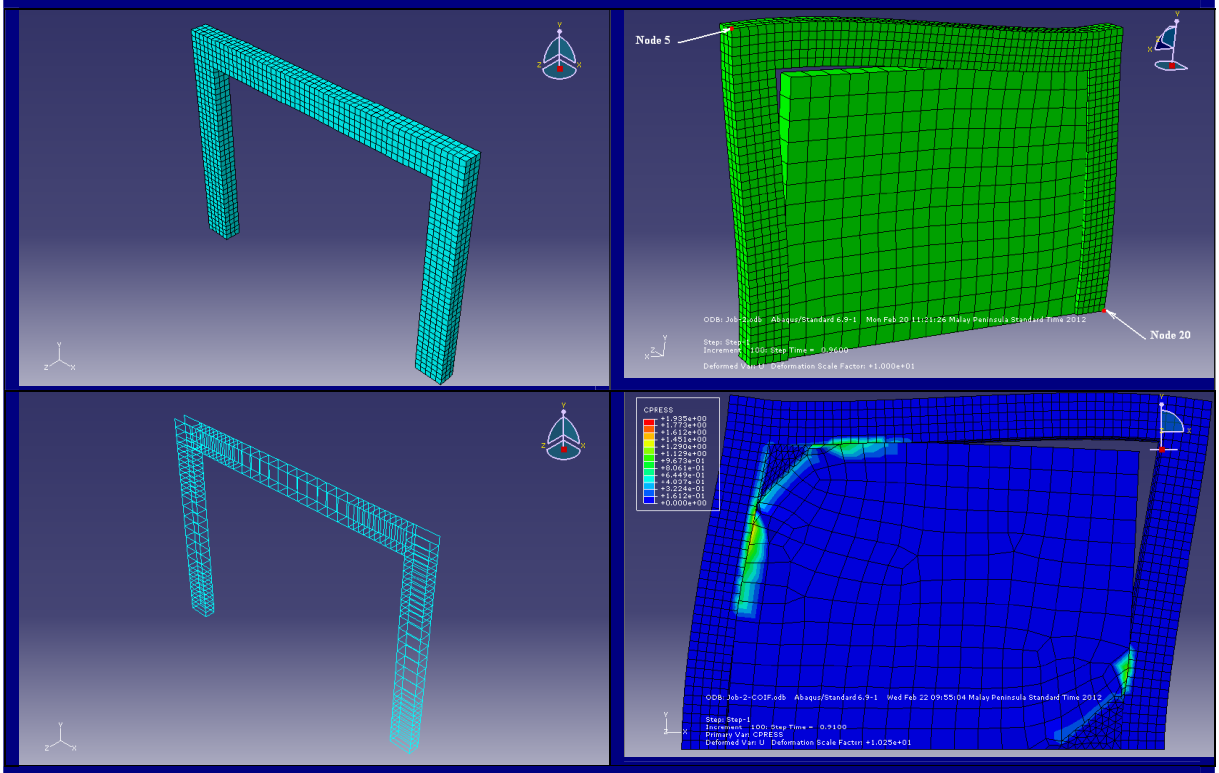
ABAQUS

For Reinforced Concrete Structures

WORKSHOP

Linear Analysis



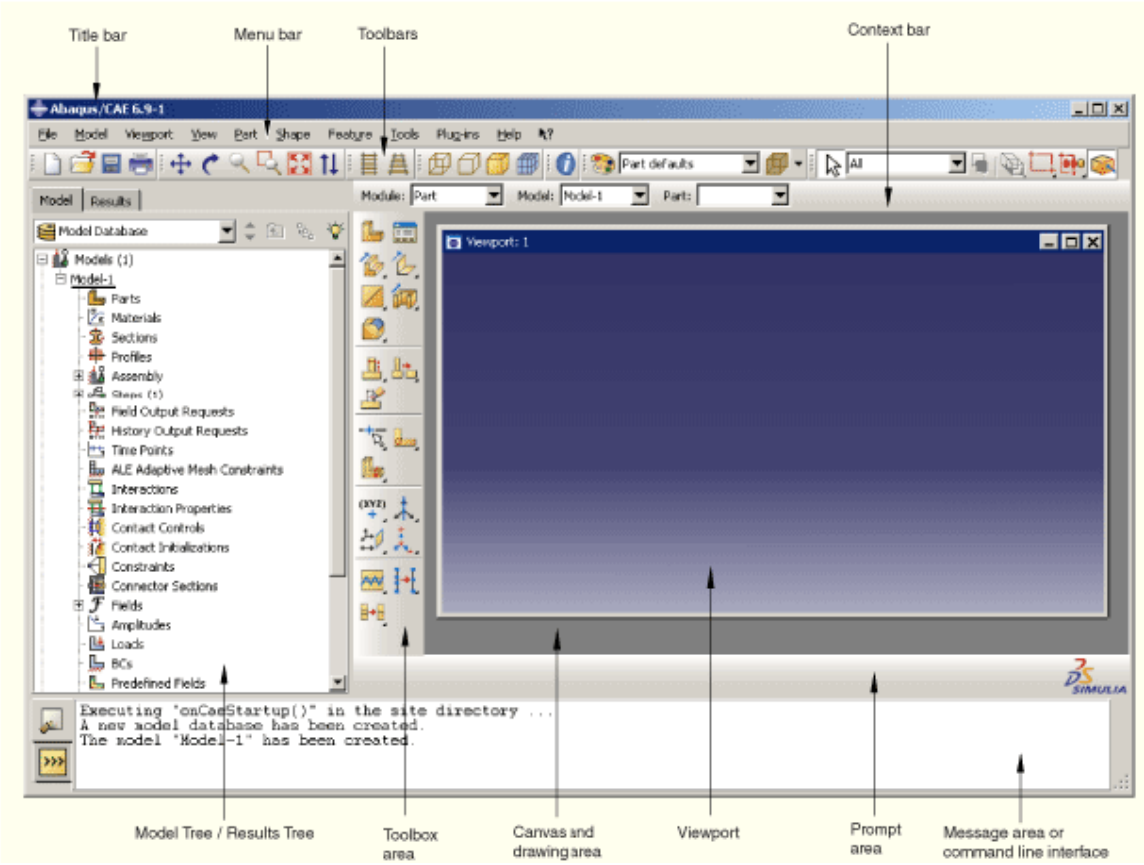
Speakers:
Hamid Reza Khoshnoud (PhD Candidate)
Dr. Abdul Kadir Marsono (Associate Professor)

Introduction

Components of the main window

Interaction with Abaqus/CAE is performed through the main window, and the appearance of the window changes as you work through the modeling process. Figure 2–1 shows the components that appear in the main window.

Figure 2–1 Components of the main window.



The components are:

Title bar

The title bar indicates the release of Abaqus/CAE you are running and the name of the current model database.

Menu bar

The menu bar contains all the available menus; the menus give access to all the functionality in the product. Different menus appear in the menu bar depending on which module you selected from the context bar. For more information, see "Components of the main menu bar," Section 2.2.2.

Toolbars

The toolbars provide quick access to items that are also available in the menus. For more information, see "Components of the toolbars," Section 2.2.3.

Context bar

Abaqus/CAE is divided into a set of modules, where each module allows you to work on one aspect of your model; the **Module** list in the context bar allows you to move between these modules. Other items in the context bar are a function of the module you are working in. For example, the context bar allows you to retrieve an existing part while creating the geometry of the model or to change the output database associated with the current viewport. Similarly, in the Mesh module you can choose whether to display the assembly or a particular part. For more information, see "The context bar," Section 2.2.4.

Model Tree

The Model Tree provides you with a graphical overview of your model and the objects that it contains, such as parts, materials, steps, loads, and output requests. In addition, the Model Tree provides a convenient, centralized tool for moving between modules and for managing objects. If your model database contains more than one model, you can use the Model Tree to move between models. When you become familiar with the Model Tree, you will find that you can quickly perform most of the actions that are found in the main menu bar, the module toolboxes, and the various managers. For more information, see "An overview of the Model Tree," Section 3.5.1.

Results Tree

The Results Tree provides you with a graphical overview of your output databases and other session-specific data such as *X*–*Y* plots. If you have more than one output database open in your session, you can use the Results Tree to move between output databases.

When you become familiar with the Results Tree, you will find that you can quickly perform most of the actions in the Visualization module that are found in the main menu bar and the toolbox. For more information, see "An overview of the Results Tree," Section 3.5.2.

Toolbox area

When you enter a module, the toolbox area displays tools in the toolbox that are appropriate for that module. The toolbox allows quick access to many of the module functions that are also available from the menu bar. For more information, see "Understanding and using toolboxes and toolbars," Section 3.3.

Canvas and drawing area

The canvas can be thought of as an infinite screen or bulletin board on which you post viewports; for more information, see Chapter 4,"Managing viewports on the canvas." The drawing area is the visible portion of the canvas.

Viewport

Viewports are windows on the canvas in which Abaqus/CAE displays your model. For more information, see Chapter 4, "Managing viewports on the canvas."

Prompt area

The prompt area displays instructions for you to follow during a procedure; for example, it asks you to select the geometry as you create a set. In the Visualization module a set of buttons is displayed in the prompt area that allow you to move between the steps and the frames of your analysis. For more information, see "Using the prompt area during procedures," Section 3.1.

Message area

Abaqus/CAE prints status information and warnings in the message area. To resize the message area, drag the top edge; to see information that has scrolled out of the message area, use the scroll bar on the right side. The message area is displayed by default, but it uses the same space occupied by the command line interface. If you have recently used the command line interface, you must click in the bottom left corner of the main window to activate the message area.

Note: If new messages are added while the command line interface is active, Abaqus/CAE changes the background color surrounding the message area icon to red. When you display the message area, the background reverts to its normal color.

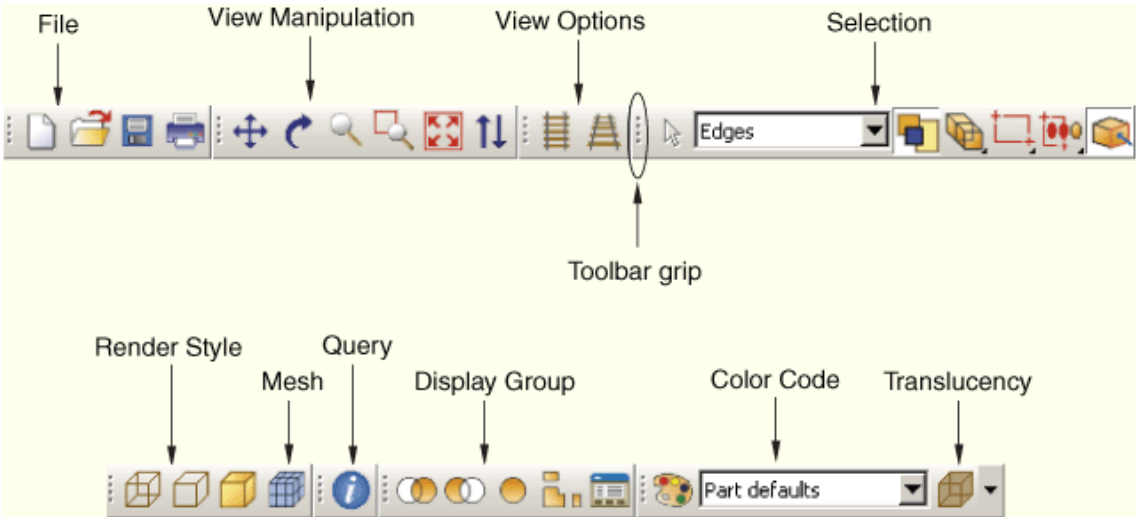
Command line interface

You can use the command line interface to type Python commands and evaluate mathematical expressions using the Python interpreter that is built into Abaqus/CAE. The interface includes primary (>>>) and secondary (...) prompts to indicate when you must indent commands to comply with Python syntax.

The command line interface is hidden by default, but it uses the same space occupied by the message area. Click in the bottom left corner of the main window to switch from the message area to the command line interface.

2.2.3 Components of the toolbars

The toolbars contain convenient sets of tools for managing your files, filtering object selection, and viewing your model. Items in a toolbar are shortcuts to functions that are also available from the main menu bar. By default, Abaqus/CAE displays all of the toolbars in a row underneath the main menu bar. Abaqus/CAE may place some toolbars in a second row depending on your display resolution and the size of the main window. The toolbars are shown in the following figure:



You can change the location of a toolbar using the toolbar's grip, as indicated in the above figure. Clicking and dragging the grip moves the toolbar around the main window. If you release the toolbar grip while the toolbar is over one of the four available docking regions of the main window (see Figure 2–2), Abaqus/CAE “docks” the toolbar; a docked toolbar has no title bar and does not obstruct any other portion of the main window.

Figure 2–2 Available docking regions for toolbars.

If you release the toolbar grip while the toolbar is not near a docking region, Abaqus/CAE creates a floating toolbar with a title bar. A floating toolbar obstructs other items in the main window (see Figure 2–3); however, a floating toolbar can be positioned outside of the Abaqus/CAE main window.

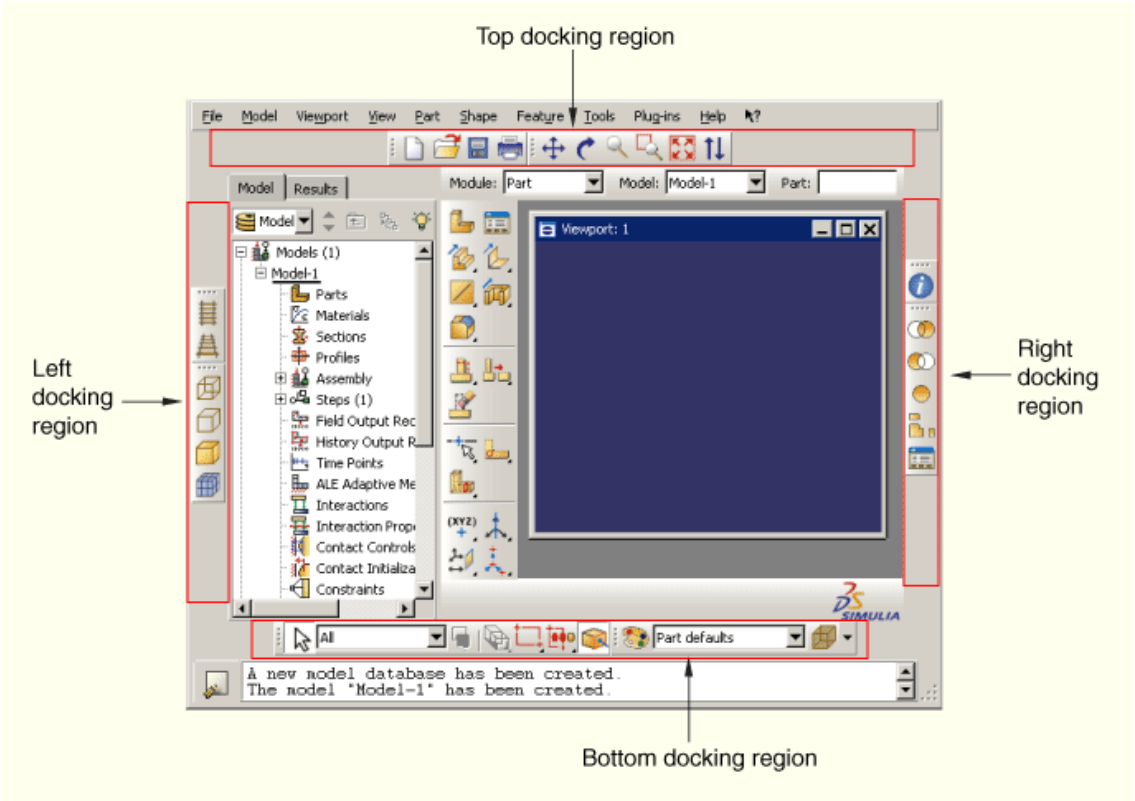
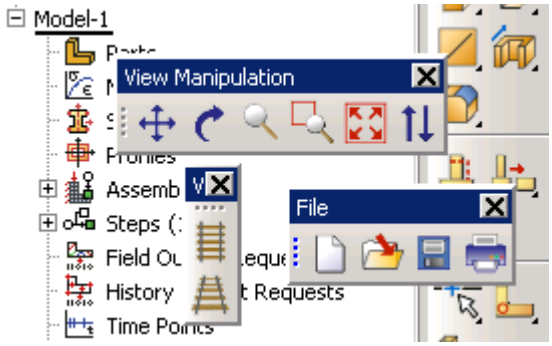


Figure 2–3 Floating toolbars.



Clicking mouse button 3 on a toolbar grip displays a menu that lets you specify the location and format of the toolbar:

Select **Top** to dock the toolbar in the top docking region.

Select **Bottom** to dock the toolbar in the bottom docking region.

Select **Left** to dock the toolbar in the left docking region.

Select **Right** to dock the toolbar in the right docking region.

Select **Float** to change a docked toolbar into a floating toolbar; this option is available only for docked toolbars.

Select **Flip** to change the orientation of a floating toolbar from horizontal to vertical, or vice versa; this option is available only for floating toolbars.

You can also hide toolbars and create custom toolbars that include shortcuts to additional functions. For more information, see Chapter 58, "TheCustomize toolset."

To obtain a short description of a tool in a toolbar, place the cursor over that tool for a moment; a small box containing a description, or "tooltip," will appear. To obtain the name of a toolbar, place the cursor over the toolbar grip for a moment.

The Abaqus/CAE toolbars contain the following functionality:

File

The **File** toolbar allows you to create, open, and save model databases; to open output databases; and to print viewports. For more information, see Part II, "Working with Abaqus/CAE model databases, models, and files," and Chapter 8, "Printing viewports."

View Manipulation

The **View Manipulation** toolbar allows you to specify different views of the model or plot. For example, you can pan, rotate, or zoom the model or plot using these tools. For more information, see Chapter 5, "Manipulating the view and controlling perspective."

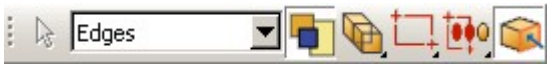
View Options

The **View Options** toolbar allows you to specify whether or not perspective is applied to your model. For more information, see"Controlling perspective," Section 5.5.

Render Style

The **Render Style** toolbar allows you to specify whether the wireframe, hidden line, or shaded render style will be used to display your model. In addition, you can switch between displaying the geometry of an Abaqus/CAE native part and the meshed representation (if it exists) of the same part. In the Visualization module the **Render Style** toolbar also includes the filled render style tool. For more information, see "Displaying a native mesh," Section 17.3.11, and "Choosing a render style," Section 52.2.1.

Selection



The **Selection** toolbar allows you to enable or disable object selection by toggling on the arrow icon. You can use the list to the right of the arrow to limit the types of objects that you can select. The **Selection** toolbar is available only when there are no active procedures running in a viewport. For more information, see "Selecting objects before choosing a procedure," Section 6.3.7.

Query

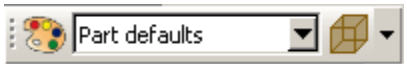
The **Query** toolbar allows you to obtain information about the geometry and features of your model, to probe model and *X–Y* plots for output data, and to perform stress linearization on your results. For more information, see Chapter 68, "The Query toolset"; Chapter 48,"Probing the model"; and Chapter 49, "Calculating linearized stresses."

Display Group

The **Display Group** toolbar allows you to selectively plot one or more model or output database items. For example, you can create a display group that contains only the

elements belonging to specified sets in your model. For more information, see Chapter 74, "Using display groups to display subsets of your model."

Color Code



The **Color Code** toolbar allows you to customize the colors of items in the viewport and change the degree of their translucency.

For color coding, you can create color mappings that assign unique colors to different elements of a display. For example, when using a part instance color mapping, each part instance in a model will appear as a different color. For more information, see Chapter 73,"Color coding geometry and mesh elements."

For translucency, you can click the arrow to the right of the tool to reveal a slider, which you can drag to make the display colors more transparent or more opaque. For more information, see "Changing the translucency," Section 73.3.

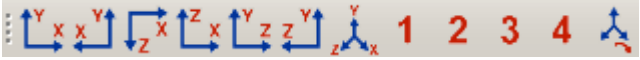
Field Output

The **Field Output** toolbar allows you to select the field output variable for display in the viewport. The selections are limited, but the tool provides access to the **Field Output** dialog box, if needed. For more information, see "Using the field output toolbar," Section 38.4.2.

Viewport

The **Viewport** toolbar allows you to create and align viewports, link viewports, and create viewport annotations. For more information, see "Managing viewports and viewport annotations from the Viewport toolbar," Section 4.2.2. The **Viewport** toolbar is not displayed by default.

Views



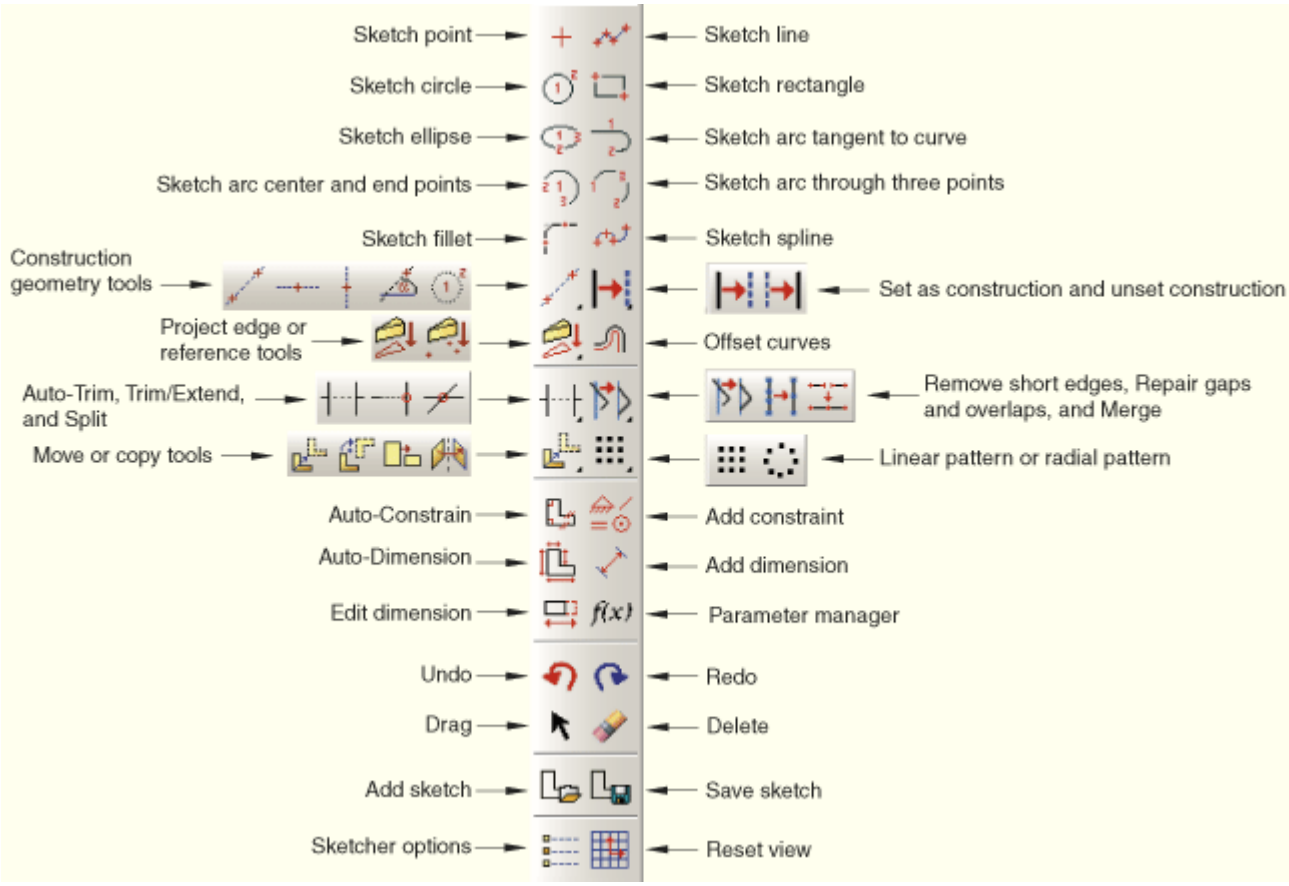
The **Views** toolbar allows you to apply a custom view to the model in the viewport. For more information, see "Custom views," Section 5.2.8. The **Views** toolbar is not displayed by default.

Chapter 1 – Sketch Module

19.4.1 The Sketcher tools

You can access all the Sketcher tools through either the main menu bar or the toolbox. Figure 19–1 shows the hidden icons for all the tools in the Sketcher toolbox.

Figure 19–1 The Sketcher toolbox.



To see a tooltip containing a brief definition of a Sketcher tool, hold the mouse over the tool for a moment. For information on using toolboxes and selecting hidden icons, see "Using toolboxes and toolbars that contain hidden icons," Section 3.3.2.

The Sketcher tools allow you to do the following:

- Create basic sketch entities, such as lines, circles, arcs, ellipses, fillets, and splines.
- Add construction geometry to help you position and align sketch entities.
- Add constraints, dimensions, and parameters to control your sketch geometry and add precision.
- Translate, rotate, scale, or mirror sketch geometry.
- Drag, trim, extend, split, or merge sketch entities.
- Create similar objects by offsetting, creating linear patterns, or creating radial patterns.

19.10.1 Sketching an isolated point

Use the point tool from the Sketcher toolbox to draw a single isolated point. You can use the resulting point as a reference, and you can create dimensions between the point and vertices on your sketch.

To sketch an isolated point:

1. From the Sketcher toolbox, select the point tool . For a diagram of the tools in the Sketcher toolbox, see "The Sketcher tools," Section 19.4.1.

   Abaqus/CAE displays prompts in the prompt area to guide you through the procedure.

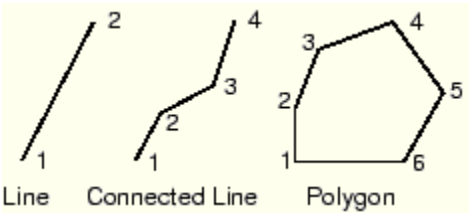
2. Click at the desired location of the point.

   The point appears.

3. To create more points, repeat the previous step.
4. When you have finished creating points, do one of the following:
   - Click mouse button 2 anywhere in the Abaqus/CAE window.
   - Select any other tool in the Sketcher toolbox.
   - Click the cancel button in the prompt area.

19.10.2 Sketching lines and polygons

Use the line tool from the Sketcher toolbox to draw lines, connected lines, or polygons. The following figure shows how you draw lines, connected lines, and polygons by clicking the locations shown, in the order indicated below:



You should take care positioning points while sketching because this positioning can affect the quality of your mesh. Points in the sketch become vertices of the part you are creating or modifying. In turn, when you mesh your model in the Mesh module, Abaqus/CAE converts these vertices into fully constrained seeds and places nodes at their location. For information on how to subsequently move vertices, see "Dragging Sketcher objects," Section 19.17.1.

To sketch lines and polygons:

1. From the line tools in the Sketcher toolbox, select the connected lines tool . For a diagram of the tools in the Sketcher toolbox, see "The Sketcher tools," Section 19.4.1.

   Abaqus/CAE displays prompts in the prompt area to guide you through the procedure.

2. To construct a simple line, click the two end points. To construct a connected line or a polygon, click each vertex.

   Tip: If necessary, you can use the text box in the prompt area to enter the precise coordinates of the vertices of the line. For more information on precisely defining the line, see "Specifying precise geometry," Section 19.6.

   The line or polygon appears as you click a vertex or enter the coordinates.

3. To complete the line or polygon, click mouse button 2.

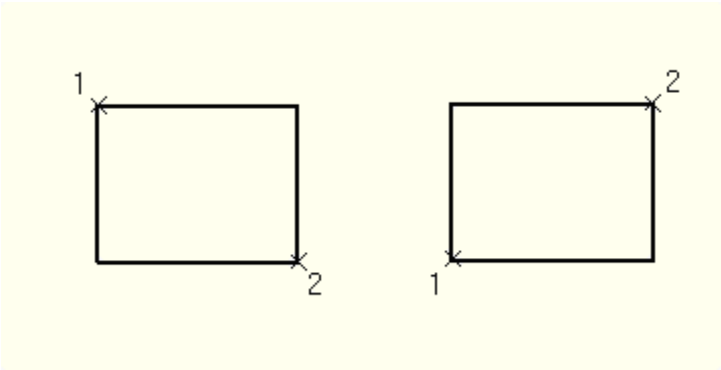
   Tip: If you make a mistake while constructing a connected line or a polygon, click the **Undo** tool in the Sketcher toolbox to delete the most recent line

segment. If you make a mistake in an earlier segment, you can delete the incorrect segments using the **Delete** tool and redraw them with the line tool.

4. To create more lines or polygons, repeat the above steps beginning with Step 2.
5. When you have finished creating lines and polygons, do one of the following:
   - Click mouse button 2 anywhere in the Abaqus/CAE window.
   - Select any other tool in the Sketcher toolbox.
   - Click the cancel button in the prompt area.

