

- "Element definition," Section 2.2.1 of the Abaqus Analysis User's Manual

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.

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.

5.12 *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 31.4.1 of the Abaqus Analysis User's Manual

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.

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.

5.13 *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 37.1.1 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines to define the emissivity of a surface:

First line:

1. Emissivity, ϵ.
2. Temperature, if temperature dependent.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

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

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

Repeat this set of data lines as often as necessary to define the emissivity as a function of temperature and user-defined field variables.

5.14 *END ASSEMBLY: End the definition of an assembly.

This option is used to end an assembly definition.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Assembly module

References:

- "Defining an assembly," Section 2.9.1 of the Abaqus Analysis User's Manual
- *ASSEMBLY

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

5.15 *END INSTANCE: End the definition of an instance.

This option is used to end an instance definition.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Assembly

Abaqus/CAE: Assembly module for part instances not imported from a previous analysis; Load module for part instances imported from a previous analysis

References:

- "Defining an assembly," Section 2.9.1 of the Abaqus Analysis User's Manual
- *INSTANCE

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

5.16 *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 6.1.3 of the Abaqus Analysis User's Manual
- *LOAD CASE

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

5.17 *END PART: End the definition of a part.

This option is used to end a part definition.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Part module

References:

- "Defining an assembly," Section 2.9.1 of the Abaqus Analysis User's Manual
- *PART

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

5.18 *END STEP: End the definition of a step.

This option is used to end a step definition.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Model

Abaqus/CAE: Step module

References:

- "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual
- *STEP

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

5.19 *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 Manual
- *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.

5.20 *ENERGY OUTPUT: Define output database requests for whole model or element set energy data.

This option is used to write whole model or element set energy requests to the output database. It must be used in conjunction with the *OUTPUT, HISTORY option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Manual
- *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 "Abaqus/Standard output variable identifiers," Section 4.2.1 of the Abaqus Analysis User's Manual, and "Abaqus/Explicit output variable identifiers," Section 4.2.2 of the Abaqus Analysis User's Manual.

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

5.21 *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 Manual

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.

5.22 *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 10.6.1 of the Abaqus Analysis User's Manual

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.

5.23 *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: Step module

Reference:

- "Modeling discontinuities as an enriched feature using the extended finite element method," Section 10.6.1 of the Abaqus Analysis User's Manual

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.

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.

5.24 *EOS: Specify an equation of state model.

This option is used to define a hydrodynamic material model in the form of an equation of state.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Reference:

- "Equation of state," Section 22.2.1 of the Abaqus Analysis User's Manual

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=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.

Data line for an ideal gas equation of state (TYPE=IDEAL GAS):

First (and only) line:

1. Gas constant, R. (Units of $\mathrm{JM}^{-1}\theta^{-1}$.)
2. The ambient pressure, p_A (Units of FL^{-2}). If this field is left blank, a default of 0.0 is used.

Data lines for an ignition and growth equation of state (TYPE=IGNITION AND GROWTH):

First line:

Material constants used in the equation of state for unreacted explosive.

1. A_s. (Units of FL^{-2}.)
2. B_s. (Units of FL^{-2}.)
3. ω_s. (Dimensionless.)
4. R_{1s}. (Dimensionless.)
5. R_{2s}. (Dimensionless.)

Second line:

Material constants used in the equation of state for reacted products.

1. A_g. (Units of FL^{-2}.)
2. B_g. (Units of FL^{-2}.)
3. ω_g. (Dimensionless.)
4. R_{1g}. (Dimensionless.)
5. R_{2g}. (Dimensionless.)

Data line for an explosive equation of state (TYPE=JWL):

First (and only) line:

1. Detonation wave speed, C_d. (Units of LT^{-1}.)
2. A. (Units of FL^{-2}.)
3. B. (Units of FL^{-2}.)
4. ω. (Dimensionless.)
5. R_1. (Dimensionless.)
6. R_2. (Dimensionless.)
7. Detonation energy density, E_0. (Units of JM^{-1}.)
8. Pre-detonation bulk modulus, K_{pd}. (Units of FL^{-2}.)

Data line for a tabulated equation of state (TYPE=TABULAR), where the volumetric strain values must be arranged in descending order:

First line:

1. f_1. (Units of FL^{-2}.)
2. f_2. (Dimensionless.)
3. Volumetric strain ε_{vol}. (Dimensionless.)

Repeat this data line as often as necessary to define the dependence of f_1 and f_2 on volumetric strain.

Data line for a linear equation of state (TYPE=USUP):

First (and only) line:

1. c_0. (Units of LT^{-1}.)
2. s. (Dimensionless.)
3. Γ_0. (Dimensionless.)

5.25 *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 22.2.1 of the Abaqus Analysis User's Manual
- *EOS

There are no parameters associated with this option.

Data line to define the plastic compaction behavior:

First (and only) line:

1. Reference sound speed in the porous material, c_e. (Units of LT^{-1}.)
2. Value of the porosity of the unloaded (virgin) material, n_0 . (Dimensionless.)
3. Pressure required to initialize plastic behavior, p_e. (Units of FL^{-2}.)
4. Compaction pressure at which all pores are crushed, p_S. (Units of FL^{-2}.)

5.26 *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 29.11.1 of the Abaqus Analysis User's Manual
- *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 Manual) 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 lines to define spud can geometry with SECTION=SPUD CAN:

First (and only) line:

1. D_o, diameter of spud can cylindrical portion.

2. θ, conical spud can cone angle in degrees. Enter a blank, zero, or 180 for a cylindrical spud can.

Include the *JOINT ELASTICITY and *JOINT PLASTICITY options as needed to define the joint behavior.

5.27 *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 31.2.1 of the Abaqus Analysis User's Manual

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 Manual, 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.

5.28 *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 13.1.2 of the Abaqus Analysis User's Manual

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.

5.29 *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 13.1.1 of the Abaqus Analysis User's Manual
- "Eulerian mesh motion," Section 13.1.3 of the Abaqus Analysis User's Manual

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 Manual) to be used to define the local directions of the bounding box. Only orientations defined with SYSTEM=RECTANGULAR or SYSTEM=Z RECTANGULAR can be specified.

VMAX FACTOR

Set this parameter equal to a fraction of the maximum velocity of the surface nodes to bound the mesh motion velocity. The default is VMAX FACTOR=1.01.

VOLFRAC MIN

Set this parameter equal to the lower bound on the volume fraction used to determine which nodes to include in the bounding box calculation for an Eulerian material surface. The default is VOLFRAC MIN=0.5.

Optional data lines to define bounding box constraints:

First line:

1. Value between 1.0 and ∞ (default): maximum scaling of the bounding box in local direction 1.
2. Value between 1.0 and ∞ (default): maximum scaling of the bounding box in local direction 2.
3. Value between 1.0 and ∞ (default): maximum scaling of the bounding box in local direction 3.
4. Value between 0.0 (default) and 1.0: minimum scaling of the bounding box in local direction 1.
5. Value between 0.0 (default) and 1.0: minimum scaling of the bounding box in local direction 2.
6. Value between 0.0 (default) and 1.0: minimum scaling of the bounding box in local direction 3.

