ABAQUS 2016

GLOSSARY

3DEXPERIENCE®

Abaqus Glossary

Preface

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

Support

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

Online support

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

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

Training

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

Feedback

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

Part I: Using the Abaqus Glossary

This glossary defines technical terms as they apply to the Abaqus Unified FEA Product Suite.

Terms may have more than one definition depending on which product they are referring to and the context in which they are used.

Cross-references to relevant sections throughout the documentation collection are provided.

Terms and definitions

Numeric

3D compass

An Abaqus/CAE viewport object that indicates the current orientation of the model in the viewport. You can manipulate the view of the model by clicking and dragging different elements of the compass. The compass performs certain view manipulations that are not available using the other view manipulation tools, including rotating the view about a fixed axis, panning the view along a fixed axis, and panning the view within a fixed plane.

For more information:

- “The 3D compass,” Section 5.3 of the Abaqus/CAE User’s Guide

3D XML

An Extensible Markup Language-based format for encoding three-dimensional images and data. You can export model images from an Abaqus/CAE viewport to a 3D XML-format file.

For more information:

- “What kinds of files can be imported and exported from Abaqus/CAE?,” Section 10.1.1 of the Abaqus/CAE User’s Guide
- “Exporting viewport data to a 3D XML-format file,” Section 10.9.5 of the Abaqus/CAE User’s Guide, in the HTML version of this guide

A

Abaqus/AMS

An add-on analysis capability for Abaqus/Standard that allows you to select the automatic multi-level substructuring (AMS) eigensolver when performing a natural frequency extraction.

For more information:

- “Automatic multi-level substructuring (AMS) eigensolver” in “Natural frequency extraction,” Section 6.3.5 of the Abaqus Analysis User’s Guide

Abaqus/Aqua

An add-on analysis capability for Abaqus/Standard and Abaqus/Explicit for calculating drag and buoyancy loads based on steady current, wave, and wind effects for modeling offshore piping and floating platform structures. Abaqus/Aqua is applicable for structures that can be idealized using line elements, including beam, pipe, and truss elements.

For more information:

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

Abaqus/CAE

The Complete Abaqus Environment: a program that provides a simple, consistent interactive graphical interface for creating, submitting, monitoring, and evaluating results from Abaqus simulations. Abaqus/CAE is divided into modules, where each module defines a logical aspect of the modeling process; for example, defining the geometry, defining material properties, generating a mesh, submitting analysis jobs, and interpreting results.

For more information:

- Part I, "Interacting with Abaqus/CAE," of the Abaqus/CAE User's Guide

Abaqus/CFD

A computational fluid dynamics program with extensive support for preprocessing, simulation, and postprocessing in Abaqus/CAE. Abaqus/CFD provides scalable parallel CFD simulation capabilities to address a broad range of nonlinear coupled fluid-thermal and fluid-structural problems.

For more information:

- "Introduction: general," Section 1.1.1 of the Abaqus Analysis User's Guide
- "Fluid dynamic analysis procedures: overview," Section 6.6.1 of the Abaqus Analysis User's Guide

Abaqus/Design

An add-on capability for Abaqus/Standard and Abaqus/Explicit that allows an Abaqus model to be defined with parametric variables. Parameter studies with such models can be performed with scripts that generate models with various values for the parametric variables, run the analyses, and gather the results. These scripts are developed using Python, an interpreted language.

For more information:

- "Parametric input," Section 1.4.1 of the Abaqus Analysis User's Guide

Abaqus/Explicit

An explicit dynamics finite element program that provides nonlinear, transient, dynamic analysis of solids and structures using explicit time integration. Its powerful contact capabilities, reliability, and computational efficiency on large models also make it highly effective for quasi-static applications involving discontinuous nonlinear behavior.

For more information:

- "Introduction: general," Section 1.1.1 of the Abaqus Analysis User's Guide

Abaqus/Foundation

An optional subset of Abaqus/Standard that provides more cost-efficient access to the linear static and dynamic analysis functionality in Abaqus/Standard.

Abaqus PDE (Abaqus Python Development Environment)

An application that allows you to create, edit, test, and debug Python scripts. It can be used with scripts containing general Python commands, Abaqus Scripting Interface commands, or Abaqus/CAE graphical user interface (GUI) commands.

For more information:

- Part III, "The Abaqus Python development environment," of the Abaqus Scripting User's Guide

Abaqus/Standard

A general-purpose finite element program that can be used for analysis of static, dynamic, heat transfer, and a variety of coupled problems. Abaqus/Standard provides both automatic and direct user control of the time step and is effective for analyzing the static, dynamic, thermal, and electrical response of both linear and nonlinear models.

For more information:

- "Introduction: general," Section 1.1.1 of the Abaqus Analysis User's Guide

Abaqus/Viewer

A subset of Abaqus/CAE that contains just the Visualization module to provide graphical display of finite element models and results. It obtains model and result information from the output database (ODB). The major capabilities of Abaqus/Viewer include undeformed and deformed shape plotting; results, contour, and symbol plotting; *X–Y* plotting and reporting; field output reporting; plot customization; and animation.

For more information:

- Part V, "Viewing results," of the Abaqus/CAE User's Guide

ACIS

An industry-standard library of geometric modeling functions that reads and writes ACIS format files (*file_name*`.sat`). ACIS files give you a way to move geometry between Abaqus/CAE and third-party modeling products. You can import the base feature of a part from an ACIS file; in addition, you can export a part or the part instances in the assembly to an ACIS file.

For more information:

- "Using the File menu," Section 9.6 of the Abaqus/CAE User's Guide, in the HTML version of this guide

active representation

Geometry that will be meshed and used in an analysis, including all geometry in a model except for reference representations and suppressed features.

For more information:

- "Understanding midsurface modeling," Section 35.1 of the Abaqus/CAE User's Guide

adaptive remeshing

An automated process to refine mesh discretization based on selected error indicators in solution results.

For more information:

- "Adaptive remeshing: overview," Section 12.3.1 of the Abaqus Analysis User's Guide

adaptivity process

A succession of analysis jobs that Abaqus/CAE creates during adaptive remeshing. Each job uses a mesh that was generated by Abaqus/CAE based on the values of the error indicator output variables in the output database and the settings in your remeshing rule.

For more information:

- "Understanding adaptivity processes," Section 19.3 of the Abaqus/CAE User's Guide

ALE (Arbitrary Lagrangian-Eulerian) adaptive meshing

A technique that combines the features of pure Lagrangian analysis and pure Eulerian analysis to control element distortion in cases where large deformation or loss of material occurs. ALE allows the mesh to move independently of the material but does not alter the topology (elements and connectivity) of the mesh.

For more information:

- "ALE adaptive meshing: overview," Section 12.2.1 of the Abaqus Analysis User's Guide

analysis input file

See *input file*.

analysis input file processor

A component of Abaqus that processes the input file and submits the resulting data to the appropriate analysis program, either Abaqus/Standard, Abaqus/Explicit, or Abaqus/CFD. The input file processor interprets the Abaqus options, performs the necessary consistency checking, and prepares the data for the analysis program.

For more information:

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

analysis procedure

See *procedure*.

Analytical Field toolset

An Abaqus/CAE toolset that allows you to define spatially varying parameters for selected interactions and prescribed conditions in the Interaction module or in the Load module.

For more information:

- Chapter 58, "The Analytical Field toolset," of the Abaqus/CAE User's Guide

analytical rigid part

See *analytical rigid surface*.

analytical rigid surface

An undeformable geometric surface with profiles that can be described with straight and curved line segments. These profiles can be swept along a generator vector or rotated about an axis to form a three-dimensional surface. Analytical rigid surfaces are not element-based and, thus, can model many surface geometries exactly and may result in decreased computational cost in an analysis.

For more information:

- "Analytical rigid surface definition," Section 2.3.4 of the Abaqus Analysis User's Guide
- "Rigid parts," Section 11.7.1 of the Abaqus/CAE User's Guide

anchor points

In Abaqus/CAE, points in a viewport that control the motion of text and arrow annotations.

For more information:

- "What are arrow and text annotations?," Section 4.1.2 of the Abaqus/CAE User's Guide

In Abaqus/Standard contact interactions, points created on a surface by the small-sliding tracking approach to define the location and orientation of planar contact constraints.

For more information:

- "Contact formulations in Abaqus/Standard," Section 38.1.1 of the Abaqus Analysis User's Guide

angle method

A process in Abaqus/CAE that allows you to select multiple faces, edges, elements, element faces, or nodes based on the spatial angles between the items.

For more information:

- "Using the angle and feature edge method to select multiple objects," Section 6.2.3 of the Abaqus/CAE User's Guide

annotation

An explanatory note or symbol that you create when working with Abaqus/CAE. Abaqus/CAE automatically generates several types of annotations. Viewport annotations generated by Abaqus/CAE include the triad as well as the legend, title block, and state block of Visualization module plots.

For more information:

- "What are arrow and text annotations?," Section 4.1.2 of the Abaqus/CAE User's Guide
- Chapter 56, "Customizing viewport annotations," of the Abaqus/CAE User's Guide

assembly

A collection of positioned part instances. A model contains only one assembly.

For more information:

- "Defining an assembly," Section 2.10.1 of the Abaqus Analysis User's Guide
- Chapter 13, "The Assembly module," of the Abaqus/CAE User's Guide

Assembly module

An Abaqus/CAE module used to create instances of parts and to construct an assembly by positioning those instances relative to each other in a global coordinate system.

For more information:

- Chapter 13, "The Assembly module," of the Abaqus/CAE User's Guide

assembly-related modules

Abaqus/CAE modules in which the assembly is displayed in the viewport. The Assembly, Step, Interaction, Load, and Mesh modules are considered to be assembly-related modules.

For more information:

- "What is a module?," Section 2.3 of the Abaqus/CAE User's Guide

assembly set

Named groupings of points, edges, surfaces, or volumes that identify regions of an Abaqus/CAE assembly.

For more information:

- "How do part sets and assembly sets differ?," Section 73.2.2 of the Abaqus/CAE User's Guide

attachment line

A feature in Abaqus/CAE that allows you to specify the location of discrete fasteners by projecting selected points through multiple faces.

For more information:

- "Understanding attachment points and lines," Section 59.1 of the Abaqus/CAE User's Guide

attachment point

A feature in Abaqus/CAE that allows you to specify the location of a *positioning point* for point-based fasteners.

For more information:

- "Understanding attachment points and lines," Section 59.1 of the Abaqus/CAE User's Guide

Attachment toolset

An Abaqus/CAE toolset available in the Property, Assembly, and Interaction modules that allows you to create and manage attachment points and lines that you can use to create point-based and discrete fasteners, respectively.

For more information:

- Chapter 59, "The Attachment toolset," of the Abaqus/CAE User's Guide

AutoCAD files

Two-dimensional profiles stored in AutoCAD files (*file_name*`.dxf`) that can be imported into Abaqus/CAE as stand-alone sketches.

For more information:

- "Imported sketches," Section 20.3.2 of the Abaqus/CAE User's Guide

B

base feature

The first feature you create when building an Abaqus/CAE part; you construct the remainder of the part by adding more features that either modify or add detail to the base feature. All other features of the part are children of the base feature; therefore, the base feature cannot be suppressed or deleted.

For more information:

- "The base feature," Section 11.3.2 of the Abaqus/CAE User's Guide

base solution

In general, the initial run or solution of a process; the meaning varies according to the context in which the term is used.

For more information:

- "Characteristics of error indicators" in "Solution-based mesh sizing," Section 12.3.3 of the Abaqus Analysis User's Guide

base solution variable

The variable selected to use in the calculation of the base solution of a process. Base solution variables are expressed as element averages of the base solution and identified by an "AVG" suffix; for example, MISESAVG.

For more information:

- "Solution accuracy" in "Error indicator output," Section 4.1.4 of the Abaqus Analysis User's Guide

basic manager

An Abaqus/CAE dialog box that contains lists of all the objects of a certain type that you have created in the current model. The basic manager also contains **Create**, **Edit**, **Copy**, **Rename**, and **Delete** buttons that you can use to manipulate existing objects and to create new ones. See also *step-dependent manager*.

For more information:

- "What are basic managers?," Section 3.4.1 of the Abaqus/CAE User's Guide

BC

See *boundary condition*.

beam sections

The area properties for the cross-section of beam elements.

For more information:

- "Meshed beam cross-sections," Section 10.6 of the Abaqus Analysis User's Guide
- "Choosing a beam cross-section," Section 29.3.2 of the Abaqus Analysis User's Guide

bias factor

A factor defined in an *adaptive remeshing* rule that determines the distribution of element sizing between the locations of minimum and maximum solution intensity.

For more information:

- “Bias factor” in “Solution-based mesh sizing,” Section 12.3.3 of the Abaqus Analysis User’s Guide

bias function

See *bias parameter*.

For more information:

- “The bias parameter” in “Mode-based steady-state dynamic analysis,” Section 6.3.8 of the Abaqus Analysis User’s Guide

bias parameter

In a steady-state dynamic analysis, a variable used to bias the frequency points for which results are required toward the ends of the user-defined frequency range. The bias parameter provides closer spacing of the results points either toward the middle or toward the ends of each frequency interval.

For more information:

- “The bias parameter” in “Mode-based steady-state dynamic analysis,” Section 6.3.8 of the Abaqus Analysis User’s Guide

blend

A geometric fillet or chamfer that can be created in Abaqus/CAE to smooth an edge of a three-dimensional solid part.

For more information:

- “Blend features,” Section 11.9.5 of the Abaqus/CAE User’s Guide

blind cut

A cut that penetrates a three-dimensional object only to a specified depth, rather than passing all the way through it. In Abaqus/CAE this depth is stored as a parameter of the cut feature and can be modified.