9.10.3 Sketching rectangles

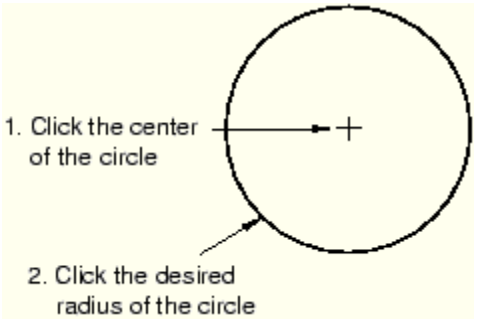
Use the rectangle tool from the Sketcher toolbox to draw rectangles. To draw a rectangle, click at any two opposite corners as indicated by the numbering in the following figure.



You should take care positioning points while sketching because this positioning can affect the quality of your mesh. Points in the sketch become vertices of the part you are creating or modifying. In turn, when you mesh your model in the Mesh module, Abaqus/CAE converts these vertices into fully constrained seeds and places nodes at their location. For information on how to subsequently move vertices, see "Dragging Sketcher objects," Section 19.17.1.

19.10.4 Sketching circles

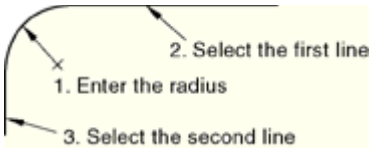
Use the circle tool from the Sketcher toolbox to draw circles based on a center point and any arbitrary point on the circumference of the circle, as shown here:



You should take care positioning points while sketching because this positioning can affect the quality of your mesh. Points in the sketch become vertices of the part you are creating or modifying. In turn, when you mesh your model in the Mesh module, Abaqus/CAE converts these vertices into fully constrained seeds and places nodes at their location. For information on how to subsequently move vertices, see "Dragging Sketcher objects," Section 19.17.1.

19.10.9 Sketching fillets between two lines

Use the fillet tool from the Sketcher toolbox to draw fillets between two lines or circles. Enter the radius of the fillet and select the two lines or circles as shown here:



"Construction geometry," Section 19.5.2, illustrates how you can create a fillet tangent to two construction circles.

You should take care positioning points while sketching because this positioning can affect the quality of your mesh. Points in the sketch become vertices of the part you are creating or modifying. In turn, when you mesh your model in the Mesh module, Abaqus/CAE converts these vertices into fully constrained seeds and places nodes at their location. For information on how to subsequently move vertices, see "Dragging Sketcher objects," Section 19.17.1.

If you create a fillet and subsequently move the selected lines or circles, Abaqus/CAE will move the fillet and maintain the tangency.

19.11 Creating construction geometry

This section describes each of the Sketcher tools used to create construction geometry. Construction geometry is used to help you create and align objects in your sketch and to define the axis of rotation for revolved solids and surfaces. The following topics are covered:

- "Creating a horizontal construction line," Section 19.11.1
- "Creating a vertical construction line," Section 19.11.2
- "Creating an oblique construction line," Section 19.11.3
- "Creating angled construction lines," Section 19.11.4
- "Creating a construction circle," Section 19.11.5
- "Setting sketch components as construction geometry," Section 19.11.6

19.15 Projecting edges onto a sketch

When you sketch the profile of a feature, you can create new edges by projecting existing edges from the part onto the sketch sheet. You can use projected edges to duplicate the shapes of existing features as they appear from the current sketch plane. To project part edges onto your sketch, select **Add⟶Edges** from the main menu bar.

To project edges:

1. From the Sketcher toolbox, select the **Project Edges** tool . For a diagram of the tools in the Sketcher toolbox, see "The Sketcher tools," Section 19.4.1.

   Abaqus/CAE displays prompts in the prompt area to guide you through the procedure.

2. If desired, toggle off **Constrain to background** in the prompt area to project the edges as independent objects.

   If projected edges are constrained to the background, their shape and position depend on the original feature edges. If projected edges are unconstrained, changes to the original feature do not affect them—you can edit the projected edges and vertices independently from the original feature.

3. Select the edges you want to project into the sketch plane.
4. Click mouse button 2 to indicate you have finished selecting edges.

   Abaqus/CAE creates the new edges in the sketch.

9.7.2 Using dimensions to control sketch geometry

Dimensions are a type of constraints that use numerical values to define sizes, angles, or distances in a sketch. Dimensions control sketch geometry by preventing changes to the dimensioned quantities. To change a dimensioned quantity, you must modify the associated dimension.

The following dimension types are available in the Sketcher:

- horizontal
- vertical
- oblique
- angular
- radial

The first three dimension types correspond to linear distances between two points (including the endpoints of a single line), two lines, or a point and a line. Angular dimensions always indicate the angle, in degrees, between two lines. Radial dimensions indicate the radius of a circle, arc, or fillet or the major and minor radii of an ellipse.

You can use the following techniques to add dimensions to a sketch:

Adding dimensions automatically

After you create a sketch, you can use the auto-dimension tool to add dimensions to the entire sketch or to a selected group of entities. Use the sketch options to control the dimension types that Abaqus/CAE can add automatically. For more information, see "Customizing the format and use of dimensions in the Sketcher," Section 19.9.9.

19.8.4 Modifying edges by trimming, extending, splitting, or merging

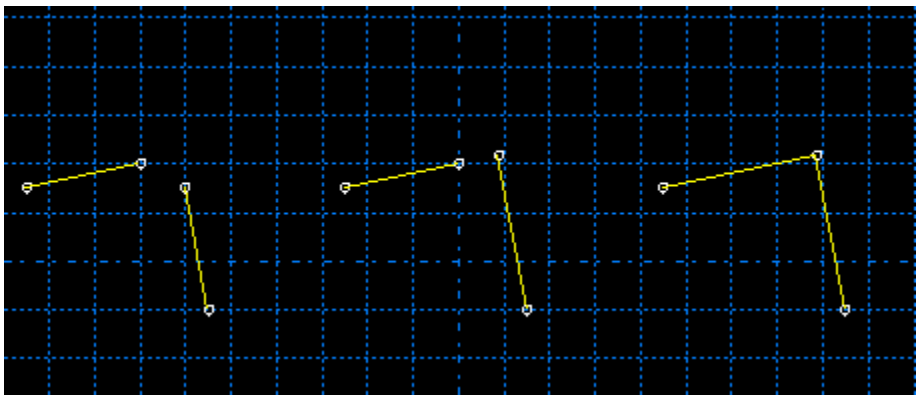
When you are modifying a sketch, you can implement your changes by modifying a dimension, by moving selected vertices, or by trimming, extending, splitting, or merging edges. Abaqus/CAE provides the following methods for trimming, extending, splitting, and merging edges:

Trim/Extend

You can trim or extend one end of a line or arc; you can also trim a spline curve, but spline curves cannot be extended. To trim or extend an edge, first select the edge near the

end that you want to modify and then select a second edge to define an intersection point. The second edge may be any object in the sketch, including construction geometry. The intersection point may lie beyond the current endpoints of either selected edge. Abaqus/CAE trims or extends the first edge at the intersection point. If you want to trim or extend the second edge to the same point, repeat the process, reversing the selection order (see Figure 19–16).

Figure 19–16 Extending two edges to create a corner.



For detailed instructions on trimming and extending edges, see "Modifying Sketcher objects by trimming or extending edges," Section 19.17.2.

9.12.6 Creating parametric equations (f(x))

Parametric equations are mathematical expressions relating different quantities in a sketch. The **Expression Builder** helps you create parametric equations using the parameters in your sketch. For example, if you want the width of a Sketcher object to be twice its length, you can associate the dimensions with parameters and use the **Expression Builder** to create the parametric equation $width = 2 * length$.

You can use the following operators to create parametric equations:

Mathematical operations:

+	Add.
–	Subtract.
*****	Multiply.
/	Divide.
1/A	Divide 1 by the parameter, value, or expression.

abs(A) Take the absolute value of the parameter, value, or expression.
sqrt(A) Take the square root of the parameter, value, or expression.

Trigonometric operations:

cos(A) Take the cosine of the parameter, value, or expression.
acos(A) Take the arccosine of the parameter, value, or expression.
cosh(A) Take the hyperbolic cosine of the parameter, value, or expression.
sin(A) Take the sine of the parameter, value, or expression.
asin(A) Take the arcsine of the parameter, value, or expression.
sinh(A) Take the hyperbolic sine of the parameter, value, or expression.
tan(A) Take the tangent of the parameter, value, or expression.
atan(A) Take the arctangent of the parameter, value, or expression.
tanh(A) Take the hyperbolic tangent of the parameter, value, or expression.

Logarithmic and exponential operations:

exp(A) Take the exponential of the parameter, value, or expression.
log(A) Take the natural log of the parameter, value, or expression.
log10(A) Take the base 10 log of the parameter, value, or expression.

To create an expression:

1. With an expression highlighted in the **Parameter Manager**, click **Expression Builder**.

   Abaqus/CAE opens the **Expression Builder** dialog box.

2. From the **Operators** list, select the desired operation.

   The operator appears within the expression window.

3. From the **Parameter Name** choices, click the name of the parameter on which to operate and click **Add to Expression**. You can choose from all parameters that appear above the parameter that you are defining in the **Parameter Manager**.

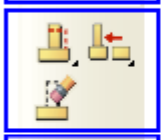
   The parameter name appears within the expression window.

4. Repeat Steps 2 and 3 as needed to complete the expression. In some cases you may need to reposition the cursor to correctly place the next portion of the expression.
5. When you are finished, click **OK** to save your changes and to close the dialog box.

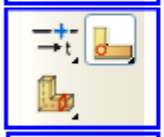
   Abaqus/CAE updates the expression and the current value in the **Parameter Manager**.

Chapter 2 – Part Module

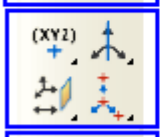
Managing Part



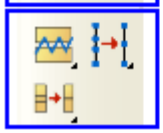
Edit Part



Partition



Datum

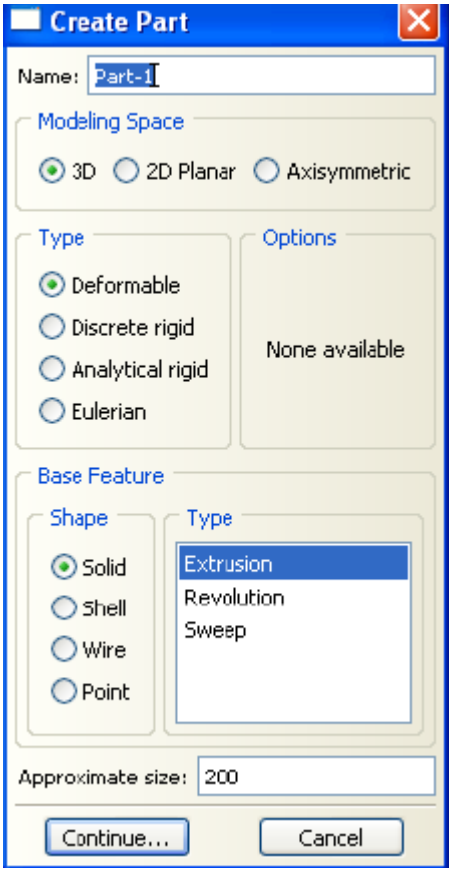


Geometry Edit

Create Part

When you create a part, you first use the **Create Part** dialog box to define the properties of the part, and then you use the **Sketch** to sketch the two-dimensional profile of the base feature. You use the **Create Part** dialog box to define the following:

Name



Use the **Name** text field at the top of the **Create Part** dialog box to name the part you are creating. To rename a part, select **Part→Rename** from the main menu bar. For information on valid names, see "Using basic dialog box components," Section 3.2.1.

After you create a part, Abaqus/CAE displays the name of the new part in title bar of the current viewport.

Modeling Space

Use the **Modeling Space** radio buttons to choose the modeling space of the new part. You can define a part to be either three-dimensional, two-dimensional (planar), or axisymmetric. If you create an axisymmetric deformable part, you can toggle on **Include twist**in the **Create Part** dialog box to include a twist degree of freedom in your model. You can change the modeling space of a part after you have created it by clicking mouse button 3 on the part in the Model Tree and selecting **Edit** from the menu that appears. For more information, see "Choosing the modeling space of a new part," Section 11.19.2.

Type

Use the **Type** radio buttons to choose the type of the new part. You can define a part to be either deformable, discrete rigid, analytical rigid, or Eulerian. You can change the type of a part after you have created it by clicking mouse button 3 on the part in the Model Tree and selecting **Edit** from the menu that appears. For more information, see "Choosing the type of a new part," Section 11.19.3.

The new part's type can be set to one of the following:

Deformable

Any arbitrarily shaped axisymmetric, two-dimensional, or three-dimensional part that you can create or import can be specified as a deformable part. A deformable part represents a part that can deform under load; the load can be mechanical, thermal, or electrical. By default, Abaqus/CAE creates parts that are deformable.

Discrete rigid

A discrete rigid part is similar to a deformable part in that it can be any arbitrary shape. However, a discrete rigid part is assumed to be rigid and is used in contact analyses to model bodies that cannot deform.

Analytical rigid

An analytical rigid part is similar to a discrete rigid part in that it is used to represent a rigid surface in a contact analysis. However, the shape of an analytical rigid part is not arbitrary and must be formed from a set of sketched lines, arcs, and parabolas.

Eulerian

Eulerian parts are used to define a domain in which material can flow for an Eulerian analysis. Eulerian parts do not deform during an analysis; instead, the material within the part deforms under load and can flow across the rigid element boundaries. For more information about Eulerian analyses, see Chapter 27, "Eulerian analyses."

Base Feature

Use the **Base Feature** field to define the shape and the type of the new part's base feature. The shape and the type options that Abaqus/CAE displays depend on the part's modeling space and type. You cannot change the type of a part's base feature after you create it. For more information, see "Choosing the base feature of a new part," Section 11.19.4.

Approximate size

Use the approximate size text field to enter the size of the part. The size that you enter is used by Abaqus/CAE to calculate the size of the Sketcher sheet and the spacing of its grid. For more information, see "Setting the approximate size of the new part," Section 11.19.5. After you create the part and start sketching its profile, you can use

the **Sketch** customization options to increase the sheet size. To display the Sketcher customization options click the tool at the bottom of the Sketcher toolbox.

11.3.2 The base feature

The first feature you create while building a part is called the base feature; you construct the remainder of the part by adding more features that either modify or add detail to the base feature. This process of building an Abaqus/CAE native part using the tools in the Part module follows a sequence of operations analogous to building a part in a machine shop. For example, you start with a piece of billet stock (the base feature) and then you do the following:

- Attach additional pieces to the billet (apply a solid extrusion, a revolved shell, or a sketched wire).
- Cut away the billet (apply an extruded cut, a revolved cut, or a circular hole; or round or chamfer an edge).

When you create a new part, you must describe the base feature. You do this by specifying two properties of the base feature: its shape and type. The shape indicates the basic topology of the feature; that is, whether it is a solid, shell, wire, or point. The type indicates which of the following methods will be used to generate the base feature:

Planar

You sketch the feature on a two-dimensional sketch plane.

Extrusion

You sketch the feature profile and then extrude it through a specified distance.

Revolution

You sketch the feature profile and then revolve it by a specified angle about an axis.

Sweep

You sketch two shapes: a sweep path and a sweep profile. The profile is then swept along the path to create the feature.

Coordinates

You enter the coordinates of a single point in the prompt area.

Before you create a part and choose the shape and the type of the base feature, you should know the sequence you will use to construct the desired part. Choosing the correct type and shape of the base feature is important.

Table 11–1 shows the base features that you can select based on the part's modeling space and type.

Table 11–1 Choosing the base feature.

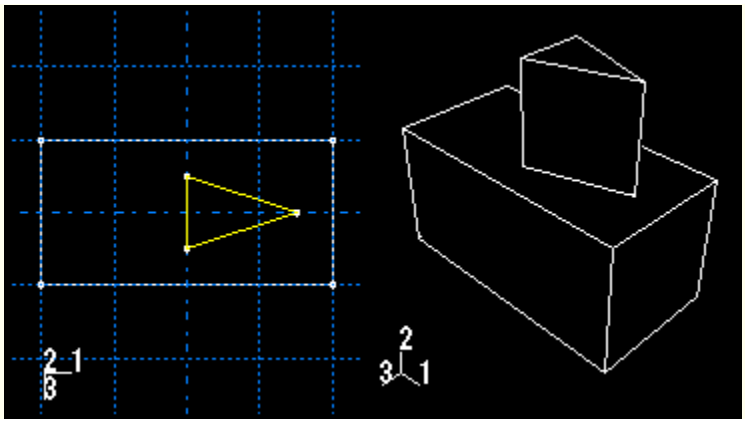
Part Type	Modeling Space	
	Three-dimensional	**Two-dimensional or Axisymmetric**
Deformable	Any	Planar shell, planar wire, or point
Discrete rigid	Any (you must convert a 3-D solid discrete rigid part to a shell before you instance it)	Planar wire or point
Analytical rigid	Extruded or revolved shell	Planar wire
Eulerian	Extruded, revolved, or swept solid	Not applicable

A part imported from a file containing third-party format geometry consists of a single feature that you import into Abaqus/CAE as the base feature of a new part. You cannot modify this base feature, but you can add additional features to it. Similarly, an orphan mesh part is created in the Mesh module or imported from an output database as the base feature of a new part. You can use the mesh editing tools to add and delete nodes and elements from an orphan mesh part.

11.21.1 Adding an extruded solid feature

Select **Shape→Solid→Extrude** from the main menu bar to add an extruded solid feature to the part in the current viewport. You can add an extruded solid feature only to three-dimensional parts.

You add an extruded solid feature by sketching a two-dimensional cross-section and defining the distance over which to extrude it. A sketch and the resulting extruded solid feature are illustrated in the following figure:



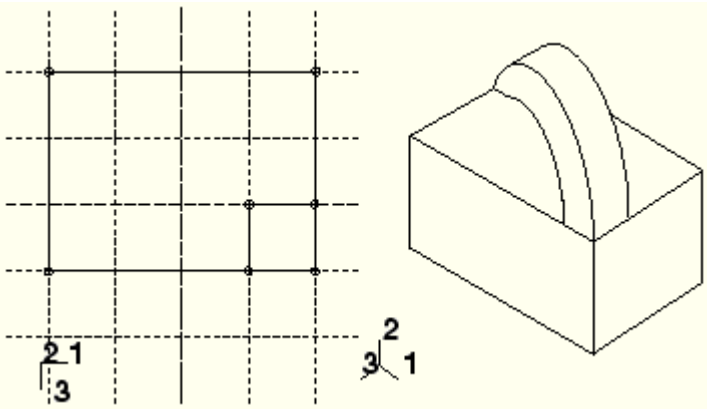
You can also define the distance over which to extrude by selecting a single face to extrude to. Abaqus/CAE extrudes the sketch until it meets the selected face.

In addition, you can select a center point and specify a pitch that Abaqus/CAE uses to twist the cross-section as it is extruded. Alternatively, Abaqus/CAE can expand or contract the cross-section along a specified draft angle as the cross-section is extruded. For more information, see "Including twist in an extrusion," Section 11.13.3, and "Including draft in an extrusion," Section 11.13.4.

11.21.2 Adding a revolved solid feature

Select **Shape→Solid→Revolve** from the main menu bar to add a revolved solid feature to the part in the current viewport. You can add a revolved solid feature only to three-dimensional parts.

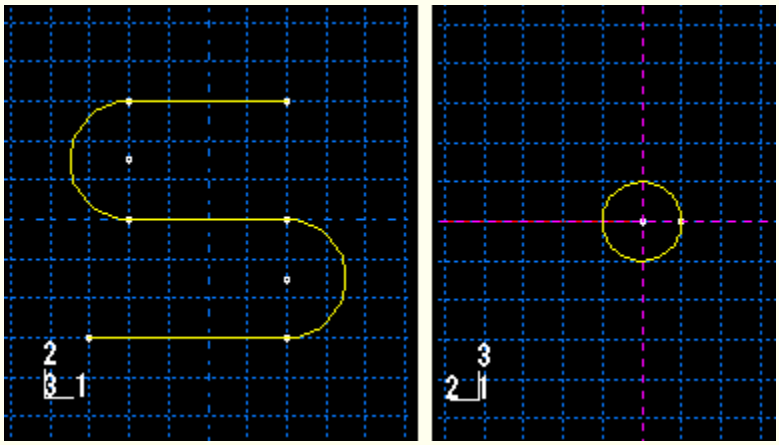
You add a revolved solid feature by sketching a two-dimensional cross-section and a construction line on a selected face. The construction line serves as an axis of revolution, and Abaqus/CAE creates the solid feature by rotating the cross-section about the axis using a specified angle of revolution. In addition, you can specify a pitch and a direction along the axis of revolution that Abaqus/CAE uses to translate the sketch along the axis of revolution as it revolves the profile. A sketch and the resulting feature, revolved through an angle of 180° with pitch, are illustrated in the following figure:

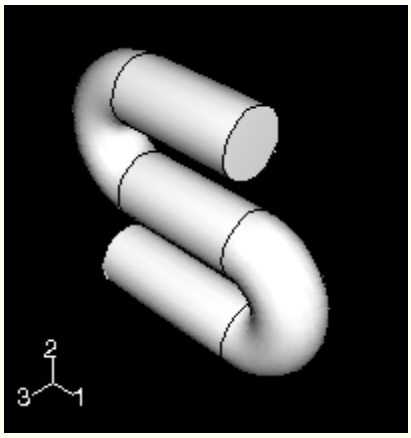


11.21.3 Adding a swept solid feature

Select **Shape→Solid→Sweep** from the main menu bar to add a swept solid feature to the part in the current viewport. You can add a swept solid feature only to three-dimensional parts.

You add a swept solid feature by sketching a sweep path on a selected face and sketching a sweep profile. The sweep profile is always perpendicular to the beginning of the path, and the profile always remains normal to the path as it is swept along its length. The sweep path, the sweep profile, and the resulting solid feature are illustrated in the following figures:



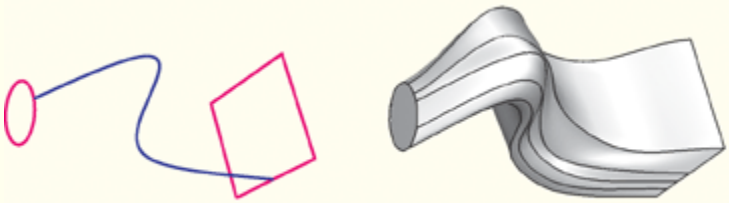


The sketch of the sweep path and the sketch of the sweep profile define the swept solid feature; both can be modified using theFeature Manipulation toolset. In addition, you can toggle on **Keep internal boundaries** in the Feature Manipulation toolset to maintain any faces or edges that are generated between the swept solid feature and the existing part. The internal boundaries may create regions that can be structured or swept meshed without having to resort to partitioning.

1.21.4 Adding a solid loft feature

Select **Shape⟶Solid⟶Loft** from the main menu bar to add a solid loft feature to the part in the current viewport. You can add a solid loft feature only to three-dimensional parts.

You add a solid loft feature by creating two or more sections from selected edges and by defining one or more loft paths. Loft sections, a loft path, and the resulting solid loft feature are illustrated in the following figure:



You can allow Abaqus/CAE to define a single loft path using a smooth path to connect the center of each loft section. If you allow Abaqus/CAE to define the path, you can apply tangency methods to the start and end sections of the loft. The curve and tangencies

define the path of the loft feature between sections. Alternatively, you can define one or more loft paths by selecting curves that connect a point on each loft section to a point on the next loft section. Each loft path must provide a continuous line connecting each consecutive loft section. If a loft path is not smooth (if there is more than one tangent to any point along the path), Abaqus/CAE will display an error message when you try to create the loft. For more information about loft sections, loft paths, and loft tangencies, see "What is lofting?," Section 11.14.

Note: You do not use the Sketcher while adding a loft feature. As a result, all of the edges that define the loft sections and the loft paths must exist in the part geometry before you create the loft. To create a loft path or to create a nonplanar loft section, you can use the tool, located with the wire tools in the Part module toolbox (see"Adding a spline wire feature," Section 11.23.3, for more information).

59.5 An overview of datum creation techniques

This section provides an overview of the methods for creating each type of datum. The following topics are covered:

- "An overview of the methods for creating a datum point," Section 59.5.1
- "An overview of the methods for creating a datum axis," Section 59.5.2
- "An overview of the methods for creating a datum plane," Section 59.5.3
- "An overview of the methods for creating a datum coordinate system," Section 59.5.4

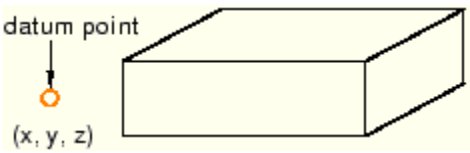
59.5.1 An overview of the methods for creating a datum point

When you choose **Point** from the **Create Datum** dialog box, the **Method** list displays the following methods for creating a datum point:

Enter coordinates

Enter the *X*-, *Y*-, and *Z*-coordinates of the datum point, as shown in Figure 59–6. For detailed instructions, see "Creating a datum point by entering its coordinates," Section 59.6.1.

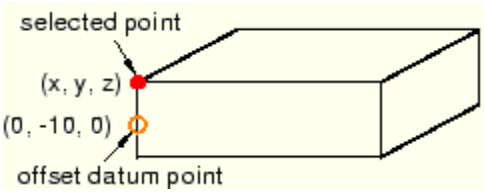
Figure 59–6 Creating a datum point by entering coordinates.



Offset from point

Enter the location of the datum point in the form of the *X*-, *Y*-, and *Z*-coordinates of an offset from a selected point, as shown in Figure 59–7. For detailed instructions, see "Creating a datum point at an offset from a selected point," Section 59.6.2.

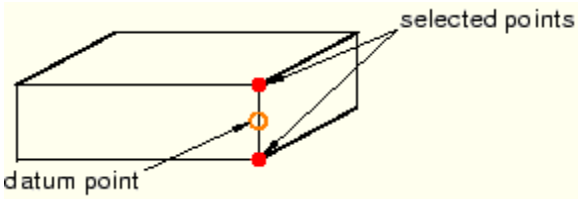
Figure 59–7 Creating a datum point by offsetting from a point.



Midway between 2 points

Select two points on the model; Abaqus/CAE creates the datum point midway between the two selected points, as shown in Figure 59–8. For detailed instructions, see "Creating a datum point midway between two points," Section 59.6.3.

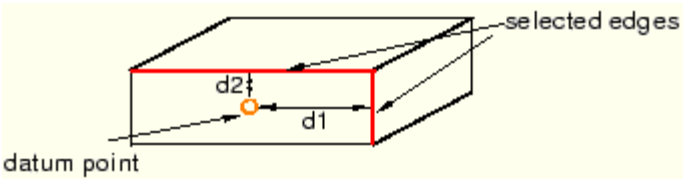
Figure 59–8 Creating a datum point by selecting two end points.



Offset from 2 edges

Select two edges on the model; and enter the distance from the datum point to each edge, as shown in Figure 59–9. For detailed instructions, see "Creating a datum point at a specified distance from two edges," Section 59.6.4.

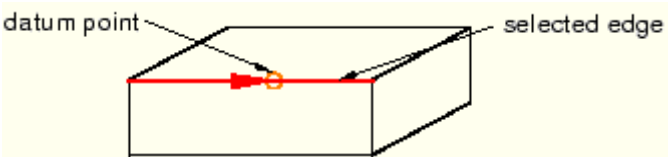
Figure 59–9 Positioning a datum point a specified distance from two edges.



Enter parameter

Select an edge on the model, and enter the location of the datum point in the form of a parameter value that represents a percentage of the edge length. An arrow along the edge indicates 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), as shown in Figure 59–10. For detailed instructions, see "Creating a datum point by entering an edge parameter," Section 59.6.5.

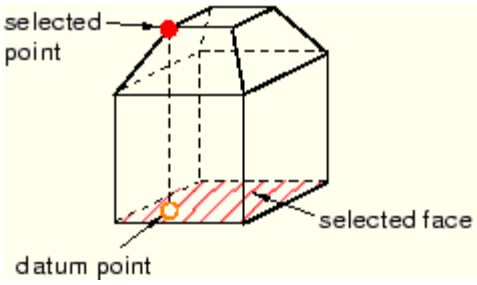
Figure 59–10 Positioning a datum point a specified distance along an edge.



Project point on face

Select a point and a plane on which to project the point. Abaqus/CAE creates the datum point where the plane intersects a line that is normal to it and passing through the selected point, as shown in Figure 59–11. The datum point also marks the shortest distance between the selected point and the selected face. For detailed instructions, see "Creating a datum point by projecting a point on a face," Section 59.6.6.

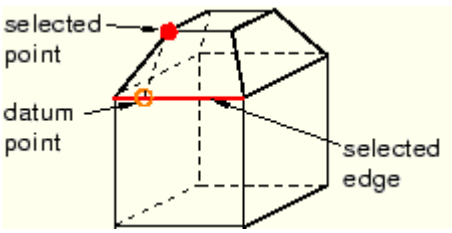
Figure 59–11 Creating a datum point by projecting a point onto a face.



Project point on line

Select a point on the model and an edge on which to project the point. Abaqus/CAE creates the datum point where the edge intersects a line that is normal to it and passing through the selected point, as shown in Figure 59–12. The datum point also marks the shortest distance between the selected point and the selected edge. For detailed instructions, see "Creating a datum point by projecting a point on a line," Section 59.6.7.

Figure 59–12 Creating a datum point by projecting a point onto an edge.