Second line:

1. FREE (default) or FIXED: constraint flag for the negative local direction 1 face of the bounding box.

2. FREE (default) or FIXED: constraint flag for the positive local direction 1 face of the bounding box.
3. FREE (default) or FIXED: constraint flag for the negative local direction 2 face of the bounding box.
4. FREE (default) or FIXED: constraint flag for the positive local direction 2 face of the bounding box.
5. FREE (default) or FIXED: constraint flag for the negative local direction 3 face of the bounding box.
6. FREE (default) or FIXED: constraint flag for the positive local direction 3 face of the bounding box.

Third line:

1. FREE (default) or FIXED: constraint flag for the center of the bounding box in local direction 1.
2. FREE (default) or FIXED: constraint flag for the center of the bounding box in local direction 2.
3. FREE (default) or FIXED: constraint flag for the center of the bounding box in local direction 3.

5.30 *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 13.1.1 of the Abaqus Analysis User's Manual
- "Eulerian elements," Section 29.15.1 of the Abaqus Analysis User's Manual

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 24.1.4 of the Abaqus Analysis User's Manual) 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.

Repeat this data line as often as necessary to define the list of all materials that may appear in the Eulerian section.

5.31 *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 2.7.1 of the Abaqus Analysis User's Manual).

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Thermal expansion," Section 23.1.2 of the Abaqus Analysis User's Manual
- "Field expansion," Section 23.1.3 of the Abaqus Analysis User's Manual
- "UEXPAN," Section 1.1.25 of the Abaqus User Subroutines Reference Manual

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables, in addition to temperature, on which the coefficients depend. If this parameter is omitted, it is assumed that the 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 2.7.1 of the Abaqus Analysis User's Manual).

FIELD

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.

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 Interface for Moldflow. Any data lines will be ignored. Material properties will be read from the ASCII neutral file identified as *jobid*`.shf`. See the Abaqus Interface for Moldflow User's Manual 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 2.7.1 of the Abaqus Analysis User's Manual.

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 θ^0. The default is ZERO=0.

This parameter is not relevant if the USER parameter is included.

Data lines to define isotropic thermal expansion coefficients (TYPE=ISO with USER parameter omitted):

First line:

1. α. (Units of θ^{-1}.)
2. Temperature.
3. First field variable.
4. Etc., up to six field variables.

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

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

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

Data lines to define orthotropic thermal expansion coefficients (TYPE=ORTHO with USER parameter omitted):

First line:

1. α_{11}. (Units of θ^{-1}.)
2. α_{22}.
3. α_{33}.
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. α_{11}. (Units of θ^{-1}.)
2. α_{22}.
3. α_{33}. (Not used for plane stress case.)
4. α_{12}.
5. α_{13}.
6. α_{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 thermal expansion coefficients as functions of temperature and other predefined field variables.

Data line to define spatially varying thermal expansion in an Abaqus/Standard analysis using a distribution:

First (and only) line:

1. Distribution name. The data defined in the distribution must be in units of θ^{-1} and must be consistent with the level of anisotropy prescribed by the TYPE parameter.

To define thermal expansion by a user subroutine (USER parameter included):

No data lines are used with this option when the USER parameter is specified. Instead, user subroutine **UEXPAN** must be used to define the thermal expansion.

Data lines to define isotropic field expansion coefficients (TYPE=ISO with USER parameter omitted):

First line:

1. α_f. (Units of $F{V_n}^{-1}$.)
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. $\alpha_{f_{11}}$. (Units of $F{V_n}^{-1}$.)
2. $\alpha_{f_{22}}$.
3. $\alpha_{f_{33}}$.
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

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

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

Repeat this set of data lines as often as necessary to define the field expansion coefficients as functions of temperature and other predefined field variables.

Data lines to define anisotropic field expansion coefficients (TYPE=ANISO with USER parameter omitted):

First line:

1. $\alpha_{f\,11}$. (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.

5.32 *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 Manual
- *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 Manual.
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.

5.33 *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 Manual
- *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 Manual.
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.

5.34 *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 Manual
- *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 20.4.1 of the Abaqus Analysis User’s Manual
- “VFABRIC,” Section 1.2.3 of the Abaqus User Subroutines Reference Manual
- *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 19.2.3 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines to define strain-based failure criteria:

First line:

1. Tensile strain limit in fiber direction, $X_{\varepsilon t}$.
2. Compressive strain limit in fiber direction, $X_{\varepsilon c}$.
3. Tensile strain limit in transverse direction, $Y_{\varepsilon t}$.
4. Compressive strain limit in transverse direction, $Y_{\varepsilon c}$.
5. Shear strain limit in the X–Y plane, S_{ε}.
6. Temperature.
7. First field variable.
8. Second field variable.

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

1. Third field variable.
2. Fourth field variable.

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

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

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 19.2.3 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, 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 σ_{biax} is given. The default is zero.
7. Biaxial stress limit, σ_{biax}. This value is used only for the Tsai-Wu theory. If this entry is nonzero, $\overset{*}{f}$ is ignored.
8. Temperature.

Subsequent lines (only needed if the DEPENDENCIES parameter has a nonzero value):

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

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

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 20.6.1 of the Abaqus Analysis User's Manual
- *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 18.1.2 of the Abaqus Analysis User's Manual, 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 31.3.4 of the Abaqus Analysis User's Manual
- *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.

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 Manual) 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 Manual) 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 31.3.4 of the Abaqus Analysis User's Manual
- *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 Manual, 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 30.6.1 of the Abaqus Analysis User's Manual
- "UFIELD," Section 1.1.27 of the Abaqus User Subroutines Reference Manual
- "VUFIELD," Section 1.2.11 of the Abaqus User Subroutines Reference Manual

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 30.1.2 of the Abaqus Analysis User's Manual).

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 Manual, 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 Manual, 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 30.6.1 of the Abaqus Analysis User's Manual.

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 Manual

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 Manual
- *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 and coupled thermal-electrical analyses.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- "Thermal loads," Section 30.4.4 of the Abaqus Analysis User's Manual
- "FILM," Section 1.1.6 of the Abaqus User Subroutines Reference Manual

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the *AMPLITUDE option that gives the variation of the sink temperature, θ^0, with time.

If this parameter is omitted in an Abaqus/Standard analysis, the reference sink temperature is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the *STEP option ("Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). 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 ignored.

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 nonuniform film coefficient is defined in user subroutine **FILM** or if a film coefficient is defined to be a function of temperature and field variables via the *FILM PROPERTY option.