For more information:

- “Adding a cut feature,” Section 11.24 of the Abaqus/CAE User’s Guide, in the HTML version of this guide

bottom-up meshing

A manual process used to create a hexahedral or hex-dominated mesh on a solid region that is unmeshable or difficult to mesh using the automated top-down meshing techniques. Bottom-up meshing allows you to select sweep, extrude, or revolve methods and parameters such as the source and connecting faces that Abaqus/CAE uses to build up a solid mesh of hexahedral elements. See also *top-down meshing*.

For more information:

- Chapter 17, "The Mesh module," of the Abaqus/CAE User's Guide

boundary condition

A prescribed value for a basic solution variable, such as displacement, rotation, or temperature.

For more information:

- "Boundary conditions in Abaqus/Standard and Abaqus/Explicit," Section 34.3.1 of the Abaqus Analysis User's Guide
- "Boundary conditions in Abaqus/CFD," Section 34.3.2 of the Abaqus Analysis User's Guide
- Chapter 16, "The Load module," of the Abaqus/CAE User's Guide

boundary face

In an Abaqus/Standard analysis, an exterior face or a face of a free surface.

For more information:

- "Eulerian analysis," Section 14.1.1 of the Abaqus Analysis User's Guide

In an Abaqus/CAE model, a face of a three-dimensional element that is shared by only a single element.

For more information:

- "Merging and cutting part instances," Section 13.7.1 of the Abaqus/CAE User's Guide

boundary mesh

The mesh generated on the boundary faces of a three-dimensional geometric region.

For more information:

- "What is a tetrahedral boundary mesh?," Section 17.10.4 of the Abaqus/CAE User's Guide

C

CAD Connection toolset

An Abaqus/CAE toolset that allows you to create a connection from Abaqus/CAE to a third-party CAD system; you can use the CAD system to modify the model or to change its position, and you can use the established connection to update the model quickly in Abaqus/CAE.

For more information:

- Chapter 60, "The CAD Connection toolset," of the Abaqus/CAE User's Guide

canvas

The region of the Abaqus/CAE main window where work takes place.

For more information:

- Chapter 4, "Managing viewports on the canvas," of the Abaqus/CAE User's Guide

CATIA

A CAD/CAM/CAE software package from Dassault Systèmes. CATIA-format parts and assemblies can be imported into Abaqus/CAE.

For more information:

- "What kinds of files can be imported and exported from Abaqus/CAE?," Section 10.1.1 of the Abaqus/CAE User's Guide, in the HTML version of this guide

chamfer

A straight blend that creates a beveled edge connecting two surfaces.

For more information:

- "Blend features," Section 11.9.5 of the Abaqus/CAE User's Guide

check box

An element of Abaqus/CAE dialog boxes that can be used to turn a particular option alternately off or on.

For more information:

- "Using basic dialog box components," Section 3.2.1 of the Abaqus/CAE User's Guide

child feature

A feature in Abaqus/CAE that, when created, depends on an existing feature called the parent feature for geometric and dimensioning information. When you modify a parent feature, the modification may change its child features. Likewise, when you delete a parent feature, Abaqus/CAE automatically deletes all of its child features.

For more information:

- "The relationship between parts and features," Section 11.3.1 of the Abaqus/CAE User's Guide

coarsening rate limit

A limit that you specified in a remeshing rule that modulates the rate at which larger elements are introduced into the mesh when Abaqus/CAE is adaptively remeshing.

For more information:

- "Refinement and coarsening rate factors" in "Solution-based mesh sizing," Section 12.3.3 of the Abaqus Analysis User's Guide
- "Choosing remeshing rule constraints," Section 17.21.4 of the Abaqus/CAE User's Guide, in the HTML version of this guide

co-execution

The co-simulation execution of two Abaqus analysis jobs that are executed in Abaqus/CAE in synchronization with one another.

For more information:

- "Structural-to-structural co-simulation," Section 17.3.1 of the Abaqus Analysis User's Guide
- "Fluid-to-structural and conjugate heat transfer co-simulation," Section 17.3.2 of the Abaqus Analysis User's Guide
- "Understanding analysis jobs," Section 19.2 of the Abaqus/CAE User's Guide
- Chapter 26, "Co-simulation," of the Abaqus/CAE User's Guide

compass

See *3D compass*.

compatible mesh

An interface between two meshes where the mesh topology is consistent across the interface.

For more information:

- "Compatible meshes between part instances," Section 17.14.3 of the Abaqus/CAE User's Guide

connector

In an Abaqus analysis, an element that models the behavior of mechanical joints or fasteners between components of a model and may include (nonlinear) force versus displacement (or velocity) behavior, friction, damage, and other phenomena.

For more information:

- Chapter 31, "Connector Elements," of the Abaqus Analysis User's Guide

In an Abaqus/CAE model, a connection that allows you to model mechanical relationships between two points in an assembly.

For more information:

- "What is a connector?," Section 24.2 of the Abaqus/CAE User's Guide

constraint

In a modeling interaction, a relationship between particular degrees of freedom that is enforced during the simulation; for example, linear constraints, general constraints, and kinematic coupling.

For more information:

- Chapter 35, "Constraints," of the Abaqus Analysis User's Guide

In a prescribed condition, a prespecified value of a degree of freedom. See also *boundary condition*.

For more information:

- "Managing prescribed conditions," Section 16.3 of the Abaqus/CAE User's Guide

In an Abaqus/CAE assembly, a feature that specifies the geometric relationships between parts to establish their relative position.

For more information:

- "How the position constraint methods differ," Section 13.5.2 of the Abaqus/CAE User's Guide

In an Abaqus/CAE sketch, a feature used to specify geometric relationships between entities in a sketch.

For more information:

- "Customizing the use of constraints in the Sketcher," Section 20.9.10 of the Abaqus/CAE User's Guide, in the HTML version of this guide

constraint control point

The point in a coupling constraint definition to which the motion of the coupled surface is constrained.

For more information:

- "Coupling constraints," Section 35.3.2 of the Abaqus Analysis User's Guide
- "Defining coupling constraints," Section 15.15.4 of the Abaqus/CAE User's Guide, in the HTML version of this guide

constraint region

An element-based or node-based surface involved in a coupling constraint.

For more information:

- "Coupling constraints," Section 35.3.2 of the Abaqus Analysis User's Guide
- "Defining coupling constraints," Section 15.15.4 of the Abaqus/CAE User's Guide, in the HTML version of this guide

construction geometry

Points, lines, or circles in the Abaqus/CAE Sketch module that help you position and align objects in your sketch. Construction geometry is visible only when you are sketching and is not visible on the part or assembly you are creating or modifying after you exit the Sketch module.

For more information:

- "Construction geometry," Section 20.5.2 of the Abaqus/CAE User's Guide
- "Connectors: overview," Section 31.1.1 of the Abaqus Analysis User's Guide, in the HTML version of this guide

contact initialization

In general contact definitions in Abaqus, a set of rules governing the adjustment of surface positions at the beginning of the analysis. Contact initialization is intended to provide minor corrections to slight overclosures or gaps caused by mesh discretization.

For more information:

- "Controlling initial contact status in Abaqus/Standard," Section 36.2.4 of the Abaqus Analysis User's Guide
- "Controlling initial contact status for general contact in Abaqus/Explicit," Section 36.4.4 of the Abaqus Analysis User's Guide
- "Contact initialization editor," Section 15.9.5 of the Abaqus/CAE User's Guide

contact pair

Two model surfaces that can interact with each other according to mechanical, thermal, electrical, or pore fluid contact properties.

For more information:

- "Defining contact pairs in Abaqus/Standard," Section 36.3 of the Abaqus Analysis User's Guide
- "Defining contact pairs in Abaqus/Explicit," Section 36.5 of the Abaqus Analysis User's Guide

context bar

A region of the Abaqus/CAE GUI located directly above the canvas and drawing area that contains a **Module** list from which you can select a module; other items in the context bar are a function of the module in which you are working. (Abaqus/Viewer contains only the Visualization module.)

For more information:

- "Components of the main window," Section 2.2.1 of the Abaqus/CAE User's Guide

context-sensitive help

A detailed set of instructions that allows you to gain immediate access to specific information in the Abaqus/CAE online guide. Invoke context-sensitive help by selecting **Help→On Context** from the main menu bar and then clicking almost any feature of an Abaqus/CAE window or dialog box.

For more information:

- "Displaying context-sensitive help," Section 2.6.1 of the Abaqus/CAE User's Guide

contour plot

Graphical output from the Visualization module of Abaqus/CAE that displays the values of a particular analysis variable at a specified step and frame. These values are shown as colored lines, colored bands, or colored faces on the model, depending on the customization options you select.

For more information:

- Chapter 44, "Contouring analysis results," of the Abaqus/CAE User's Guide

CORM (components of relative motion)

Relative displacements and rotations that are local to a connector.

For more information:

- "Connection-type library," Section 31.1.5 of the Abaqus Analysis User's Guide
- "Creating connector sections," Section 15.12.11 of the Abaqus/CAE User's Guide, in the HTML version of this guide

co-simulation

A multiphysics capability that provides several functions, available within Abaqus or as separate add-on analysis capabilities, for runtime coupling of Abaqus and another analysis program. An Abaqus analysis can be coupled to another Abaqus analysis or to a third-party analysis program to perform multidisciplinary simulations and multidomain (multimodel) coupling.

For more information:

- "Co-simulation: overview," Section 17.1.1 of the Abaqus Analysis User's Guide, in the HTML version of this guide
- Chapter 26, "Co-simulation," of the Abaqus/CAE User's Guide

CSYS (coordinate system)

An Abaqus/CAE datum coordinate system, either rectangular, cylindrical, or spherical.

For more information:

- "Creating datum coordinate systems," Section 62.9 of the Abaqus/CAE User's Guide, in the HTML version of this guide

current viewport

A window on the canvas, identified by a different color title bar than other viewports, in which work in Abaqus/CAE takes place. There is only one current viewport at any time.

For more information:

- “What is a viewport?,” Section 4.1.1 of the Abaqus/CAE User’s Guide

custom view

A predefined view (particular combinations of position, orientation, and scale factor) that Abaqus/CAE allows you to apply to an object in a selected viewport. The custom views are front, back, left, right, top, bottom, and isometric, as well as four user-defined views.

For more information:

- “Custom views,” Section 5.2.8 of the Abaqus/CAE User’s Guide

D

damage evolution

A law that defines how material degrades after one or more damage initiation criteria are met. Multiple forms of damage evolution may act on a material at the same time—one for each damage initiation criterion that was defined.

For more information:

- “Damage evolution and element removal for ductile metals,” Section 24.2.3 of the Abaqus Analysis User’s Guide
- “Damage evolution and element removal for fiber-reinforced composites,” Section 24.3.3 of the Abaqus Analysis User’s Guide
- “Damage evolution for ductile materials in low-cycle fatigue,” Section 24.4.3 of the Abaqus Analysis User’s Guide
- “Damage evolution” in “Defining damage,” Section 12.9.3 of the Abaqus/CAE User’s Guide, in the HTML version of this guide

damage initiation criteria

The conditions that will initiate a crack. They can be based on either maximum principal stress or maximum principal strain to represent the behavior of a material.

For more information:

- Chapter 24, “Progressive Damage and Failure,” of the Abaqus Analysis User’s Guide
- “Defining damage,” Section 12.9.3 of the Abaqus/CAE User’s Guide, in the HTML version of this guide

data check

An abbreviated Abaqus/Standard, Abaqus/Explicit, or Abaqus/CFD execution that checks only that the model is consistent and that all required model options have been set.

For more information:

- “Abaqus/Standard, Abaqus/Explicit, and Abaqus/CFD execution,” Section 3.2.2 of the Abaqus Analysis User’s Guide
- “Performing a data check on a model,” Section 19.7.3 of the Abaqus/CAE User’s Guide, in the HTML version of this guide
- “Performing a data check on an adaptivity process,” Section 19.9.2 of the Abaqus/CAE User’s Guide, in the HTML version of this guide

data line

A line in an Abaqus/Standard, Abaqus/Explicit, or Abaqus/CFD input file used to provide data that are more easily given in lists than as parameters on an option. If a data line is required, it must immediately follow the keyword line introducing the option.

For more information:

- “Input syntax rules,” Section 1.2.1 of the Abaqus Analysis User’s Guide

datum

A feature of an Abaqus/CAE model that represents auxiliary geometry that can be used for reference when modeling a part or assembly. The following types of datum are available: points, axes, planes, and coordinate systems.

For more information:

- “Understanding the role of datum geometry,” Section 62.1 of the Abaqus/CAE User’s Guide

Datum toolset

An Abaqus/CAE toolset used to create datum points, axes, planes, and coordinate systems with respect to a combination of existing geometry—such as vertices, planes, and edges—and existing datum geometry.

For more information:

- Chapter 62, “The Datum toolset,” of the Abaqus/CAE User’s Guide”

decoration

The Abaqus/CAE viewport title and the viewport border.

For more information:

- “Working with viewports,” Section 4.4 of the Abaqus/CAE User’s Guide, in the HTML version of this guide

deformable part

A part that can deform under load. See also *discrete rigid part*.

For more information:

- "Part types," Section 11.4.2 of the Abaqus/CAE User's Guide

deformation scale factor

The factor that is applied to the deformation field when you display a plot of a deformed model in Abaqus/CAE. You can scale the deformations to magnify, reduce, or otherwise distort the deformed model shape.

For more information:

- "Scaling deformations," Section 55.4.1 of the Abaqus/CAE User's Guide

deformed field output variable

The variable whose values control the shape of the model in a deformed shape plot in Abaqus/CAE. Deformed field output variables can only be vector quantities such as displacement or velocity.

For more information:

- "Selecting the deformed field output variable," Section 42.5.4 of the Abaqus/CAE User's Guide, in the HTML version of this guide

deformed shape plot

An Abaqus/CAE plot that displays the shape of a model at a specified step and frame of the analysis results according to the values of the deformed field output variable that you specify.