For information on related topics, click any of the following items:

- "Creating datum points," Section 59.6
- "Controlling datum display," Section 72.7

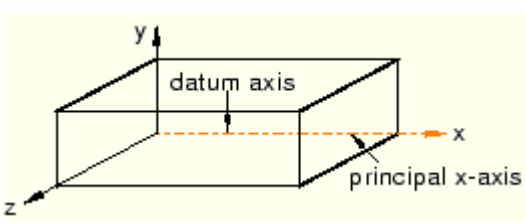
59.5.2 An overview of the methods for creating a datum axis

When you choose **Axis** from the **Create Datum** dialog box, the **Method** list displays the following methods for creating a datum axis:

Principal axis

Select one of the three principal axes with which the datum axis must be colinear, as shown in Figure 59–13. For detailed instructions, see "Creating a datum axis along a principal axis," Section 59.7.1.

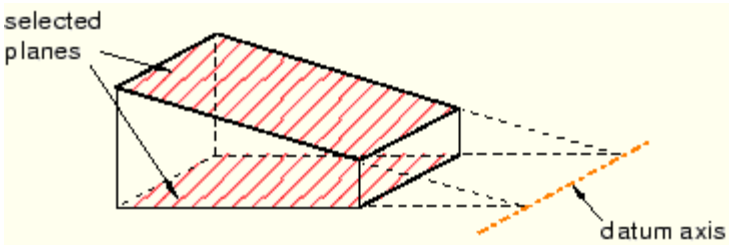
Figure 59–13 Defining a datum axis as one of the three principal axes.



Intersection of 2 planes

Select two non-parallel planar surfaces. Abaqus/CAE creates the datum axis where the two planes (or extensions of the two planes) intersect, as shown in Figure 59–14. For detailed instructions, see "Creating a datum axis along the intersection of two planes," Section 59.7.2.

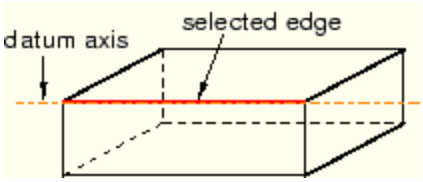
Figure 59–14 Defining a datum axis as the intersection of two planes.



Straight edge

Select a straight edge on the model with which the datum axis must be colinear, as shown in Figure 59–15. For detailed instructions, see "Creating a datum axis along a straight edge," Section 59.7.3.

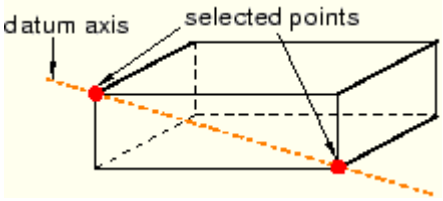
Figure 59–15 Defining a datum axis as a straight edge on the model.



2 points

Select any two points on the model through which the datum axis must pass, as shown in Figure 59–16. For detailed instructions, see"Creating a datum axis through two points," Section 59.7.4.

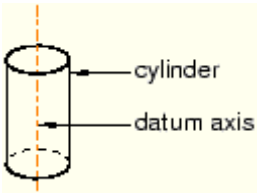
Figure 59–16 Defining a datum axis by selecting two points.



Axis of cylinder

Select a cylindrical face on the model. Abaqus/CAE creates a datum axis that lies along the axis of the cylindrical face, as shown inFigure 59–17. For detailed instructions, see "Creating a datum axis along the axis of a cylinder," Section 59.7.5.

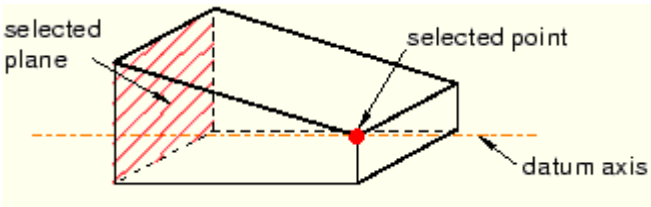
Figure 59–17 Defining a datum axis as the axis of a cylinder.



Normal to plane, thru point

Select a plane and a point that is not on the plane. Abaqus/CAE creates a datum axis that is normal to the plane and passes through the point, as shown in Figure 59–18. For detailed instructions, see "Creating a datum axis normal to a plane and passing through a point,"Section 59.7.6.

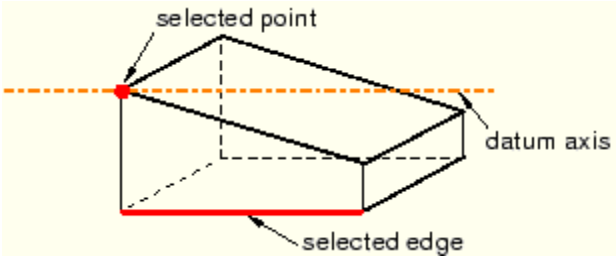
Figure 59–18 Defining a datum axis by selecting a point and a plane.



Parallel to line, thru point

Select an edge of the model and a point outside the edge. Abaqus/CAE creates a datum axis that is parallel to the edge and passes through the point, as shown in Figure 59–19. For detailed instructions, see "Creating a datum axis parallel to a line and passing through a point," Section 59.7.7.

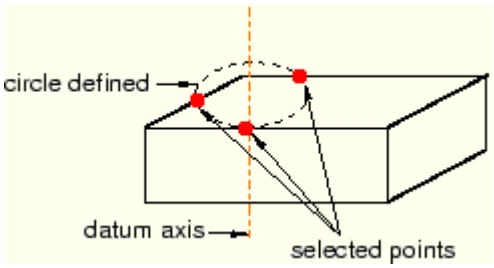
Figure 59–19 Defining a datum axis by selecting a point and an edge.



3 points on circle

Select three points on the model that define a circle. Abaqus/CAE creates a datum axis along the axis of the circle, as shown in Figure 59–20. For detailed instructions, see "Creating a datum axis running along the axis of a circle defined by three points," Section 59.7.8.

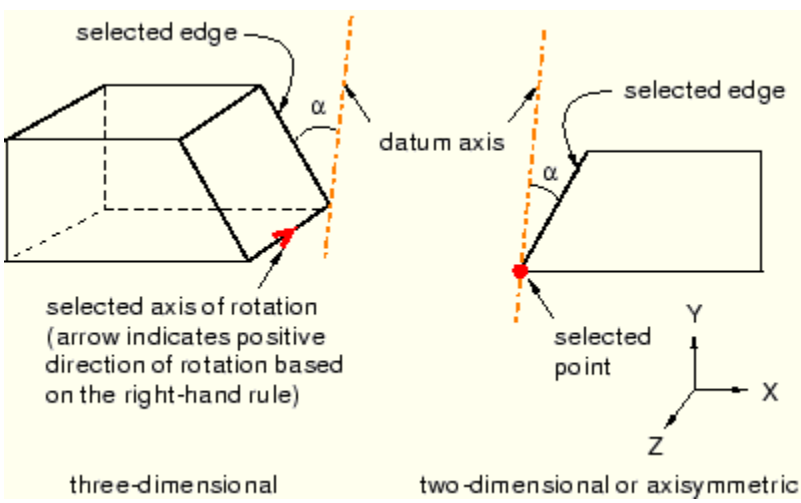
Figure 59–20 Defining a datum axis as the axis of a circle.



Rotate from line

Select an edge and an axis of rotation, and specify the angle through which the edge will be rotated. Abaqus/CAE creates a datum axis by rotating the edge about the axis through the specified angle, as shown in Figure 59–21. For detailed instructions, see "Creating a datum axis by rotating an existing edge through a specified angle," Section 59.7.9.

Figure 59–21 Defining a datum axis by rotating an edge through a specified angle.



For information on related topics, click any of the following items:

- "Creating datum axes," Section 59.7
- "Controlling datum display," Section 72.7

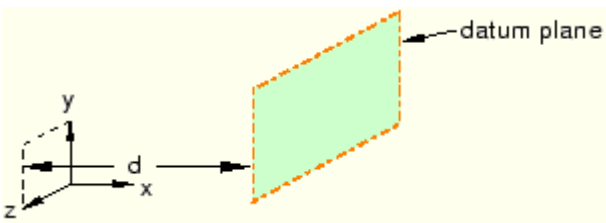
59.5.3 An overview of the methods for creating a datum plane

When you choose **Plane** from the **Create Datum** dialog box, the **Method** list displays the following methods for creating a datum plane:

Offset from principal plane

Select one of the three principal planes; and provide the location of the datum plane in the form of an offset from the selected plane, as shown in Figure 59–22. A positive value indicates an offset in the positive direction along the axis normal to the selected plane; for example, along the *X*-axis normal to the *Y*–*Z* plane. For detailed instructions, see "Creating a datum plane offset from a principal plane," Section 59.8.1.

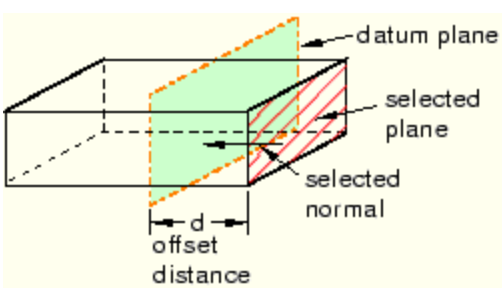
Figure 59–22 Creating a datum plane by offsetting from one of the three principal planes.



Offset from plane

Select any plane on the model; and provide the location of the datum plane by specifying the direction of the normal and an offset from the selected plane along the normal, as shown in Figure 59–23. You can specify the offset by entering a value or selecting a point. For detailed instructions, see "Creating a datum plane at an offset from a selected plane," Section 59.8.2.

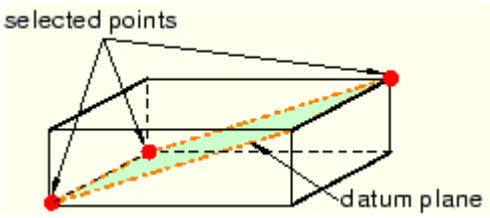
Figure 59–23 Creating a datum plane by offsetting from any plane.



3 points

Select three points through which the datum plane must pass, as shown in Figure 59–24. For detailed instructions, see "Creating a datum plane passing through three points," Section 59.8.3.

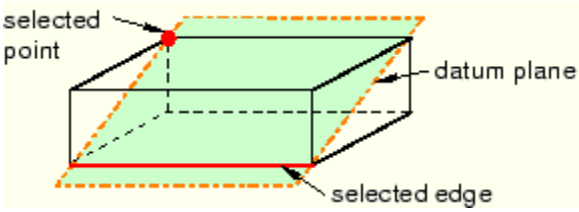
Figure 59–24 Creating a datum plane by selecting three points.



Line and point

Select an edge and a point through which the datum plane must pass, as shown in Figure 59–25. For detailed instructions, see"Creating a datum plane through a line and a point," Section 59.8.4.

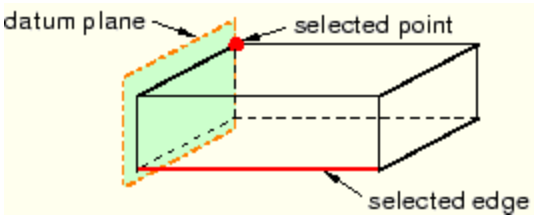
Figure 59–25 Creating a datum plane by selecting a point and an edge.



Point and normal

Select a point and an edge; the datum plane passes through the point and is normal to the selected edge, as shown in Figure 59–26. For detailed instructions, see "Creating a datum plane passing through a point and normal to an edge," Section 59.8.5.

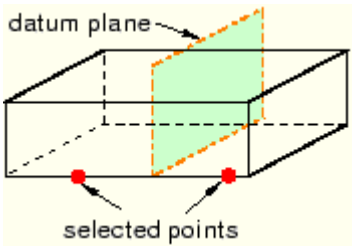
Figure 59–26 Creating a datum plane by selecting a point and a normal edge.



Midway between 2 points

Select two points. Abaqus/CAE creates a datum plane midway between the two selected points and normal to the line connecting them, as shown in Figure 59–27. For detailed instructions, see "Create a datum plane midway between two points and normal to the line connecting the two points," Section 59.8.6.

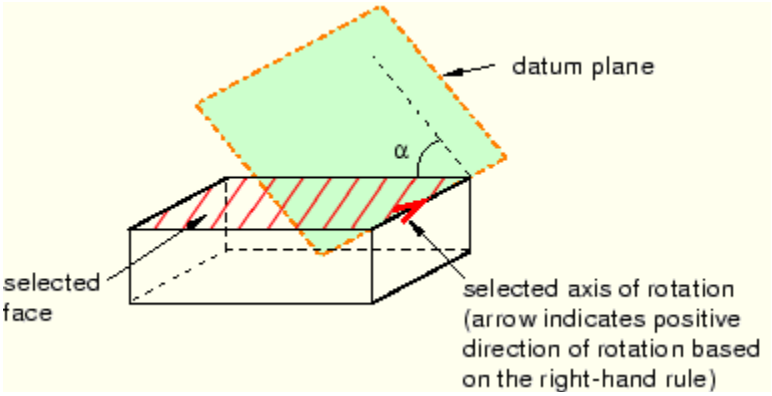
Figure 59–27 Positioning a datum plane midway between two points.



Rotate from plane

Select a face and an axis of rotation, and specify the angle through which the face will be rotated. Abaqus/CAE creates a datum plane by rotating the face about the axis through the specified angle, as shown in Figure 59–28. For detailed instructions, see "Creating a datum plane by rotating an existing face through a specified angle," Section 59.8.7.

Figure 59–28 Defining a datum plane by rotating a face through a specified angle.



For information on related topics, click the following item:

- "Creating datum planes," Section 59.8

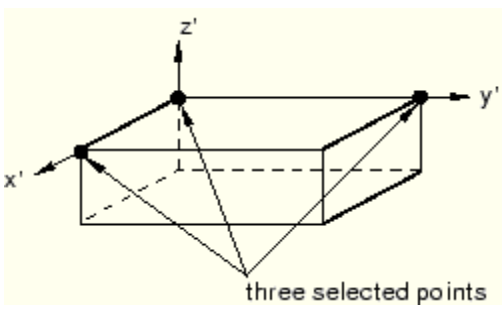
59.5.4 An overview of the methods for creating a datum coordinate system

Datum coordinates systems are used throughout Abaqus/CAE; for example, to define material orientations and to define connector orientations. To help you keep track of your datum coordinate systems, you can name the systems when you create them, and the name appears alongside their entry in the Model Tree. When you choose **CSYS** from the **Create Datum** dialog box, the **Method** list displays the following methods for creating a datum coordinate system:

3 points

Define a rectangular, cylindrical, or spherical coordinate system by selecting the origin and, optionally, two additional points. In the case of a rectangular system, the second point defines the *X*-axis, and the *X*–*Y* plane passes through the second and third points, as shown in Figure 59–29. This is the most versatile tool for creating a datum coordinate system, and you should use it when possible. For detailed instructions, see "Creating a datum coordinate system defined by three points," Section 59.9.1.

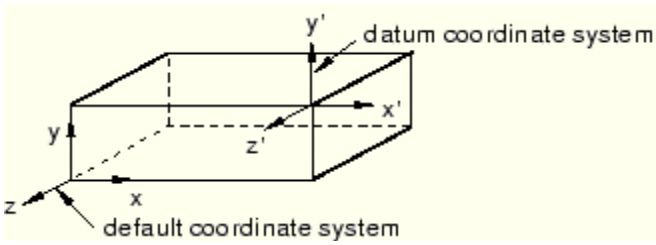
Figure 59–29 Positioning a rectangular datum coordinate system by selecting the origin and two points.



Offset from CSYS

Select a coordinate system; and provide the location of the rectangular, cylindrical, or spherical datum coordinate system by specifying an offset, as shown by the example involving a rectangular system in Figure 59–30. You can specify the offset by entering a value or by selecting a point. For detailed instructions, see "Creating a datum coordinate system at an offset from another coordinate system,"Section 59.9.2.

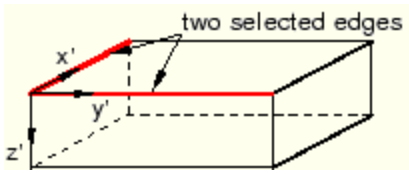
Figure 59–30 Positioning a rectangular datum coordinate system by offsetting from another coordinate system.



2 lines

Select two edges that define the rectangular, cylindrical, or spherical coordinate system. In the case of a rectangular system, the first edge defines the *X*-axis and the *X*–*Y* plane passes through the second edge, as shown in Figure 59–31. For detailed instructions, see"Creating a datum coordinate system defined by two lines," Section 59.9.3.

Figure 59–31 Positioning a rectangular datum coordinate system by selecting two edges.



For information on related topics, click any of the following items:

- "Creating datum planes," Section 59.8
- "Controlling datum display," Section 72.7

67.4 An overview of partitioning techniques

This section provides an overview of the different partitioning techniques. The following topics are covered:

- "An overview of the methods for partitioning edges," Section 67.4.1
- "An overview of the methods for partitioning faces," Section 67.4.2
- "An overview of the methods for partitioning cells," Section 67.4.3

67.4.1 An overview of the methods for partitioning edges

Select **Tools→Partition** from the main menu bar to display the **Create Partition** dialog box. When you choose **Edge** from the **Create Partition** dialog box, the **Method** list displays the following methods for partitioning edges:

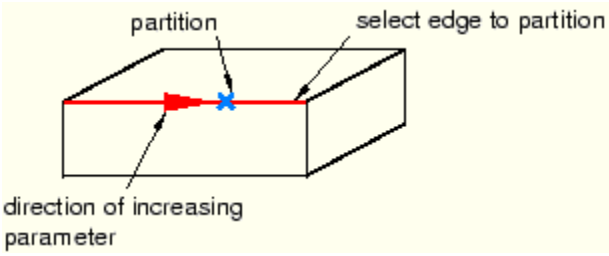
Specify parameter by location

Pick a point anywhere along the edge. For detailed instructions, see "Using the specify parameter by location method to partition edges," Section 67.5.1.

Enter parameter

Enter a parameter in the prompt area, as shown in Figure 67–5. An arrow along the edge indicates 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 detailed instructions, see "Using the enter parameter method to partition edges," Section 67.5.2.

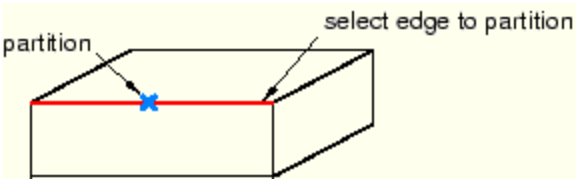
Figure 67–5 Entering a parameter to partition an edge.



Select midpoint/datum point

Select the midpoint of the edge or a datum point along the edge, as shown in Figure 67–6. For detailed instructions, see "Using the pick midpoint/datum point method to partition an edge," Section 67.5.3.

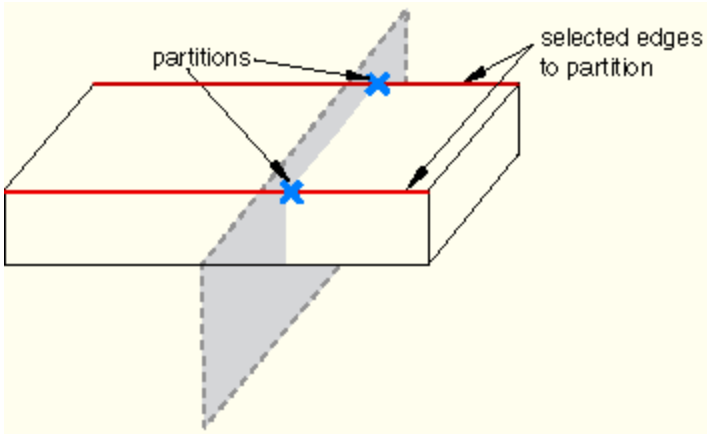
Figure 67–6 Selecting the midpoint or a datum to partition an edge.



Use datum plane

Select a datum plane. Abaqus/CAE creates the partition where the datum plane intersects the edges, as shown in Figure 67–7. Partitioning a group of selected edges with a datum plane is a useful technique for aligning a group of partitions. For detailed instructions, see "Using the datum plane method to partition edges," Section 67.5.4.

Figure 67–7 Selecting a datum plane to partition edges.



For information on related topics, click any of the following items:

- "An overview of partitioning techniques," Section 67.4
- "Partitioning edges," Section 67.5
- "Understanding partitions," Section 67.3

67.4.2 An overview of the methods for partitioning faces

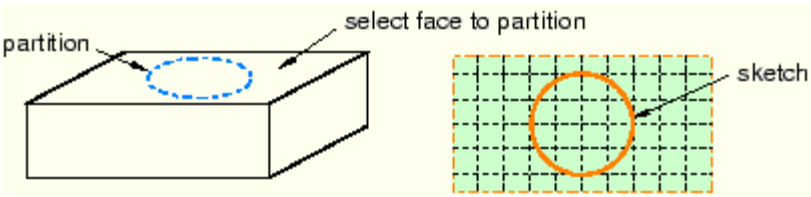
When you choose **Face** from the **Create Partition** dialog box, the **Method** list displays the following methods for partitioning faces:

Sketch planar partition

Partition a selected face by sketching a partition with the Sketcher, as shown in Figure 67–8. For detailed instructions, see "Using the sketch method to partition faces," Section 67.6.1.

You can sketch directly on the face to be partitioned, or you can sketch on a second face or datum plane and then project the sketch onto the face that you want to partition. For an example of projecting a sketch from a datum plane, see "Using the Datum toolset in thePart module," Section 11.16.1.

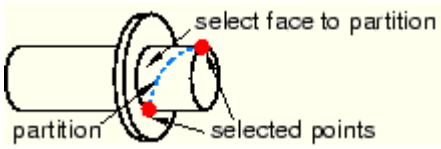
Figure 67–8 Partitioning a face using the Sketcher.



Shortest path between 2 points

Partition the face along the shortest path connecting two selected points; the resulting partition will be curved if the face being partitioned is curved, as shown in Figure 67–9. You can select points that are not associated with the face being partitioned; for example, the points can be located on a different face or even a different part instance. For detailed instructions, see "Using the shortest path method to partition faces," Section 67.6.2.

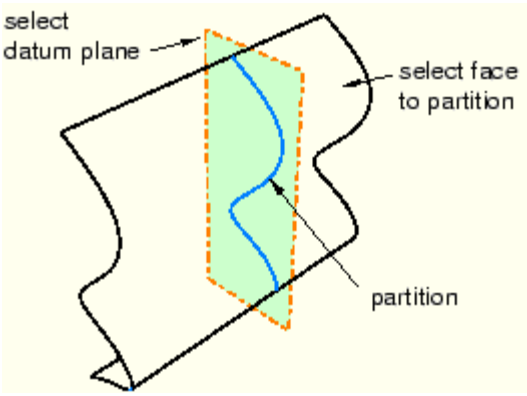
Figure 67–9 Partitioning a face using the shortest path between two points.



Use datum plane

Partition a face using the intersection with the extension of a datum plane, as shown in Figure 67–10. For detailed instructions, see"Using the datum plane method to partition faces," Section 67.6.3.

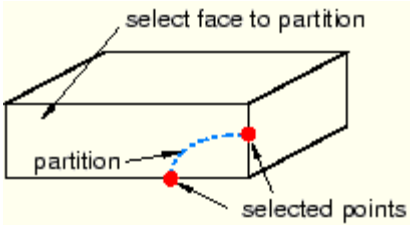
Figure 67–10 Partitioning a face using a datum plane.



Curved path normal to 2 edges

Partition the face along a Bézier curve that is normal to two of the face's edges, as shown in Figure 67–11. Position the curve by selecting two points anywhere along the two edges. The arc subtended by the two edges must be less than 180°. For detailed instructions, see "Using the curved path method to partition a face," Section 67.6.4.

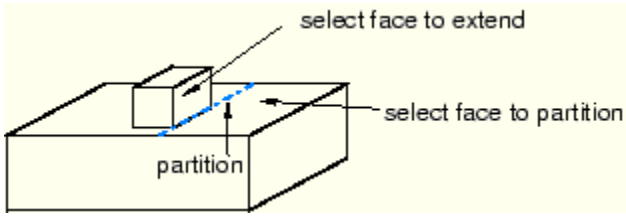
Figure 67–11 Partitioning a face using a Bézier curve.



Extend another face

Partition the face using the intersection with the extension of another face, as shown in Figure 67–12. The face being extended can be either planar, cylindrical, conical, or spherical; and it need not belong to the part containing the face to be partitioned. For detailed instructions, see "Using the extended face method to partition faces," Section 67.6.5.

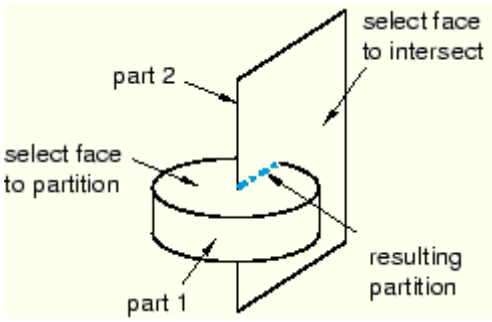
Figure 67–12 Partitioning a face using the extension of another face.



Intersect by other faces

Partition the face using the intersection of the target face with one or more other faces, as shown in Figure 67–13. The faces can be intersecting or tangential. For detailed instructions, see "Using the intersection method to partition faces," Section 67.6.6.

Figure 67–13 Partitioning a face using an intersection of faces.



Auto-partition

When you mesh a face with quadrilateral elements using the free meshing technique, the Mesh module internally partitions the face into regions with three to five logical sides before meshing the face. For more information, see "Free meshing with quadrilateral and quadrilateral-dominated elements," Section 17.10.2. However, if you want to view and perhaps modify the automatically generated regions before generating the mesh, you can use the auto-partitioning tool to partition the face without meshing it. This tool is available only in the Mesh module. For detailed instructions, see "Using the automatic generation method to partition faces," Section 67.6.7.

For information on related topics, click any of the following items:

- "An overview of partitioning techniques," Section 67.4
- "Understanding partitions," Section 67.3
- "Partitioning faces," Section 67.6

67.4.3 An overview of the methods for partitioning cells

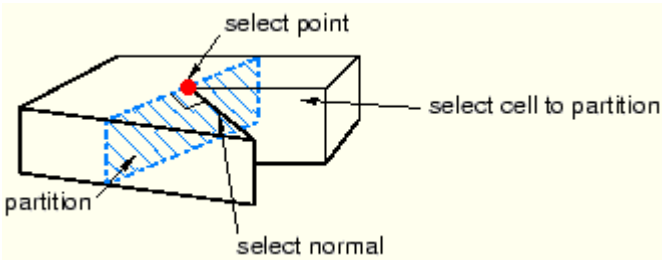
When you choose **Cell** from the **Create Partition** dialog box, the **Method** list displays the following methods for partitioning cells:

Define cutting plane

Partition a cell by cutting it with a plane; the plane will pass completely through the cell. Use one of the following three methods to define the cutting plane:

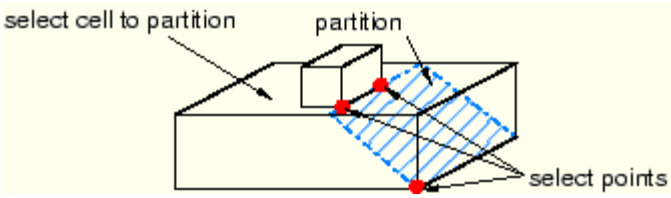
- Select a point on the cutting plane; then pick an edge or datum axis that defines the normal to this plane, as shown in Figure 67–14.

  Figure 67–14 Defining the cutting plane with a point and a normal.



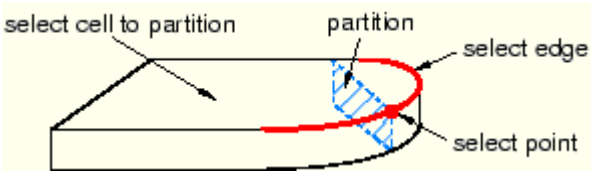
- Select three distinct and noncolinear points, as shown in Figure 67–15.

  Figure 67–15 Defining the cutting plane with three points.



- Select an edge and a point along the edge; the cutting plane will be normal to the edge at the selected point, as shown in Figure 67–16.

  Figure 67–16 Defining the cutting plane with an edge and a point.

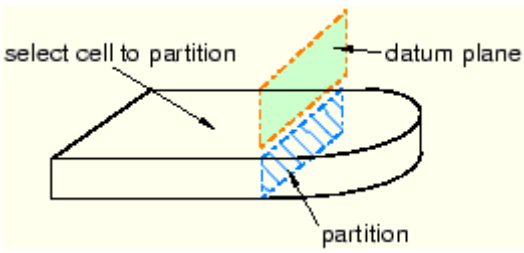


For detailed instructions, see "Using the cutting plane method to partition cells," Section 67.7.1.

Use datum plane

Partition a cell using the intersection with the extension of a datum plane, as shown in Figure 67–17. For detailed instructions, see"Using the datum plane method to partition cells," Section 67.7.2.

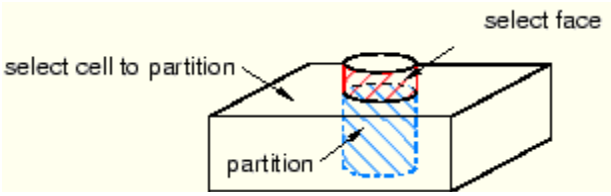
Figure 67–17 Partitioning a cell using a datum plane.