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 Manual).
3. Reference sink temperature value, θ^0. (Units of θ.) For nonuniform film coefficients the sink temperature must be defined in user subroutine **FILM**. If given, this value will be passed into the user subroutine.
4. Reference film coefficient value, h (units of $\mathrm{JT}^{-1}\mathrm{L}^{-2}\theta^{-1}$), 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 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 and coupled thermal-electrical 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 30.4.4 of the Abaqus Analysis User’s Manual
- *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 18.1.2 of the Abaqus Analysis User’s Manual, 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 $\mathrm{JT^{-1}L^{-2}}\theta^{-1}$.)
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 Manual
- *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.

See "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Manual, for more information.

LIMIT

Include this parameter if you want to set a limit (cap value) to the output variables.

This parameter must be used in conjunction with the OPERATOR parameter.

OPERATOR

This parameter can be used with or without the TYPE parameter. When it is used with a filter type, it will operate on filtered data; and when it is used without a filter type, it will operate on raw (unfiltered) data.

Set OPERATOR=MAX if you want to obtain the maximum value for the variables for which this filter is used. You can put a cap value on the maximum value by using the LIMIT parameter.

Set OPERATOR=MIN if you want to obtain the minimum value for the variables for which this filter is used. You can put a cap value on the minimum value by using the LIMIT parameter.

Set OPERATOR=ABSMAX if you want to obtain the absolute maximum value for the variables for which this filter is used. You can put a cap value on this value by using the LIMIT parameter.

START CONDITION

This parameter must be used with the TYPE parameter.

Set START CONDITION=DC (default) to pre-charge the filter with constant values equal to the first raw data value.

Set START CONDITION=USER DEFINED to pre-charge the filter with constant values equal to the user-defined value.

TYPE

Set TYPE=BUTTERWORTH (which is the default value when OPERATOR is omitted) to define a Butterworth filter.

Set TYPE=CHEBYS1 to define a Type I Chebyshev filter.

Set TYPE=CHEBYS2 to define a Type II Chebyshev filter.

Data lines to define a Butterworth filter:

First line:

1. Cutoff frequency, fc. (Units of T^{-1}.)
2. Order of the filter, N. Abaqus expects an even number; if an odd number is specified, it will be internally changed to the closest greater even number. The default value is two.

Second line (only needed if START CONDITION=USER DEFINED):

1. Starting value.

Data lines to define a Type I Chebyshev filter:

First line:

1. Cutoff frequency, fc. (Units of T^{-1}.)
2. Ripple factor, ϵ.
3. Order of the filter, N. Abaqus expects an even number; if an odd number is specified, it will be changed internally to the closest greater even number. The default value is two.

Second line (only needed if START CONDITION=USER DEFINED):

1. Starting value.

Data lines to define a Type II Chebyshev filter:

First line:

1. Cutoff frequency, fc. (Units of T^{-1}.)
2. Ripple factor, $1/A$.
3. Order of the filter, *N*. Abaqus expects an even number; if an odd number is specified, it will be changed internally to the closest greater even number. The default value is two.

Second line (only needed if START CONDITION=USER DEFINED):

1. Starting value.

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/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Mass scaling," Section 11.7.1 of the Abaqus Analysis User's Manual
- "Output," Section 4.1.1 of the Abaqus Analysis User's Manual

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 30.4.6 of the Abaqus Analysis User's Manual
- "FLOW," Section 1.1.7 of the Abaqus User Subroutines Reference Manual

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 "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). 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 Manual).
3. Reference pore pressure value, u_w^∞. (Units of FL^{-2}.)
4. Reference seepage coefficient value, k_s. (Units of $F^{-1}L^3T^{-1}$.)

Repeat this data line as often as necessary to define uniform seepage for various elements or element sets.

Data lines to define drainage-only seepage:

First line:

1. Element number or element set label.
2. Seepage flow type label (see Part VI, "Elements," of the Abaqus Analysis User's Manual).
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 Manual).
3. Optional reference pore pressure value. If given, this value is passed into user subroutine **FLOW** in the variable used to define the sink pore pressure.
4. Optional reference seepage coefficient. If given, this value is passed into user subroutine **FLOW** in the variable used to define the seepage coefficient.

The reference pore pressure value, u_w^∞, and reference seepage coefficient, k_s, are defined in user subroutine **FLOW** for nonuniform flow.

Repeat this data line as often as necessary to define nonuniform seepage for various elements or element sets.

6.15 *FLUID BEHAVIOR: Define fluid behavior for a fluid cavity.

This option is used to define the fluid behavior for a surface-based fluid cavity.

Product: Abaqus/Explicit

Type: Model data

Level: Part, Part instance

Reference:

- "Fluid cavity definition," Section 11.6.2 of the Abaqus Analysis User's Manual

Required parameter:

NAME

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

There are no data lines associated with this option.

6.16 ∗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 or the ∗FLUID PROPERTY option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- "Fluid cavity definition," Section 11.6.2 of the Abaqus Analysis User's Manual
- "Hydrostatic fluid models," Section 23.4.1 of the Abaqus Analysis User's Manual
- ∗FLUID BEHAVIOR
- ∗FLUID PROPERTY

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 18.1.2 of the Abaqus Analysis User's Manual, 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.

6.17 *FLUID CAVITY: Define a fluid cavity.

This option is used to define a surface-based fluid cavity.

Product: Abaqus/Explicit

Type: Model data

Level: Part, Part instance, Assembly

Reference:

- "Fluid cavity definition," Section 11.6.2 of the Abaqus Analysis User's Manual

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

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 calculated by Abaqus/Explicit.

ADIABATIC

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 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

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 parameter ADDED VOLUME 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.

6.18 *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 and pneumatic fluids and should not be used for user-defined fluids. The *FLUID DENSITY option can be used only in conjunction with the *FLUID BEHAVIOR option or the *FLUID PROPERTY option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- "Fluid cavity definition," Section 11.6.2 of the Abaqus Analysis User's Manual
- "Hydrostatic fluid models," Section 23.4.1 of the Abaqus Analysis User's Manual
- *FLUID BEHAVIOR
- *FLUID PROPERTY

Optional parameters when the *FLUID DENSITY option is used in conjunction with the *FLUID PROPERTY option:

PRESSURE

Set this parameter equal to the value of the reference gauge pressure, p_R. It is relevant only for a pneumatic fluid. The default is PRESSURE=0.

TEMPERATURE

Set this parameter equal to the value of the reference temperature, θ_R, for the temperature scale in use. It is relevant only for a pneumatic fluid. The default is TEMPERATURE=0.

Data line to define the reference fluid density:

First (and only) line:

1. Reference fluid density, ρ_R.

6.19 *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.

Product: Abaqus/Explicit

Type: Model data

Level: Model

References:

- "Fluid exchange definition," Section 11.6.3 of the Abaqus Analysis User's Manual
- *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:

CONSTANTS

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

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** 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

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. Optional second reference node number of fluid cavity. If only one node is specified, fluid exchange will occur 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.