For more information:

- Chapter 43, "Plotting the undeformed and deformed shapes," of the Abaqus/CAE User's Guide

degrees of freedom (dof)

The fundamental variables calculated during an analysis: the set of independent displacements and/or rotations that specify completely the displaced or deformed position and orientation of the body or system. For example, in a stress/displacement analysis the degrees of freedom are the translations. For shell and beam elements the degrees of freedom are the rotations at each node.

For more information:

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

Discrete Field toolset

An Abaqus/CAE toolset that allows you to create and manage discrete fields in the Property module, the Interaction module, or the Load module. Discrete fields associate unique values with individual nodes or elements in a meshed model. You can use discrete fields to define spatially varying parameters for selected model attributes, such as a temperature that varies by node over a region in an initial temperature predefined field.

For more information:

- Chapter 63, "The Discrete Field toolset," of the Abaqus/CAE User's Guide

discrete orientation

A definition of a spatially varying orientation for each native or orphan mesh element. A discrete orientation can be based on the topology of the part. You define the *normal axis* and the *primary axis*, and Abaqus/CAE uses these axes to construct a right-handed Cartesian coordinate system. A variety of selection methods are available to define the desired axes. Discrete orientations can be used for material orientations and composite layup orientations.

For more information:

- "Using discrete orientations for material orientations and composite layup orientations," Section 12.16 of the Abaqus/CAE User's Guide, in the HTML version of this guide

discrete rigid part

A part that is assumed to be rigid and is used in contact analyses to model bodies that cannot deform. See also *deformable part*.

For more information:

- "Rigid parts," Section 11.7.1 of the Abaqus/CAE User's Guide

discrete set

An Abaqus/CAE set that is made up of either nodes or elements that you have selected from an orphan mesh.

For more information:

- Chapter 73, "The Set and Surface toolsets," of the Abaqus/CAE User's Guide

display group

A collection of selected model components in Abaqus/CAE that can contain any combination of part instances, geometry (cells, faces, or edges), nodes, elements, and surfaces or the default (entire) model. A display group allows you to reduce clutter on your screen and to focus on an area of interest within your model.

For more information:

- Chapter 78, "Using display groups to display subsets of your model," of the Abaqus/CAE User's Guide

display list

A feature in Abaqus/CAE that helps you display repeated images faster. When an object is displayed repeatedly (for example, in an animation), the system must perform many computations to render each animation frame. If you enable the display list option, the results of these computations are stored in a display list the first time you display the animation. The next time you display the animation, Abaqus/CAE refers to the display list instead of performing the calculations again; as a result, the animation is faster.

For more information:

- "Using display lists," Section 7.2 of the Abaqus/CAE User's Guide, in the HTML version of this guide

DMP (distributed memory parallel)

A mode of parallel execution where each processor has its own memory. Abaqus supports DMP using specific MPI libraries and interconnects. See also *MPI*.

double buffering

A graphics rendering technique used in Abaqus/CAE to prevent screen flicker when the viewport is refreshed.

For more information:

- "GraphicsOptions object," Section 17.9 of the Abaqus Scripting Reference Guide

double precision

The storage and manipulation of floating point numbers in 8 B, as opposed to single precision which uses a standard 4 B. Abaqus/Explicit performs calculations using double precision by default.

For more information:

- "Abaqus/Standard, Abaqus/Explicit, and Abaqus/CFD execution," Section 3.2.2 of the Abaqus Analysis User's Guide
- "Using the Abaqus environment settings," Section 3.3.1 of the Abaqus Analysis User's Guide

drag mode

A setting in Abaqus/CAE that allows you to display an image as a simple wireframe during mouse manipulations such as panning, zooming, and rotating; this mode allows faster manipulation of very large models in the shaded render style instead of the current render style (wireframe, filled, hidden line, or shaded).

For more information:

- "Controlling drag mode," Section 7.6 of the Abaqus/CAE User's Guide, in the HTML version of this guide

drawing area

The visible portion of the Abaqus/CAE *canvas*.

For more information:

- "Components of the main window," Section 2.2.1 of the Abaqus/CAE User's Guide

E

eddy currents

Electric currents generated in a conductor that is placed in a time-varying magnetic field. Abaqus/Standard provides electromagnetic capabilities that can solve time-harmonic and transient eddy current problems.

For more information:

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

edge parameter

A position along an edge, expressed as a fraction of its length. An edge parameter is used to partition an edge and to position a datum along an edge. Abaqus/CAE displays an arrow along the edge indicating the direction of increasing parameter value from the start vertex (corresponding to an edge parameter value of zero) to the end vertex (corresponding to a value of one).

For more information:

- "Creating a datum point by entering an edge parameter," Section 62.6.5 of the Abaqus/CAE User's Guide, in the HTML version of this guide
- "Using the enter parameter method to partition edges," Section 70.5.2 of the Abaqus/CAE User's Guide, in the HTML version of this guide

element set

A named collection of elements.

For more information:

- "Element definition," Section 2.2.1 of the Abaqus Analysis User's Guide
- Chapter 73, "The Set and Surface toolsets," of the Abaqus/CAE User's Guide

Elysium Neutral file

A file format used to import geometry from the following CAD applications using the appropriate Elysium translator plug-in:

- NX using the Abaqus/CAE Associative Interface for NX.
- Pro/ENGINEER using the Pro/ENGINEER Associative Interface.

You can import parts in Elysium Neutral File format; however, you cannot export parts in Elysium Neutral File format.

For more information:

- "What kinds of files can be imported and exported from Abaqus/CAE?," Section 10.1.1 of the Abaqus/CAE User's Guide
- "Importing parts," Section 10.7.2 of the Abaqus/CAE User's Guide, in the HTML version of this guide

encastre

A fixed displacement and rotation boundary condition.

encrypt

To convert an Abaqus data file into an encoded, password-protected format that only authorized parties can access. The **abaqus encrypt** utility is intended for the encryption of data that you include by reference in input (`.inp`) files or other data files.

For more information:

- "Encrypting and decrypting Abaqus input data," Section 3.2.42 of the Abaqus Analysis User's Guide

EPS (Encapsulated PostScript)

A variation of PostScript that describes a single graphic designed to be included in a larger document without modification. Abaqus/CAE allows you to save images of selected viewports in EPS-format files.

For more information:

- "Printed image formats," Section 8.1.1 of the Abaqus/CAE User's Guide

equation solver

The solution procedure for a linear system of equations in Abaqus/Standard. Available linear equation system solution techniques include the direct linear equation solver, the iterative linear equation solver, the subspace iteration eigenvalue solver, the Lanczos eigenvalue solver, and the AMS eigenvalue solver.

For more information:

- "Direct linear equation solver," Section 6.1.5 of the Abaqus Analysis User's Guide
- "Iterative linear equation solver," Section 6.1.6 of the Abaqus Analysis User's Guide
- "Selecting the eigenvalue extraction method" in "Eigenvalue buckling prediction," Section 6.2.3 of the Abaqus Analysis User's Guide
- "Dynamic analysis procedures: overview," Section 6.3.1 of the Abaqus Analysis User's Guide
- "Natural frequency extraction," Section 6.3.5 of the Abaqus Analysis User's Guide

error indicator

A metric used during adaptive meshing to measure error in the base solution. An error indicator determines where a mesh needs refinement to achieve an error goal.

For more information:

- "Selection of error indicators influencing adaptive remeshing," Section 12.3.2 of the Abaqus Analysis User's Guide

error indicator output variable

An output variable selected to use in the calculation of the error indicator. Error indicator output variables have an "ERI" suffix; for example, MISESERI.

For more information:

- "Selection of error indicators influencing adaptive remeshing," Section 12.3.2 of the Abaqus Analysis User's Guide

error target

A user-defined remeshing goal described as a percentage target for a normalized form of the error indicator output variable. The normalized form of the error indicator for a particular variable is the ratio of the value of the error indicator to the value of the base solution.

For more information:

- "Selection of error indicators influencing adaptive remeshing," Section 12.3.2 of the Abaqus Analysis User's Guide

Eulerian analysis

A finite element technique in which materials can flow through the elements in a model's mesh. Abaqus offers two implementations of the Eulerian technique: pure Eulerian analysis and Arbitrary Lagrangian-Eulerian (ALE) adaptive meshing. In a pure Eulerian analysis the mesh is rigid and the model behavior is defined by the flow of material through the mesh. ALE adaptive meshing is a technique that combines the features of Lagrangian analysis and Eulerian analysis within the same mesh to maintain a high mesh quality during analyses involving large deformations.

For more information:

- "ALE adaptive meshing: overview," Section 12.2.1 of the Abaqus Analysis User's Guide
- "Eulerian analysis," Section 14.1.1 of the Abaqus Analysis User's Guide
- Chapter 28, "Eulerian analyses," of the Abaqus/CAE User's Guide

Eulerian-Lagrangian analysis

A finite element technique that involves the interaction between pure Eulerian part instances and pure Lagrangian part instances within the same model. Contact can be defined between the elements in the Lagrangian part and the material boundaries in the Eulerian part.

For more information:

- “Eulerian analysis,” Section 14.1.1 of the Abaqus Analysis User’s Guide
- Chapter 28, “Eulerian analyses,” of the Abaqus/CAE User’s Guide

Eulerian part

A part that acts as a domain in which materials can flow in an Eulerian analysis. Any arbitrarily shaped three-dimensional part that you can create or import in Abaqus/CAE can be specified as an Eulerian part.

For more information:

- “Part types,” Section 11.4.2 of the Abaqus/CAE User’s Guide

F

facet

An individual element face or edge.

fastener

A point-to-point connection between two or more surfaces such as a spot weld or rivet connection. When you model fasteners, the attachment to each of the surfaces being connected is distributed to several nodes to be connected in the neighborhood of the *fastening point*.

For more information:

- “Mesh-independent fasteners,” Section 35.3.4 of the Abaqus Analysis User’s Guide
- “About fasteners,” Section 29.1 of the Abaqus/CAE User’s Guide

fastening point

The actual point where a fastener layer attaches to the surfaces that are being connected with fasteners. The location is determined by considering the *positioning point* location, projection method, and surfaces to be fastened.

For more information:

- “Mesh-independent fasteners,” Section 35.3.4 of the Abaqus Analysis User’s Guide

features

The basic definitions that combine to make up an Abaqus/CAE native part and assembly, such as geometry operations and positioning constraints. Each feature contains parameters, such as size, location, and depth. Abaqus/CAE retains the parameters that define each feature and uses this information to regenerate a part or assembly if a feature is modified.

For more information:

- “What is feature-based modeling?,” Section 11.3 of the Abaqus/CAE User’s Guide

feature angle

An angle formed between the normals of the two facets connected to an edge.

For more information:

- "Assigning surface properties for general contact in Abaqus/Explicit," Section 36.4.2 of the Abaqus Analysis User's Guide
- "Surface properties for general contact in Abaqus/Standard," Section 36.2.2 of the Abaqus Analysis User's Guide
- "Defining model feature edges," Section 55.3.2 of the Abaqus/CAE User's Guide
- "Defining mesh feature edges," Section 76.5 of the Abaqus/CAE User's Guide

Feature Manipulation toolset

An Abaqus/CAE toolset that contains tools that you use to create and manage features.

For more information:

- Chapter 65, "The Feature Manipulation toolset," of the Abaqus/CAE User's Guide

field

A definition of input or output data representing the value of a variable such as displacement, temperature, or pressure over a domain.

field output

The output of variables that are written relatively infrequently to the output database. Typically, you request field output from your entire model or a large region of your model; Abaqus/Standard and Abaqus/Explicit write every component of the variable to the output database at the selected frequency. In the Abaqus/CAE Visualization module you can view field output in the form of a deformed, contour, or symbol plot and you can produce a report of field output.

For more information:

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Guide

fillet

A circular arc that joins two lines in a continuous manner. When you are using the Sketch module, you can create a fillet between two lines meeting at an angle. When you are creating or modifying a three-dimensional solid part, you can fillet, or round, selected edges.

For more information:

- "Sketching fillets between two lines," Section 20.10.9 of the Abaqus/CAE User's Guide, in the HTML version of this guide
- "Blending edges," Section 11.27 of the Abaqus/CAE User's Guide, in the HTML version of this guide

fixed-interface substructure dynamic modes

Substructure dynamic modes in which all retained degrees of freedom have been constrained.

For more information:

- "Defining substructures," Section 10.1.2 of the Abaqus Analysis User's Guide

fixed part instance

The *part instance* whose position remains fixed during the application of an assembly constraint in Abaqus/CAE.

For more information:

- "How the position constraint methods differ," Section 13.5.2 of the Abaqus/CAE User's Guide

frame

In a structure, a mechanical skeleton structure.

In a finite element model, a frame element provides efficient modeling for design calculations of frame-like structures composed of initially straight, slender members.

For more information:

- "Frame elements," Section 29.4.1 of the Abaqus Analysis User's Guide

In history output data, a container in the output database structure that corresponds to a snapshot in the history of the simulation. In addition, the term applies to a single plot in an animating series.

For more information:

- "Generating output database reports," Section 3.2.22 of the Abaqus Analysis User's Guide
- "Selecting the results step and frame," Section 42.3 of the Abaqus/CAE User's Guide
- Chapter 49, "Animating plots," of the Abaqus/CAE User's Guide

free body cross-section

The set of nodes and elements that make up the surface across which you want Abaqus/CAE to display the resultant forces and moments.

For more information:

- "Resultant forces and moments on free body cuts in Abaqus/CAE," Section 67.1 of the Abaqus/CAE User's Guide
- "Creating or editing a free body cut," Section 67.2 of the Abaqus/CAE User's Guide, in the HTML version of this guide

free body cut

In the Visualization module of Abaqus/CAE, a tool that displays the resultant forces and moments transmitted across a selected surface of your model.