Extend face

Partition a cell by cutting it with a shell, where the shell is the extended geometry of a face, as shown in Figure 67–18. The face being extended can be planar, cylindrical, conical, or spherical. For detailed instructions, see "Using the extended face method to partition cells," Section 67.7.3.

Figure 67–18 Partitioning a cell using an extension of a face.



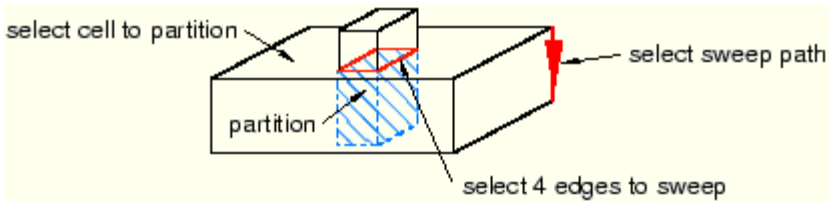
Extrude/Sweep edges

Partition a cell by sweeping selected edges (that form the sweep profile) along a selected path (known as the sweep path). You can select any number of edges to be swept, although all the edges must be connected, must lie on the same plane, and must belong to the same part instance.

Use either of the following two methods to define the sweep path:

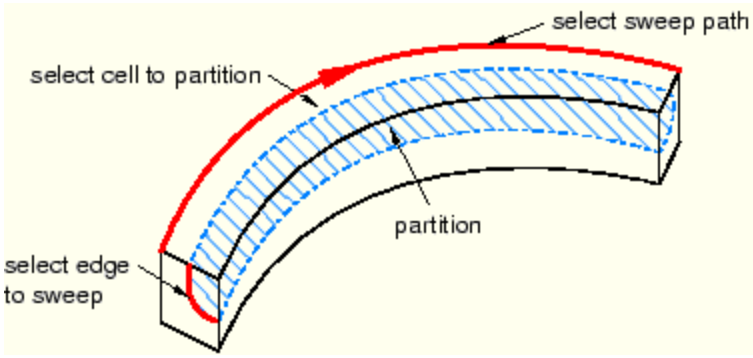
- Create a straight partition through the cell by extending the sweep profile infinitely in a direction parallel to a selected straight edge or datum axis that acts as a sweep path; the partition is created where the swept edge(s) pass through the selected cell, as shown in Figure 67–19. The sweep path must be straight and perpendicular to the set of edges being swept.

  Figure 67–19 Sweeping a profile along a direction.



- Create a straight or curved partition through the cell by extending the sweep profile along or parallel to a selected edge. The partition extends only as far as the selected edge; and the partition is created where the swept edge(s) pass through the selected cell, as shown in Figure 67–20. The sweep path must begin in the plane containing the edges to be swept, and its tangent must be perpendicular to the same plane.

  Figure 67–20 Sweeping a profile along an edge.



For detailed instructions, see "Using the extrude/sweep method to partition cells," Section 67.7.4.

Use n-sided patch

Partition a cell by dividing it with a surface patch formed from a loop of connected edges. The edges can be curved or straight, must be connected, and must belong to the same part as the cell to be partitioned. In addition, the patch must pass completely through the cell. Choose from the following methods to define the patch:

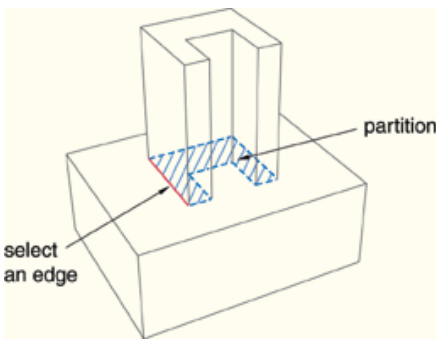
Select Edges

You can choose from the following methods to select the edges that form the N-sided patch:

Loop

Select a single edge, and allow Abaqus/CAE to search for a continuous loop of connected edges that will partition the cell, as shown in Figure 67–21. The resulting patch can have any number of edges.

Figure 67–21 Allowing Abaqus/CAE to define a patch after selecting an edge.



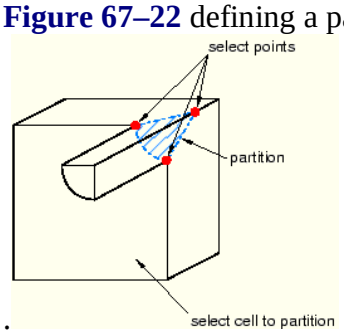
Edges

Manually select the edges that will partition the cell. You can select any number of edges, and the selected edges must form a closed loop.

Select Corner Points

Select three, four, or five points that define the corners of the patch. If two of the points are connected by an existing edge, the resulting partition will follow the curve of the edge, as shown in Figure 67–22. The points must be on the boundary edges of the cell being partitioned.

Figure 67–22 defining a patch with corner points

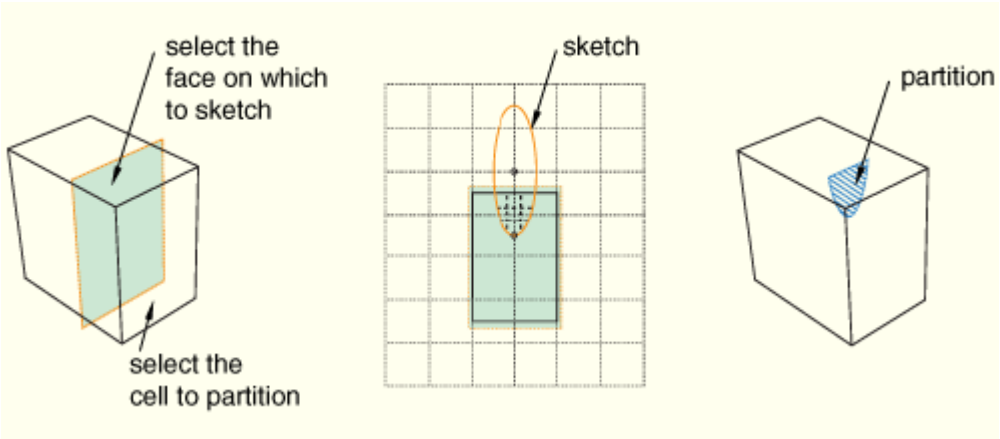


For detailed instructions, see "Using the N-sided patch method to partition a cell," Section 67.7.5.

Sketch planar partition

Partition a selected cell by sketching a partition with the Sketcher, as shown in Figure 67–23. In most cases you will sketch on a datum plane that intersects the selected cell. You can also select an existing face on which to sketch and draw the sketch outside the boundaries of the face. Abaqus/CAE creates the partition wherever the sketch intersects the cell.

Figure 67–23 Partitioning a cell using the Sketcher.



For detailed instructions, see "Using the sketch planar partition method to partition a cell," Section 67.7.6.

For information on related topics, click any of the following items:

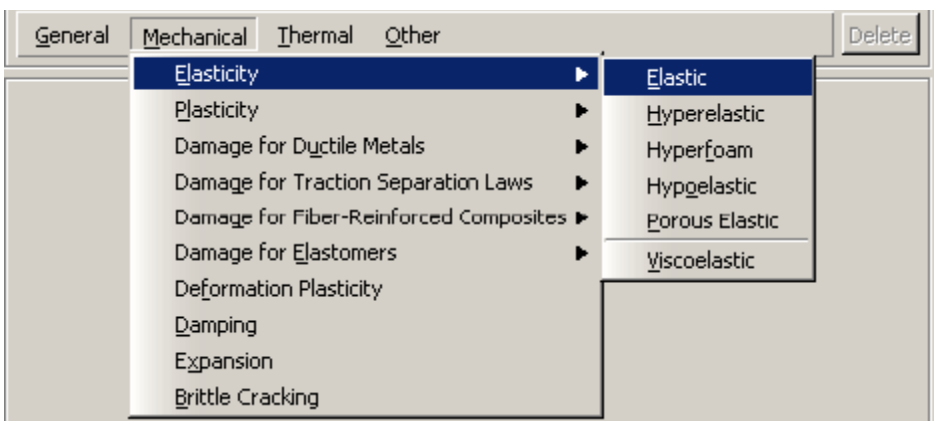
- "An overview of partitioning techniques," Section 67.4
- "Understanding partitions," Section 67.3
- "Partitioning cells," Section 67.7

Chapter 3 – Property Module

12.7.1 Creating or editing a material

You use the **Edit Material** dialog box to create a new material or edit an existing material. When you select **Material⟶Create** from the main menu bar, you can enter the name of your choice for the material or accept the default name, you can provide a description for the material, and you can define the material properties. When you select **Material⟶Edit**, you can redefine the material description or properties, but you must use **Material⟶Rename** to change the name of an existing material.

Use the menu bar under the **Material Behaviors** list to add properties to a material. Some of the menu items contain submenus; for example, the following figure shows the behaviors available under the **Mechanical⟶Elasticity** menu item:



Note: To display information on a particular material behavior, click and hold that behavior and then press **F1**. A help window appears that contains information about the parameters and data associated with the behavior.

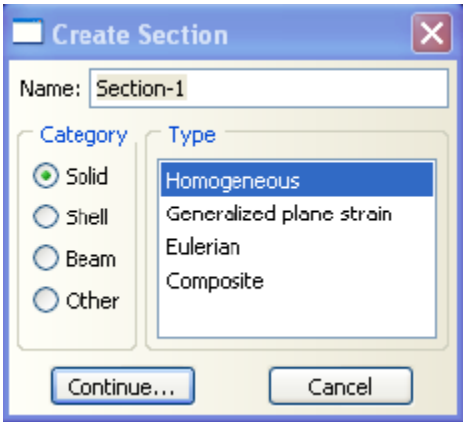
Use the **Material Behaviors** list to select an existing material behavior to edit.

Caution: Abaqus/CAE does not check for missing or invalid material behaviors until you submit the job for analysis. (Any warnings and errors are reported by the Job module.) Therefore, you must be careful to supply valid data for all of the material behaviors that the analysis requires.

12.4.3 Creating sections

You can use the Property module to create the following types of sections:

- Homogeneous solid sections
- Generalized plane strain sections
- Eulerian sections
- Composite solid sections
- Homogeneous shell sections
- Composite shell sections
- Membrane sections
- Surface sections
- General shell stiffness sections
- Beam sections
- Truss sections
- Gasket sections
- Cohesive sections
- Acoustic infinite sections
- Acoustic interface sections



To create a section, select **Section⟶Create** from the main menu bar. A **Create Section** dialog box appears in which you can name the section and specify the type of section that you want to create. Once you have specified a section name and type, click **Continue** in the **Create Section** dialog box to display the section editor, which allows you to create and edit sections.

Choosing a profile type

The **Create Profile** dialog box allows you to specify which type of profile you want to define. You can define a shape-based profile by providing the geometric data from which Abaqus can calculate the engineering properties of the section. Alternatively, you can define a generalized profile by providing the engineering properties of the section directly. For more information, see "Beam section behavior," Section 25.3.5 of the Abaqus Analysis User's Manual.

After you have assigned the beam section and beam orientation to the part, you can use the part display options to view an idealized representation of the shape-based or generalized beam profile. Displaying beam profiles is useful for checking that the correct profile has been assigned to a particular region and that the assigned beam orientation

results in the expected orientation of the profile. For more information, see "Controlling beam profile display," Section 72.6.

To choose a profile type:

1. From the main menu bar, select **Profile→Create**.

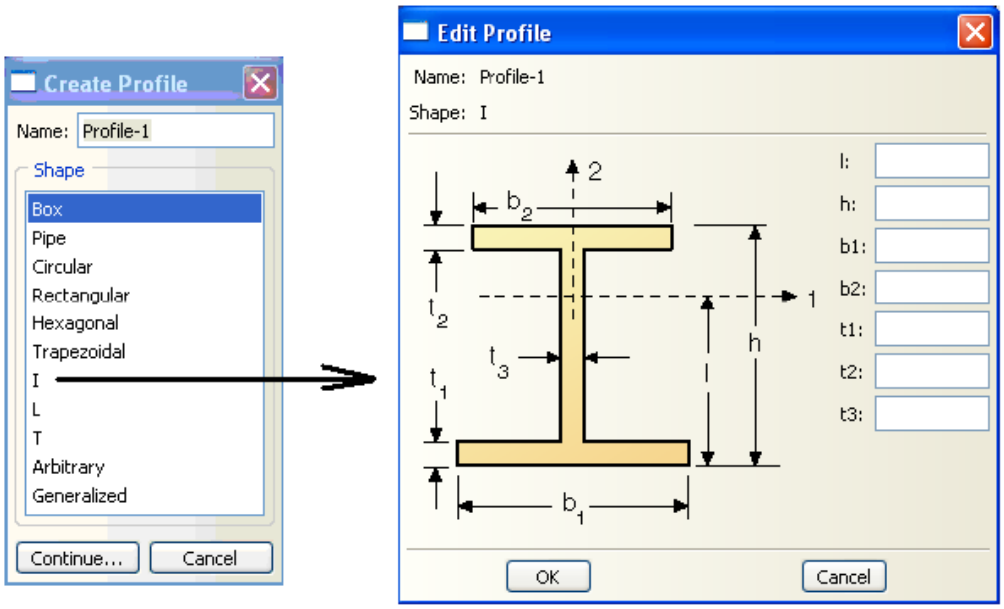
   The **Create Profile** dialog box appears.

   Tip: You can also click **Create** in the **Profile Manager** or select the create profile tool in the Property module toolbox.

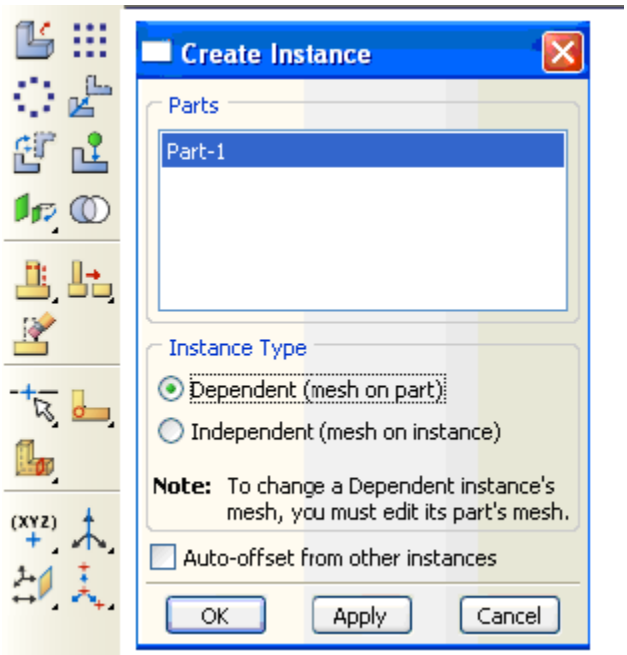
2. Enter a profile name. For more information on naming objects, see "Using basic dialog box components," Section 3.2.1.
3. Select a profile shape, and click **Continue**.

   The **Edit Profile** dialog box for the profile shape you have chosen appears.

4. In the **Edit Profile** dialog box, enter the required profile data. See the following sections for details:
   - "Defining a box profile"
   - "Defining a pipe profile"
   - "Defining a circular profile"
   - "Defining a rectangular profile"
   - "Defining a hexagonal profile"
   - "Defining a trapezoidal profile"
   - "Defining an I-shaped profile"
   - "Defining an L-shaped profile"
   - "Defining a T-shaped profile"
   - "Defining an arbitrary profile"
   - "Defining a generalized profile"


Create Profile
Name: Profile-1
Shape
Box
Pipe
Circular
Rectangular
Hexagonal
Trapezoidal
I
L
T
Arbitrary
Generalized
Continue...
Cancel
Edit Profile
Name: Profile-1
Shape: I
2
1
b2
t2
t3
t1
h
l
b1
l:
h:
b1:
b2:
t1:
t2:
t3:
OK
Cancel

Chapter 4 – Assembly Module



13.9.3 Creating a part instance

To create a part instance, select **Instance→Create** from the main menu bar and select the parts to instance from the **Create Instance** dialog box that appears. You can select from any of the existing parts in the current model. You can create multiple instances of the same part, but you cannot assemble instances of parts that were created in different modeling spaces (three-dimensional, two-dimensional, or axisymmetric).

When you create the first part instance, the Assembly module displays a graphic symbol indicating the origin and orientation of the assembly's global coordinate system. This symbol is a datum coordinate system. If desired, you can hide it using the assembly display options; for more information, see "Controlling datum display," Section 72.7.

By default, Abaqus/CAE creates a dependent part instance. A dependent instance is only a pointer to the geometry of the original part. As a result, many operations are not allowed on a dependent part instance; for example, you cannot add partitions, create virtual topology, or mesh the instance. In contrast, an independent part instance is a copy

of the geometry of the original part. You can perform most operations on an independent instance; for example, you can add partitions, create virtual topology, and mesh the instance. You cannot create both an independent and a dependent instance of the same part. You can select the instance from the Model Tree and change it from independent to dependent or vice versa. For more information, see "What is the difference between a dependent and an independent part instance?," Section 13.3.2.

When you create an instance of a part, by default Abaqus/CAE positions the instance so that the origin of the original geometry aligns with the origin of the assembly coordinate system. When you create multiple part instances, a new instance can be positioned over an existing instance. However, if you toggle on **Auto-offset from other instances** in the **Create Instance** dialog box, Abaqus/CAE translates each new part instance along the *X*–axis until it does not overlap any existing part instances. If the assembly is axisymmetric, Abaqus/CAE translates the new part instance along the axis of revolution instead of along the *X*–axis.

To create a part instance:

1. From the main menu bar, select **Instance→Create** to create a part instance from the parts in the model.

   Abaqus/CAE displays the **Create Instance** dialog box and a list of all the existing parts in the model.

   Tip: You can also create a part instance using the tool from the Assembly module toolbox. For a diagram of the tools in the Assembly toolbox, see "Using the Assembly module toolbox," Section 13.8.

2. From the list of parts, select the parts to instance. You can use a combination of **[Ctrl]+Click** and **[Shift]+Click** to select multiple parts.

   A temporary image of the selected part instances appears in the current viewport. Abaqus/CAE positions the temporary images so that their origins coincide with the origin of the global coordinate system.

3. By default, Abaqus/CAE creates a **Dependent** part instance. If desired, toggle on **Independent** to create an independent part instance.
4. If desired, toggle on **Auto-offset from other instances** to offset the new part instances.
5. If you are satisfied that you have selected the correct part instances, click **Apply** from the **Create Instance** dialog box.

   Abaqus/CAE creates the part instances and applies an auto-offset if selected.

6. To create additional part instances, repeat this procedure from Step 2. When you have finished creating part instances, click **Cancel** to close the **Create Instance** dialog box.

For information on related topics, click any of the following items:

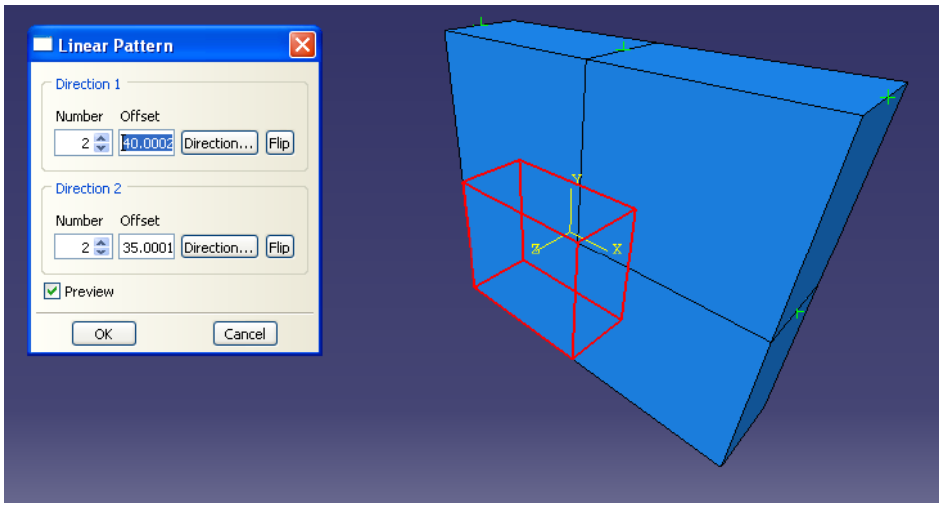
- "Using the **Instance** menu," Section 13.9.1
- "Working with part instances," Section 13.3

13.9.4 Creating a linear pattern of part instances

To create multiple copies of a selected part instance in a linear pattern, select **Instance→ Linear Pattern** from the main menu bar. You can create a pattern that extends in one direction (for example, horizontally or vertically), or you can create a pattern that extends in two directions (for example, both horizontally and vertically). You cannot edit a pattern after you create it. You can specify the following:

- The number of instances to create in each direction, including the selected instance. You can create any number of instances.
- The spacing between each instance along the specified direction.
- A line that defines the direction along which Abaqus/CAE generates the instances.

For more information, see "Creating patterns of part instances," Section 13.5.



To create a linear pattern of part instances:

1. From the main menu bar, select **Instance→Linear Pattern**.

   Tip: You can also create a linear pattern of part instances using the tool from the Assembly module toolbox. For a diagram of the tools in the Assembly toolbox, see "Using the Assembly module toolbox," Section 13.8.

2. Select the instances that you want to copy.

   Tip: To select more than one instance, hold down the **[Shift]** key as you click each instance or drag a rectangle around the instances. To unselect an instance, use **[Ctrl]+Click**. For more information, see Chapter 6, "Selecting objects within the viewport."

3. Click mouse button 2 to indicate that you have finished selecting instances.

   Abaqus/CAE displays the **Linear Pattern** dialog box.

4. From the **Linear Pattern** dialog box, configure the pattern in **Direction-1** (by default, **Direction-1** is the *X*-direction):
   a. Click the arrows to the right of **Number** to increase or decrease the number of copies to create, including the selected instances. The number of copies in the assembly updates when you click the arrows and provides a preview of the setting.

      Alternatively, you can type in a number and press **[Enter]** to preview the setting. You can enter any number greater than or equal to 1. If you enter a value of 1, Abaqus/CAE displays only the selected instances and does not create any copies of the selected instances; in effect, you are disabling copies in **Direction-1**.

   b. Enter the **Spacing** between each copy along the specified direction.
   c. By default, Abaqus/CAE creates the copies along the *X*-direction. If you want to change the direction in which Abaqus/CAE creates the copies, click **Direction** and select a line from the assembly to define the new direction. You must pick a straight edge or a datum axis.
   d. By default, Abaqus/CAE creates the copies in the positive direction. Click **Flip** to reverse the direction in which Abaqus/CAE creates the copies.
5. To create copies in a second direction, enter a **Number** greater than 1 and specify the **Spacing**, the **Direction**, and the **Flip** direction for **Direction-2** (by default, **Direction-2** is the *Y*-direction). You must enter a **Number** greater than 1 for at least one direction.

6. In most cases you will want to preview the linear pattern that Abaqus/CAE will create as you enter values in the **Linear Pattern** dialog box. However, if you choose to create a large number of copies, the preview capability may impact the performance of Abaqus/CAE. In this case you should toggle off the **Preview** button.
7. To copy more instances, repeat the above steps beginning with Step 1.

For information on related topics, click any of the following items:

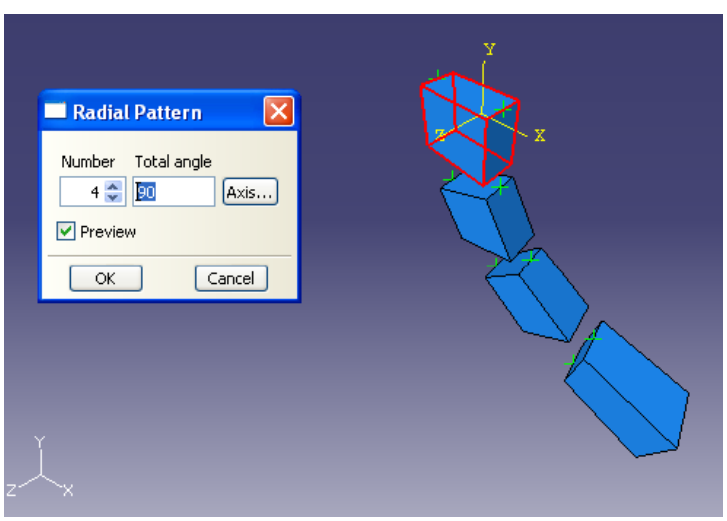
- "Creating patterns of part instances," Section 13.5
- "Creating a radial pattern of part instances," Section 13.9.5

13.9.5 Creating a radial pattern of part instances

To create multiple copies of a selected part instance in a radial pattern, select **Instance→ Radial Pattern** from the main menu bar. You cannot edit a pattern after you create it. You can specify the following:

- The number of copies to create in the radial pattern, including the selected instances. You can create any number of instances.
- The total angle between the original instance and the last copy in the pattern.
- The position of the axis of the circular pattern.

For more information, see "Creating patterns of part instances," Section 13.5.



To create radial patterns of instances:

1. From the main menu bar, select **Instance→Radial Pattern**.

   Tip: You can also create a radial pattern of part instances using the tool from the Assembly module toolbox. For a diagram of the tools in the Assembly toolbox, see "Using the Assembly module toolbox," Section 13.8.

2. Select the instances that you want to copy.

   Tip: To select more than one instance, hold down the **[Shift]** key as you click each instance or drag a rectangle around the instances. To unselect an instance, use **[Ctrl]+Click**. For more information, see Chapter 6, "Selecting objects within the viewport."

3. Click mouse button 2 to indicate that you have finished selecting instances.

   Abaqus/CAE displays the **Radial Pattern** dialog box.

4. From the **Radial Pattern** dialog box, configure the radial pattern:
   a. Click the arrows to the right of **Number** to increase or decrease the number of copies to create, including the selected instances. The number of copies in the assembly updates when you click the arrows and provides a preview of the setting.

      Alternatively, you can type in a number and press **[Enter]** to preview the setting. You can enter any number greater than or equal to 2.

   b. Enter the **Total angle** between the original instances that you selected and the final copy. The angle must be between –360° and +360°. A positive angle corresponds to a counterclockwise direction.
   c. By default, Abaqus/CAE rotates the selected instance about the *Z*-axis to create the pattern. To define a new axis of rotation, click **Axis** and select a line from the assembly that represents the new axis of rotation. You can also select an axis from the global coordinate system triad.
5. In most cases you will want to preview the radial pattern that Abaqus/CAE will create as you enter values in the **Radial Pattern** dialog box. However, if you choose to create a large number of copies, the preview capability may impact the performance of Abaqus/CAE. In this case you should toggle off the **Preview** button.
6. To copy more instances, repeat the above steps beginning with Step 1.

For information on related topics, click any of the following items:

- "Creating patterns of part instances," Section 13.5
- "Creating a linear pattern of part instances," Section 13.9.4

13.9.6 Translating part instances

Select **Instance→Translate** from the main menu bar to move selected part instances along a selected vector. The direction and magnitude of the vector are arbitrary except that you can translate axisymmetric part instances only along the axis of rotation. If the translation conflicts with a previous position constraint; for example, a constraint that aligns two faces, Abaqus/CAE applies the components of translation only along the unconstrained degrees of freedom. If all of the degrees of freedom are constrained, Abaqus/CAE displays an error message and the translation fails.

When you create the first part instance, Abaqus/CAE displays a graphic indicating the origin and orientation of the assembly's default coordinate system. You can use this graphic to help you decide how to translate your part instances. In addition, you can use the Query toolset to review the sum of the translations and rotations previously applied to a part instance and the distance between selected vertices. Translations and rotations are not considered features of the assembly and cannot be edited or deleted.

To translate part instances:

1. From the main menu bar, select **Instance→Translate**.

   Tip: You can also translate part instances using the tool from the Assembly module toolbox. For a diagram of the tools in the Assembly toolbox, see "Using the Assembly module toolbox," Section 13.8.

   Abaqus/CAE displays prompts in the prompt area to guide you through the procedure.

2. Select the part instances to translate. You can also click the **Instance List** button on the right of the prompt area and select the instances to translate from the **Instance List** dialog box that appears.

   Tip: If you are unable to select the desired part instance, you can use the **Selection** toolbar to change the selection behavior. For more information, see "Using the selection options," Section 6.3.

   You can use a combination of **[Ctrl]+Click** and **[Shift]+Click** to select multiple part instances.

Abaqus/CAE highlights the selected part instances.

3. Select the start point of the translation vector. You can select any existing vertices or datum points, or you can enter the coordinates in the text box in the prompt area.
4. Select the end point of the translation vector. Again, you can select any existing vertices or datum points, or you can enter the coordinates in the text box in the prompt area.

   Abaqus/CAE displays a temporary image indicating the translation that will be applied to the selected part instance. You cannot edit or delete a translation after it is applied. Attempting to translate a part instance may result in a conflict with existing position constraints. Abaqus/CAE applies the components of translation only along the unconstrained degrees of freedom. If all of the degrees of freedom are constrained, Abaqus/CAE displays an error message and the translation fails.

5. Do one of the following:
   a. If you are satisfied the translation is correct, click the **OK** button in the prompt area.

      Abaqus/CAE translates the part instance and positions it at the same location as the temporary image of the part instance.

   b. If you are not satisfied with the translation, click the **Previous** button ( ) and specify a new translation vector.
   c. Abort the translation by clicking the cancel button ( ).