6.20 *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 11.6.2 of the Abaqus Analysis User's Manual
- "Fluid exchange definition," Section 11.6.3 of the Abaqus Analysis User's Manual
- *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 30.1.2 of the Abaqus Analysis User's Manual.

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.

6.21 *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.

Product: Abaqus/Explicit

Type: Model data

Level: Model

References:

- "Fluid exchange definition," Section 11.6.3 of the Abaqus Analysis User's Manual
- *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.

Set TYPE=ENERGY RATE LEAKAGE to define fluid exchange by specifying the heat energy flow rate as a function of temperature difference and pressure.

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.

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 to define fluid exchange by specifying the mass flow rate and/or heat energy flow rate.

Optional parameters:

CONSTANTS

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

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.

6.22 *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 or the *FLUID PROPERTY option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- "Fluid cavity definition," Section 11.6.2 of the Abaqus Analysis User's Manual
- "Hydrostatic fluid models," Section 23.4.1 of the Abaqus Analysis User's Manual
- *FLUID BEHAVIOR
- *FLUID PROPERTY

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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

ZERO

Set this parameter equal to the value of θ^0. The default is ZERO=0.

Data lines to define the thermal expansion coefficient:

First line:

1. Mean coefficient of thermal expansion, α.
2. Temperature, θ.
3. First field variable.
4. Etc., up to six field variables.

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

1. Seventh field variable.

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

Repeat this set of data lines as often as necessary to specify α as a function of θ and field variables.

6.23 *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:

- "Modeling fluid-filled cavities," Section 11.5.1 of the Abaqus Analysis User's Manual

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 30.1.2 of the Abaqus Analysis User's Manual). 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}.)

6.24 *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 11.6.4 of the Abaqus Analysis User’s Manual
- *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.

6.25 *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 11.6.4 of the Abaqus Analysis User's Manual
- *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.

6.26 *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 11.6.2 of the Abaqus Analysis User's Manual
- "Inflator definition," Section 11.6.4 of the Abaqus Analysis User's Manual
- *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.

6.27 *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 11.6.4 of the Abaqus Analysis User's Manual
- *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.

6.28 *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 29.5.7 of the Abaqus Analysis User's Manual
- "UFLUIDLEAKOFF," Section 1.1.29 of the Abaqus User Subroutines Reference Manual

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 18.1.2 of the Abaqus Analysis User's Manual, 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.

6.29 *FLUID LINK: Define properties for fluid link elements.

This option is used to define the properties of fluid link elements.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- "Modeling fluid-filled cavities," Section 11.5.1 of the Abaqus Analysis User's Manual
- "Fluid link elements," Section 29.8.3 of the Abaqus Analysis User's Manual

Required parameter:

ELSET

Set this parameter equal to the name of the element set containing the fluid link elements for which properties are being defined.

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the specification of the mass flow rate. If this parameter is omitted, the mass flow rate is assumed not to depend on any field variables but may still depend on average pressure and average temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

TYPE

Set TYPE=FUNCTION (default) to specify a functional relationship between mass flow rate and pressure difference.

Set TYPE=TABULAR to specify a table of mass flow rate versus pressure difference.

Data lines for TYPE=FUNCTION:

First line:

1. C_V.
2. C_H.
3. Average pressure, $\overline{p}$, if pressure dependent.

4. Average temperature, $\overline{\theta}$, 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 C_V and C_H as functions of $\overline{p}$, $\overline{\theta}$, and field variables.

Data lines for TYPE=TABULAR:

First line:

1. Mass flow rate, q.
2. Pressure difference, Δp.
3. Average pressure, $\overline{p}$, if pressure dependent.
4. Average temperature, $\overline{\theta}$, 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 q as a function of Δp, $\overline{p}$, $\overline{\theta}$, and field variables.

6.30 *FLUID PROPERTY: Define properties for hydrostatic fluid elements.

This option is used to define properties for hydrostatic fluid elements associated with a given fluid cavity.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- "Modeling fluid-filled cavities," Section 11.5.1 of the Abaqus Analysis User's Manual
- "Hydrostatic fluid elements," Section 29.8.1 of the Abaqus Analysis User's Manual
- "UFLUID," Section 1.1.28 of the Abaqus User Subroutines Reference Manual

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the hydrostatic fluid elements for which properties are being defined.

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.

TYPE

This parameter applies only to Abaqus/Standard analyses. Abaqus/Explicit uses only pneumatic fluids to model the fluid within a cavity (see "Hydrostatic fluid models," Section 23.4.1 of the Abaqus Analysis User's Manual, for more information).

Set TYPE=HYDRAULIC to specify an incompressible or approximately incompressible fluid.

Set TYPE=PNEUMATIC to specify a compressible fluid that behaves as an ideal gas.

Set TYPE=USER to specify a fluid in which the fluid constitutive model is defined in user subroutine **`UFLUID`**.

Optional parameters:

AMBIENT

This parameter specifies the magnitude of the ambient pressure, which will typically be atmospheric pressure. It is relevant only for TYPE=PNEUMATIC. The default is AMBIENT=0.

NAME

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to a label that will be used to refer to the fluid property. The label given can be used to identify the fluid property in user subroutine **UFLUID**.

Data line for two-dimensional elements:

First (and only) line:

1. Element thickness.

To define fluid properties for three-dimensional and axisymmetric elements:

There are no data lines required.

To define the fluid model by using a user subroutine:

No data lines are used with this option when TYPE=USER is specified. Instead, user subroutine **UFLUID** must be used to define the fluid model.

6.31 *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 Manual

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.

6.32 *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 Manual
- "Modeling discontinuities as an enriched feature using the extended finite element method," Section 10.6.1 of the Abaqus Analysis User's Manual
- *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=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 18.1.2 of the Abaqus Analysis User’s Manual, for more information.

MIXED MODE BEHAVIOR

This parameter is relevant only for 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=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.

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=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=VCCT.

Set this parameter equal to the value of the viscosity coefficient used in the viscous regularization. The default value is 0.0.

Data lines to define the critical stress criterion (TYPE=CRITICAL STRESS):

First line:

1. Normal failure stress, σ^f.
2. Shear failure stress, τ_1^f.
3. Shear failure stress, τ_2^f. (Not applicable in two dimensions.)
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

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

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

Repeat this set of data lines as often as necessary to define the critical stress criterion as a function of temperature and/or field variables.

Data lines to define the crack opening displacement criterion (TYPE=COD):

First line:

1. Critical crack opening displacement, δ_c.
2. Cumulative crack length.

3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

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

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

Repeat this set of data lines as often as necessary to define the crack opening displacement criterion as a function of temperature and/or field variables.

Data lines to define the crack length versus time criterion (TYPE=CRACK LENGTH):

First line:

1. Total time (not step time).
2. Crack length, l, from the reference point.
3. Etc., up to four time/length pairs per line. Crack length must be given as an increasing function of time.

Repeat this data line as often as necessary to define the crack length as a function of time.