For more information:

- “Resultant forces and moments on free body cuts in Abaqus/CAE,” Section 67.1 of the Abaqus/CAE User’s Guide
- “Creating or editing a free body cut,” Section 67.2 of the Abaqus/CAE User’s Guide, in the HTML version of this guide

free meshing

A meshing technique that uses no preestablished mesh patterns and allows more flexibility than structured meshing. When you mesh a region using the structured meshing technique, you can predict the pattern of the mesh based on the region topology. In contrast, it is impossible to predict a free mesh pattern before creating the mesh.

For more information:

- “Free meshing,” Section 17.10 of the Abaqus/CAE User’s Guide

free-interface substructure dynamic modes

Substructure dynamic modes in which none of the retained degrees of freedom have been constrained.

For more information:

- “Defining substructures,” Section 10.1.2 of the Abaqus Analysis User’s Guide

free rotation handle

The point at the top of the *3D compass* that allows you to rotate the view of a model in any direction.

For more information:

- “Rotating the view using the 3D compass,” Section 5.3.1 of the Abaqus/CAE User’s Guide

frustum

The three-dimensional space visible in a viewport in movie camera mode; it is a truncated pyramid with its apex at the camera position. The frustum begins at the near plane (a plane parallel to the pyramid base but closer to the camera) and extends to the far plane (the pyramid base). The near plane and far plane positions are determined by the view options in Abaqus/CAE.

For more information:

- “Understanding camera modes and view options,” Section 5.1 of the Abaqus/CAE User’s Guide

G

general contact

A simple interface that can be used to enforce contact between all exterior surfaces in a model with a single interaction definition; also known as “automatic” contact.

For more information:

- “Defining general contact in Abaqus/Standard,” Section 36.2 of the Abaqus Analysis User’s Guide
- “Defining general contact in Abaqus/Explicit,” Section 36.4 of the Abaqus Analysis User’s Guide
- “Understanding interactions,” Section 15.3 of the Abaqus/CAE User’s Guide

general release

The first general availability release of Abaqus that introduces enhancements and new functionality. See also *maintenance delivery*.

geometric nonlinearity (NLGEOM)

Nonlinear stiffness variations caused by large deformations or rotations.

For more information:

- “General and linear perturbation procedures,” Section 6.1.3 of the Abaqus Analysis User’s Guide
- “Linear and nonlinear procedures,” Section 14.3.2 of the Abaqus/CAE User’s Guide

geometry set

Geometric objects—such as cells, faces, edges, and vertices—that you select in Abaqus/CAE from an unmeshed part or assembly. The geometric objects can be of different types; for example, you can include a face and an edge in the same set.

For more information:

- Chapter 73, “The Set and Surface toolsets,” of the Abaqus/CAE User’s Guide

geometry surface

A surface you create in Abaqus/CAE by selecting faces or edges from native or imported geometry in an assembly.

For more information:

- Chapter 73, “The Set and Surface toolsets,” of the Abaqus/CAE User’s Guide

grid coordinates

The cursor coordinates that appear in the upper-left corner of the Abaqus/CAE Sketcher as you create new sketch geometry. They are tied to the current origin and rotation of the sketch grid.

For more information:

- “Realigning the sketch grid relative to the sketch,” Section 20.4.4 of the Abaqus/CAE User’s Guide

H

heal

A process performed during the import of a part into Abaqus/CAE to improve the accuracy of the part's geometry.

For more information:

- "Importing parts," Section 10.7.2 of the Abaqus/CAE User's Guide, in the HTML version of this guide

history

The activity of a step-dependent object through the course of an analysis. In Abaqus/CAE you can view the histories of step-dependent objects by displaying the appropriate step-dependent manager.

For more information:

- "Defining a model in Abaqus," Section 1.3.1 of the Abaqus Analysis User's Guide
- "What are step-dependent managers?," Section 3.4.2 of the Abaqus/CAE User's Guide

history data

The portion of an Abaqus input file that defines what happens to the model—the sequence of events or loadings for which the model's response is sought. History data are combined with model data in the input file that defines a model.

For more information:

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

history output

The output of variables that are written relatively frequently to the output database—as often as every increment. You can use history output in the Visualization module of Abaqus/CAE to generate *X*–*Y* plots.

For more information:

- "Output," Section 4.1.1 of the Abaqus Analysis User's Guide

homogeneous section

Solid and shell sections that define the section properties of solid and shell elements and refer to a single material.

For more information:

- "Using a shell section integrated during the analysis to define the section behavior," Section 29.6.5 of the Abaqus Analysis User's Guide

- “Using a general shell section to define the section behavior,” Section 29.6.6 of the Abaqus Analysis User’s Guide
- “Solid (continuum) elements,” Section 28.1.1 of the Abaqus Analysis User’s Guide
- “Defining sections,” Section 12.2.3 of the Abaqus/CAE User’s Guide

I

IGES (Initial Graphics Exchange Specification) file

A neutral data format designed for graphics exchange between computer-aided design (CAD) systems. Using Abaqus/CAE, you can import IGES-format parts, and you can export parts in IGES format. In addition, you can import and export a sketch from an IGES file.

For more information:

- Chapter 10, “Importing and exporting geometry data and models,” of the Abaqus/CAE User’s Guide

increment

An interval of an analysis step.

For more information:

- “Defining an analysis,” Section 6.1.2 of the Abaqus Analysis User’s Guide
- “Specifying model attributes,” Section 9.8.4 of the Abaqus/CAE User’s Guide, in the HTML version of this guide

initial condition

A condition you prescribe to define the initial value of a solution, state, or field variable such as stress or temperature. You can define initial conditions by creating predefined fields in the Load module.

For more information:

- “Initial conditions in Abaqus/Standard and Abaqus/Explicit,” Section 34.2.1 of the Abaqus Analysis User’s Guide
- “Initial conditions in Abaqus/CFD,” Section 34.2.2 of the Abaqus Analysis User’s Guide
- “Creating and modifying prescribed conditions,” Section 16.4 of the Abaqus/CAE User’s Guide

input file

An ASCII file that is read and processed by Abaqus/Standard, Abaqus/Explicit, or Abaqus/CFD and contains keywords and data that define your model. When you submit a job for analysis using Abaqus/CAE, it generates an input file based on the model you have defined. If necessary, you can modify the input file generated by Abaqus/CAE using the Keywords Editor. In addition, you can import Abaqus/Standard or Abaqus/Explicit input files into Abaqus/CAE; Abaqus/CAE translates the options and data lines in the imported input file into a new Abaqus/CAE model.

For more information:

- "Defining a model in Abaqus," Section 1.3.1 of the Abaqus Analysis User's Guide
- "Understanding the files generated by creating and analyzing a model," Section 9.4 of the Abaqus/CAE User's Guide
- "Importing a model from an Abaqus input file," Section 10.5.2 of the Abaqus/CAE User's Guide

input parameter

A capability in Abaqus/Design that allows you to create an input file in which parameters are used in place of input quantities. The parameters are evaluated according to their definition and are substituted for the parametrized quantities before an analysis is performed by Abaqus/Standard or Abaqus/Explicit. See also *keyword parameter* to learn how keyword parameters are used in an Abaqus input file.

For more information:

- "Scripting parametric studies," Section 20.1.1 of the Abaqus Analysis User's Guide

instance

See *part instance*.

interaction

Step-dependent objects that define how bodies or regions in a model act with each other or with their surrounding environment; examples include contact, kinematic relationships associated with mechanical connectors, and thermal constraints such as convection or radiation boundary conditions. Interactions also establish dependencies between fields such as coupled thermomechanical, coupled pore fluid-mechanical, and coupled thermal-electrical effects. In Abaqus/CAE an elastic foundation is also considered a form of interaction.

For more information:

- Part IX, "Interactions," of the Abaqus Analysis User's Guide
- Chapter 15, "The Interaction module," of the Abaqus/CAE User's Guide

Interaction module

An Abaqus/CAE module used to define *interactions* between regions of a model or between a region of a model and its surroundings.

For more information:

- Chapter 15, "The Interaction module," of the Abaqus/CAE User's Guide

interaction property

A collection of data that are necessary to define certain types of interactions completely.

For more information:

- “Contact interaction analysis: overview,” Section 36.1.1 of the Abaqus Analysis User’s Guide
- Chapter 15, “The Interaction module,” of the Abaqus/CAE User’s Guide

J

job

A process you submit for execution on any computer or network. You submit an Abaqus model for analysis in the form of an analysis job.

For more information:

- “Abaqus/Standard, Abaqus/Explicit, and Abaqus/CFD execution,” Section 3.2.2 of the Abaqus Analysis User’s Guide
- Chapter 19, “The Job module,” of the Abaqus/CAE User’s Guide

Job module

An Abaqus/CAE module used to create a job, submit it for analysis, and monitor its progress.

For more information:

- Chapter 19, “The Job module,” of the Abaqus/CAE User’s Guide

job monitor

The process of observing the status of Abaqus/CAE analysis jobs in the model database using the job monitor dialog box.

For more information:

- “Output,” Section 4.1.1 of the Abaqus Analysis User’s Guide
- “Monitoring the progress of an analysis job,” Section 19.2.6 of the Abaqus/CAE User’s Guide

journal (`.jnl`) file

A file (*model_database_name*`.jnl`) containing commands that Abaqus/CAE can use to replicate the model database if it becomes corrupted. When you save a model database, Abaqus/CAE also saves the journal file automatically.

For more information:

- “Recreating an unsaved model database,” Section 9.5.3 of the Abaqus/CAE User’s Guide

K

kernel

The brains behind Abaqus/CAE that interpret the Python commands generated by the GUI and use the options and settings to create an internal representation of your model.

For more information:

- "Abaqus/CAE and the Abaqus Scripting Interface," Section 2.1 of the Abaqus Scripting User's Guide

kernel matrix

The Jacobian (i.e., stiffness) matrix used in the full Newton technique to solve nonlinear problems. The quasi-Newton technique is an alternative solution approach that can be less expensive than the default full Newton technique. The difference between the two approaches lies in the number of times that the Jacobian is reformed. The quasi-Newton technique reforms this matrix less often than the full Newton method, allowing for the possibility of a faster solution. The user has no control over how often the Jacobian is reformed in the full Newton technique, whereas with the quasi-Newton approach this can be adjusted. The kernel matrix is the mathematical derivation of the quasi-Newton technique, in which the kernel matrix is an approximate representation of the inverse of the full Jacobian.

For more information:

- "Convergence criteria for nonlinear problems," Section 7.2.3 of the Abaqus Analysis User's Guide
- "Configuring general analysis procedures," Section 14.11.1 of the Abaqus/CAE User's Guide, in the HTML version of this guide
- "Quasi-Newton solution technique," Section 2.2.2 of the Abaqus Theory Guide

keyword

An input option in Abaqus/Standard, Abaqus/Explicit, and Abaqus/CFD that indicates the kind of data definitions specified in option blocks. For example, if you want to use a particular material in an analysis, you must add an option block beginning with the keyword *MATERIAL to the input file. Keywords are always preceded by an asterisk and appear in uppercase characters in the Abaqus documentation.

For more information:

- *Abaqus Keywords Reference Guide*
- "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Guide

keyword line

The first line of an option block in an Abaqus input file. Keyword lines begin with a particular keyword, followed, in some cases, by parameters associated with the keyword. For example, the keyword line of an option block describing a material might appear as ∗MATERIAL, NAME=*name*. In this case ∗MATERIAL is the keyword and NAME is a parameter that allows you to specify the name of the material being defined.

For more information:

- “Input syntax rules,” Section 1.2.1 of the Abaqus Analysis User’s Guide

keyword parameter

Words or phrases on a keyword line that are used to define the behavior of an option. Parameters can stand alone or have a value, and they may be required or optional. An example is the keyword parameter TYPE used with the keyword ∗ELEMENT to specify what type of element (such as solid, beam, or shell) is being defined. See also *input parameter* to learn how input parameters are used in Abaqus/Design.

keywords editor

A specialized text editor that allows you to modify an Abaqus input file generated by Abaqus/CAE before submitting it for analysis.

For more information:

- “Adding unsupported keywords to your Abaqus/CAE model,” Section 9.10.1 of the Abaqus/CAE User’s Guide, in the HTML version of this guide

L

Lagrangian analysis

A finite element technique in which materials are fixed within elements and the materials deform as the element mesh deforms.

For more information:

- “Eulerian analysis,” Section 14.1.1 of the Abaqus Analysis User’s Guide
- Chapter 28, “Eulerian analyses,” of the Abaqus/CAE User’s Guide

LEFM (linear elastic fracture mechanics)

A crack propagation technique that assumes the material is elastic or that the material is subjected to small-scale yielding. The technique is appropriate for brittle fracture analysis.

For more information:

- “Modeling moving cracks based on the principles of linear elastic fracture mechanics (LEFM) and phantom nodes” in “Modeling discontinuities as an enriched feature using the extended finite element method,” Section 10.7.1 of the Abaqus Analysis User’s Guide

load

Any of the forces; moments; concentrated or distributed tractions; fluxes; or, more generally, influences—including predefined fields imposed upon a structure or body—that cause deformation, displacement, or change the state of a structure from its unloaded state.

For more information:

- “Applying loads: overview,” Section 34.4.1 of the Abaqus Analysis User’s Guide
- Chapter 16, “The Load module,” of the Abaqus/CAE User’s Guide

load case

A set of loads, boundary conditions, and base motions used to define a particular loading condition within a step in a model. Multiple load cases can be defined in a single step.

For more information:

- “Multiple load case analysis,” Section 6.1.4 of the Abaqus Analysis User’s Guide
- Chapter 34, “Load cases,” of the Abaqus/CAE User’s Guide

Load module

An Abaqus/CAE module used to define prescribed conditions such as loads, boundary conditions, and predefined fields.