For information on related topics, click any of the following items:

- "Using the **Instance** menu," Section 13.9.1
- "Creating the assembly," Section 13.4

3.9.8 Rotating part instances

Select **Instance→Rotate** from the main menu bar to rotate selected part instances about a selected axis. To rotate a three-dimensional part instance, you must select two points that define the axis about which the part instance will rotate. To rotate a two-dimensional part instance, you must select a single point about which the part instance will rotate. You cannot rotate axisymmetric part instances. If the rotation conflicts with a previous

position constraint (for example, a constraint that aligns two faces), Abaqus/CAE displays an error message and the rotation fails.

When you create the first part instance, Abaqus/CAE displays a graphic indicating the origin and orientation of the assembly's global coordinate system. You can use this graphic to help you decide how to rotate your part instances. In addition, you can use the Query toolset to review the sum of the translations and rotations previously applied to a part instance and the distance between selected vertices. Rotations and translations are not considered features of the assembly and cannot be edited or deleted.

To rotate part instances:

1. From the main menu bar, select **Instance→Rotate**.

   Tip: You can also rotate a part instance using the tool from the Assembly module toolbox. For a diagram of the tools in the Assembly toolbox, see "Using the Assembly module toolbox," Section 13.8.

   Abaqus/CAE displays prompts in the prompt area to guide you through the procedure.

2. From the assembly, select the part instances to rotate. You can also click the **Instance List** button on the right of the prompt area and select the instances to rotate from the **Instance List** dialog box that appears.

   Tip: If you are unable to select the desired part instance, you can use the **Selection** toolbar to change the selection behavior. For more information, see "Using the selection options," Section 6.3.

   You can use a combination of **[Ctrl]+Click** and **[Shift]+Click** to select multiple part instances.

   Abaqus/CAE highlights the selected part instances.

3. Select the start point of the vector that defines the axis of rotation. You can select any existing vertices or datum points, or you can enter the coordinates in the text box in the prompt area.
4. Select the end point of the vector that defines the axis of rotation. Again, you can select any existing vertices or datum points, or you can enter the coordinates in the text box in the prompt area.
5. In the text box that appears in the prompt area, type the angle of rotation. A positive angle indicates a counterclockwise rotation; a negative angle indicates a clockwise rotation.

Abaqus/CAE displays a temporary image indicating the rotation that will be applied to the selected part instances. You cannot edit or delete a rotation after it is applied. Attempting to rotate a part instance may result in a conflict with existing position constraints. If a conflict occurs, Abaqus/CAE displays an error message and the rotation fails.

6. Do one of the following:
   a. If you are satisfied that the rotation is correct, click the **OK** button in the prompt area.

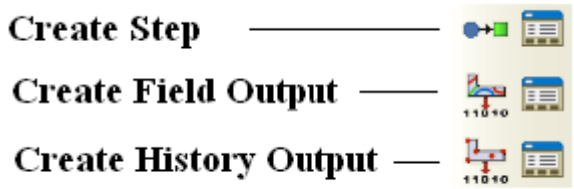
      Abaqus/CAE rotates the part instance and positions it at the same location as the temporary image of the part instances.

   b. If you are not satisfied with the rotation, click the **Previous** button ( ) and specify a new rotation.
   c. Abort the rotation by clicking the cancel button ( ).

For information on related topics, click any of the following items:

- "Using the **Constraint** menu," Section 13.10.1
- "Creating the assembly," Section 13.4

Chapter 5 – Step Module



14.9.2 Creating a step

You can create any sequence of procedures that is allowed by Abaqus/Standard or Abaqus/Explicit; the procedure list in the **Create Step** dialog box is updated to show only the available procedures for the new step. For example, if your first step contains a static stress/displacement procedure, you cannot follow it with a new step containing a heat transfer procedure.

To create a step:

1. From the main menu bar, select **Step⟶Create**.

   The **Create Step** dialog box appears.

   Tip: You can initiate the **Create** procedure in two other ways:

   - Click **Create** in the **Step Manager**. (You can display the **Step Manager** by selecting **Step⟶Manager** from the main menu bar.)
   - Click the tool in the Step module toolbox.
2. If desired, use the **Name** text field to change the name of the new step.

   All steps must have unique names, and you cannot name a step "Initial".

3. From the list of existing steps, select the step after which the new step will be inserted.
4. Click the arrow next to the **Procedure type** field, and select either **General** or **Linear perturbation** from the list that appears.

   The lower half of the dialog box displays a list of available procedures.

5. Select the desired procedure and click **Continue**.

The **Edit Step** dialog box appears.

6. Use the **Edit Step** dialog box to modify the settings from their default values and to provide values for optional settings. (For detailed help on a particular editor feature, select **Help⟶On Context** from the main menu bar and then click the feature of interest.)
7. Click **OK**.

   Abaqus/CAE closes the **Edit Step** dialog box, and the new step appears in the **Step Manager**.

For information on related topics, click any of the following items:

- "Understanding steps," Section 14.3
- "General and linear perturbation procedures," Section 6.1.2 of the Abaqus Analysis User's Manual

Configuring a static, general procedure

A static stress procedure is one in which inertia effects are neglected. The analysis can be linear or nonlinear and ignores time-dependent material effects. For more information, see "Static stress analysis," Section 6.2.2 of the Abaqus Analysis User's Manual.

To create or edit a static, general procedure:

1. Display the **Edit Step** dialog box following the procedure outlined in "Creating a step," Section 14.9.2 (**Procedure type:** `General; Static, General`), or "Editing a step," Section 14.9.3.
2. On the **Basic**, **Incrementation**, and **Other** tabbed pages, configure settings such as the time period for the step, the maximum number of increments, the increment size, the default load variation with time, and whether to account for geometric nonlinearity as described in the following procedures.

To configure settings on the Basic tabbed page:

1. In the **Edit Step** dialog box, display the **Basic** tabbed page.

2. In the **Description** field, enter a short description of the analysis step. Abaqus stores the text that you enter in the output database, and the text is displayed in the state block by the Visualization module.
3. In the **Time period** field, enter the time period of the step. For more information, see "Time period" in "Static stress analysis," Section 6.2.2 of the Abaqus Analysis User's Manual.
4. Select an **Nlgeom** option:
   - Toggle **Nlgeom Off** to perform a geometrically linear analysis during the current step.
   - Toggle **Nlgeom On** to indicate that Abaqus/Standard should account for geometric nonlinearity during the step. Once you have toggled **Nlgeom** on, it will be active during all subsequent steps in the analysis.

   For more information, see "Linear and nonlinear procedures," Section 14.3.2.

5. Select an automatic stabilization method if you expect the problem to have local instabilities such as surface wrinkling, material instability, or local buckling. Abaqus/Standard can stabilize this class of problems by applying damping throughout the model. For more information, see "Unstable problems" in "Static stress analysis," Section 6.2.2 of the Abaqus Analysis User's Manual, and "Automatic stabilization of static problems with a constant damping factor" in "Solving nonlinear problems," Section 7.1.1 of the Abaqus Analysis User's Manual

To configure settings on the Incrementation tabbed page:

1. In the **Edit Step** dialog box, display the **Incrementation** tabbed page.

   (For information on displaying the **Edit Step** dialog box, see "Creating a step," Section 14.9.2, or "Editing a step," Section 14.9.3.)

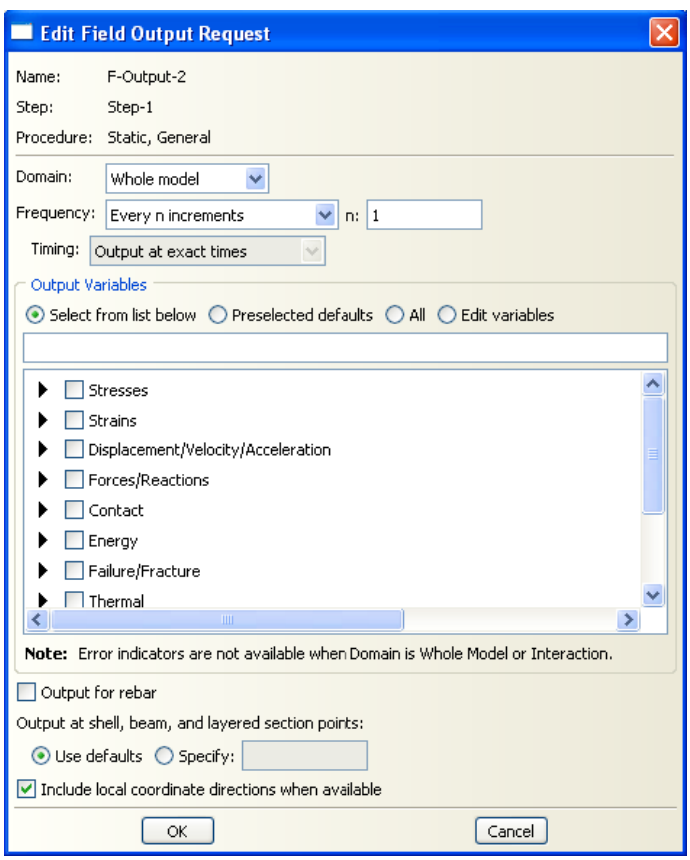
2. Choose a **Type** option:
   - Choose **Automatic** to allow Abaqus/Standard to choose the size of the time increments based on computational efficiency.
   - Choose **Fixed** to specify direct user control of the incrementation. Abaqus/Standard uses an increment size that you specify as the constant increment size throughout the step.
3. In the **Maximum number of increments** field, enter the upper limit to the number of increments in the step. The analysis stops if this maximum is exceeded before Abaqus/Standard arrives at the complete solution for the step.
4. If you selected **Automatic** in Step 2, enter values for **Increment size**:

a. In the **Initial** field, enter the initial time increment. Abaqus/Standard modifies this value as required throughout the step.
b. In the **Minimum** field, enter the minimum time increment allowed. If Abaqus/Standard needs a smaller time increment than this value, it terminates the analysis.
c. In the **Maximum** field, enter the maximum time increment allowed.

5. If you selected **Fixed** in Step 2, enter a value for the constant time increment in the **Increment size** field.
6. When you have finished configuring settings for the static, general step, click **OK** to close the **Edit Step** dialog box.

14.12.2 Modifying field output requests

You can use the field output editor to modify existing field output requests. If you modify a field output request during a step into which it was propagated, you can modify only the output variables and the output frequency.



14.12.1 Creating an output request

When you create the first step in a sequence of steps, Abaqus/CAE generates default field and history output requests based on the analysis procedure that you selected for the step. The output requests are propagated to subsequent steps in the analysis. For more information, see"Propagation of output requests," Section 14.4.3. You can use the output request editors to edit the default output requests in the step in which they were created. You can also use the output request editors to create new output requests. A new output request overrides any propagated requests.

To create an output request:

1. From the main menu bar, select **Output→Field Output Requests→Create** or **Output→History Output Requests→Create**.

   Tip: You can also create field and history output requests using the and tools, located in the module toolbox. For a diagram of the tools in the Step module toolbox, see "Using the Step module toolbox," Section 14.8.

   Abaqus/CAE displays the **Create Field Output** or **Create History Output** dialog box.

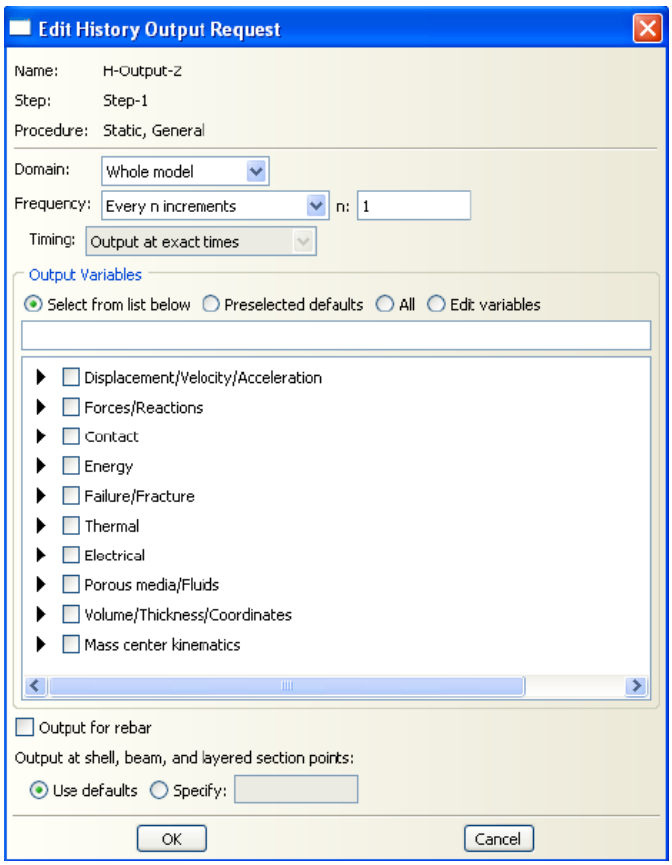
2. In the dialog box, do the following:
   a. Type a name for the output request or accept the default name.
   b. Click the arrow next to the **Step** text field, and select the step from the list that appears. Abaqus/CAE creates the output request during the selected step. After you create an output request, you can use the output requests manager to move it to an earlier or later step.
   c. Click **Continue**.

   Abaqus/CAE displays the **Edit Field Output Request** or the **Edit History Output Request** dialog box.

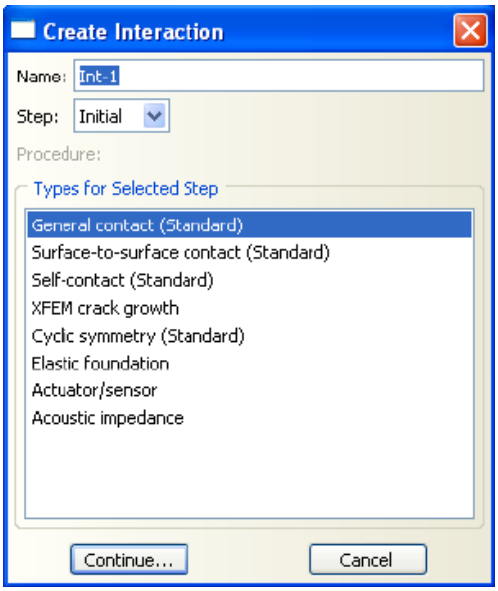
3. In the editor, enter the data necessary to define the output request. For more information on using the editor, see "Modifying field output requests," Section 14.12.2, or "Modifying history output requests," Section 14.12.3.
4. When you have finished configuring your output request, click **OK** to save your data and to exit the editor.

For information on related topics, click any of the following items:

- "Understanding output requests," Section 14.4
- "Defining output requests," Section 14.12



Chapter 6– Interaction Module



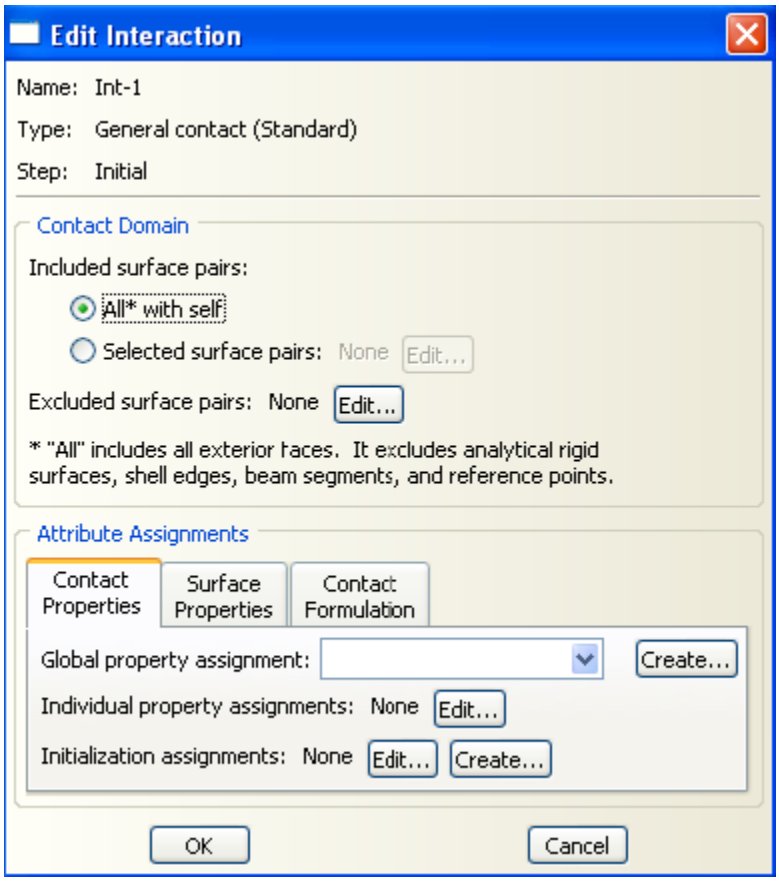
15.12.1 Creating interactions

When you create an interaction, you must specify the name of the interaction, the step in which to activate the interaction, the type of interaction, and the region of the assembly to which you want to apply the interaction. The region selection does not apply to general contact interactions. The available types of interactions depend on the procedure selected for the step. For example, you can define heat flux on a surface only during a heat transfer, coupled temperature-displacement, or coupled thermal-electrical step. Similarly, you can define interactions with a user-defined actuator/sensor only during the initial step.

15.13.1 Defining general contact

In Abaqus/Standard a general contact definition can create interactions for all exterior faces in the model. In Abaqus/Explicit a general contact definition can create interactions for all exterior faces, analytical rigid surfaces, shell perimeter edges, edges based on beams and trusses, and Eulerian material boundaries in

the model; however, you cannot specify general contact (including self-contact) between two analytical rigid surfaces.



To define general contact:

1. From the main menu bar, select **Interaction⟶Create**.

   Tip: You can also create a general contact interaction using the tool in the Interaction module toolbox.

2. In the **Create Interaction** dialog box that appears, do the following:
   - Name the interaction. For more information about naming objects, see "Using basic dialog box components," Section 3.2.1.
   - Select the step in which the interaction will be created. In Abaqus/Standard general contact can be created only in the initial step.

- Select the **General contact (Standard)** or **General contact (Explicit)** type of interaction, depending on the analysis steps being defined in your model.

3. Click **Continue** to close the **Create Interaction** dialog box.

   The **Edit Interaction** dialog box appears.

4. Specify the contact domain using either of the following methods:
   - Choose **All* with self** to specify contact (including self-contact) for all allowable element faces and model entities. This is the simplest way to define the contact domain.

     Note: If you are defining contact for an Eulerian-Lagrangian analysis, you must choose the **All* with self** contact domain. For more information, see "Defining contact in Eulerian-Lagrangian models," Section 27.3.

   - To specify individual contact surface pairings:
     a. Choose **Selected surface pairs**, and click **Edit**.

        The **Edit Included Pairs** dialog box appears. By default, when you select a surface from the list or the table, Abaqus/CAE highlights the surface in the viewport; however, highlighting does not apply for **(All*)** and **(Self)**. You can toggle off **Highlight selected regions** at the bottom of the dialog box to turn off selection highlighting.

     b. Select one or more surfaces from the list of existing surfaces in the first column on the left side. Select **(All*)** to specify a surface that includes all allowable element faces and model entities.
     c. Select the second surface or surfaces from the list of existing surfaces in the second column to define the surface pairings.
        - When multiple surfaces are selected in either column, all possible combinations will be generated in the table.
        - To specify self-contact, select either the same surface name or **(Self)** in the second column.
        - The order in which the surfaces are specified does not matter for the analysis.
     d. Click the arrows in the middle of the dialog box to transfer the surface pair to the list of pairings that will be included in the contact domain.

The table on the right side of the dialog box is updated to reflect your selections (the order of the surface pairings is irrelevant).

e. Repeat the above steps as needed to completely define the contact domain inclusions. If you want to delete included pairs, select the rows and click **Delete Selected Rows**.
f. Click **OK** to save your selections and to close the **Edit Included Pairs** dialog box.

The interaction editor reappears with updated information on the number of selected surface pairs for inclusion in the contact domain.

5. If necessary, select the surface pairs to exclude from the contact domain.

Click **Edit** next to **Excluded surface pairs**.

The **Edit Excluded Pairs** dialog box appears. By default, when you select a surface from the list or the table, Abaqus/CAE highlights the surface in the viewport; however, highlighting does not apply for **(All*)** and **(Self)**. You can toggle off **Highlight selected regions**at the bottom of the dialog box to turn off selection highlighting.

Select one or more surfaces from the list of existing surfaces in the first column on the left side. Select **(All*)** to specify a surface that includes all allowable element faces and model entities.

Select the second surface or surfaces from the list of existing surfaces in the second column to define the surface pairings.

When multiple surfaces are selected in either column, all possible combinations will be generated in the table.

To specify that self-contact should be excluded, select either the same surface name or **(Self)** in the second column.

The order in which the surfaces are specified does not matter for the analysis.

If the excluded regions overlap with the included regions, the contact exclusions will take precedence over the contact inclusions.

Click the arrows in the middle of the dialog box to transfer the surface pair to the list of pairings that will be excluded from the contact domain.

The table on the right side of the dialog box is updated to reflect your selections (the order of the surface pairings is irrelevant).

Repeat the above steps as needed to completely define the contact domain exclusions. If you want to delete excluded pairs, select the rows and click **Delete Selected Rows**.

Click **OK** to save your selections and to close the **Edit Excluded Pairs** dialog box.

The interaction editor reappears with updated information on the number of selected surface pairs for exclusion from the contact domain.

Specify the **Attribute Assignments** at the bottom of the interaction editor. In Abaqus/Standard you can modify the contact properties in any step in which the general contact interaction is active, but all other attributes are assigned for the entire analysis. In Abaqus/Explicit you can specify or modify the attributes in any step in which the general contact interaction is active. You can specify the following assignments:

Contact Properties. For detailed instructions, see "Specifying and modifying contact property assignments for general contact,"Section 15.13.2.

Contact Initializations. For detailed instructions, see "Specifying and modifying contact initialization assignments for general contact," Section 15.13.3.

Surface Properties. For detailed instructions, see "Specifying surface property assignments for general contact," Section 15.13.4.

Contact Formulation. For detailed instructions, see "Specifying master-slave assignments for general contact," Section 15.13.5.

Click **OK** to create the interaction and to close the editor.

Defining surface-to-surface contact in an Abaqus/Standard analysis

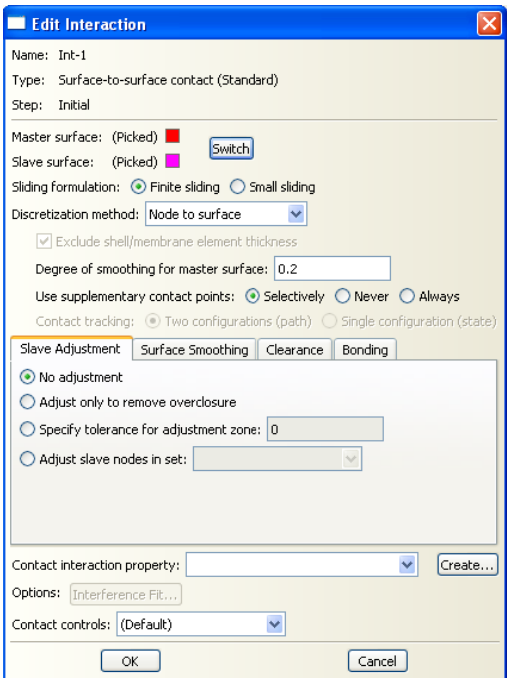
Certain interaction behaviors can be defined in Abaqus/Standard only by using surface-to-surface contact; see "Contact simulation capabilities in Abaqus/Standard" in "Contact interaction analysis: overview," Section 31.1.1 of the Abaqus Analysis User's Manual, for more information.

To define surface-to-surface contact in an Abaqus/Standard analysis:

1. From the main menu bar, select **Interaction→Create**.

   Tip: You can also create a surface-to-surface contact interaction using the tool in the Interaction module toolbox.

2. In the **Create Interaction** dialog box that appears, do the following:
   - Name the interaction. For more information about naming objects, see "Using basic dialog box components," Section 3.2.1.
   - Select the step in which the interaction will be created.
   - Select the **Surface-to-surface contact (Standard)** type of interaction.

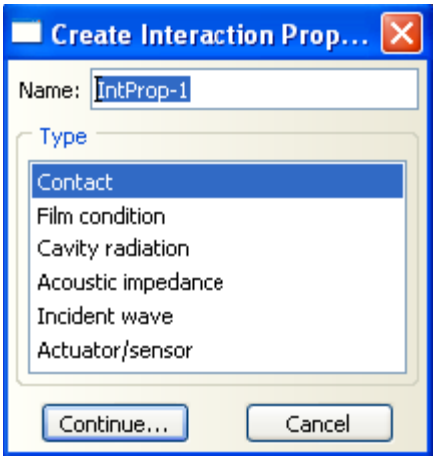


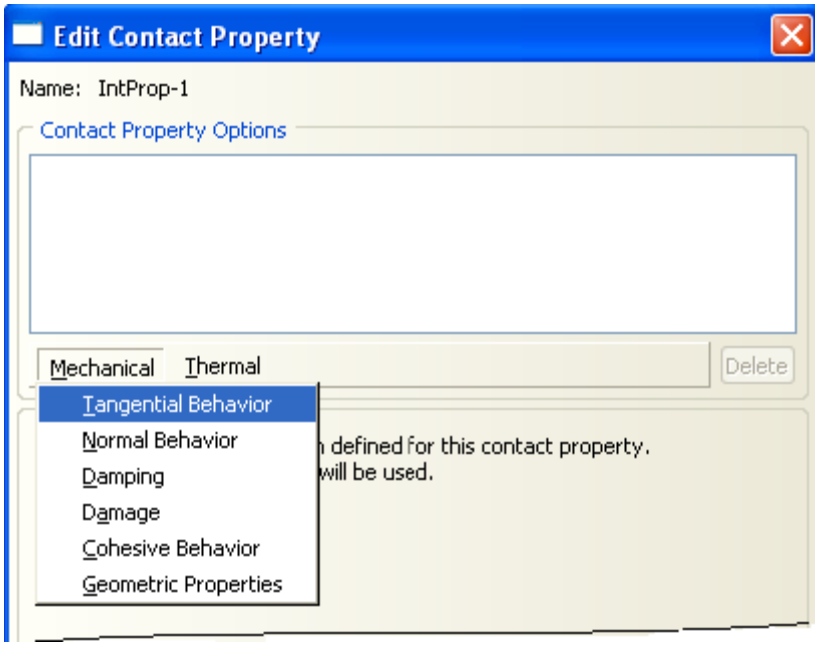
3. Click **Continue** to close the **Create Interaction** dialog box.

15.14.1 Defining a contact interaction property

The contact property editor contains the following menus from which you can choose options to include in the property definition:

- **Mechanical**; see "Defining mechanical contact property options."
- **Thermal**; see "Defining thermal contact property options."





Specifying frictional behavior for mechanical contact property options

You can specify a friction model that defines the force resisting the relative tangential motion of the surfaces in a mechanical contact analysis. For more information, see "Frictional behavior," Section 32.1.5 of the Abaqus Analysis User's Manual.

To specify frictional behavior:

1. From the main menu bar, select **Interaction⟶Property⟶Create**.
2. In the **Create Interaction Property** dialog box that appears, do the following:
   - Name the interaction property. For more information about naming objects, see "Using basic dialog box components," Section 3.2.1.
   - Select the **Contact** type of interaction property.
3. Click **Continue** to close the **Create Interaction Property** dialog box.
4. From the menu bar in the contact property editor, select **Mechanical⟶ Tangential Behavior**.
5. In the editor that appears, click the arrow to the right of the **Friction formulation** field, and select how you want to define friction between the contact surfaces:
   - Select **Frictionless** if you want Abaqus to assume that surfaces in contact slide freely without friction.
   - Select **Penalty** to use a stiffness (penalty) method that permits some relative motion of the surfaces (an "elastic slip") when they should be sticking. While the surfaces are sticking (i.e., $\bar{\tau} < \bar{\tau}_{crit}$), the magnitude of sliding is limited to this elastic slip. Abaqus will continually adjust the magnitude of the penalty constraint to enforce this condition. For more information, see "Stiffness method for imposing frictional constraints" in "Frictional behavior," Section 32.1.5 of the Abaqus Analysis User's Manual.
   - Select **Static-Kinetic Exponential Decay** to specify static and kinetic friction coefficients directly. In this model it is assumed that the friction coefficient decays exponentially from the static value to the kinetic value. Alternatively, you can enter test data to fit the exponential model. (This **Friction formulation** option also allows you to specify elastic slip.) For more information, see "Specifying static and kinetic friction coefficients" in "Frictional behavior," Section 32.1.5 of the Abaqus Analysis User's Manual.
   - Select **Rough** to specify an infinite coefficient of friction. For more information, see "Preventing slipping regardless of contact pressure" in "Frictional behavior," Section 32.1.5 of the Abaqus Analysis User's Manual.
   - Select **Lagrange Multiplier (Standard only)** to enforce the sticking constraints at an interface between two surfaces using the

Lagrange multiplier implementation. With this method there is no relative motion between two closed surfaces until $\bar{\tau} = \bar{\tau}_{crit}$. For more information, see "Lagrange multiplier method for imposing frictional constraints in Abaqus/Standard" in "Frictional behavior,"Section 32.1.5 of the Abaqus Analysis User's Manual.