Data lines to define the 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, η.
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, η.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

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

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

Repeat this set of data lines as often as necessary to define the critical energy rates and exponent as a function of temperature and field variables.

Data lines to define the VCCT criterion (TYPE=VCCT) 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.

6.33 *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 26.4.1 of the Abaqus Analysis User's Manual
- "Frame section behavior," Section 26.4.2 of the Abaqus Analysis User's Manual

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 18.1.2 of the Abaqus Analysis User's Manual, 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 26.3.9 of the Abaqus Analysis User's Manual). The following cross-sections are available for elastic frame elements (when elastic-plastic and buckling strut response are omitted):

- BOX, for a rectangular, hollow box section.
- CIRC, for a solid circular section.
- GENERAL, for a general cross-section (default).
- I, for an I-beam section.
- PIPE, for a hollow, circular section.
- RECT, for a solid rectangular section.

For elastic-plastic material response the only available plastic interaction surface is an ellipsoid, which is recommended for PIPE cross-sections only. Other cross-section types, except the GENERAL section, can be used at the user's discretion.

For buckling strut response only the PIPE cross-section is available.

YIELD STRESS

Set this parameter equal to the yield stress for the material making up the cross-section.

This parameter is required when defining default elastic-plastic material response with the PLASTIC DEFAULTS parameter and when modeling buckling strut response by using the *BUCKLING ENVELOPE option or the BUCKLING parameter.

ZERO

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

Data lines for SECTION=GENERAL:

First line:

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

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

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

The entries on this line must be (0, 0, −1) for FRAME2D elements. The default for FRAME3D elements is (0, 0, −1) if the first element section axis is not defined by an additional node in the element's connectivity. See "Frame elements," Section 26.4.1 of the Abaqus Analysis User's Manual, 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 26.3.9 of the Abaqus Analysis User's Manual, 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 26.4.1 of the Abaqus Analysis User's Manual, 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.

6.34 *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 Manual

Optional parameters:

ACOUSTIC COUPLING

For the Lanczos eigensolver that is not based on the SIM architecture, set ACOUSTIC COUPLING=ON (default) 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.

For the AMS eigensolver and Lanczos eigensolver based on the SIM architecture, set ACOUSTIC COUPLING=PROJECTION (default) to project the acoustic-structural coupling operator during the natural frequency extraction procedure in models with acoustic and structural elements coupled using the *TIE option.

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 the subspace iteration eignensolver and for the Lanczos eigensolver 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 and for the Lanczos eigensolver 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 eigensolver.

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. Only mode-based transient and mode-based or subspace-based steady-state dynamic procedures are supported with this option. 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 containing 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_{cutoff_1}, the first AMS parameter. AMS_{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_{cutoff_2}, the second AMS parameter. AMS_{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_{cutoff_3}, the third AMS parameter. AMS_{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, $2n$), 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.

6.35 *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 28.2.1 of the Abaqus Analysis User's Manual
- "Mechanical contact properties: overview," Section 33.1.1 of the Abaqus Analysis User's Manual
- "Frictional behavior," Section 33.1.5 of the Abaqus Analysis User's Manual
- "FRIC," Section 1.1.8 of the Abaqus User Subroutines Reference Manual
- "FRIC_COEF," Section 1.1.9 of the Abaqus User Subroutines Reference Manual
- "VFRIC," Section 1.2.4 of the Abaqus User Subroutines Reference Manual
- "VFRIC_COEF," Section 1.2.5 of the Abaqus User Subroutines Reference Manual
- "VFRICTION," Section 1.2.6 of the Abaqus User Subroutines Reference Manual
- *CHANGE FRICTION
- *CONNECTOR FRICTION
- *GAP
- *INTERFACE
- *ITS
- *SURFACE INTERACTION

Optional, mutually exclusive parameters:

ELASTIC SLIP

This parameter applies only to Abaqus/Standard analyses.

In a steady-state transport analysis set this parameter equal to the absolute magnitude of the allowable elastic slip velocity ($\dot{\gamma}_i$) to be used in the stiffness method for sticking friction. In all other analysis procedures set this parameter equal to the absolute magnitude of the allowable elastic slip (γ_i) to be used in the stiffness method for sticking friction. If this parameter is omitted, the elastic slip (or elastic slip velocity) is defined by the SLIP TOLERANCE value.

LAGRANGE

This parameter applies only to Abaqus/Standard analyses and cannot be used when friction is defined for connector elements.

Include this parameter to choose the Lagrange multiplier formulation for friction.

ROUGH

This parameter cannot be used when friction is defined for connector elements.

Include this parameter to specify completely rough (no slipping) friction.

SLIP TOLERANCE

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the value of F_f (defined as the ratio of allowable maximum elastic slip velocity to angular velocity times the diameter of the spinning body in a steady-state transport analysis or as the ratio of allowable maximum elastic slip to characteristic contact surface face dimension in all other analysis procedures). The default is SLIP TOLERANCE=.005.

When friction is defined for connector elements, F_f is defined (when possible) as the ratio of allowable maximum elastic slip to a characteristic element dimension in the model. In this case the default is SLIP TOLERANCE=.0001.

USER

This parameter cannot be used when friction is defined for connector elements.

In an Abaqus/Standard analysis, set USER=FRIC (default) if the friction model is to be defined in user subroutine **FRIC**. Set USER=COEFFICIENT if the friction coefficient is to be defined in user subroutine **FRIC_COEF**.

In an Abaqus/Explicit analysis, set USER=FRIC (default) if the friction model is to be defined in user subroutine **VFRIC**. Set USER=FRICTION if the friction model is to be defined in user subroutine **VFRICTION**. **VFRIC** is applicable to contact pairs, whereas **VFRICTION** is applicable to general contact. Set USER=COEFFICIENT if the friction coefficient is to be defined in user subroutine **VFRIC_COEF**. **VFRIC_COEF** can be used only with general contact.

Optional parameters:

ANISOTROPIC

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 18.1.2 of the Abaqus Analysis User's Manual, 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** 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, μ.
2. Slip rate, $\dot{\gamma}_{eq}$. If this value is omitted, the friction coefficient is assumed to be independent of the slip rate.
3. Contact pressure, p. If this value is omitted, the friction coefficient is assumed to be independent of the contact pressure.

4. Average temperature at the contact point, $\bar{\theta}$, 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, μ_1.
2. Friction coefficient in the second slip direction, μ_2.
3. Slip rate, $\dot{\gamma}_{eq}$. If this value is omitted, the friction coefficient is assumed to be independent of the slip rate.
4. Contact pressure, p. If this value is omitted, the friction coefficient is assumed to be independent of the contact pressure.
5. Average temperature at the contact point, $\bar{\theta}$, 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, μ_s.