For more information:

- Chapter 16, “The Load module,” of the Abaqus/CAE User’s Guide

M

main window

The GUI with which you interact with Abaqus/CAE. The main window contains a menu bar, prompt area, toolbars, and a variety of other components that allow you to perform the tasks necessary for creating and analyzing a model and viewing analysis results. Certain aspects of the main window, such as the menu bar and the toolbars, can change as you work through the modeling process.

For more information:

- “Overview of the main window,” Section 2.2 of the Abaqus/CAE User’s Guide

maintenance delivery

A software update following a general release that addresses issues identified in status reports but does not include enhancements or new functionality. See also *general release*.

manager

See *basic manager* and *step-dependent manager*.

material orientation triad

An Abaqus/CAE viewport annotation that indicates the material directions of elements in your model at the element integration points in the Visualization module.

For more information:

- “Customizing material orientation plot triads,” Section 46.4.1 of the Abaqus/CAE User’s Guide, in the HTML version of this guide

membrane sections

Thin surfaces in space that offer strength in the plane of the surface but have no bending stiffness.

For more information:

- “Membrane elements,” Section 29.1 of the Abaqus Analysis User’s Guide
- “Defining sections,” Section 12.2.3 of the Abaqus/CAE User’s Guide

mesh

An arrangement of finite elements defined on an FEA model. You can define a mesh on a part or on the assembly.

Meshing is the activity of discretizing geometry into a finite element representation.

For more information:

- “Element definition,” Section 2.2.1 of the Abaqus Analysis User’s Guide
- Chapter 17, “The Mesh module,” of the Abaqus/CAE User’s Guide

mesh-geometry association

Abaqus/CAE techniques used to associate a mesh with geometry for the proper transfer of loads and boundary conditions from geometric entities. Top-down meshing techniques automatically associate the mesh with the geometry that was used to create it. If you use bottom-up meshing techniques or edit the association of a top-down mesh, you may need to check the association.

For more information:

- “Mesh-geometry association,” Section 17.12 of the Abaqus/CAE User’s Guide

Mesh module

An Abaqus/CAE module that contains tools used to generate meshes on parts and assemblies created in or imported into Abaqus/CAE. In addition, the Mesh module contains query functions that provide information about existing meshes.

For more information:

- Chapter 17, "The Mesh module," of the Abaqus/CAE User's Guide

mesh surface

A named collection of element faces or edges selected from either native or orphan meshes in an assembly that can be used to request output or add loads to specific areas of the model.

For more information:

- Chapter 73, "The Set and Surface toolsets," of the Abaqus/CAE User's Guide

message area

A section of the main window in Abaqus/CAE used to display information and warnings.

For more information:

- "Overview of the main window," Section 2.2 of the Abaqus/CAE User's Guide

message (`.msg`) file

A file (*job_name*`.msg`) that contains diagnostic or informative messages about the progress of an Abaqus/Standard or Abaqus/Explicit analysis.

For more information:

- "Output," Section 4.1.1 of the Abaqus Analysis User's Guide
- "Understanding the files generated by creating and analyzing a model," Section 9.4 of the Abaqus/CAE User's Guide
- "Diagnostic printing," Section 14.5.3 of the Abaqus/CAE User's Guide

midsurface model

A shell model with thickness and offset definitions, used in place of a solid model in an analysis. The midsurface model is intended to provide a simplified model that reduces the expense of an analysis without compromising the results.

For more information:

- Chapter 35, "Midsurface modeling," of the Abaqus/CAE User's Guide

mixed-interface substructure dynamic modes

Substructure dynamic modes in which only some of the retained degrees of freedom have been constrained, while others have not been constrained.

For more information:

- "Defining substructures," Section 10.1.2 of the Abaqus Analysis User's Guide

model

A collection of the data that are needed to conduct an analysis; also refers to the physical object being analyzed.

For more information:

- "Defining a model in Abaqus," Section 1.3.1 of the Abaqus Analysis User's Guide
- "What is an Abaqus/CAE model?," Section 9.2 of the Abaqus/CAE User's Guide

model data

A portion of an Abaqus/Standard, Abaqus/Explicit, or Abaqus/CFD input file that defines a finite element model: the elements, nodes, element properties, material definitions, and so on—any data that specify the model itself. Model data include all data in an Abaqus input file that appear before the *STEP option.

For more information:

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

model database

A database used by Abaqus/CAE to store your models and analysis jobs. While you may have multiple model databases stored on your computer or network, Abaqus/CAE can work on only one at any time. The model database in use is known as the current model database, and Abaqus/CAE displays its name across the top of the main window.

For more information:

- "What is an Abaqus/CAE model database?," Section 9.1 of the Abaqus/CAE User's Guide

model definition

The internal Abaqus representation of the model.

For more information:

- "Defining a model in Abaqus," Section 1.3.1 of the Abaqus Analysis User's Guide
- "Controlling the input file generated by Abaqus/CAE," Section 9.10 of the Abaqus/CAE User's Guide, in the HTML version of this guide

modeling space

The space a part inhabits. A part can inhabit three-dimensional, two-dimensional, or axisymmetric modeling space.

For more information:

- "Part modeling space," Section 11.4.1 of the Abaqus/CAE User's Guide

module

A functional unit of Abaqus/CAE. Each module defines a logical aspect of the modeling process, such as defining the geometry, defining material properties, and generating a mesh, and contains only those tools that are relevant to a specific portion of the modeling task.

For more information:

- "What is a module?," Section 2.3 of the Abaqus/CAE User's Guide

monitor

See *job monitor*.

movable part instance

The part instance whose position can change during the application of an assembly constraint in Abaqus/CAE.

For more information:

- "How the position constraint methods differ," Section 13.5.2 of the Abaqus/CAE User's Guide

MPI (message passing interface)

A software library defined by the MPI standard that is used to parallelize software. MPI is typically used to enable execution on compute clusters or networks of workstations but provides effective parallelization on SMP (shared memory parallel) machines as well. See also *MPI-based parallel* and *SMP*.

For more information:

- MPI Forum (www.mpi-forum.org)

MPI-based parallel

DMP processing supported by Abaqus on multiprocessor workstations or compute clusters using MPI libraries.

For more information:

- "Parallel execution," Section 3.5 of the Abaqus Analysis User's Guide

multiphysics

A coupled approach in the numerical solution of multiple interacting physical domains. Abaqus provides built-in fully coupled procedures, sequential coupling, and co-simulation as solution techniques for multiphysics simulation. See the Co-simulation page at www.3ds.com/simulia.

For more information:

- "Multiphysics analyses" in "Solving analysis problems: overview," Section 6.1.1 of the Abaqus Analysis User's Guide

multi-point constraint

A constraint that allows you to constrain the degrees of freedom of the slave nodes of a region to the degrees of freedom of a single point or multiple points.

For more information:

- "General multi-point constraints," Section 35.2.2 of the Abaqus Analysis User's Guide

N

native mesh

The meshed representation of a part instance. You create a native mesh by discretizing geometry using the Abaqus/CAE Mesh module. You use the Mesh module to mesh native parts that you positioned in the assembly. A native mesh and its attributes are feature based, and a native mesh maintains its association with the original parts and with the assembly.

For more information:

- "What is feature-based modeling?," Section 11.3 of the Abaqus/CAE User's Guide

native part

A part created using the Abaqus/CAE Part module. Abaqus/CAE stores each native part in the form of an ordered list of features. The parameters that define each feature—extruded depth, hole diameter, sweep path, etc.—define the geometry of the part.

For more information:

- "What is feature-based modeling?," Section 11.3 of the Abaqus/CAE User's Guide

NLGEOM

An abbreviation for *geometric nonlinearity*.

node set

A named collection of nodes.

For more information:

- "Node definition," Section 2.1.1 of the Abaqus Analysis User's Guide
- Chapter 73, "The Set and Surface toolsets," of the Abaqus/CAE User's Guide

non-manifold edge

An exterior element edge at which more than two exterior element faces meet. You can use the Query toolset in the Abaqus/CAE Mesh module to highlight free and non-manifold mesh edges in a solid or shell mesh. Non-manifold edges may indicate problems, especially in a solid mesh.

For more information:

- "Obtaining mesh information," Section 17.19.2 of the Abaqus/CAE User's Guide, in the HTML version of this guide

normal axis

A spatially varying orientation at the centroid of each native or orphan mesh element. The normal axis for a *discrete orientation* represents the normal axis of the material orientation. Abaqus/CAE uses the normal axis and the *primary axis* to construct a right-handed Cartesian coordinate system. You choose the coordinate system axis that you want the normal axis to represent in the resultant orientation and select topology or datums or enter vector values to define the desired axis.

For more information:

- "Using discrete orientations for material orientations and composite layup orientations," Section 12.16 of the Abaqus/CAE User's Guide, in the HTML version of this guide

O

online documentation

The Abaqus HTML documentation, which contains all the available content including context-sensitive help pages and is the primary format.

For more information:

- Part II, "Using the Abaqus HTML Documentation," of Using Abaqus Online Documentation

option

See *keyword*. Also refers to choices presented to you by the Abaqus/CAE graphical user interface.

For more information:

- "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Guide

option block

A group of keywords that must be used in conjunction with other keywords. An option block contains data concerning a particular option (or keyword) that describes part of the problem definition. An option block begins with a keyword line and is often followed by one or more data lines. You choose those options that are relevant for a particular application. Parameters available on the command line are also referred to as options.

For more information:

- "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Guide
- *Abaqus Keywords Reference Guide*

orphan mesh

A collection of nodes, elements, surfaces, and sets with no associated geometry. In effect, the mesh information has been orphaned from its parent geometry. You can import a part into Abaqus/CAE from an output database or from an input file in the form of an orphan mesh. You can also create an orphan mesh part from the meshed assembly in the Mesh module. An orphan mesh part appears in the model's list of parts; however, you cannot add geometric features to it. You can use the Edit Mesh toolset in the Part module to edit the mesh definition, and you can change the element type assigned to the mesh in the Mesh module.

For more information:

- "What can I do with the Edit Mesh toolset?," Section 64.1 of the Abaqus/CAE User's Guide

output database (`.odb`)

A file (*job_name*`.odb`) that contain the results from your analysis. You use the Visualization module to open an output database and to view a graphical representation of the contents. In addition, you can import a part from an output database in the form of an orphan mesh.

For more information:

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Guide
- "Opening a model database or an output database," Section 9.7.2 of the Abaqus/CAE User's Guide, in the HTML version of this guide
- "Understanding the files generated by creating and analyzing a model," Section 9.4 of the Abaqus/CAE User's Guide

output request

An instruction for Abaqus to write data of interest to various output files, such as the data (`.dat`) file, the output database (`.odb`) file, and the restart (`.res`) file. The variables that Abaqus writes during a step, the rate at which they are written, the region of the model associated with the output, and the section points of interest define an output request.

For more information:

- "Output," Section 4.1.1 of the Abaqus Analysis User's Guide
- "Understanding output requests," Section 14.4 of the Abaqus/CAE User's Guide

P

parameter

A variable quantity that restricts or gives particular form to the thing that it characterizes. In Abaqus/CAE parameter refers to modifiable parameters that define features (for example, the length of an extrusion). It is also used in the term *edge parameter* to describe a position along an edge, expressed as a fraction of its length. See also *keyword parameter* to learn how parameters are used in Abaqus input files, and see *input parameter* to learn how they are used in Abaqus/Design.

For more information:

- “Input syntax rules,” Section 1.2.1 of the Abaqus Analysis User’s Guide
- Chapter 65, “The Feature Manipulation toolset,” of the Abaqus/CAE User’s Guide

Parasolid

A solid modeling system developed by Unigraphics. You can import Parasolid-format parts generated by Parasolid Version 7 through Version 13. You cannot export parts in Parasolid format.

For more information:

- “What kinds of files can be imported and exported from Abaqus/CAE?,” Section 10.1.1 of the Abaqus/CAE User’s Guide
- “Importing parts,” Section 10.7.2 of the Abaqus/CAE User’s Guide, in the HTML version of this guide

parent feature

See *child feature*.

part

The geometry building blocks of an Abaqus/CAE model. You assemble instances of parts to create an assembly that you can then mesh and analyze.

For more information:

- “Defining an assembly,” Section 2.10.1 of the Abaqus Analysis User’s Guide
- “Understanding the role of the Part module,” Section 11.1 of the Abaqus/CAE User’s Guide

part instance

A reusable copy of the original Abaqus/CAE part, except that an instance also maintains its association with the original part; if you modify the part, the part instance is also modified at the same time. When you assemble your model, you work with part instances, not with the original parts.

For more information:

- “Defining an assembly,” Section 2.10.1 of the Abaqus Analysis User’s Guide
- “Working with part instances,” Section 13.3 of the Abaqus/CAE User’s Guide

partition

A feature used to divide an Abaqus/CAE part or assembly into regions. Such regions have many uses; for example, applying loads or assigning mesh attributes.

For more information:

- Chapter 70, “The Partition toolset,” of the Abaqus/CAE User’s Guide

Partition toolset

An Abaqus/CAE toolset that allows you to divide a part or assembly into regions.

For more information:

- Chapter 70, “The Partition toolset,” of the Abaqus/CAE User’s Guide

Part module

An Abaqus/CAE module used to create, edit, and manage the parts in the current model.

For more information:

- Chapter 11, “The Part module,” of the Abaqus/CAE User’s Guide

part-related modules

Abaqus/CAE modules in which parts are displayed in the viewport. The Part and Property modules are considered part-related modules.

For more information:

- “What is a module?,” Section 2.3 of the Abaqus/CAE User’s Guide

part set

An Abaqus/CAE set consisting of a region of a part. Part sets are available only when you are in the Part or Property modules.

For more information:

- “How do part sets and assembly sets differ?,” Section 73.2.2 of the Abaqus/CAE User’s Guide

path

A line you define by specifying a series of points through your model. You can view results along the path in the form of an *X–Y* plot.