- Select **User-defined** to define the shear interaction between the contact surfaces with user subroutine `FRIC` or `VFRIC`. For more information, see "User-defined friction model" in "Frictional behavior," Section 32.1.5 of the Abaqus Analysis User's Manual.

6. If you selected the **Penalty** or **Lagrange Multiplier (Standard only)** friction formulation, perform the following steps:

a. Display the **Friction** tabbed page.

b. Choose the **Directionality**:

- Choose **Isotropic** to enter a uniform friction coefficient.
- Choose **Anisotropic (Standard only)** to allow for different friction coefficients in the two orthogonal directions on the contact surface. For more information, see "Using the anisotropic friction model in Abaqus/Standard" in "Frictional behavior," Section 32.1.5 of the Abaqus Analysis User's Manual.

c. Toggle on **Use slip-rate-dependent data** if the friction coefficient is dependent on slip rate.

d. Toggle on **Use contact-pressure-dependent data** if the friction coefficient is dependent on the contact pressure.

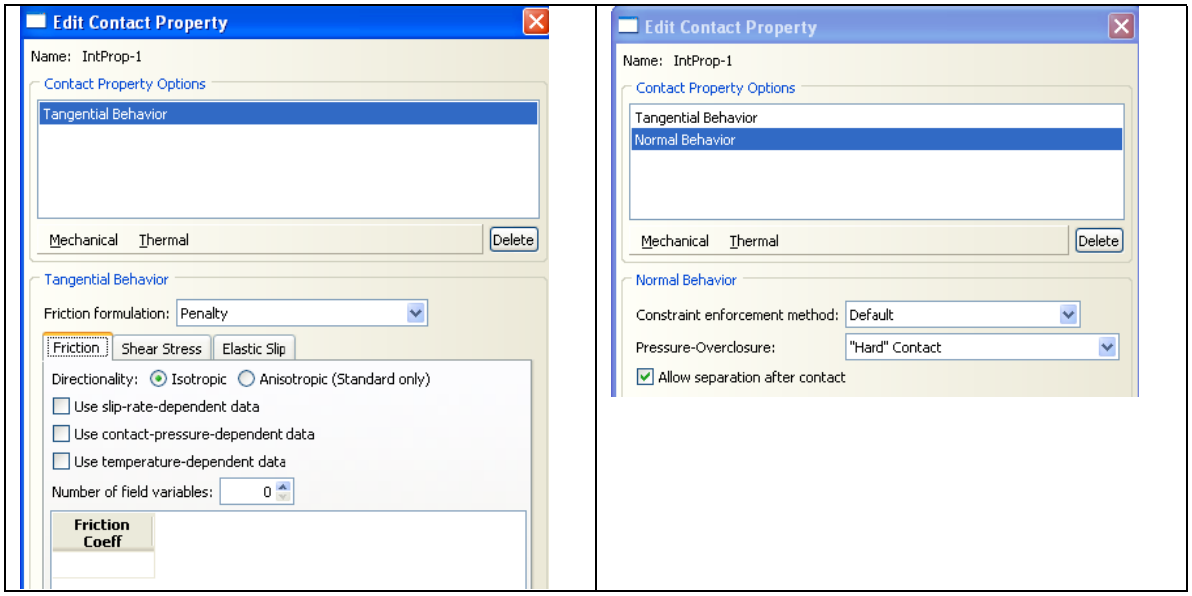
e. Toggle on **Use temperature-dependent data** if the friction coefficient is dependent on temperature.

f. Click the arrows to the right of the **Number of field variables** field to specify the number of field variables on which the friction coefficient depends.

g. Enter the required data in the data table provided.

h. Display the **Shear Stress** tabbed page, and choose a **Shear stress limit** option:

- Choose **No limit** if you do not want to limit the shear stress that can be carried by the interface before the surfaces begin to slide.
- Choose **Specify** to enter an equivalent shear stress limit, $\bar{\tau}_{max}$. If you choose this option, sliding will occur if the magnitude of the equivalent shear stress reaches this value, regardless of the magnitude of the contact pressure stress. For more information, see "Using the optional shear stress limit" in "Frictional behavior," Section 32.1.5 of the Abaqus Analysis User's Manual.

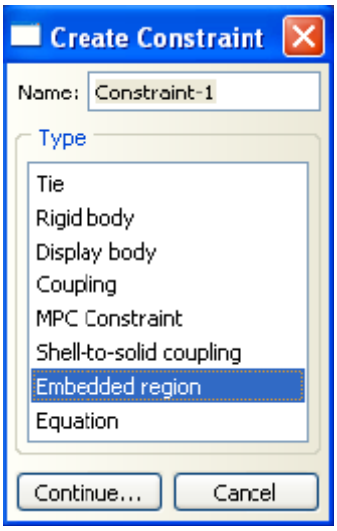


In “Hard” Contact two surfaces can not penetrate to each others.

15.12.5 Creating constraints

You can create the following constraints:

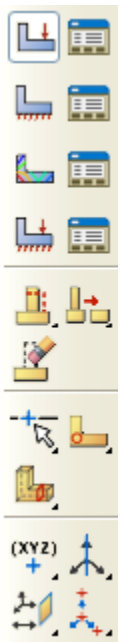
- Tie constraints that tie two separate surfaces together so that there is no relative motion between them.
- Rigid body constraints that allow you to designate a collection of regions as a rigid body.
- Display body constraints that allow you to designate a part instance that will be used for display only.
- Coupling constraints that allow you to constrain the motion of a surface to the motion of a reference node.
- Multi-point constraints that allow you to constrain the motion of the slave nodes of a region to the motion of a single point.
- Shell-to-solid coupling constraints that allow you to couple the motion of a shell edge to the motion of an adjacent solid face.
- Embedded region constraints that allow you to embed a region of the model within a "host" region of the model or within the whole model.
- Equation constraints that describe linear constraints between individual degrees of freedom.

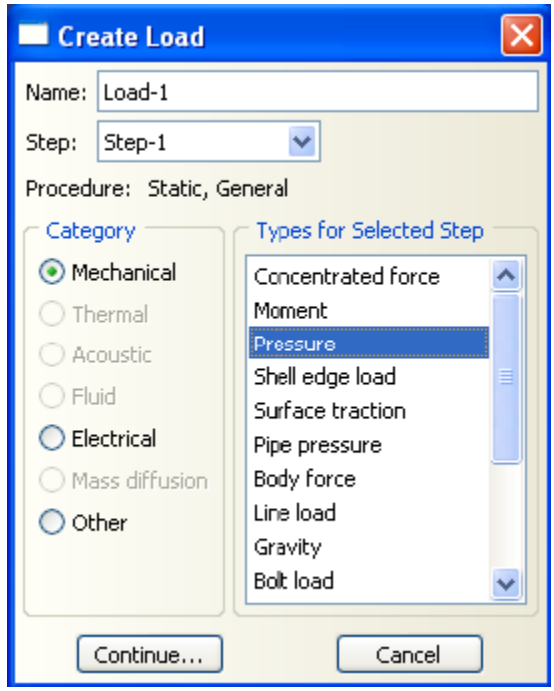


The embedded element technique:

- is used to specify an element or a group of elements that lie embedded in a group of host elements whose response will be used to constrain the translational degrees of freedom of the embedded nodes (i.e., nodes of embedded elements);
- can be used in geometrically linear or nonlinear analysis;
- is not available for host elements with rotational degrees of freedom;
- can be used to model a set of rebar-reinforced membrane, shell, or surface elements that lie embedded in a set of three-dimensional solid (continuum) elements; a set of truss or beam elements that lie embedded in a set of solid elements; or a set of solid elements that lie embedded in another set of solid elements;
- will not constrain rotational degrees of freedom of the embedded nodes when shell or beam elements are embedded in solid elements; and

Chapter 7– Load Module





16.8.1 Creating loads

When you create a load, you must specify the name of the load, the step in which to activate the load, the type of load, and the region of the assembly to which you want to apply the load.

To create a load:

1. From the main menu bar, select **Load⟶Create**.

   A **Create Load** dialog box appears with a default name displayed in the **Name** text field.

   Tip: You can also create a load using the tool in the Load module toolbox.

2. Type a name for the load. For more information on naming objects, see "Using basic dialog box components," Section 3.2.1.

3. Select the step in which to activate the load. Click the arrow next to the **Step** text field, and select from the list that appears. Loads can be created only in an analysis step; you cannot create a load in the initial step.
4. From the **Category** list on the left side of the dialog box, choose the desired category. The **Category** choices available are dependent upon the type of analysis procedures you are performing.

   The **Types for Selected Step** list on the right side of the dialog box changes to a list of all the available load types.

5. From the **Types for Selected Step** list, select the load type and click **Continue**.
6. If you are creating a load other than gravity or inertia relief, select the region to which you want to apply the load.

   If you are creating a connector force or connector moment, you must select wires that are associated with a connector section assignment. The best approach for selecting wires is to use the default geometry set name for the wire feature (see "Creating or modifying wire features for multiple connectors," Section 15.12.8, for more information). If you select multiple wires, you must ensure that the connector sections assigned to the wires in the connector section assignments have the available components of relative motion for which you want to define forces or moments. If there are insufficient available components of relative motion for the connector force or connector moment, a message appears asking you to select different wires or to change the connection type.

   Use one of the following methods to select the region for the load:

   - Select a region in the viewport. You can use the angle method to select a group of faces or edges from a native geometric part instance or a group of element faces from an orphan mesh part instance. For more information, see "Using the angle method to select multiple objects," Section 6.2.3. When you have finished selecting, click mouse button 2.

     Tip: You can limit the types of objects that you can select in the viewport by clicking the selection options tool in the prompt area and then clicking the selection filter of your choice in the dialog box that appears. See "Using the selection options," Section 6.3, for more information.

     If the model contains a combination of orphan mesh instances and native part instances, click one of the following from the prompt area:

     - Click **Geometry** to apply the load to a native part instance or to a reference point.
     - Click **Mesh** to apply the load to an orphan mesh instance.

- To select from a list of existing sets or surfaces, do the following:

a. Click **Sets** or **Surfaces** on the right side of the prompt area. (The name of the button depends on the type of object you are creating. For example, if you are creating a pressure load, a **Surfaces** button appears.)

Abaqus/CAE displays the **Region Selection** dialog box containing a list of available sets or surfaces.

b. Select the set or surface of interest and click **Continue**.

Note: The default selection method is based on the selection method you most recently employed. To revert to the other method, click **Select in Viewport** or **Sets** or **Surfaces** on the right side of the prompt area.

The load editor appears. The region to which you are applying the load is highlighted in the viewport.

7. If you are creating a gravity load or an inertia relief load, the load editor appears.
8. Enter all of the data necessary to define the load and click **OK**.

Note: If you create a connector force or connector moment that exceeds the failure criteria for a connector, the connector force or connector moment will still be applied.

For detailed information on a particular feature of the editor, select **Help⟶On Context** from the main menu bar and then click the feature of interest or see "Using the load editors," Section 16.9.

Symbols appear in the viewport that represent the load that you just created. For more information, see "Understanding symbols that represent prescribed conditions," Section 16.5.

For information on related topics, click any of the following items:

- "Understanding and using toolboxes and toolbars," Section 3.3
- "What are step-dependent managers?," Section 3.4.2
- Chapter 6, "Selecting objects within the viewport"
- "Using the load editors," Section 16.9
- Chapter 23, "Connectors"
- Chapter 70, "The Set and Surface toolsets"

16.8.2 Creating boundary conditions

When you create a boundary condition, you must specify the name of the boundary condition, the step in which to activate the boundary condition, the type of boundary condition, and the region of the assembly to which you want to apply the boundary condition.

To create a boundary condition:

1. From the main menu bar, select **BC→Create**.

   A **Create Boundary Condition** dialog box appears with a default name displayed in the **Name** text field.

   Tip: You can also create a boundary condition using the tool in the Load module toolbox.

2. Type a name for the boundary condition. For more information on naming objects, see "Using basic dialog box components," Section 3.2.1.
3. Select the step in which to activate the boundary condition. Click the arrow next to the **Step** text field, and select from the list that appears.
4. From the **Category** list on the left side of the dialog box, choose the desired category. The **Category** choices available are dependent upon the type of analysis procedures you are performing.

   The **Types for Selected Step** list on the right side of the dialog box changes to a list of all the available boundary condition types.

5. From the **Types for Selected Step** list, select the boundary condition type and click **Continue**.
6. Select the region to which you want to apply the boundary condition.

   If you are creating a connector displacement, connector velocity, or connector acceleration boundary condition, you must select wires that are associated with a connector section assignment. The best approach for selecting wires is to use the default geometry set name for the wire feature (see "Creating or modifying wire features for multiple connectors," Section 15.12.8, for more information). If you select multiple wires, you must ensure that the connector sections assigned to the wires in the connector section assignments have the available components of relative motion for which you want to define displacement, velocity, or acceleration. If there are insufficient available components of relative motion for the connector boundary condition, a message appears asking you to select different wires or to change the connection type.

If you are creating a connector material flow boundary condition, you must select endpoints of wires that are associated with a connector section assignment.

Use one of the following methods to select the region for the boundary condition:

- Select a region in the viewport. You can use the angle method to select a group of faces or edges from a native geometric part instance or a group of element faces from an orphan mesh part instance. For more information, see "Using the angle method to select multiple objects," Section 6.2.3. When you have finished selecting, click mouse button 2.

  Tip: You can limit the types of objects that you can select in the viewport by clicking the selection options tool in the prompt area and then clicking the selection filter of your choice in the dialog box that appears. See "Using the selection options," Section 6.3, for more information.

  If the model contains a combination of orphan mesh instances and native part instances, you must choose the type of region to which you want to apply the boundary condition. From the prompt area, select one of the following:

  - Click **Geometry** to apply the boundary condition to a native part instance or to a reference point.
  - Click **Mesh** to apply the boundary condition to an orphan mesh instance.
- To select from a list of existing sets or surfaces, do the following:

a. Click **Sets** or **Surfaces** on the right side of the prompt area. (The name of the button depends on the type of object you are creating. For example, if you are creating a pressure load, a **Surfaces** button appears.)

Abaqus/CAE displays the **Region Selection** dialog box containing a list of available sets or surfaces.

b. Select the set or surface of interest and click **Continue**.

Note: The default selection method is based on the selection method you most recently employed. To revert to the other method, click **Select in Viewport** or **Sets** or **Surfaces** on the right side of the prompt area.

The boundary condition editor appears. The region to which you are applying the boundary condition is highlighted in the viewport.

7. Enter all of the data necessary to define the boundary condition and click **OK**.

Note: If you create a connector displacement boundary condition that exceeds the failure criteria for a connector, the connector displacement will be ignored.

For detailed information on a particular feature of the editor, select **Help⟶On Context** from the main menu bar and then click the feature of interest or see "Using the boundary condition editors," Section 16.10.

Symbols appear in the viewport that represent the boundary condition that you just created. For more information, see "Understanding symbols that represent prescribed conditions," Section 16.5.

For information on related topics, click any of the following items:

- "Understanding and using toolboxes and toolbars," Section 3.3
- "What are step-dependent managers?," Section 3.4.2
- Chapter 6, "Selecting objects within the viewport"
- "Using the boundary condition editors," Section 16.10
- Chapter 23, "Connectors"
- Chapter 70, "The Set and Surface toolsets"

16.10.1 Defining a symmetry/antisymmetry/encastre boundary condition

You can define a boundary condition by selecting one of the common types listed in the symmetry/antisymmetry/encastre boundary condition editor.

To create or edit a symmetry/antisymmetry/encastre boundary condition:

1. Display the symmetry/antisymmetry/encastre boundary condition editor using one of the following methods:
   - To create a new symmetry/antisymmetry/encastre boundary condition, follow the procedure outlined in "Creating boundary conditions," Section 16.8.2 (**Category**: `Mechanical`; **Types for Selected Step**: `Symmetry/Antisymmetry/Encastre`).
   - To edit an existing symmetry/antisymmetry/encastre boundary condition using menus or managers, see "Editing step-dependent objects," Section 3.4.12. You can edit the symmetry/antisymmetry/encastre boundary condition only in the step in which it was created.
2. If you are creating the boundary condition in a buckling step, select the **Use BC for** option that specifies the calculations for which you want the boundary

condition used. For more information, see "Boundary conditions," in "Eigenvalue buckling prediction," Section 6.2.3 of the Abaqus Analysis User's Manual.

3. Select one of the following options:

   XSYMM Symmetry about a plane X = constant (U1 = UR2 = UR3 = 0).

   YSYMM Symmetry about a plane Y = constant (U2 = UR1 = UR3 = 0).

   ZSYMM Symmetry about a plane Z = constant (U3 = UR1 = UR2 = 0).

   XASYMM Antisymmetry about a plane with X = constant (U2 = U3 = UR1 = 0;Abaqus/Standard only).

   YASYMM Antisymmetry about a plane with Y = constant (U1 = U3 = UR2 = 0;Abaqus/Standard only).

   ZASYMM Antisymmetry about a plane with Z = constant (U1 = U2 = UR3 = 0;Abaqus/Standard only).

   PINNED Pinned (U1 = U2 = U3 = 0).

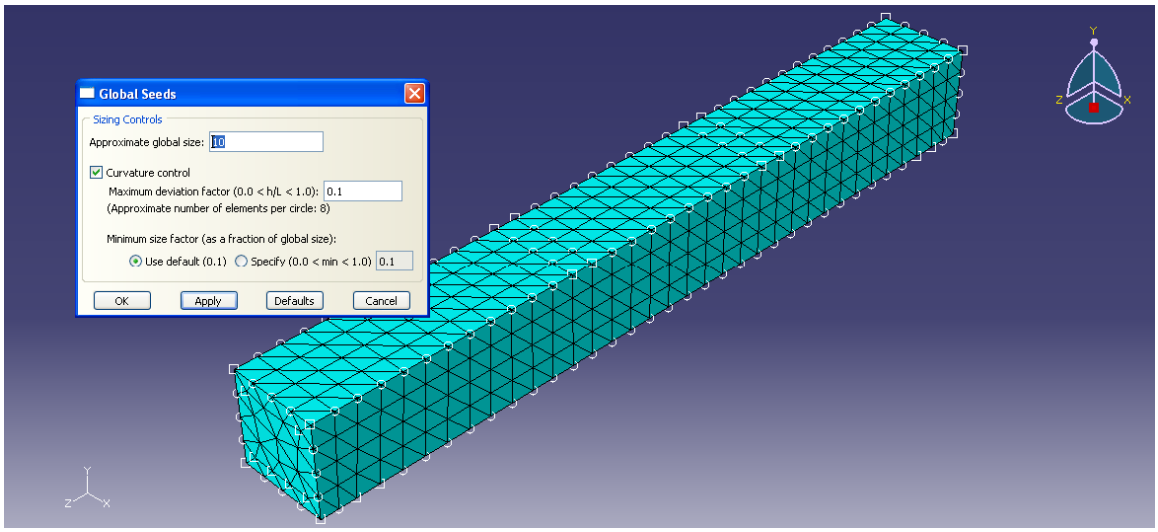
   ENCASTRE Fully built-in (U1 = U2 = U3 = UR1 = UR2 = UR3 = 0).

4. Click **OK** to save your data and to exit the editor.

For information on related topics, click any of the following items:

- "Using the Load module," Section 16.8
- "Creating and modifying prescribed conditions," Section 16.4
- "Boundary conditions," Section 29.3.1 of the Abaqus Analysis User's Manual

Chapter 8– Mesh Module



7.15.1 Defining seed density for an entire part or part instance

You can select **Seed⟶Part** or **Seed⟶Instance** from the main menu to define the approximate element size for all edges of a part or part instance that do not already have magenta-colored edge seeds. Seeds defined in this way are called part seeds or instance seeds and are colored white. (For more information, see "Controlling the seed density," Section 17.4.3.)

To create part or instance seeds:

1. From the main menu bar, select **Seed⟶Part** or **Seed⟶Instance**.

   Abaqus/CAE displays prompts in the prompt area to guide you through the procedure.

   Tip: You can also seed a part or part instance using the tool, located with the seed tools in the Mesh module toolbox. (For more information, see "Using the Mesh module toolbox," Section 17.14.)

2. If you are seeding a part instance and your assembly contains more than one part instance, select the part instance to seed and click mouse button 2.

Note: Edge seeds always override part and instance seeds; therefore, if you have already individually seeded all the edges of the part or part instance, the instance seeds are unused and do not appear. If necessary, use the seed deletion tool, described in "Deleting edge seeds," Section 17.15.8, to remove any unwanted edge seeds; if you have assigned part or instance seeds, these seeds automatically appear on the edges where you delete edge seeds.

3. In the **Global Seeds** dialog box that appears, enter an approximate element size.
4. By default, Abaqus/CAE applies curvature control to the seeding of your part or part instance to allow for small holes or regions of high curvature that you wish to model. Curvature control allows Abaqus/CAE to calculate the seed distribution based on the curvature of the edge along with the target element size. To control the effect of curvature on seeding, do the following:
    a. Enter a value for the deviation factor. The deviation factor is a measure of how much the element edges deviate from the original geometry. To help you visualize the influence of the deviation factor, Abaqus/CAE displays the number of elements it would create around a circle corresponding to the setting that you enter.
    b. If desired, specify a minimum size factor as a fraction of the global element size. Specifying a minimum size factor prevents Abaqus/CAE from creating very fine meshes in areas of high curvature that you have no interest in modeling.
5. Click **Apply** to view the seeding that Abaqus/CAE will use, and adjust the values that you entered in the **Global Seeds** dialog box if necessary.
6. Click **OK** to commit the element size and to close the dialog box.

   White seeds appear on all edges of the part or part instance except those already assigned magenta edge seeds.

7. To exit the part or instance seeding procedure, press **[Enter]** or click mouse button 2.

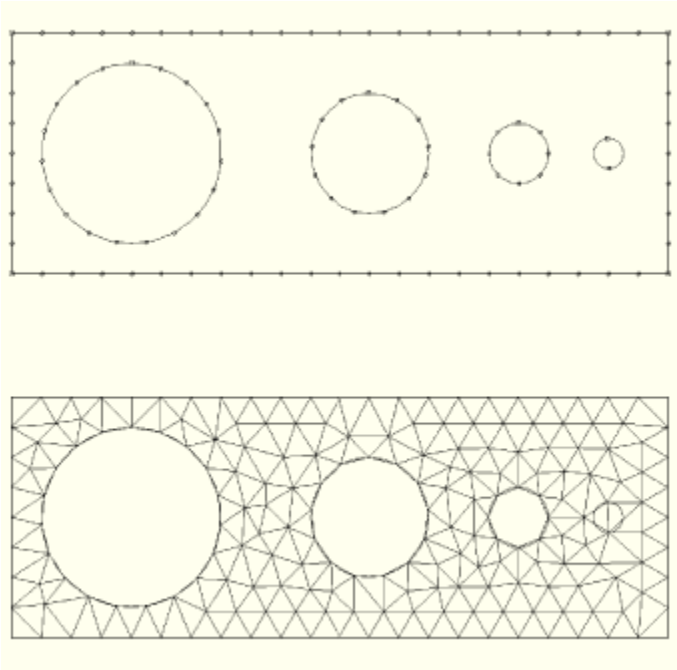
For information on related topics, click any of the following items:

- "Seeding a model," Section 17.15
- "Understanding seeding," Section 17.4
- "Using the Mesh module toolbox," Section 17.14
- Chapter 6, "Selecting objects within the viewport"

17.4.4 Applying curvature control to your seeding

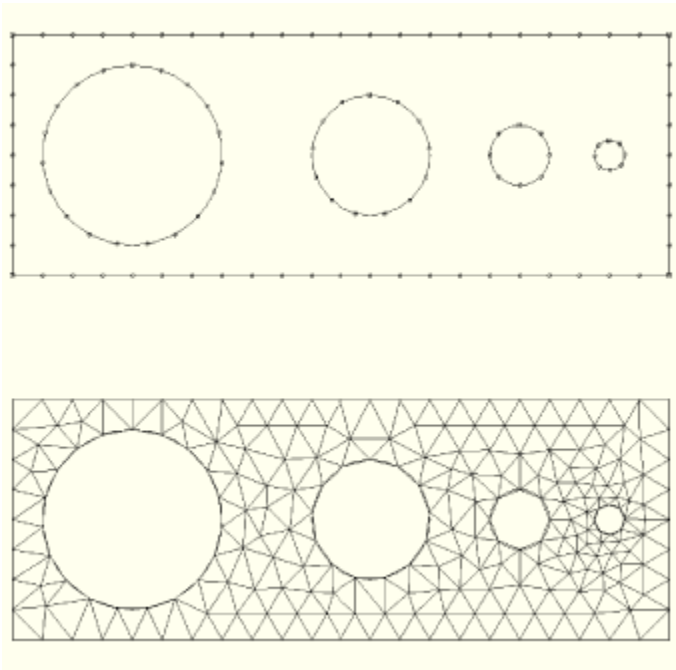
The part seeding tool allows you to specify a target element size when you are seeding a part or a part instance. If the geometry of the part is relatively regular, specifying a single target element size can result in an acceptable mesh. However, if you specify a single target element size and the geometric features that make up the part vary in size, the resulting mesh may be too coarse to adequately represent any small features, as shown in Figure 17–9.

Figure 17–9 Seeding and the resulting mesh with no curvature control.



To avoid the problem of inadequate seeding around small curved features, Abaqus/CAE applies curvature control when it seeds a part or part instance. Curvature control allows Abaqus/CAE to calculate the seed distribution based on the curvature of the edge along with the target element size. Figure 17–10 shows the same part seeded and meshed with curvature control enabled.

Figure 17–10 Seeding and the resulting mesh with curvature control enabled.

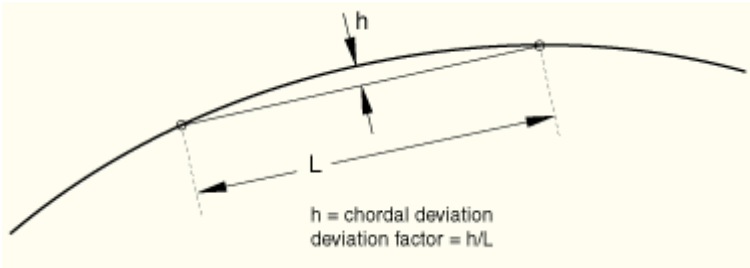


The **Global Seeds** dialog box allows you to specify how curvature control will influence the seeding. You can configure the following:

Deviation factor

The deviation factor is a measure of how much the element edges deviate from the original geometry, as shown in Figure 17–11.

Figure 17–11 Deviation factor.

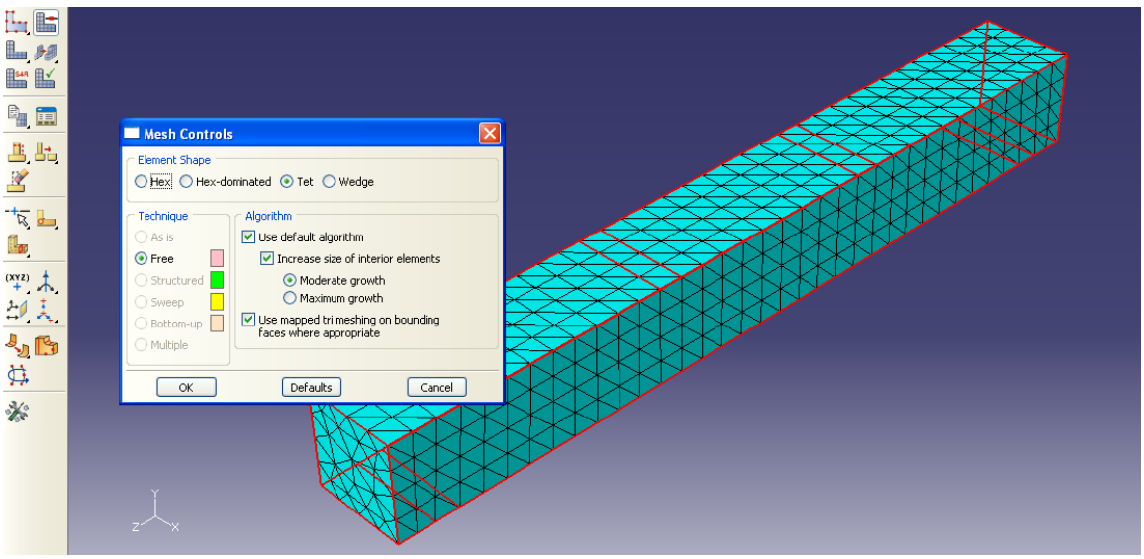


To help you visualize the deviation factor, Abaqus/CAE displays the approximate number of elements it would create around a circle corresponding to the setting that you enter. As you reduce the deviation factor, the number of elements that Abaqus/CAE would create around a circle increases. This number is only a visual aid; for example, if you are seeding a spline or an ellipse, Abaqus/CAE creates a different number of elements, depending on the local curvature along the edge.

Specify minimum size factor

Specifying a minimum size factor prevents Abaqus/CAE from creating very fine meshes in areas of high curvature that you have no interest in modeling; for example, kinks in spline curves or fillets with a very small radius. The number that you enter representing the minimum size is the fraction of the global seed size. As a result, if you change the global seed size, you do not have to change the minimum size factor.