2. Kinetic friction coefficient, μ_k.
3. Decay coefficient, d_c. The default value is zero.

Data lines if the EXPONENTIAL DECAY and TEST DATA parameters are used:

First line:

1. Friction coefficient for the first data point, μ_1. This value corresponds to the static friction coefficient.

Second line:

1. Friction coefficient for the second data point, μ_2. This value corresponds to the dynamic friction coefficient measured at the reference slip rate, $\dot{\gamma}_2$.
2. Slip rate of the second data point, $\dot{\gamma}_2$. This value corresponds to the reference slip rate used to measure the dynamic friction coefficient.

Third line (optional):

1. Kinetic friction coefficient, μ_∞. This value corresponds to the asymptotic value of the friction coefficient at infinite slip rate, $\dot{\gamma}_\infty$. If this data line is omitted, Abaqus/Standard automatically calculates μ_∞ such that $(\mu_2 - \mu_\infty)/(\mu_1 - \mu_\infty) = 0.05$.

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`**, **`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 36.2.1 of the Abaqus Analysis User's Manual

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 33.2.1 of the Abaqus Analysis User's Manual
- *GAP
- *INTERFACE
- *SURFACE INTERACTION
- "GAPCON," Section 1.1.10 of the Abaqus User Subroutines Reference Manual

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 simulation. It must be used in conjunction with the *SURFACE INTERACTION option.

Product: Abaqus/Standard

Type: Model data

Level: Model in Abaqus/Standard; Step in Abaqus/Explicit

References:

- "Electrical contact properties," Section 33.3.1 of the Abaqus Analysis User's Manual
- *SURFACE INTERACTION
- "GAPELECTR," Section 1.1.11 of the Abaqus User Subroutines Reference Manual

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables on which σ_g depends.

USER

Include this parameter to define σ_g in user subroutine **GAPELECTR**. In this case the DEPENDENCIES parameter and any data lines are ignored.

Data lines to define the gap electrical conductance directly:

First line:

1. Electrical conductivity, σ_g. (Units of $\mathrm{CT}^{-1}\mathrm{L}^{-2}\varphi^{-1}$.)
2. Surface separation, *d*.
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.

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 29.5.7 of the Abaqus Analysis User's Manual

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 18.1.2 of the Abaqus Analysis User's Manual, 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 (μ) (TYPE=NEWTONIAN):

First line:

1. μ.
2. Temperature, θ.
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, α (TYPE=POWER LAW):

First line:

1. K.
2. α.
3. Temperature, θ.
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 33.1.5 of the Abaqus Analysis User's Manual
- "Thermal contact properties," Section 33.2.1 of the Abaqus Analysis User's Manual
- "Electrical contact properties," Section 33.3.1 of the Abaqus Analysis User's Manual
- *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. η, fraction of dissipated energy converted into heat, including any unit conversion factor. The default value is 1.0.
2. *f*, weighting factor for the distribution of heat between the interacting surfaces. The heat flux into the 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 33.2.1 of the Abaqus Analysis User's Manual
- *GAP
- *INTERFACE
- *SURFACE INTERACTION

There are no parameters associated with this option.

Data lines to define surface constants for radiative heat transfer:

First line:

1. Emissivity, ϵ_A.
2. Emissivity, ϵ_B.

Second line:

1. Effective viewfactor, F. $(0 \leq F \leq 1)$
2. Gap clearance, d.

Repeat this data line as often as necessary to define the dependence of the viewfactor on gap clearance.

7.7 *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 29.6.6 of the Abaqus Analysis User's Manual

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.

7.8 *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 29.6.6 of the Abaqus Analysis User's Manual

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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines to define a contact area for average pressure output:

First line:

1. Contact area or width. (This value cannot be negative.)
2. Closure. (This value cannot be negative.)
3. Temperature, θ.
4. First field variable.
5. Etc., up to five field variables.

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

1. Sixth field variable.

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

Repeat this set of data lines as often as necessary to define the dependence of the contact area or width versus closure curves on temperature and field variables.

7.9 *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 29.6.6 of the Abaqus Analysis User's Manual

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 18.1.2 of the Abaqus Analysis User's Manual, for more information.

VARIABLE

This parameter is used only with COMPONENT=TRANSVERSE SHEAR to specify the unit system in which the transverse shear behavior will be defined.

Set VARIABLE=FORCE to define the transverse shear stiffness in terms of force per unit displacement or force per unit length per unit displacement, depending on the element type to which this behavior refers.

Set VARIABLE=STRESS (default) to define the transverse shear stiffness in terms of stress per unit displacement.

Data lines for COMPONENT=TRANSVERSE SHEAR:

First line:

1. Shear stiffness. (This value cannot be negative.)
2. Temperature, θ.
3. First field variable.
4. Etc., up to six field variables.

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

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

Repeat this set of data lines as often as necessary to define the dependence of the shear stiffness on temperature and field variables.

Data lines for COMPONENT=MEMBRANE:

First line:

1. Young's modulus, E.
2. Poisson's ratio, ν.
3. Temperature, θ.
4. First field variable.
5. Etc., up to five field variables.

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

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

Repeat this set of data lines as often as necessary to define the dependence of Young's modulus and Poisson's ratio on temperature and field variables.

7.10 *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 29.6.1 of the Abaqus Analysis User's Manual
- "Defining the gasket behavior using a material model," Section 29.6.5 of the Abaqus Analysis User's Manual
- "Defining the gasket behavior directly using a gasket behavior model," Section 29.6.6 of the Abaqus Analysis User's Manual

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 Manual) 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}).

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.

7.11 *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 29.6.6 of the Abaqus Analysis User's Manual

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 18.1.2 of the Abaqus Analysis User's Manual, 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, θ.
4. First field variable.
5. Etc., up to five field variables.

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

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

Repeat this set of data lines as often as necessary to define the dependence of the loading curve on temperature and field variables.

Data lines to define the loading in terms of force or force per unit length per closure (DIRECTION=LOADING and VARIABLE=FORCE):

First line:

1. Force or force per unit length. (This value cannot be negative.)
2. Closure. (This value cannot be negative.)
3. Temperature, θ.
4. First field variable.
5. Etc., up to five field variables.

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

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

Repeat this set of data lines as often as necessary to define the dependence of the loading curve on temperature and field variables.

Data lines to define the unloading in terms of pressure versus closure for an elastic-plastic model (TYPE=ELASTIC-PLASTIC, DIRECTION=UNLOADING, and VARIABLE=STRESS):

First line:

1. Pressure. (This value cannot be negative.)
2. Closure. (This value must be positive.)
3. Plastic closure. (This value must be positive.)
4. Temperature, θ.
5. First field variable.
6. Etc., up to four field variables.

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

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

Repeat this set of data lines as often as necessary to define the dependence of the unloading curve of the elastic-plastic model on temperature and field variables.