For more information:

- Chapter 48, “Viewing results along a path,” of the Abaqus/CAE User’s Guide

plot state

A combination of all the active customization options that produce a plot in the viewport. You enter a particular plot state by producing a plot of the corresponding type. For example, if you produce an undeformed plot, the current viewport will then be in the undeformed plot state. The plot state of a viewport persists, and Abaqus/CAE updates it with any changes you make to the customization options until you produce a plot in some other state in that viewport.

For more information:

- “What is a plot state?,” Section 40.3.1 of the Abaqus/CAE User’s Guide

plug-in

A piece of software that installs itself into another application to extend the capabilities of that application.

For more information:

- “What is a plug-in?,” Section 81.1 of the Abaqus/CAE User’s Guide
- Chapter 82, “Abaqus plug-ins,” of the Abaqus/CAE User’s Guide, in the HTML version of this guide

PNG (Portable Network Graphics)

An industry standard for storing bitmap images. A PNG file consists of color information and a compressed bitmap representation of the image. Abaqus/CAE allows you to save images of selected viewports in PNG-format files.

For more information:

- “Printed image formats,” Section 8.1.1 of the Abaqus/CAE User’s Guide

position constraint

A constraint used to prescribe the relative positions of part instances during assembly in Abaqus/CAE.

For more information:

- “How the position constraint methods differ,” Section 13.5.2 of the Abaqus/CAE User’s Guide

positioning point

A point that locates the first *fastening point* of mesh-independent fasteners. A positioning point can reside on the first surface to be fastened, or it can reside close to the first surface and be projected along a normal or a user-specified direction.

For more information:

- "Specifying the positioning points, projection method, and fastening points" in "Mesh-independent fasteners," Section 35.3.4 of the Abaqus Analysis User's Guide

PostScript

A page-description language developed by Adobe Systems that offers a flexible font capability and high-quality graphics. PostScript uses English-like commands to control page layout and to load and scale outline fonts. Abaqus/CAE allows you to print images of selected viewports directly to a PostScript printer or to save the same image in a PostScript-format file.

For more information:

- "Printed image formats," Section 8.1.1 of the Abaqus/CAE User's Guide

predefined field

A time-dependent, non-solution-dependent condition that exists over the spatial domain of the model.

For more information:

- "Predefined fields," Section 34.6.1 of the Abaqus Analysis User's Guide
- "Creating and modifying prescribed conditions," Section 16.4 of the Abaqus/CAE User's Guide

prescribed condition

An external condition such as a load, an initial condition, or a boundary condition applied to a model.

For more information:

- "Prescribed conditions: overview," Section 34.1.1 of the Abaqus Analysis User's Guide
- Chapter 16, "The Load module," of the Abaqus/CAE User's Guide

preselection

A change in the appearance of objects on the screen to help you make the desired selections for a particular Abaqus/CAE procedure. Two types of preselection are available:

- Preselection highlighting appears in an Abaqus/CAE viewport when you stop moving the cursor. Any objects that would be selected at the current cursor position are highlighted in orange.
- Preselection symbols appear on an Abaqus/CAE sketch as you move the cursor around to select a point, such as the center of a circle or the end of a line. Preselection symbols help you position the cursor and indicate a point on the sketch that can be selected, such as a vertex or a midpoint.

If you click when preselection highlighting or a preselection symbol is visible, Abaqus/CAE selects the indicated object.

For more information:

- “Selecting and unselecting individual objects,” Section 6.2.1 of the Abaqus/CAE User’s Guide
- “The Sketcher cursors and preselection,” Section 20.4.5 of the Abaqus/CAE User’s Guide

primary axis

An approximate material direction for a *discrete orientation*. Abaqus/CAE uses the primary axis and the *normal axis* to construct a right-handed Cartesian coordinate system. You choose the coordinate system axis that you want the primary axis to represent in the resultant orientation and select topology or datums or enter vector values to define the desired axis.

For more information:

- “Using discrete orientations for material orientations and composite layup orientations,” Section 12.16 of the Abaqus/CAE User’s Guide, in the HTML version of this guide

primary cursor

An active cursor that you use with most applications on your computer. The primary cursor usually appears as an arrow pointer; you position this cursor by moving the mouse. See also *secondary cursor*.

For more information:

- “The Sketcher cursors and preselection,” Section 20.4.5 of the Abaqus/CAE User’s Guide

printed output (`.dat`) file

A file (*job_name*`.dat`) that contains information generated by the Abaqus analysis input file preprocessor, including the model definition and any error or warning messages that were detected while processing the input data. In addition, the printed output file contains any printed output data written during the analysis.

For more information:

- “Output,” Section 4.1.1 of the Abaqus Analysis User’s Guide
- “Output to the data and results files,” Section 4.1.2 of the Abaqus Analysis User’s Guide
- “Understanding output requests,” Section 14.4 of the Abaqus/CAE User’s Guide

privileged plane

The base of the *3D compass*. By default, Abaqus/CAE draws the *3D compass* so that the X–Z plane is the privileged plane. You can customize the appearance of the *3D compass* to change the privileged plane to any of the three major compass planes (X–Y, Y–Z, or X–Z).

For more information:

- "Customizing the 3D compass," Section 5.3.4 of the Abaqus/CAE User's Guide

probe

A mode in which Abaqus/CAE displays information as you move the cursor around the current viewport. Probing a model plot displays model data and analysis results; probing an *X–Y* plot displays *X–Y* curve data.

For more information:

- Chapter 51, "Probing the model," of the Abaqus/CAE User's Guide

procedure

The type of analysis to be performed during an analysis step. Static stress, dynamic stress, eigenvalue buckling, and transient heat transfer are examples of analysis procedures.

In Abaqus/CAE many tasks that you perform are broken into step-by-step procedures. When you perform one of these procedures, Abaqus/CAE displays instructions for each step at the appropriate time in the prompt area near the bottom of the main window.

For more information:

- "Solving analysis problems: overview," Section 6.1.1 of the Abaqus Analysis User's Guide
- "Using the prompt area during procedures," Section 3.1 of the Abaqus/CAE User's Guide

profile

A definition of the engineering properties of a beam section that are related to its cross-sectional shape and size (for example, cross-section area and moments of inertia). When you define a beam section, you must include a reference to a profile in the section definition.

For more information:

- "Defining profiles," Section 12.2.2 of the Abaqus/CAE User's Guide

A surface profile is the collection of line segments defining the cross-section of analytical rigid surfaces.

For more information:

- "Analytical rigid surface definition," Section 2.3.4 of the Abaqus Analysis User's Guide

Profile also refers to the distribution of a variable over space or time such as a temperature profile or a wind velocity profile.

For more information:

- "Uncoupled heat transfer analysis," Section 6.5.2 of the Abaqus Analysis User's Guide
- "Abaqus/Aqua analysis," Section 6.11.1 of the Abaqus Analysis User's Guide

prompt area

An area located at the bottom of the Abaqus/CAE main window that displays instructions for you to follow during a procedure.

For more information:

- “Using the prompt area during procedures,” Section 3.1 of the Abaqus/CAE User’s Guide

Property module

An Abaqus/CAE module that allows you to define the material and section properties of a model by creating section definitions and assigning them to parts or to regions of parts. Most section definitions refer to material definitions, which you also create using the Property module.

For more information:

- Chapter 12, “The Property module,” of the Abaqus/CAE User’s Guide

Q

quantity type

A category that describes the data in one of the columns of an *X–Y* data object. Abaqus/CAE includes over 80 predefined quantity types that describe the output of every variable in Abaqus, including commonly used quantity types such as temperature, time, stress, and strain.

For more information:

- “Understanding quantity types,” Section 47.1.4 of the Abaqus/CAE User’s Guide

Query toolset

An Abaqus/CAE toolset that allows you to obtain information about your model. Abaqus/CAE displays the requested information in the message area; and, in most cases, the same information is written to the replay file.

For more information:

- Chapter 71, “The Query toolset,” of the Abaqus/CAE User’s Guide

queue

An option that executes jobs on a computer in a sequential manner.

For more information:

- “Submitting a job remotely,” Section 19.2.7 of the Abaqus/CAE User’s Guide
- “Customizing Abaqus/CAE startup,” Section 4.3.3 of the Abaqus Installation and Licensing Guide

R

radio button

An option button that allows you to choose between mutually exclusive options in some Abaqus/CAE dialog boxes. When a particular option is controlled by radio buttons, you can choose only one of the buttons at a time.

For more information:

- "Using basic dialog box components," Section 3.2.1 of the Abaqus/CAE User's Guide

rake

A line segment with a series of points specified along its length that controls the locations where Abaqus/CAE displays streamlines for a fluid flow analysis.

For more information:

- "Creating a stream," Section 74.2 of the Abaqus/CAE User's Guide, in the HTML version of this guide

recovery (`.rec`) file

A file (*model_database_name*`.rec`) containing commands that Abaqus/CAE can use to replicate the model database currently in memory if it becomes lost due to a catastrophic interruption of your Abaqus/CAE session, such as a power outage. The recovery file contains only those commands that were executed since the last time the model database was saved; all remaining commands are saved in the *journal file*.

For more information:

- "Recreating an unsaved model database," Section 9.5.3 of the Abaqus/CAE User's Guide

reference dimension

A type of constraint in Abaqus/CAE that annotates quantities that are already controlled elsewhere in the sketch or quantities such as projected reference geometry that cannot be controlled within the current sketch. A reference dimension indicates the size of geometry in a sketch while allowing the size to change, as opposed to regular dimensions that fix the geometry at the dimensioned size. You can use reference dimensions to indicate the size of reference geometry or to provide alternate dimensions without overconstraining a sketch.

For more information:

- "Constraining, dimensioning, and parameterizing a sketch," Section 20.12 of the Abaqus/CAE User's Guide, in the HTML version of this guide

reference geometry

In Abaqus/CAE, the edges and vertices of the existing part or assembly that are in the same plane as the sketch plane. Abaqus/CAE projects the reference geometry onto the sketch sheet when you enter the Sketch module; you can select reference geometry to help position objects and to constrain the sketch to the underlying geometry.

For more information:

- "Reference geometry," Section 20.5.1 of the Abaqus/CAE User's Guide

reference representation

An alternative representation of a part or a subset (one or more cells) of a part that is not used in an Abaqus/CAE analysis. The reference representation replaces a selected part or portion of a part. The replaced geometry is removed from the active representation of the part—the geometry that will be meshed and used in an analysis—leaving the reference representation to display a translucent view of the same geometry. The reference representation is typically used as a tool to create a new active representation, such as a midsurface model.

For more information:

- "Understanding the reference representation," Section 35.2 of the Abaqus/CAE User's Guide

reference surface

The spatial geometry defined by the nodes and normal directions of conventional shell elements. The reference surface is typically positioned at the center of the shell's thickness, but it is also possible to offset the reference surface from the midsurface.

For more information:

- "Shell elements: overview," Section 29.6.1 of the Abaqus Analysis User's Guide
- "Defining the initial geometry of conventional shell elements," Section 29.6.3 of the Abaqus Analysis User's Guide
- "Defining a temperature field," Section 16.11.9 of the Abaqus/CAE User's Guide, in the HTML version of this guide

refinement rate limit

An upper bound on the total number of elements calculated by the remesh algorithm in Abaqus/CAE that modulates the aggressivity of the sizing method and controls the introduction of smaller elements. When Abaqus/CAE is remeshing your model adaptively, the coarsening rate limit that you specified in the remeshing rule modulates the rate at which smaller elements are introduced into the mesh. The refinement factor has a significant effect on the convergence of the adaptive meshing procedure and may help you achieve faster and more efficient mesh convergence.

For more information:

- "Refinement and coarsening rate factors" in "Solution-based mesh sizing," Section 12.3.3 of the Abaqus Analysis User's Guide

regenerate

A process that recalculates model geometry after a feature of an Abaqus/CAE model has been modified; by default, Abaqus/CAE automatically regenerates all dependent features if you modify a feature. For example, if you modify a feature of a part, Abaqus/CAE regenerates the part, any instances of the part in the assembly, and the final mesh.

For more information:

- "Using the Feature Manipulation toolset," Section 65.1 of the Abaqus/CAE User's Guide

region

Any particular portion of an Abaqus/CAE model. A region can be a vertex, edge, face, cell, node, element, or a collection of these entities. You can specify and name specific regions of a part or assembly by creating sets and surfaces that contain those regions. You can divide a part or an assembly into more regions by partitioning it.

For more information:

- Chapter 73, "The Set and Surface toolsets," of the Abaqus/CAE User's Guide

remeshing rule

A rule used by Abaqus/CAE to adapt your mesh iteratively to meet specified error targets. A remeshing rule describes all aspects of your adaptive meshing specification:

- The region to which the rule will be applied
- The step during which the rule will be applied
- The error indicator output variables
- The sizing method
- Sizing constraints

For more information:

- "What are remeshing rules?," Section 17.13.1 of the Abaqus/CAE User's Guide

render style

The style in which you display an object in a viewport in Abaqus/CAE. Examples of render style are **Wireframe**, **Hidden**, and **Shaded**.

For more information:

- "Choosing a render style," Section 76.2 of the Abaqus/CAE User's Guide

replay

An option that specifies the name of the file from which Abaqus/CAE commands are to be replayed. Almost every operation that you perform in Abaqus/CAE is recorded automatically in the replay file (**`abaqus.rpy`**) in the form of Abaqus commands. Executing the replay file is equivalent to replaying the original sequence of operations.

For more information:

- "Recreating an unsaved model database," Section 9.5.3 of the Abaqus/CAE User's Guide

restart (`.res`) file

A file (*job_name*`.res`) used to continue an analysis job.

In addition, a capability that allows you to run complex multistep simulations in stages rather than in a single job so that you can examine the results and confirm that the analysis is performing as expected before continuing with the next stage. The Abaqus restart analysis capability uses restart data from a prior solution to determine the model's response to additional load history.