For detailed instructions on applying curvature control, see "Defining seed density for an entire part or part instance," Section 17.15.1.



17.17.2 Choosing an element shape

You can control the shape of the elements in your mesh by selecting **Mesh⟶Controls** from the main menu bar. The **Element Shape** options are located at the top of the **Mesh Controls** dialog box that appears.

To specify the element shape to be used in a region:

1. From the main menu bar, select **Mesh⟶Controls**.

   Abaqus/CAE displays prompts in the prompt area to guide you through the procedure.

Tip: You can set the element shape using the tool, located in the Mesh module toolbox.

2. If your part or assembly contains more than one region, select those regions whose element shapes you want to view or modify and then press mouse button 2. All the selected regions must have the same dimensionality.

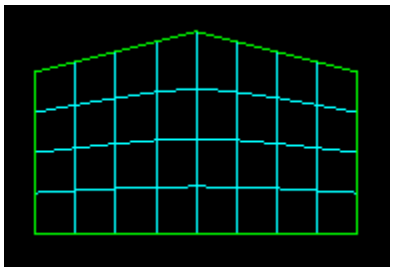
   The **Mesh Controls** dialog box appears.

3. From the list of **Element Shape** options, select the element shape of your choice.

   If you selected a two-dimensional region, you can choose from the following element shape options:

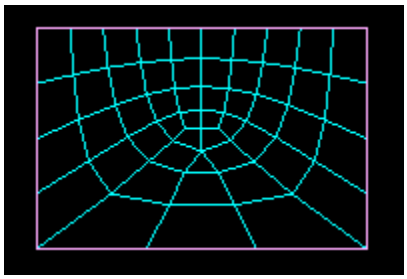
   Quad

   Use exclusively quadrilateral elements. The following figure shows an example of a mesh that was constructed using this setting:



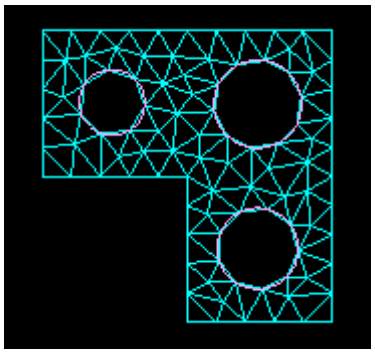
   Quad-dominated

   Use primarily quadrilateral elements, but allow triangles in transition regions. This setting is the default. The following figure shows an example of a mesh that was constructed using this setting:



   Tri

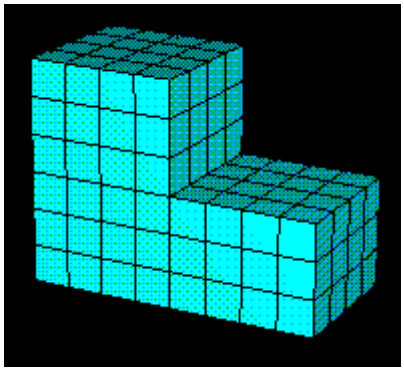
Use exclusively triangular elements. The following figure shows an example of a mesh that was constructed using this setting:



If you selected a three-dimensional region, you can choose from the following element shape options:

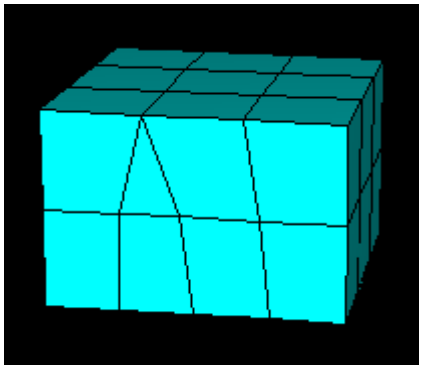
Hex

Use exclusively hexahedral elements. This setting is the default. The following figure shows an example of a mesh that was constructed using this setting:



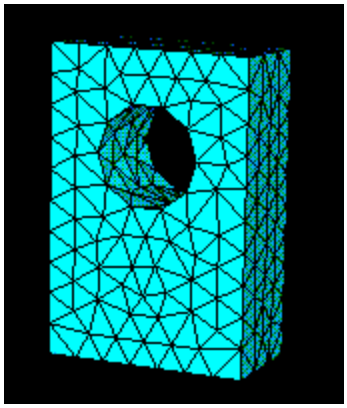
Hex-dominated

Use primarily hexahedral elements, but allow some triangular prisms (wedges) in transition regions. The following figure shows an example of a mesh that was constructed using this setting:



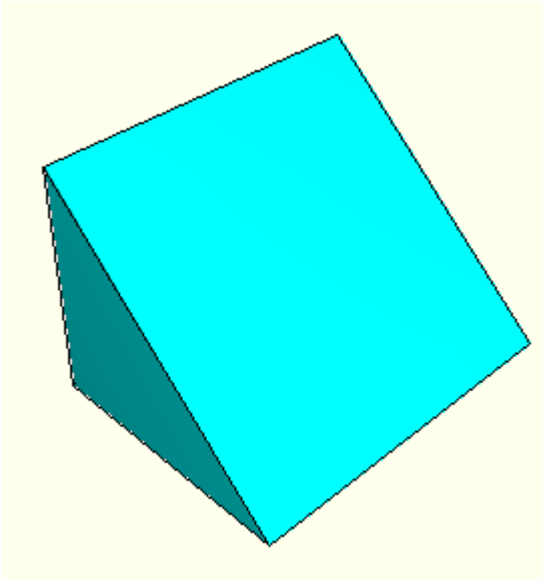
Tet

Use exclusively tetrahedral elements. The following figure shows an example of a mesh that was constructed using this setting:



Wedge

Use exclusively wedge elements. The following figure shows an example of a single-element mesh that was constructed using this setting:



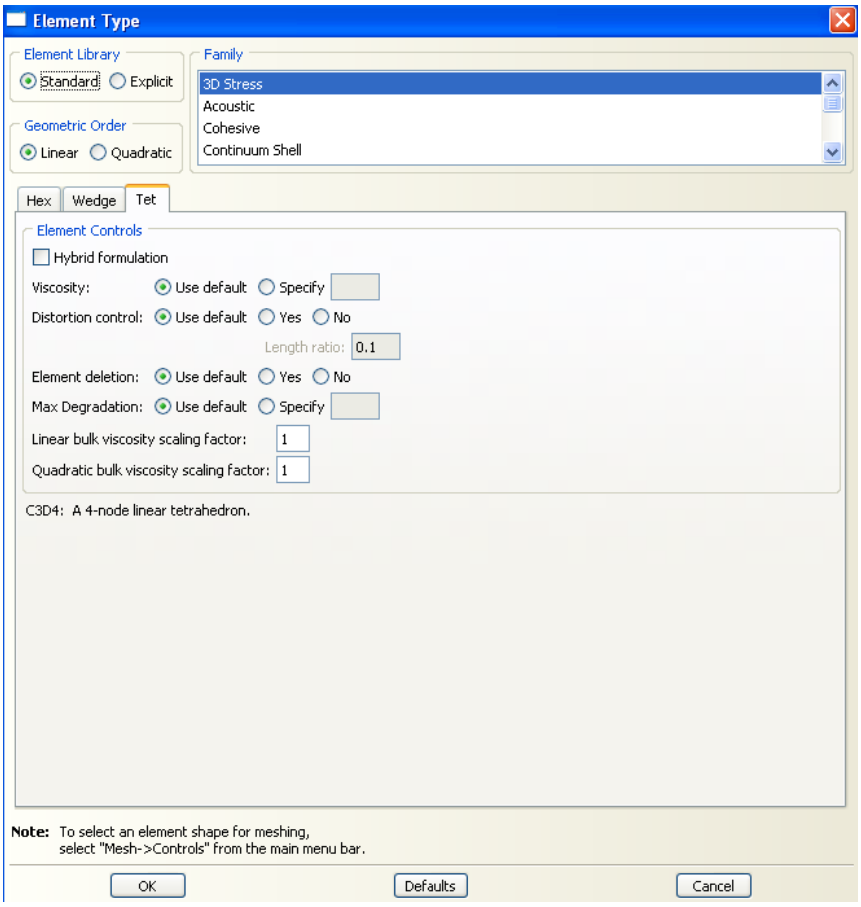
4. Click **OK**.

   The next time you generate a mesh on the selected regions, your selections will be honored.

   If the selected regions are already meshed, you will be prompted to delete the mesh or to cancel the mesh control procedure.

For information on related topics, click any of the following items:

- "Controlling mesh characteristics," Section 17.17
- "Understanding mesh generation," Section 17.7
- "Using the Mesh module toolbox," Section 17.14
- Chapter 6, "Selecting objects within the viewport"



17.17.9 Associating Abaqus elements with mesh regions

To associate particular Abaqus elements with mesh regions or with an orphan mesh, select **Mesh⟶Element Type** from the main menu bar. Then select those regions whose element type you want to assign, and make the assignment using the **Element Type** dialog box that appears.

You can use the dialog box to specify Abaqus element settings for all the element shapes that could conceivably appear in the regions you select, even if the regions currently contain only a few different element shapes. For example, even though a selected region may contain only quadrilateral elements, you can associate an element type with other shapes, too, such as triangles.

The element type setting behaves like a feature. For example, if you assign element types to a region and then later partition that region into several more regions, the new regions will inherit the element type settings of the original parent region.

To associate Abaqus elements with mesh regions:

1. From the main menu bar, select **Mesh⟶Element Type**.

   Abaqus/CAE displays prompts in the prompt area to guide you through the procedure.

   Tip: You can also choose element types using the tool, located in the Mesh module toolbox. (For more information, see "Using the Mesh module toolbox," Section 17.14.)

2. If your assembly contains both orphan meshes and native part instances, choose **geometry** in the prompt area to assign an element type to native geometry.

   Orphan meshes in an assembly are dependent part instances; you cannot assign an element type to a dependent part instance. To assign an element type to an orphan mesh you must select the orphan mesh part from the parts list in the **Object** field of the context bar and assign the element type to the desired elements of the part.

3. If you are selecting from multiple regions of Abaqus/CAE native geometry or if you are selecting elements from a part imported from an output database, use the following selection techniques:

   Abaqus/CAE native geometry

   Use the mouse to select the desired regions in the viewport, and then click mouse button 2 when your selection is complete.

   You can select only regions from parts of the same type; for example, you cannot select both a rigid surface and a deformable body. Likewise, the regions that you choose must have the same dimensionality.

   An orphan mesh part

   Use the mouse to select the desired elements of the part, and then click mouse button 2 when your selection is complete.

   Alternatively, you can click **Sets** on the right side of the prompt area. A dialog box appears with a list of all of the elements sets associated with the meshed part.

Select the element set of your choice and then click **Continue**. For information on creating sets, see Chapter 70, "The Set and Surface toolsets."

All elements that you select, regardless of the selection method, must be of the same order. In addition, the elements must belong to parts of the same type.

The **Element Type** dialog box appears.

4. In the upper left corner of the dialog box, select the **Element Library** option of your choice.

   Select **Standard** to choose from the list of Abaqus/Standard elements, or select **Explicit** to choose from the list of Abaqus/Explicit elements.

5. Select the **Geometric Order** of your choice: **Linear** (first order) or **Quadratic** (second order).
6. From the **Family** list on the right side of the dialog box, select an appropriate element family for the type of analysis you will perform on the model. For example, if you plan to do a heat transfer analysis, select the **Heat Transfer** family.

   The name of the default element for the element library, geometric order, and family that you specified appears in the lower half of the dialog box.

   Note: You can set the element type corresponding to only a single family. For example, you cannot set the element type for linear triangles in both the plane strain and heat transfer families; you must select either heat transfer or plane strain. If, after setting element types for one family, you switch to another family, the settings for the first family are lost.

7. Choose the Abaqus element type of your choice for each element shape.
   a. Click the tab corresponding to the element shape of interest.
   b. Select the element characteristics of your choice.

      The name of the Abaqus element that meets all your criteria appears at the bottom of the tabbed page with a brief description.

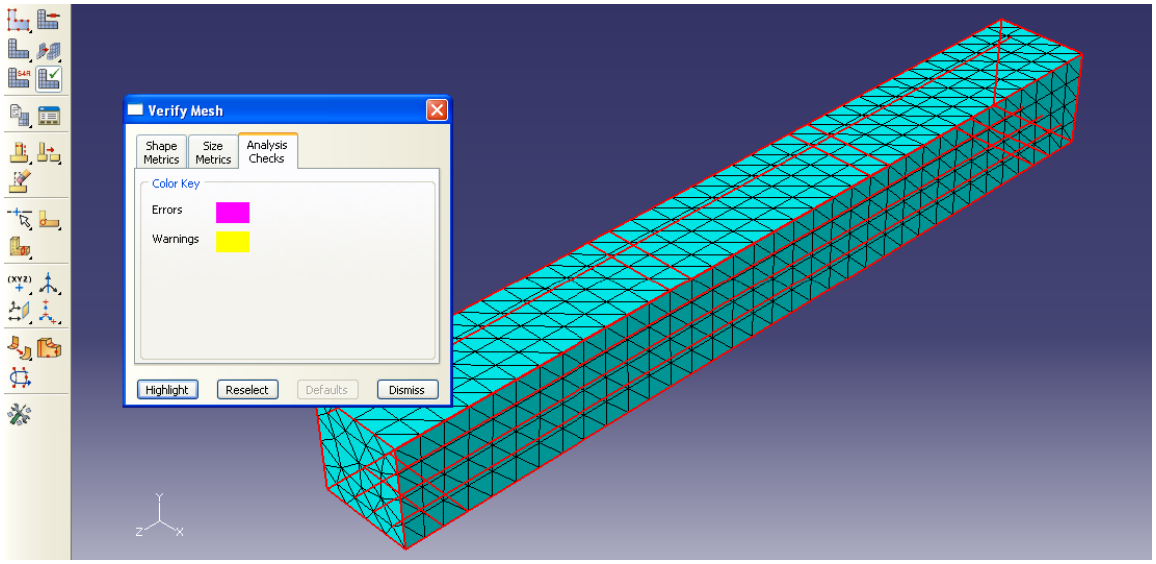
8. Click **OK** to commit your element type assignments, or click **Defaults** and then **OK** to return all element settings to their default values.

   The element types are changed according to your specifications.

9. To set element types for additional regions, repeat this procedure starting from Step 2.

For information on related topics, click any of the following items:

- "How do mesh elements correspond to Abaqus elements?," Section 17.5.1
- "What kinds of elements must be generated outside the Mesh module?," Section 17.5.2
- "Element type assignment," Section 17.5.3
- "Importing parts," Section 10.7.2
- "Using the prompt area during procedures," Section 3.1
- "Using the Mesh module toolbox," Section 17.14
- Chapter 6, "Selecting objects within the viewport"



17.18.1 Verifying element quality

To verify the quality of a mesh, select **Mesh⟶Verify** from the main menu bar. The mesh verify tool allows you to do the following:

- Select a part, or select one or more part instances or regions; and highlight elements that do not meet specified criteria, such as aspect ratio. You can also obtain mesh statistics for each selected part, part instance, or region, such as the total number of elements, the number of highlighted elements, and the average and worst values of the selection criterion.

- Select a part, or select one or more part instances or regions; and highlight elements that do not pass the mesh quality tests that are included with the input file processor in Abaqus/Standard and Abaqus/Explicit.

You can also obtain quality information for individual elements. For more information, see "Verifying your mesh," Section 17.6.1.

To verify selected elements:

1. To verify the quality of selected elements, select **Mesh⟶Verify** from the main menu bar.

   Abaqus/CAE displays prompts in the prompt area to guide you through the procedure.

   Tip: You can also verify selected elements using the tool, located in the Mesh module toolbox. (For more information, see "Using the Mesh module toolbox," Section 17.14.)

2. From the **Select the regions to verify by** field in the prompt area, select **Element**.
3. Select the element that you want to verify. Abaqus/CAE displays the following in the message area:
   - The name of the part or part instance
   - The element index
   - The element shape
   - The shape factor for triangle and tetrahedra elements
   - The minimum and maximum face corner angles
   - The aspect ratio
   - The geometric deviation factor
   - The stable time increment
   - The maximum allowable frequency for acoustic elements
   - The shortest edge and longest edge
   - Whether the element passes the checks found in the input file processor in Abaqus/Standard and Abaqus/Explicit
4. Continue selecting elements, as desired.
5. When you have finished selecting elements, either
   - Click mouse button 2 in the viewport, or
   - Select any other tool from the toolbox, or
   - Click the cancel button in the prompt area, or
   - Click the verify mesh tool in the Mesh module toolbox.

To verify a part, a part instance, or a region:

1. From the **Object** field in the context bar, select a part or select the assembly.

2. From the main menu bar, select **Mesh⟶Verify** from the main menu bar.

   Abaqus/CAE displays prompts in the prompt area to guide you through the procedure.

   Tip: You can also verify a mesh using the tool, located in the Mesh module toolbox. (For more information, see "Using the Mesh module toolbox," Section 17.14.)

3. From the text field in the prompt area, select the type of region to verify:
   - Select **Part** or **Part Instances** and select the part or part instances whose mesh you want to verify, and press mouse button 2.
   - **Geometric Regions**. Select the cells, faces, or edges whose mesh you want to verify, and press mouse button 2.

   Abaqus/CAE displays the **Verify Mesh** dialog box.

4. From the top of the **Verify Mesh** dialog box, click the tab corresponding to the desired verification checks. The following verification types are available:
   - **Shape metrics**
   - **Size metrics**
   - **Analysis checks**
5. If you selected **Shape metrics**, do the following:

a. Select the element shape to verify.

b. Choose one of the following selection criteria and enter a value:
   - **Shape factor**
   - **Smaller face corner angle**
   - **Larger face corner angle**
   - **Aspect ratio**

   For a detailed description of the selection criteria, see "Verifying your mesh," Section 17.6.1.

c. Click **Highlight**.

   Abaqus/CAE highlights the elements that fail the element checks. In addition Abaqus/CAE displays information in the message area, such as the name of the part instance, the total number of elements, the number of highlighted elements, and the average and worst value of the selection criterion.

6. If you selected **Size metrics**, do the following:

. Choose one of the following selection criteria and enter a value:
   - **Geometric deviation factor**

- **Shortest edge**
- **Longest edge**
- **Stable time increment**
- **Maximum allowable frequency for acoustic elements**

Stable time increment is unavailable if the selected element types are not in the Abaqus/Explicit element library. **Maximum allowable frequency for acoustic elements** is unavailable if the selected element types are not acoustic elements in the Abaqus/Standard element library.

For a detailed description of the selection criteria, see "Verifying your mesh," Section 17.6.1.

a. Click **Highlight**.

Abaqus/CAE highlights the elements that fail the element checks. In addition Abaqus/CAE displays information in the message area, such as the name of the part instance, the total number of elements, the number of highlighted elements, and the average and worst value of the selection criterion.

7. If you selected **Analysis checks**, click **Highlight** to verify the mesh using the checks found in the input file processor in Abaqus/Standard and Abaqus/Explicit.

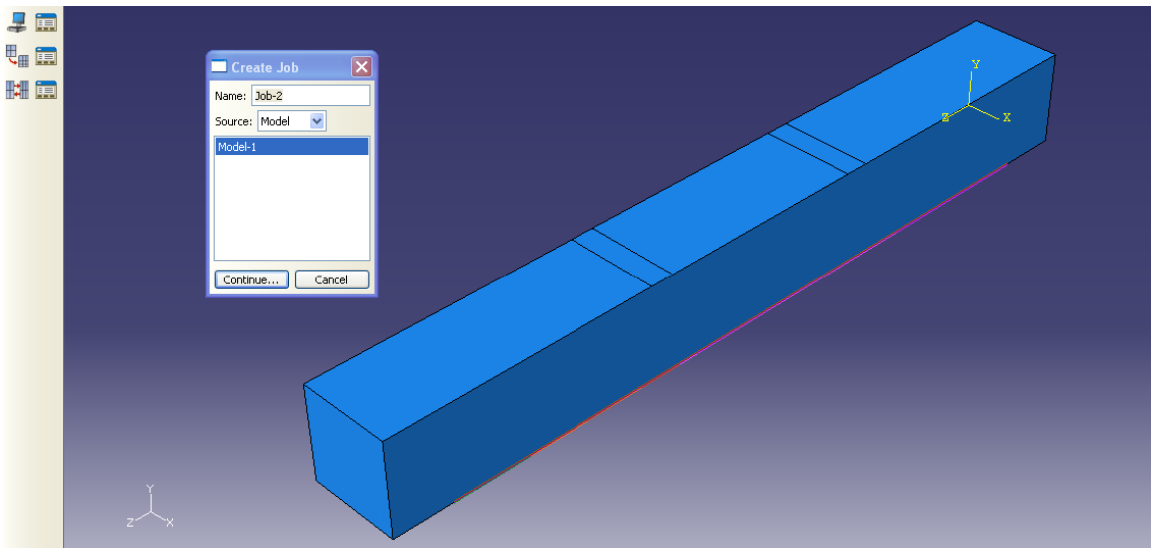
   Abaqus/CAE highlights any elements that generated error or warning messages during the mesh quality tests. Abaqus/CAE also displays in the message area the number of elements tested along with the number of errors and warnings. In most cases, it will be obvious from the element shape why the input file processor issued an error or a warning. If neccessary, you can submit a datacheck analysis from the Job module and review the messages that Abaqus writes to the data file. Abaqus/CAE does not support analysis checks for beam, gasket, or cohesive elements.

8. From the buttons along the bottom of the **Verify Mesh** dialog box, do the following:
   - Click **Reselect** to select different part instances or regions.
   - Click **Defaults** to restore the default element failure criteria on all of the tabs.
   - Click **Dismiss** to close the **Verify Mesh** dialog box.

Your changes to the mesh verification criteria are saved for use in future Abaqus/CAE sessions.

- ---

Chapter 9– Load Module



8.6.1 Creating a new analysis job

To create a new analysis job, select **Job⟶Create** from the main menu bar. Analysis jobs are stored in the model database and are maintained between sessions.

To create a new analysis job:

1. From the main menu bar, select **Job⟶Create**.

   The **Create Job** dialog box appears.

   Tip: You can also create a new analysis job by clicking the icon in the module toolbox or by clicking **Create** in the **Job Manager**.

2. Type the name of the new job in the **Name** text field. The name that you specify must adhere to the file name rules of your operating system.
3. Click the arrow to the right of the **Source** field to choose the source for the job.
   - Select **Model** to create a job based on a model created in Abaqus/CAE.

     The **Model** list displays all the models defined in the model database. From this list, select the model to associate with the new job.

   - Select **Input file** to create a job based on an input file that may or may not have been created in Abaqus/CAE.

     Click **Select**; Abaqus/CAE lists all the files in the selected directory with the file extension `.inp`. Select the input file to associate with the new job, and click **OK**.

     Note: You cannot create a job based on an input file that contains references to other results files, such as a restart analysis, an import analysis, or a submodel analysis.

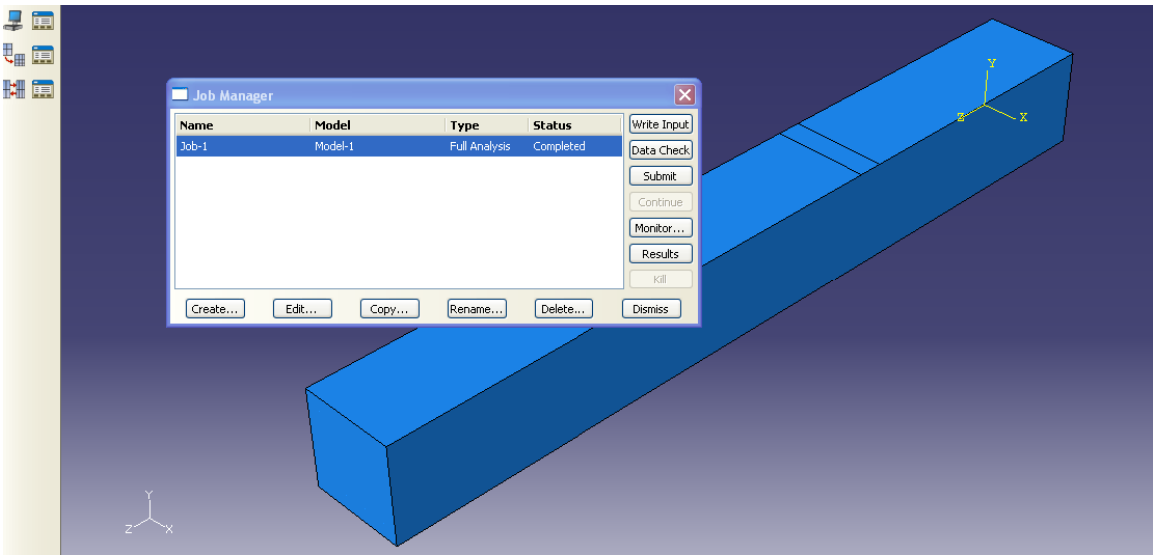
4. Click **Continue**.

   The job editor appears.

5. If desired, enter the job description. When you submit the job for analysis, Abaqus/CAE writes the job description immediately following the input file header. The job description is not written to the output database and is not retained upon import into Abaqus/CAE. For more information, see "Importing descriptions" in "Importing a model from an Abaqus/Standard or an Abaqus/Explicit input file," Section 10.5.2.
6. In the editor, enter all data necessary to define the job and click **OK**. (For more information, see "Using the job editor," Section 18.7.)

For information on related topics, click any of the following items:

- "Understanding analysis jobs," Section 18.2
- "Creating, editing, and manipulating jobs," Section 18.6



18.2.3 The Job Manager

The **Job Manager**, which is similar to other managers in Abaqus/CAE, allows you to do the following:

- Create an analysis job and associate the new job with a selected model or input file.
- Edit the selected analysis job.
- Copy, rename, or delete the selected analysis job.

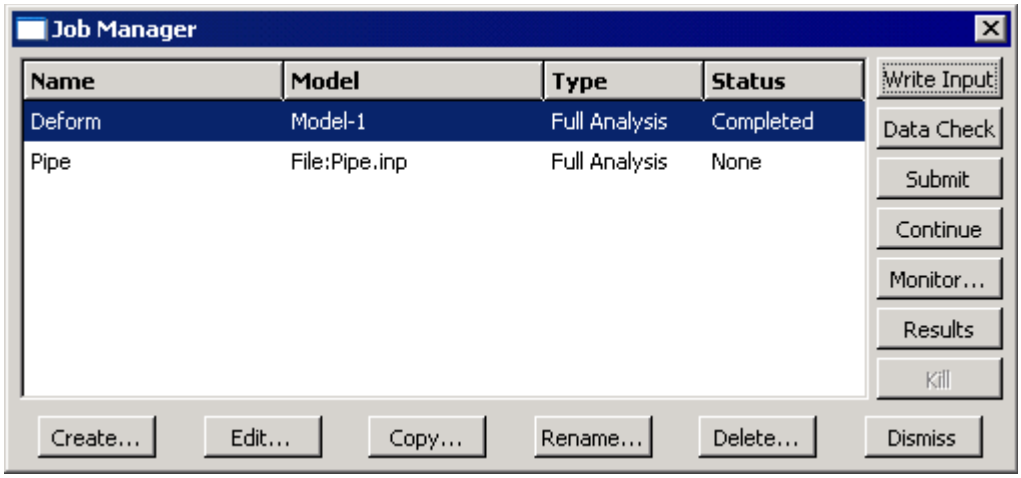
  Note: A job associated with a model can be copied only to a job associated with a model. A job associated with an input file can be copied only to a job associated with an input file.

In addition, the **Job Manager** allows you to do the following:

- Write an input file for a model-based job without submitting it for analysis.
- Perform a data check on a model.
- Submit a job for analysis.
- Continue an analysis to completion after performing a data check.
- Monitor the analysis as it progresses.
- View the results from a job.
- Kill a job that is currently running.

You can display the **Job Manager** by selecting **Job⟶Manager** from the main menu bar. Figure 18–1 shows the layout of the **Job Manager**.

Figure 18–1 The Job Manager.



The four columns of the **Job Manager** display the following:

Name

The **Name** column displays the name of the job. Click **Rename** to rename the selected job.

Model

The **Model** column displays the name of the model or input file associated with the job.

Type

The **Type** column displays the job type that you selected when you configured the job using the job editor. The job type can be one of the following:

- **Full Analysis**
- **Recover**
- **Restart**

(See "Selecting a job type," Section 18.2.5, for more information.) You can use the job editor to change the job type as long as the job is not running.

Status

The **Status** column displays the current status of the analysis job and is updated continually while your job is running. The status can be one of the following:

None

The job has not been submitted for analysis.

Check Submitted

The input file has been written, and the model is being submitted for a data check.

Check Running

The data check of the model is running.

Check Completed

The data check of the model has completed successfully; you can now continue with the full analysis.

Submitted

The input file has been written, and the job is being submitted for a full analysis.

Running

The job has been submitted for a full analysis and is running.

Completed

The analysis is complete. You can click **Results** to view the contents of the output database and graphically verify your results.

Aborted

The job has been aborted due to problems such as fatal errors in the input file or lack of disk space.