Data lines to define the unloading in terms of force or force per unit length versus closure for an elastic-plastic model (TYPE=ELASTIC-PLASTIC, DIRECTION=UNLOADING, and VARIABLE=FORCE):

First line:

1. Force or force per unit length. (This value cannot be negative.)
2. Closure. (This value must be positive.)
3. Plastic closure. (This value must be positive.)
4. Temperature, θ.
5. First field variable.
6. Etc., up to four field variables.

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

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

Repeat this set of data lines as often as necessary to define the dependence of the unloading curve of the elastic-plastic model on temperature and field variables.

Data lines to define the unloading in terms of pressure versus closure for a damage model (TYPE=DAMAGE, DIRECTION=UNLOADING, and VARIABLE=STRESS):

First line:

1. Pressure. (This value cannot be negative.)

2. Closure. (This value cannot be negative.)
3. Maximum closure reached while loading the gasket. (This value must be positive.)
4. Temperature, θ.
5. First field variable.
6. Etc., up to four field variables.

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

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

Repeat this set of data lines as often as necessary to define the dependence of the unloading curve of the damage model on temperature and field variables.

Data lines to define the unloading in terms of force or force per unit length versus closure for a damage model (TYPE=DAMAGE, DIRECTION=UNLOADING, and VARIABLE=FORCE):

First line:

1. Force or force per unit length. (This value cannot be negative.)
2. Closure. (This value cannot be negative.)
3. Maximum closure reached while loading. (This value must be positive.)
4. Temperature, θ.
5. First field variable.
6. Etc., up to four field variables.

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

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

Repeat this set of data lines as often as necessary to define the dependence of the unloading curve of the damage model on temperature and field variables.

7.12 *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 22.2.1 of the Abaqus Analysis User's Manual

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 $\mathrm{JM}^{-1}\theta^{-1}$.)
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.

7.13 *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 23.7.5 of the Abaqus Analysis User's Manual

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, $r_a^{\rm dry}$. (Units of L.)
2. Fully swollen radius of gel particles, r_a^f. (Units of L.)
3. Number of gel particles per unit volume, k_a. (Units of L^{-3}.)
4. Relaxation time constant for long-term swelling of gel particles, τ_1. (Units of T.)

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 Manual

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

Product: Abaqus/Standard

Type: History data

Level: Step

References:

- "Material damping," Section 23.1.1 of the Abaqus Analysis User's Manual
- "Damping in dynamic analysis" in "Dynamic analysis procedures: overview," Section 6.3.1 of the Abaqus Analysis User's Manual
- "Acoustic, shock, and coupled acoustic-structural analysis," Section 6.10.1 of the Abaqus Analysis User's Manual

Optional parameters:

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.

ALPHA

Set this parameter equal to the α_{global} factor to create global Rayleigh mass proportional damping $D_\alpha = \alpha_{global} * M$, where M denotes the model mass matrix. The default is ALPHA=0. (Units of T^{-1}.)

BETA

Set this parameter equal to the β_{global} factor to create global Rayleigh stiffness proportional damping $D_\beta = \beta_{global} * K$, where K denotes the model stiffness matrix. The default is BETA=0. (Units of T.)

STRUCTURAL

Set this parameter equal to the s_{global} factor to create frequency-independent stiffness proportional structural damping $D_s = i s_{global} * K$, 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/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 Manual

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, or coupled temperature-displacement 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 Manual
- "HETVAL," Section 1.1.13 of the Abaqus User Subroutines Reference Manual

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/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 Manual
- "Cavity radiation," Section 37.1.1 of the Abaqus Analysis User's Manual

Optional parameters:

DELTMX

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

Set END=PERIOD (default) to analyze a specific time period. Set END=SS to end the analysis when steady state is reached.

This parameter is relevant only for transient analysis.

STEADY STATE

Include this parameter to choose steady-state analysis. Transient analysis is assumed if this parameter is omitted.

Optional parameter for cavity radiation analysis:

MXDEM

Set this parameter equal to the maximum allowable emissivity change with temperature and field variables during an increment. If this value is exceeded, Abaqus/Standard will cut back the increment until the maximum change in emissivity is less than the specified value. If this parameter is omitted, a default value of 0.1 is used.

Data line to control incrementation and steady-state conditions in a pure heat transfer analysis:

First (and only) line:

1. Initial time increment. If automatic incrementation is used, this should be a reasonable suggestion for the initial increment size and will be adjusted as necessary. If direct incrementation is used, this will be the fixed time increment size.
2. Total time period. If END=SS is chosen, the step ends when steady state is reached or after this time period, whichever occurs first.
3. Minimum time increment allowed. If Abaqus/Standard finds it needs a smaller time increment than this value, the analysis is terminated. If a value is given, Abaqus/Standard will use the minimum of the given value and 0.8 times the initial time increment. If no value is given, Abaqus/Standard sets the minimum increment to the minimum of 0.8 times the initial time increment (first data item on this data line) and 1×10^{-5} times the total time period (second data item on this data line). This value is used only for automatic time incrementation.
4. Maximum time increment allowed. If this value is not specified, no upper limit is imposed. This value is used only for automatic time incrementation.
5. Temperature change rate (temperature per time) used to define steady state; only needed if END=SS is chosen. When all nodal temperatures are changing at less than this rate, the solution terminates.

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 27.4.1 of the Abaqus Analysis User's Manual

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 18.1.2 of the Abaqus Analysis User's Manual, 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 24.1.4 of the Abaqus Analysis User's Manual
- *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 19.5.1 of the Abaqus Analysis User's Manual
- "UHYPER," Section 1.1.33 of the Abaqus User Subroutines Reference Manual
- *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 β while the other coefficients of the Van der Waals model are fitted from the test data given by the user. If this parameter is omitted, β will be determined from a nonlinear, least-squares fit of the test data. Allowable values of BETA are $0 \leq \beta \leq 1$. It is recommended to set $\beta = 0$ if only one type of test data is available.

MODULI

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

Set MODULI=INSTANTANEOUS to indicate that the hyperelastic material constants define the instantaneous behavior. This 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, ν, to account for compressibility. This parameter cannot be used if the material coefficients are specified directly or if volumetric behavior is defined by entering nonzero values for D_i on the data line or by specifying the *VOLUMETRIC TEST DATA option. In addition, this parameter cannot be used for the Marlow model 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. μ.
2. λ_m.
3. D.
4. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

Data lines to define the material constants for the MOONEY-RIVLIN model:

First line:

1. C_{10}.
2. C_{01}.
3. D_1.
4. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

Data lines to define the material constants for the NEO HOOKE model:

First line:

1. C_{10}.
2. D_1.
3. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

Data lines to define the material constants for the OGDEN strain energy potential:

First line if N=1:

1. μ_1.
2. α_1.
3. D_1.
4. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

First line if N=2:

1. μ_1.
2. α_1.
3. μ_2.
4. α_2.
5. D_1.
6. D_2.
7. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

First line if N=3:

1. μ_1.
2. α_1.
3. μ_2.

4. α_2.
5. μ_3.
6. α_3.
7. D_1.
8. D_2.