For more information:

- "Restarting an analysis," Section 9.1.1 of the Abaqus Analysis User's Guide
- "Restart output requests," Section 14.5.2 of the Abaqus/CAE User's Guide

Results Tree

A component of the Abaqus/CAE GUI that provides a visual description of the output data available in your session, including all open output databases and session-specific data such as *X–Y* data and *X–Y* plots. This tool shares the left side of the Abaqus/CAE interface with the Model Tree.

For more information:

- "An overview of the Results Tree," Section 3.5.2 of the Abaqus/CAE User's Guide

resume

The process of restoring a feature that was temporarily deleted from an Abaqus/CAE model to simplify the appearance of a part or assembly. See also *suppress*.

For more information:

- "Using the Feature Manipulation toolset," Section 65.1 of the Abaqus/CAE User's Guide

rigid body

A collection of nodes, elements, and/or surfaces that is so much stiffer than the rest of the model that its deformation can be considered negligible. In Abaqus/Standard and Abaqus/Explicit a rigid body is a collection of rigid elements. See also *analytical rigid surface* or *discrete rigid part* for information about rigid bodies in Abaqus/CAE.

For more information:

- "Analytical rigid surface definition," Section 2.3.4 of the Abaqus Analysis User's Guide
- "Modeling rigid bodies and display bodies," Section 11.7 of the Abaqus/CAE User's Guide

rigid body reference node

The node located at the rigid body reference point. When constraining the rigid body, you apply constraints to the degrees of freedom of the rigid body reference node.

For more information:

- "Analytical rigid surface definition," Section 2.3.4 of the Abaqus Analysis User's Guide
- "The reference point," Section 11.8.1 of the Abaqus/CAE User's Guide

rigid body reference point

A selected point that is used to define the motion of a rigid body (a rigid part in Abaqus/CAE) or to apply constraints to a rigid body.

For more information:

- "Analytical rigid surface definition," Section 2.3.4 of the Abaqus Analysis User's Guide
- "The reference point," Section 11.8.1 of the Abaqus/CAE User's Guide

round

A concave easing of an interior corner of a part used to reduce stress concentration; also known as a fillet. You can use the blend tools in the Part module to round selected edges of the part in the current viewport to the desired radius.

For more information:

- "Blend features," Section 11.9.5 of the Abaqus/CAE User's Guide

S

scale factor animation

A series of plots in Abaqus/CAE created from a single step and frame of the output database (ODB). The different plots are formed by multiplying the deformation scale factor by a range of animation scale factors.

For more information:

- "Scale factor animation," Section 49.1.2 of the Abaqus/CAE User's Guide

scratch output database (`.ods`) file

A file (*job_name*`.ods`) that contains a *session step* and is deleted automatically when the original output database file is closed or when the Abaqus/CAE session ends.

For more information:

- "Understanding the files generated by creating and analyzing a model," Section 9.4 of the Abaqus/CAE User's Guide

script

A type of program that consists of a set of instructions to an application. In Abaqus/CAE almost every operation that you perform during a session can be duplicated by a script containing a set of Abaqus/CAE commands. You can find examples of Abaqus/CAE commands in the replay file (**`abaqus.rpy`**) that is written automatically during every Abaqus/CAE session.

For more information:

- "Understanding the files generated by creating and analyzing a model," Section 9.4 of the Abaqus/CAE User's Guide

secondary cursor

An active cursor within the Abaqus/CAE Sketch module that looks like a plus sign (+) and appears near the primary cursor whenever the Sketch module prompts you to select a point. By default, if you move the primary cursor near a point that is eligible for selection, the secondary cursor jumps directly to the point while the primary cursor remains fixed; therefore, you can easily see exactly which point is selected before committing the selection. See also *primary cursor*.

For more information:

- "The Sketcher cursors and preselection," Section 20.4.5 of the Abaqus/CAE User's Guide

section definition

The data that specify the properties of regions in an Abaqus/CAE assembly or in a set of elements in an Abaqus/Standard, Abaqus/Explicit, or Abaqus/CFD model. A section definition can contain information such as a material name, Poisson's ratio, transverse shear data, and various other parameters.

For more information:

- Chapter 12, "The Property module," of the Abaqus/CAE User's Guide

section point

An integration point. When you define shell or beam sections that are integrated during an analysis, you must specify the number of section integration points through the thickness of the section. A group of section points is located at each material integration point over the surface of a shell element or along the length of a beam element.

For more information:

- Chapter 12, “The Property module,” of the Abaqus/CAE User’s Guide

seed

Markers that you place along the edges of an unmeshed assembly in Abaqus/CAE to indicate the desired density of the mesh. By default, mesh seeds provide only a target mesh density; if necessary, the mesh generator alters the original seed distribution to generate the mesh successfully. You can prevent this redistribution by constraining seeds.

For more information:

- “Understanding seeding,” Section 17.4 of the Abaqus/CAE User’s Guide

session

The time during which a program accepts input, processes information, and responds to user commands. An Abaqus/CAE session begins when you start Abaqus/CAE and continues until you exit.

For more information:

- “Starting and exiting Abaqus/CAE,” Section 2.1 of the Abaqus/CAE User’s Guide

session step

A step in the *scratch output database file* (*job_name*`.ods`) in which Abaqus/CAE saves field output that you have created by either operating on existing field output variables or combining results from several analysis frames.

For more information:

- “Starting and exiting Abaqus/CAE,” Section 2.1 of the Abaqus/CAE User’s Guide

set

A named region or collection of objects on which you can perform various operations. See also *node set*, *element set*, *geometry set* and *discrete set*.

For more information:

- “Input syntax rules,” Section 1.2.1 of the Abaqus Analysis User’s Guide
- “Understanding sets and surfaces,” Section 73.2 of the Abaqus/CAE User’s Guide

Set toolset

An Abaqus/CAE toolset that allows you to create and manage sets in all Abaqus/CAE modules except the Visualization module.

For more information:

- Chapter 73, “The Set and Surface toolsets,” of the Abaqus/CAE User’s Guide

shell

In reference to an element formulation, a shell is an element formulation that captures bending and membrane behavior suitable for structures where the thickness is small relative to other dimensions.

For more information:

- “Shell elements,” Section 29.6 of the Abaqus Analysis User’s Guide

In reference to a geometry type, a shell is a surface geometry.

For more information:

- “The relationship between parts and features,” Section 11.3.1 of the Abaqus/CAE User’s Guide

shell sections

The section properties of shell regions. Shells model structures in which one dimension (the thickness) is significantly smaller than the other two dimensions and in which the stresses in the thickness direction are negligible.

For more information:

- “Shell section behavior,” Section 29.6.4 of the Abaqus Analysis User’s Guide
- “Defining sections,” Section 12.2.3 of the Abaqus/CAE User’s Guide

sizing method

The method used by Abaqus/CAE to generate new element sizes during the adaptive remesh process. For a particular variable, a sizing method reads and operates on a field of base solution values and their corresponding error indicator output variables from a region defined by the remeshing rule. Abaqus uses the sizing method to calculate new element sizes. Abaqus/CAE provides two sizing methods: **Uniform error distribution** and **Minimum/maximum control**.

For more information:

- “Sizing methods” in “Solution-based mesh sizing,” Section 12.3.3 of the Abaqus Analysis User’s Guide
- “Choosing the remeshing rule sizing method,” Section 17.21.3 of the Abaqus/CAE User’s Guide, in the HTML version of this guide

sketch

A two-dimensional profile that is used to help form the geometry defining an Abaqus/CAE native part. You use sketches to create planar or axisymmetric features; or you can extrude, revolve, or sweep a sketch to form a three-dimensional feature.

For more information:

- Chapter 20, “The Sketch module,” of the Abaqus/CAE User’s Guide

sketch coordinates

The cursor coordinates that appear in the upper-right corner of the Sketcher as you create new sketch geometry. Sketch coordinates appear only when they differ from the current grid coordinates—when the grid origin or rotation has been changed.

For more information:

- “Realigning the sketch grid relative to the sketch,” Section 20.4.4 of the Abaqus/CAE User’s Guide

Sketcher

An Abaqus/CAE toolset that allows you to sketch the lines and curves that form the two-dimensional profile of a feature, to add constraints to the sketch, and to modify the sketch in Abaqus/CAE.

For more information:

- Chapter 20, “The Sketch module,” of the Abaqus/CAE User’s Guide

Sketch module

An Abaqus/CAE module used to create a sketch that defines a planar part, a beam, or a partition or to create a sketch that might be extruded, swept, or revolved to form a three-dimensional part.

For more information:

- Chapter 20, “The Sketch module,” of the Abaqus/CAE User’s Guide

slider

An indicator on a gauge in Abaqus/CAE dialog boxes that you drag to set the value of an option that has a continuous range of possible values.

For more information:

- “Using basic dialog box components,” Section 3.2.1 of the Abaqus/CAE User’s Guide

SMP (shared memory parallel)

A mode of parallel execution where multiple processors share a single address space. Abaqus supports thread-based SMP.

For more information:

- “Parallel execution,” Section 3.5 of the Abaqus Analysis User’s Guide

solid sections

A collection of information that defines the section properties of two-dimensional, three-dimensional, and axisymmetric solid regions.

For more information:

- "Defining sections," Section 12.2.3 of the Abaqus/CAE User's Guide

solver

A term used to refer to Abaqus products: Abaqus/Standard (implicit), Abaqus/Explicit (explicit), and Abaqus/CFD (computational fluid dynamics). See also *equation solver*.

spline

A curve defined by a mathematical function that connects separate points with a high degree of smoothness. Use the spline tool in the Abaqus/CAE Sketch module to sketch a smooth curve that connects a series of points. Abaqus/CAE calculates the shape of the curve using a cubic spline fit between all the points along the spline; as a result, the first and second derivatives of the spline are continuous.

For more information:

- "Sketching splines," Section 20.10.10 of the Abaqus/CAE User's Guide, in the HTML version of this guide

stand-alone sketch

An Abaqus/CAE sketch that is independent of any particular feature; you can incorporate a stand-alone sketch into the current sketch.

For more information:

- "Stand-alone sketches," Section 20.3.1 of the Abaqus/CAE User's Guide

status (`.sta`) file

A file (*job_name*`.sta`) generated during an Abaqus/Standard, Abaqus/Explicit, or Abaqus/CFD analysis job that contains information about the progress of the analysis.

For more information:

- "Output," Section 4.1.1 of the Abaqus Analysis User's Guide
- "Degree of freedom monitor requests," Section 14.5.4 of the Abaqus/CAE User's Guide

STEP [STandard for the Exchange of Product model data (STEP ISO 10303–1)]

A replacement standard for IGES that attempts to overcome some of the shortcomings of IGES. The STEP format is designed to provide computer-interpretable representation of a product throughout its lifecycle, independent of any particular system. You can import STEP format parts, and you can export parts in STEP-format. In addition, you can import and export a sketch from a STEP file.

For more information:

- “Importing files into and exporting files from Abaqus/CAE,” Section 10.1 of the Abaqus/CAE User’s Guide
- “Importing parts,” Section 10.7.2 of the Abaqus/CAE User’s Guide, in the HTML version of this guide
- “Exporting geometry, model, and mesh data,” Section 10.9 of the Abaqus/CAE User’s Guide, in the HTML version of this guide

step

A sequence that provides a convenient way to capture changes in the loading and boundary conditions of the model, changes in the way parts of the model interact with each other, and any other changes that may occur in the model during the course of the analysis. In addition, steps allow you to change the analysis procedure, the data output, and various controls. You can also use steps to define linear perturbation analyses about nonlinear base states.

For more information:

- “Defining a model in Abaqus,” Section 1.3.1 of the Abaqus Analysis User’s Guide
- “What is a step?,” Section 14.3.1 of the Abaqus/CAE User’s Guide

step-dependent manager

An Abaqus/CAE dialog box that contains a list of all of the objects of a certain type that you have created, as well as **Create**, **Edit**, **Copy**, **Rename**, and **Delete** buttons that you can use to manipulate existing objects and to create new ones. A step-dependent manager is similar to a *basic manager*, but it contains additional information concerning the history of each object listed in the manager.

For more information:

- “What are step-dependent managers?,” Section 3.4.2 of the Abaqus/CAE User’s Guide

step-dependent object

An object that you can create and, in some cases, modify and deactivate in particular steps of an analysis. Loads, boundary conditions, and interactions are step-dependent objects.

For more information:

- “What are step-dependent managers?,” Section 3.4.2 of the Abaqus/CAE User’s Guide

Step module

An Abaqus/CAE module used to create and define analysis steps and to request output for each step. You can also use the Step module to specify adaptive meshing as well as contact and general solution controls.

For more information:

- Chapter 14, “The Step module,” of the Abaqus/CAE User’s Guide

stream

A set of streamlines for visualization of data in a fluid flow analysis.

For more information:

- Chapter 74, “The Stream toolset,” of the Abaqus/CAE User’s Guide

streamline

A curve that is instantaneously tangent to the velocity vector of the flow. Streamlines show the direction a fluid element will travel in at any point in time in a fluid flow analysis.

For more information:

- Chapter 74, “The Stream toolset,” of the Abaqus/CAE User’s Guide

structured meshing

An Abaqus/CAE technique that generates structured meshes using simple predefined mesh topologies. Abaqus/CAE transforms the mesh of a regularly shaped region, such as a square or a cube, onto the geometry of the region you want to mesh.

For more information:

- “Structured meshing and mapped meshing,” Section 17.8 of the Abaqus/CAE User’s Guide

substructure

A collection of elements from which the internal degrees of freedom have been eliminated.

For more information:

- “Using substructures,” Section 10.1.1 of the Abaqus Analysis User’s Guide
- “Defining substructures,” Section 10.1.2 of the Abaqus Analysis User’s Guide
- Chapter 39, “Substructures,” of the Abaqus/CAE User’s Guide

substructure database

A set of files that describe the geometry of a substructure. Abaqus writes all substructure data to the substructure database during the analysis.