Second line if N=3:

1. D_3.
2. Temperature.

Repeat this pair of data lines as often as necessary to define the material constants as a function of temperature.

Data lines for higher values of N (up to 6):

1. The data lines for higher values of *N* follow the same pattern. First, give the μ_i and α_i for i from *1* to *N*. Then, give the *N* coefficients D_i. Finally, give the temperature. Exactly eight data values should be given on each line.

Data lines to define the material constants for the POLYNOMIAL strain energy potential:

First line if N=1:

1. C_{10}.
2. C_{01}.
3. D_1.
4. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

First line if N=2:

1. C_{10}.
2. C_{01}.
3. C_{20}.
4. C_{11}.
5. C_{02}.
6. D_1.
7. D_2.
8. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

First line if N=3:

1. C_{10}.
2. C_{01}.

3. C_{20}.
4. C_{11}.
5. C_{02}.
6. C_{30}.
7. C_{21}.
8. C_{12}.

Second line if N=3:

1. C_{03}.
2. D_1.
3. D_2.
4. D_3.
5. Temperature.

Repeat this pair of data lines as often as necessary to define the material constants as a function of temperature.

Data lines for higher values of N (up to 6):

1. The data lines for higher values of *N* follow the same pattern. For each value of $(i+j)$ from *1* to *N*, give the C_{ij}, with *i* decreasing from $(i+j)$ to zero and *j* increasing from zero to $(i+j)$. Then, give the N coefficients D_i. Finally, give the temperature. Exactly eight data values should be given on each line.

Data lines to define the material constants for the REDUCED POLYNOMIAL strain energy potential:

First line if N=1:

1. C_{10}.
2. D_1.
3. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

First line if N=2:

1. C_{10}.
2. C_{20}.
3. D_1.
4. D_2.
5. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

First line if N=3:

1. C_{10}.
2. C_{20}.
3. C_{30}.
4. D_1.
5. D_2.
6. D_3.
7. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

Data lines for higher values of N (up to 6):

1. The data lines for the higher values of N follow the same pattern. First, give the C_{i0} for i from 1 to N. Then, give the N coefficients D_i. Finally, give the temperature. Exactly eight data values should be given on each line.

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

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

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

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

Data lines to define the material constants for the VAN DER WAALS model:

First line:

1. μ.
2. λ_m.
3. a.
4. β.
5. D.
6. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

Data lines to define the material constants for the YEOH model:

First line:

1. C_{10}.
2. C_{20}.

3. C_{30}.
4. D_1.
5. D_2.
6. D_3.
7. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

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 19.5.2 of the Abaqus Analysis User's Manual
- *BIAXIAL TEST DATA
- *PLANAR TEST DATA
- *SIMPLE SHEAR TEST DATA
- *UNIAXIAL TEST DATA
- *VOLUMETRIC TEST DATA

Optional parameters:

MODULI

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

Set MODULI=INSTANTANEOUS to indicate that the hyperfoam material constants define the instantaneous behavior. This parameter value is not available for frequency domain viscoelasticity in an Abaqus/Standard analysis.

Set MODULI=LONG TERM (default) to indicate that the hyperfoam material constants define the long-term behavior.

N

Set this parameter equal to the order of the strain energy potential. The maximum value is N=6. The default is N=1.

POISSON

Set this parameter equal to the effective Poisson's ratio, ν, of the material. This parameter is valid only when the TEST DATA INPUT parameter is specified.

If this parameter is included, it is assumed that $\nu_i = \nu$ for all i and that lateral strain test data and volumetric test data are not required. If this parameter is omitted, the ν_i are calculated from lateral strain data and/or volumetric test data.

TEST DATA INPUT

Include this parameter if the μ_i, the α_i, and the ν_i material constants are to be computed by Abaqus from data taken from simple tests on a material specimen. If this parameter is omitted, μ_i, α_i, and ν_i must be given directly on the data lines.

To define elastomeric foam behavior by giving test data:

No data lines are used with this option when the TEST DATA INPUT parameter is specified. The data are given instead under the *BIAXIAL TEST DATA, the *PLANAR TEST DATA, the *SIMPLE SHEAR TEST DATA, the *UNIAXIAL TEST DATA, and the *VOLUMETRIC TEST DATA options.

Data lines to define the elastic properties directly:

First line if N=1:

1. μ_1.
2. α_1.
3. ν_1.
4. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

First line if N=2:

1. μ_1.
2. α_1.
3. μ_2.
4. α_2.
5. ν_1.
6. ν_2.
7. Temperature.

Repeat this data line as often as necessary to define the material constants as a function of temperature.

First line if N=3:

1. μ_1.
2. α_1.
3. μ_2.
4. α_2.
5. μ_3.

6. α_3.
7. ν_1.
8. ν_2.

Second line if N=3:

1. ν_3.
2. Temperature.

Repeat this pair of data lines as often as necessary to define the material constants as a function of temperature.

Data lines for higher values of N (up to 6):

1. The data lines for higher values of *N* follow the same pattern. First, give the μ_i and α_i for i from *1* to *N*. Then, give the $N\,\nu_i$. Finally, give the temperature. Exactly eight data values should be given on each line.

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 19.4.1 of the Abaqus Analysis User's Manual
- "UHYPEL," Section 1.1.32 of the Abaqus User Subroutines Reference Manual

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, ν.
3. First strain invariant, I_1.
4. Second strain invariant, I_2.
5. Third strain invariant, I_3.

Repeat this data line as often as necessary to define the variation of the moduli with the strain invariants.

To define hypoelasticity by a user subroutine:

No data lines are used with this option when the USER parameter is specified. Instead, user subroutine **UHYPEL** must be used to define the hypoelasticity.

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 19.8.1 of the Abaqus Analysis User's Manual

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 19.8.1 of the Abaqus Analysis User's Manual.

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis; multiphysics solutions for insight into challenging engineering problems; and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Vélizy, France, SIMULIA provides sales, services, and support through a global network of regional offices and distributors. For more information, visit www.simulia.com.

About Dassault Systèmes

As a world leader in 3D and Product Lifecycle Management (PLM) solutions, Dassault Systèmes brings value to more than 100,000 customers in 80 countries. A pioneer in the 3D software market since 1981, Dassault Systèmes develops and markets PLM application software and services that support industrial processes and provide a 3D vision of the entire lifecycle of products from conception to maintenance to recycling. The Dassault Systèmes portfolio consists of CATIA for designing the virtual product, SolidWorks for 3D mechanical design, DELMIA for virtual production, SIMULIA for virtual testing, ENOVIA for global collaborative lifecycle management, and 3DVIA for online 3D lifelike experiences. Dassault Systèmes' shares are listed on Euronext Paris (#13065, DSY.PA) and Dassault Systèmes' ADRs may be traded on the US Over-The-Counter (OTC) market (DASTY). For more information, visit www.3ds.com.

DASSAULT SYSTEMES