For more information:

- “Defining substructures,” Section 10.1.2 of the Abaqus Analysis User’s Guide
- Chapter 39, “Substructures,” of the Abaqus/CAE User’s Guide

substructure dynamic modes

Eigenmodes and residual modes that are retained in a substructure definition.

For more information:

- "Defining substructures," Section 10.1.2 of the Abaqus Analysis User's Guide
- Chapter 39, "Substructures," of the Abaqus/CAE User's Guide

suppress

The process of temporarily deleting a feature from an Abaqus/CAE model to simplify the appearance of a part or assembly. In addition, suppressing a feature can increase the speed of regeneration. See also *resume*.

For more information:

- "Using the Feature Manipulation toolset," Section 65.1 of the Abaqus/CAE User's Guide

surface

A named region that can be defined on the faces, edges, or nodes of a geometric rigid body or a discrete finite element model. A surface definition can also contain information to distinguish between the positive and negative sides of the surface.

For more information:

- "Surfaces: overview," Section 2.3.1 of the Abaqus Analysis User's Guide
- Chapter 73, "The Set and Surface toolsets," of the Abaqus/CAE User's Guide

Surface toolset

An Abaqus/CAE toolset that allows you to create and manage surfaces in all Abaqus/CAE modules except the Visualization module.

For more information:

- Chapter 73, "The Set and Surface toolsets," of the Abaqus/CAE User's Guide

swept meshing

A technique used by Abaqus/CAE to mesh complex extruded or revolved solid regions as well as revolved surface regions. The swept meshing technique involves two phases:

- Abaqus/CAE creates a mesh on one side of the region, known as the source side.
- Abaqus/CAE copies the nodes of that mesh, one element layer at a time, until the final side, known as the target side, is reached.

For more information:

- "Swept meshing," Section 17.9 of the Abaqus/CAE User's Guide

symbol plot

An Abaqus/CAE plot that shows the magnitude and direction of a particular vector or tensor variable at a specified step and frame of the analysis. Abaqus/CAE represents the values as symbols (arrows) drawn at the locations in the model where the results were obtained.

For more information:

- Chapter 45, “Plotting analysis results as symbols,” of the Abaqus/CAE User’s Guide

synchronization

The process by which Abaqus/CAE animates data in every animation-eligible viewport in your session at the same time so that the viewports play, stop, and increment together when you examine data frame by frame, animate a model plot, or animate a time-dependent *X*–*Y* plot.

For more information:

- “Controlling animations in multiple viewports,” Section 49.4.6 of the Abaqus/CAE User’s Guide, in the HTML version of this guide

T

tab

A labeled group of options in an Abaqus/CAE dialog box.

For more information:

- “Using dialog boxes separated by tabs,” Section 3.2.6 of the Abaqus/CAE User’s Guide

text field

An area in an Abaqus/CAE dialog box in which you can enter information.

For more information:

- “Using basic dialog box components,” Section 3.2.1 of the Abaqus/CAE User’s Guide

thread-based execution

A mode of parallel operation using multiple simultaneous threads of execution that enables Abaqus to split computationally intensive operations into multiple simultaneous tasks. On multiprocessor or multi-core systems, threads run simultaneously on different processors or cores. Abaqus supports thread-based execution using SMP.

For more information:

- “Parallel execution: overview,” Section 3.5.1 of the Abaqus Analysis User’s Guide

through hole

A hole that passes completely through a three-dimensional solid object. The path of the hole continues to infinity and cuts the object anywhere it intersects.

For more information:

- “Adding a cut feature,” Section 11.24 of the Abaqus/CAE User’s Guide, in the HTML version of this guide

TIFF (Tag Image File Format)

A standard bitmap graphics file format commonly used for storage of graphic images. Depending on the display device, the TIFF format provides options to adjust both color and greyscale images and can encode very high-quality images. Abaqus/CAE allows you to save images of selected viewports in TIFF-format files. Abaqus/CAE does not compress the data stored in TIFF-format files; as a result, the files can consume large amounts of disk space.

For more information:

- “Printed image formats,” Section 8.1.1 of the Abaqus/CAE User’s Guide

time history

A sequence of plots that vary over time according to actual analysis results. If the time history is frame-based, the animation increments frame by frame; if the time history is time-based, the animation increments along a common time line.

For more information:

- “Customizing the time history of synchronized viewports,” Section 49.4.7 of the Abaqus/CAE User’s Guide, in the HTML version of this guide

toolbar

An area in Abaqus/CAE that contains a convenient sets of tools for managing your files and for viewing your model. Items in a toolbar are shortcuts to functions that are also available from the main menu bar. By default, all the standard toolbars appear in a row directly under the Abaqus/CAE main menu bar; however, you can configure the toolbars to appear anywhere on your screen.

For more information:

- “Components of the toolbars,” Section 2.2.3 of the Abaqus/CAE User’s Guide

toolbox

A collection of icons that provide quick access to commonly used Abaqus/CAE functions. The tools available in a toolbox are also available from the main menu bar.

For more information:

- "Understanding and using toolboxes and toolbars," Section 3.3 of the Abaqus/CAE User's Guide

toolset

A functional unit that allows you to perform a specific modeling task in Abaqus/CAE.

For more information:

- "What is a toolset?," Section 2.4 of the Abaqus/CAE User's Guide

top-down meshing

The automated meshing process used by Abaqus/CAE to create a mesh by working down from the geometry of a region to the elements and nodes. See also *bottom-up meshing*.

For more information:

- Chapter 17, "The Mesh module," of the Abaqus/CAE User's Guide

triad

See *view orientation triad* or *material orientation triad*.

truss sections

The section properties of truss regions. Trusses model slender, rod-like structures that provide axial strength but—unlike beams—provide no bending stiffness.

For more information:

- "Truss elements," Section 29.2.1 of the Abaqus Analysis User's Guide
- "Defining sections," Section 12.2.3 of the Abaqus/CAE User's Guide
- "Creating truss sections," Section 12.13.12 of the Abaqus/CAE User's Guide, in the HTML version of this guide

U

unmeshable region

A solid region that Abaqus/CAE cannot mesh automatically using hexahedral elements. An unmeshable region can be made meshable by partitioning, by assigning the bottom-up meshing technique, or by assigning tetrahedral elements to the region. See also *mesh*.

For more information:

- Chapter 17, "The Mesh module," of the Abaqus/CAE User's Guide

user-defined view

A custom view to an Abaqus/CAE model in a selected viewport. You can save the current position, orientation, and scale factor as one of four user-defined views and subsequently apply the view to any viewport. A user-defined view is saved for the duration of the session. See also *view*.

For more information:

- "Saving a user-defined view," Section 5.6.9 of the Abaqus/CAE User's Guide, in the HTML version of this guide

user subroutine

Code you write to increase the functionality of several Abaqus options for which data line usage may be too restrictive. A user subroutine is typically written as Fortran code.

For more information:

- "User subroutines and utilities," Section 18.1 of the Abaqus Analysis User's Guide
- *Abaqus User Subroutines Reference Guide*

V

view

The position, orientation, and scale factor define the view of an object in an Abaqus/CAE viewport.

For more information:

- "Understanding the view manipulation tools," Section 5.2 of the Abaqus/CAE User's Guide

view orientation triad

A set of three perpendicular axes that indicate the orientation of your view of the model currently being displayed.

For more information:

- "Customizing the view triad," Section 5.4 of the Abaqus/CAE User's Guide

viewport

An area on the Abaqus/CAE canvas where you can display models or analysis results. You can create multiple viewports that can be moved, resized, and deleted.

For more information:

- "What is a viewport?," Section 4.1.1 of the Abaqus/CAE User's Guide

Virtual Topology toolset

An Abaqus/CAE toolset that allows you to ignore small details, such as very small faces and edges, when you mesh a part or a part instance.

For more information:

- Chapter 75, "The Virtual Topology toolset," of the Abaqus/CAE User's Guide

Visualization module

An Abaqus/CAE module that provides graphical display of finite element models and results. It obtains model and result information from the output database (**`.odb`**). Major capabilities of the Visualization module include undeformed and deformed shape plotting, results contour and symbol plotting, *X–Y* plotting and reporting, field output reporting, plot customization, and animation.

The Visualization module is also licensed separately as Abaqus/Viewer.

For more information:

- Chapter 40, "Visualization module basics," of the Abaqus/CAE User's Guide

volume fraction

In an Eulerian analysis, the amount of any given material in an element's volume expressed in terms of a percentage between zero and one.

For more information:

- "Eulerian analysis," Section 14.1.1 of the Abaqus Analysis User's Guide
- Chapter 28, "Eulerian analyses," of the Abaqus/CAE User's Guide

volume fraction tool

An Abaqus/CAE tool that creates a discrete field that specifies a volume fraction for each element in an Eulerian part instance based on the spatial overlap between that element and a reference part instance in the model assembly. The discrete field can be used to create a material assignment predefined field (in the Eulerian part instance) that corresponds to the geometry of the reference part instance.

For more information:

- "The volume fraction tool," Section 28.5 of the Abaqus/CAE User's Guide

W

wedge angle

An angle used to define initial curvature of a model when you create a generalized plane strain section. A wedge angle is given about the global 1- and 2-axes at the reference point indicating the reference node required by generalized strain plane elements.

For more information:

- "Generalized plane strain elements" in "Choosing the element's dimensionality," Section 27.1.2 of the Abaqus Analysis User's Guide
- "Creating generalized plane strain sections," Section 12.13.2 of the Abaqus/CAE User's Guide, in the HTML version of this guide

wire edge

A geometric region in Abaqus/CAE represented by a line or curve, as opposed to the edge of a solid or shell feature.

For more information:

- "Removing wire edges," Section 69.6.6 of the Abaqus/CAE User's Guide, in the HTML version of this guide

work directory

The directory into which Abaqus/CAE writes files during a session and the directory from which you started Abaqus/CAE unless you specified the directory using **File→Set Work Directory**.

For more information:

- "Setting the work directory," Section 9.7.8 of the Abaqus/CAE User's Guide, in the HTML version of this guide
- "Using file selection dialog boxes," Section 3.2.10 of the Abaqus/CAE User's Guide

X, Y, Z

XFEM

The extended finite element method. XFEM allows the presence of discontinuities in an element by enriching degrees of freedom with special displacement functions. XFEM does not require the mesh to match the geometry of the discontinuities. As a result, XFEM is a very attractive and effective way to simulate initiation and propagation of a discrete crack along an arbitrary, solution-dependent path without the requirement of remeshing in the bulk materials.

For more information:

- “Modeling discontinuities as an enriched feature using the extended finite element method,” Section 10.7.1 of the Abaqus Analysis User’s Guide

X–Y data object

A two-dimensional array that Abaqus/CAE stores in two columns: an *X*-column and a *Y*-column. You can use the Visualization module to display *X–Y* data in the form of an *X–Y* plot.

For more information:

- “What is an *X–Y* data object, and what is an *X–Y* plot?,” Section 47.1.1 of the Abaqus/CAE User’s Guide

X–Y plot

A two-axis graph of one variable versus another. Abaqus/CAE can display *X–Y* data objects in the form of an *X–Y* plot.

For more information:

- Chapter 47, “*X–Y* plotting,” of the Abaqus/CAE User’s Guide

X–Y report

A tabular listing of *X* and *Y* data values. Abaqus/CAE can generate an *X–Y* report from the data contained in *X–Y* objects.

For more information:

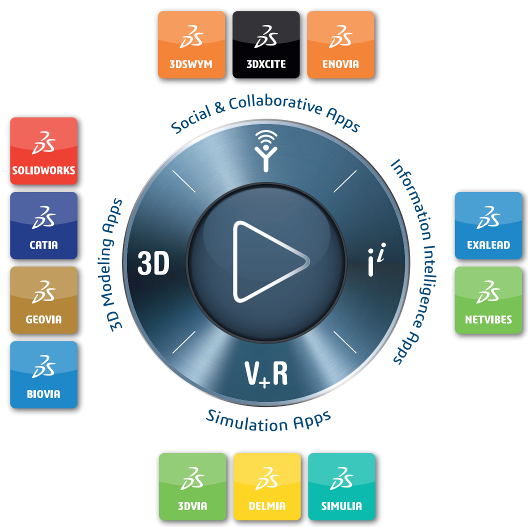
- Chapter 54, “Generating tabular data reports,” of the Abaqus/CAE User’s Guide

About SIMULIA

Dassault Systèmes SIMULIA applications, including Abaqus, Isight, Tosca, and Simulation Lifecycle Management, enable users to leverage physics-based simulation and high-performance computing to explore real-world behavior of products, nature, and life. As an integral part of Dassault Systèmes' **3D**EXPERIENCE platform, SIMULIA applications accelerate the process of making highly informed, mission-critical design and engineering decisions before committing to costly and time-consuming physical prototypes. **www.3ds.com/simulia**

Our **3D**EXPERIENCE Platform powers our brand applications, serving 12 industries, and provides a rich portfolio of industry solution experiences.

Dassault Systèmes, the **3D**EXPERIENCE Company, provides business and people with virtual universes to imagine sustainable innovations. Its world-leading solutions transform the way products are designed, produced, and supported. Dassault Systèmes' collaborative solutions foster social innovation, expanding possibilities for the virtual world to improve the real world. The group brings value to over 170,000 customers of all sizes in all industries in more than 140 countries. For more information, visit **www.3ds.com**.



DASSAULT SYSTEMES | The **3D**EXPERIENCE® Company

Europe/Middle East/Africa
Dassault Systèmes
10, rue Marcel Dassault
CS 40501
78946 Vélizy-Villacoublay Cedex
France

Asia-Pacific
Dassault Systèmes K.K.
ThinkPark Tower
2-1-1 Osaki, Shinagawa-ku,
Tokyo 141-6020
Japan

Americas
Dassault Systèmes
175 Wyman Street
Waltham, Massachusetts
02451-1223
USA