Terminated

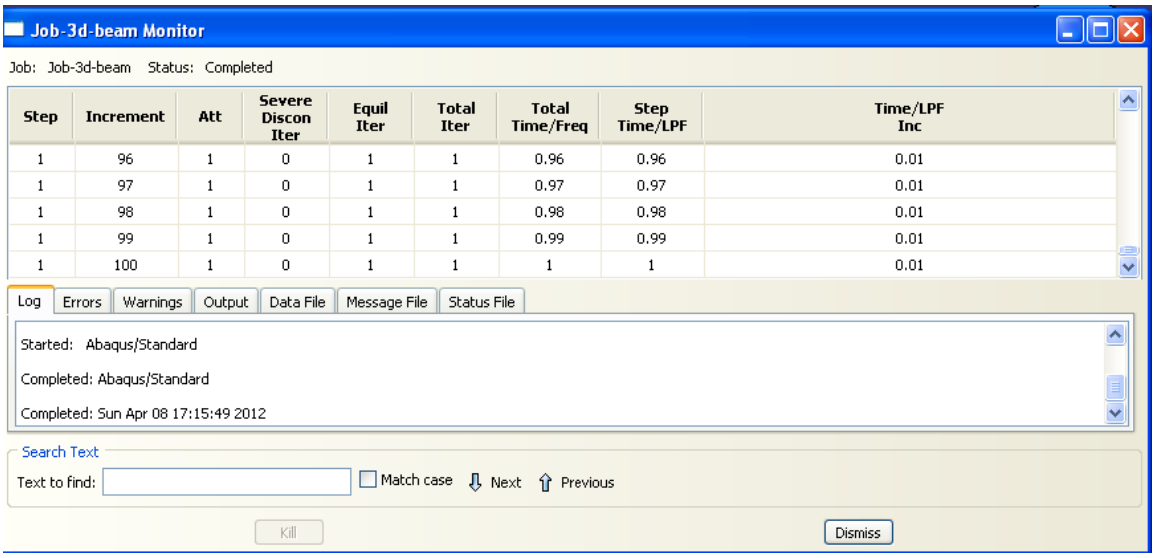
The job has been killed by the user.

For detailed instructions on using the **Job Manager** to create, edit, and manipulate jobs, see the following sections:

- "Creating a new analysis job," Section 18.6.1
- "Writing the input file only," Section 18.6.2
- "Performing a data check on a model," Section 18.6.3
- "Submitting an analysis job," Section 18.6.4
- "Terminating an analysis job," Section 18.6.6
- "Viewing the results of your job," Section 18.6.7

For information on related topics, click any of the following items:

- "Understanding analysis jobs," Section 18.2
- "Creating, editing, and manipulating jobs," Section 18.6
- "Using the job editor," Section 18.7



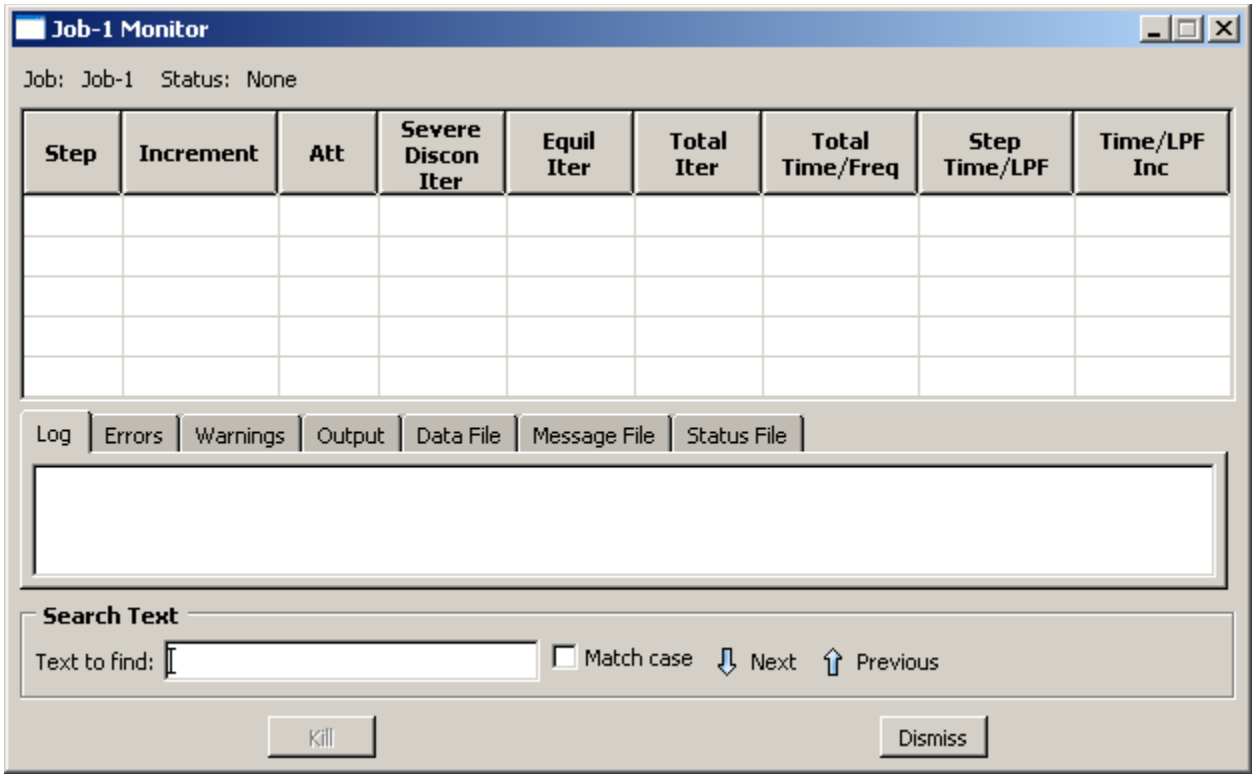
18.2.6 Monitoring the progress of an analysis job

The **Job Manager** and **Co-execution Manager** continually update the status of analysis jobs in the model database. In addition, Abaqus/CAE prints error messages from the analysis products to the message area and creates diagnostic files in your current working directory.

You can monitor information concerning a submitted job by selecting **Job→Monitor→** ***job of your choice*** from the main menu bar or by selecting the job of your choice and clicking **Monitor** in the **Job Manager**. The job monitor dialog box for that job appears,

as shown in Figure 18–2. You can display as many job monitors as necessary to view information on multiple jobs.

Figure 18–2 The job monitor.



The jobs submitted for a co-execution appear in the Model Tree in the **Jobs** container under the co-execution in the **Co-executions** container. You can monitor these jobs by clicking mouse button 3 on the job in the Model Tree and selecting **Monitor**.

The top half of the job monitor dialog box displays the information available in the status file that Abaqus creates for the analysis. The bottom half of the dialog box displays the following information:

- Click the **Log** tab to display the start and end times that appear in the log file for the analysis.
- Click the **Errors** and **Warnings** tabs to display the errors or the warnings associated with the analysis. Abaqus/CAE indicates the presence of errors or warnings by prepending an exclamation point before the **Errors** and **Warnings** tab. If a particular region of the model is causing the error or warning, a node or element set will be created automatically that contains that region. The name of the node or element set appears with the error or warning message, and you can view the set using display groups in the Visualization module. (For more

information on display groups, see Chapter 74, "Using display groups to display subsets of your model.")

Abaqus/CAE may not perform all consistency checks when it creates an input file, which can result in warnings or error messages during an analysis. If your analysis generates warning or error messages, consider running a data check analysis (see "Performing a data check on a model," Section 18.6.3) to diagnose and fix possible problems in your model.

The number of error and warning messages that appear in the job monitor is limited by the environment parameters **cae_error_limit** and **cae_warning_limit**, respectively (see "Job customization parameters," Section 4.1.3 of the Abaqus Installation and Licensing Guide, for details). If the number of errors or warnings exceeds the job monitor limit, consult the data, message, or status file for a complete list of messages.

- Click the **Output** tab to display a record of each output data entry as it is written to the output database. In addition, if you requested that Abaqus monitor the values of a degree of freedom of a particular node to the message and status files, the **Output** tabbed page records each time this information is written and the value of the degree of freedom at that point of the analysis. (For more information, see "Understanding output requests," Section 14.4, and "Degree of freedom monitor requests," Section 14.5.4.)
- As the analysis proceeds, Abaqus creates the data file, the message file, and, for Abaqus/Explicit analyses, the status file; and Abaqus/CAE activates the **Data File**, **Message File**, and **Status File** tabs accordingly. You can click any of these tabs to browse or search the corresponding file for additional error and warning messages.

  Note: Abaqus/CAE populates the **Data File**, **Message File**, and **Status File** tabbed pages only for locally submitted analyses; this information will not be displayed in the job monitor for remote jobs. In addition, although Abaqus/CAE updates the contents of these pages periodically as the analysis runs, the data might not always be synchronized with the latest data in the files.

For detailed information on the different output files that Abaqus creates during an analysis, see "Output," Section 4.1.1 of the Abaqus Analysis User's Manual.

You can search for specific error or warning messages within any of the files displayed in tabbed pages in the job monitor. Select the desired file tab, enter a search string in the **Text to find** field, and click **Next** or **Previous** to step through the file from one hit to the next. Toggle on **Match case** to perform a case-sensitive search.

The information presented in the job monitor dialog box is updated continually as the analysis progresses. If the job fails, the **Errors** tabbed page appears in front of the other

tabbed pages automatically to help you determine the cause of the failure. In addition, an exclamation point appears on the tab if any error or warning messages are output.

If you start monitoring a job and then exit Abaqus/CAE, close the current model database, or open a new model database, Abaqus/CAE will stop updating the job monitor. The job will continue to run; however, the job monitor will not report the status of the job or update the increment information.

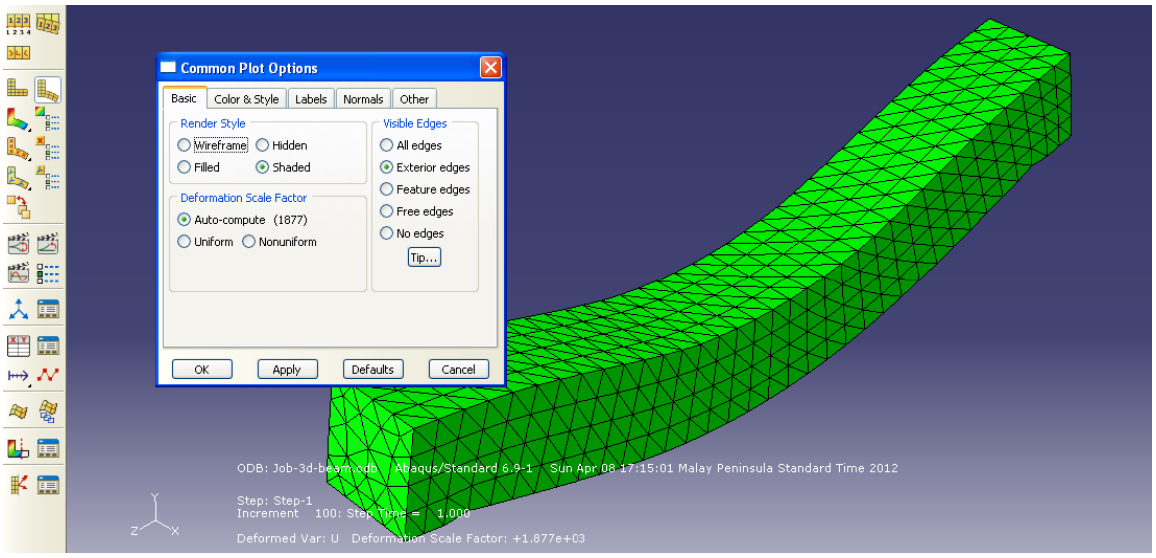
If you requested **DOF Monitor** output on a particular degree of freedom for a particular node, Abaqus/CAE provides another opportunity to monitor the job by plotting the values of the degree of freedom over time. The plot appears in a new viewport that is generated automatically when you submit the job. If the visible part of the canvas is already filled with one or more viewports, the new viewport may be placed on a part of the canvas that is not visible; in this case you should tile or cascade the viewports or enlarge the canvas to bring the viewport into view. (For information on requesting output for a particular degree of freedom for a particular node, see "Degree of freedom monitor requests," Section 14.5.4.)

If necessary, you can terminate the analysis job by clicking **Kill** at the bottom of the job monitor dialog box.

For information on related topics, click any of the following items:

- "Understanding analysis jobs," Section 18.2
- "Understanding co-executions," Section 18.4
- "Creating, editing, and manipulating jobs," Section 18.6
- "Creating, editing, and manipulating co-executions," Section 18.10

Chapter 10– Visualization Module



39.3 Overview of common plot options

You can use the common plot options to customize the appearance of undeformed and deformed plots. The common options are also used in conjunction with the other plot state–independent options in the **View** menu and the dependent and independent customization options for contour, symbol, and material orientation plots. Select **Options⟶Common** from the main menu bar or use the tool in the toolbox to access the **Common Plot Options** dialog box. Click the following tabs to customize the appearance of plots in the current viewport:

- **Basic**: Choose render style, edge visibility, and deformation scale factor (deformed plot only).

- **Color & Style**: Control model edge color and style, model face color, edge style, and edge thickness.
- **Labels**: Control element, face, and node labels and node symbols.
- **Normals**: Control element and surface normals.
- **Other**: The **Other** page contains the following tabs:
    - **Scaling**: Control model scaling and shrinking.
    - **Translucency**: Control shaded and filled render style translucency.

If you choose to superimpose the undeformed and deformed model shapes, the common options control the display of the deformed shape and the superimpose plot options control the display of the undeformed shape (for more information, see "Superimposing deformed and undeformed model plots," Section 39.6). Plot state–dependent options may override some common options; for example, the color options for contour plots will always override the common color options.

To learn how to customize the render style and other display characteristics of your plots, see Chapter 52, "Customizing plot display."

40.3 Producing a contour plot

A contour plot displays the values of an analysis variable at a specified step and frame. Abaqus/CAE represents the values as customized colored lines, colored bands, or colored faces on your model or as colored tick marks on lines drawn normal to your model. For more information on selecting an analysis variable, see "Selecting the primary field output variable," Section 38.4.3. For more information on selecting a specific step and frame, see "Selecting a specific results step and frame," Section 38.2.1. To learn how to display the minimum and maximum values associated with your contour plot, see "Customizing the legend," Section 53.1.

To produce a contour plot:

1. Use the **File** menu to open the output database containing your analysis results.
2. Use the **Result** menu to select the following:
    a. The step and frame to display.
    b. The primary field output variable to display.
    c. The deformed field output variable to display (for contours on the deformed shape only).
    d. The quantity to plot and averaging options.
3. Select the plot state–independent and contour plot customization options that you want.

4. If your chosen field output variable includes complex numbers, select the **Complex Form** tab in the **Result Options** dialog box to control the numeric form to display.
5. From the main menu bar, select **Plot⟶Contours** and choose whether to plot the contours on the undeformed shape, the deformed shape, or both.

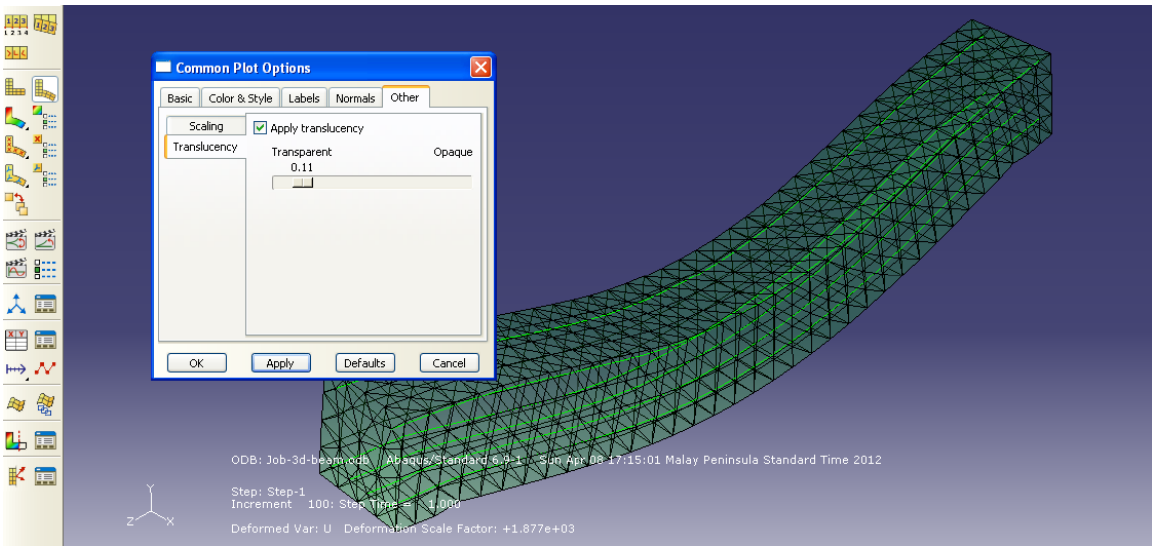
   Tip: You can also produce a contour plot using the deformed , undeformed , or superimposed contour tools in the toolbox.

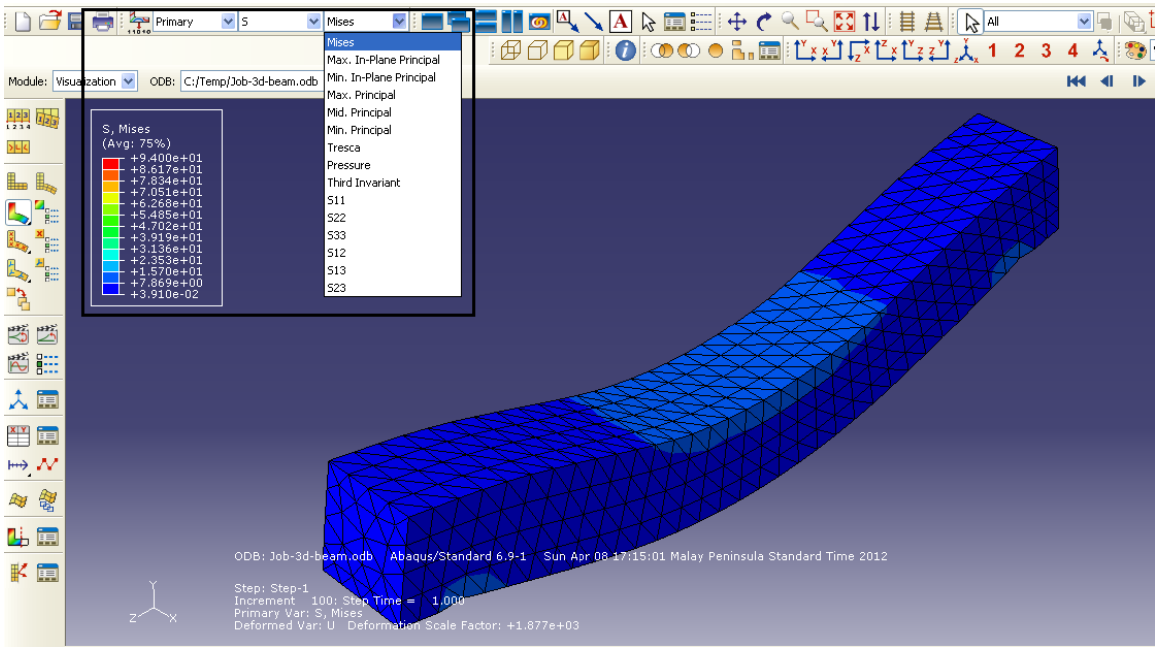
   The current viewport displays a customized contour plot of the specified field output variable at the specified step and frame of the current output database.

   Abaqus automatically refreshes your contour plot each time you click **Apply** in the step and frame selector, field output options, plot state–independent options, superimpose plot options (if applicable), or contour plot options dialog boxes.

For information on related topics, click any of the following items:

- "Overview of contour plot options," Section 40.2
- Chapter 38, "Selecting model data and analysis results to plot"





38.4.2 Using the field output toolbar

You can use the **Field Output** toolbar to access the basic functionality of the **Field Output** dialog box. From the toolbar, you can

- choose the type of field output variables to manipulate (**Primary**, **Deformed**, or **Symbol**),
- choose the variable name from a list of the available field output variables, and
- choose the refinement level, such as invariants and components for the selected primary variable, if available.

The **Status** variable type is the only field output that you cannot select from the toolbar. The **Status** variable type is available in the **Field Output** dialog box; it allows you to specify criteria that Abaqus/CAE uses to remove failed elements from the model display.

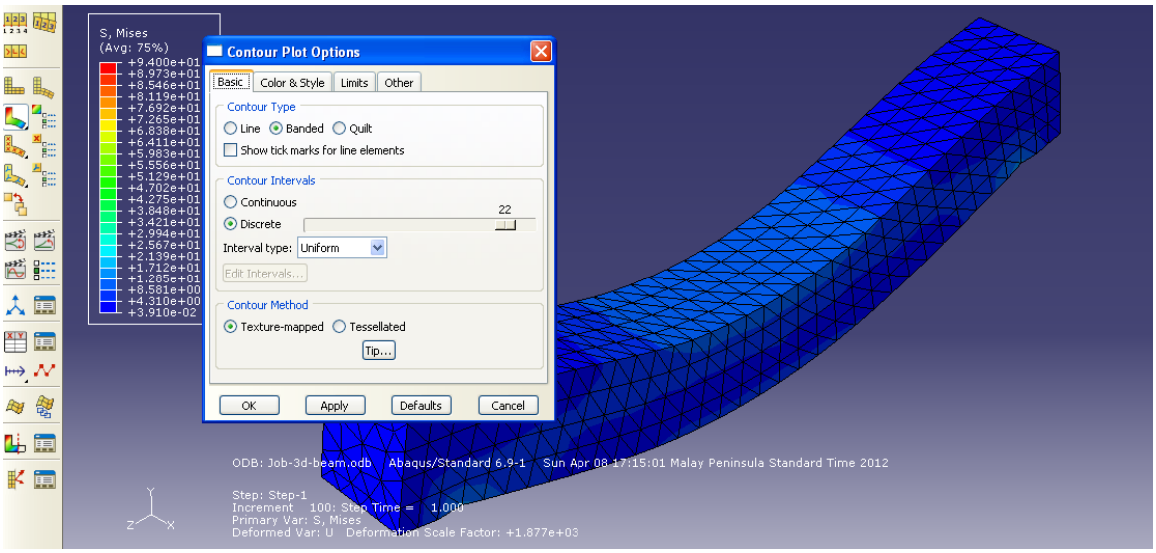
To open the **Field Output** dialog box, click , located on the left side of the **Field Output** toolbar.

Figure 38–2 The Field Output toolbar.

As you make selections from the toolbar, Abaqus/CAE updates the current viewport to display the output; the viewport plot state is also updated, as needed. For example, selecting **Primary** as the variable type changes the plot state to display contours on the deformed model if the viewport does not already contain a contour plot.

For information on related topics, click any of the following items:

- Chapter 38, "Selecting model data and analysis results to plot"
- "Overview of field output variable selection," Section 38.4.1



40.2 Overview of contour plot options

You can use the contour plot options to customize the appearance of contour plots. Select **Options→Contour** from the main menu bar or click in the toolbox to access the **Contour Plot Options** dialog box. Click the following tabs to customize the appearance of contour plots in the current viewport:

- **Basic**: Choose the contour type (including whether to show tick marks for line elements), contour intervals, and contour method.
- **Color & Style**: The **Color & Style** page contains the following tabs:
  - **Model Edges**: Control the color of model edges.
  - **Spectrum**: Choose contour colors.

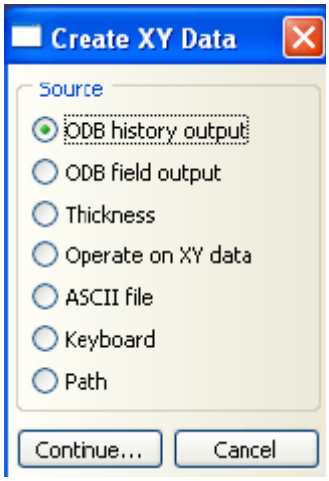
    - **Line**: For line-type contours control the style and thickness of each line.
    - **Banded**: For banded-type contours control the color, style, and thickness of contour edges.
- **Limits**: Control the computation of contour limits and the display of annotations for the maximum and minimum contour values.
- **Other**: Control tick mark plot display options, the display of nodal averaged vector or tensor orientations, and the display of section point or envelope plots.

Other options such as the render style, edge visibility, deformation scale factor, and translucency are located in the **Common Plot Options** dialog box. If you choose to plot contours on both the undeformed and the deformed shape, you can use the **Superimpose Plot Options** to customize the appearance of the undeformed shape. See "Superimposing deformed and undeformed model plots," Section 39.6, for more information about superimpose plots. To learn how to customize the render style and underlying model of your contour plot, see Chapter 52, "Customizing plot display." For information on the computation of result values, see "Understanding how results are computed," Section 38.5.1.

For information on related topics, click the following item:

- "Customizing a contour plot," Section 40.4

Tools>XY Data> Create



43.1.2 Understanding how to specify an X–Y data object

To specify an *X–Y* data object, you first choose the source of the data and then provide any necessary details. Possible sources are:

ODB history output

Select this method to specify an *X–Y* data object by reading history output results from an output database. You can specify which variables to read from the output database, from which steps of the analysis to read, and the frequency at which to read the data; for example, you can read every third data point. You can also specify a numeric form for any complex-valued output data. For more information, see "Reading X–Y data from output database history output," Section 43.2.1.

ODB field output

Select this method to specify an *X–Y* data object by reading field output results from an output database. You can specify which variables to read from the output database and for which elements or nodes to read the data. Abaqus/CAE extracts results from the currently active steps and frames; see "Activating and deactivating steps and frames," Section 38.3.1, for more information. You can also specify a numeric form for any complex-valued output data. For more information, see "Reading X–Y data from output database field output," Section 43.2.2.

Thickness

Select this method to specify an *X–Y* data object by reading field output results from elements through the thickness of a shell region of your model. Abaqus/CAE extracts results from the current step and frame. You can specify which variables to read from the output database and for which elements to read the data. For more information, see "Reading X–Y data through the thickness of a shell," Section 43.2.3.

Operate on X–Y data

Select this method to derive a new *X–Y* data object by manipulating previously saved *X–Y* data objects. You specify the new *X–Y* data object by applying functions and mathematical operations to existing data. An example of a function is **Combine**. If you **Combine** an *X–Y* data object containing stress versus time with an *X–Y* data object containing strain versus time, you produce an *X–Y* data object containing stress versus strain at equivalent times. For more information, see "Operating on saved X–Y data objects," Section 43.4.

ASCII file

Select this method to read *X*- and *Y*-values from an existing text file. The file can contain more than two columns of data, separated by commas, spaces, or tabs; and you can specify which columns correspond to the *X*- and *Y*-axis data. In addition, you can specify the frequency at which the data should be read from the file; for example, every third row. For more information, see "Reading X–Y data from an ASCII file," Section 43.2.4.

Keyboard

Select this method to manually type *X*- and *Y*-values into a simple table editor. Within this method, Abaqus supports several special editing techniques, as well as an option to read data from a file. For more information on this topic, see "Entering X–Y data from the keyboard," Section 43.2.5.

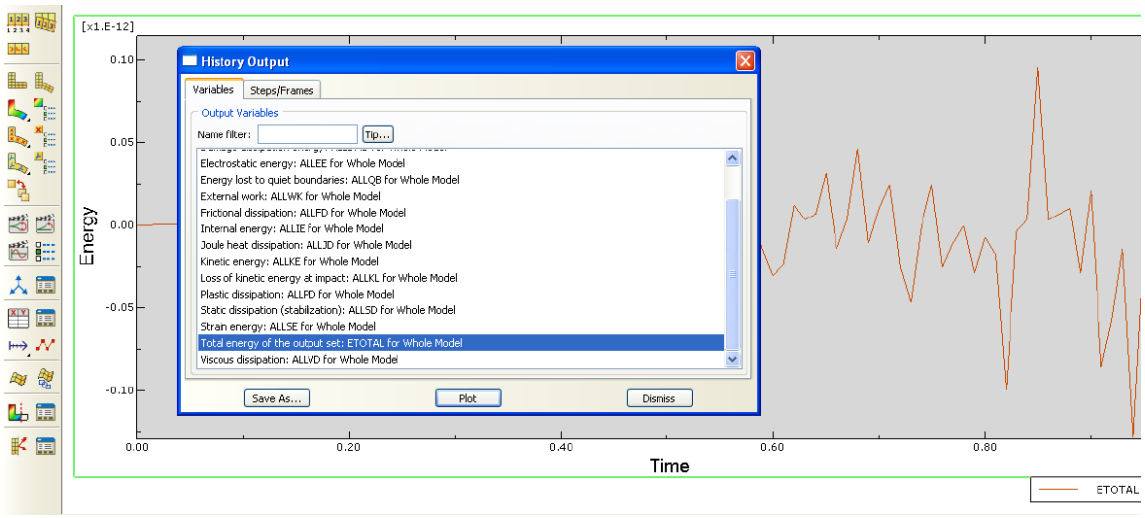
Path

Select this method to specify an *X–Y* data object by reading field output results at locations along a path through your model. Abaqus obtains results from an output database. You can specify the points, elements, or edges that make up the path and the step, frame, and variable for which to obtain results. For more information, see Chapter 44, "Viewing results along a path."

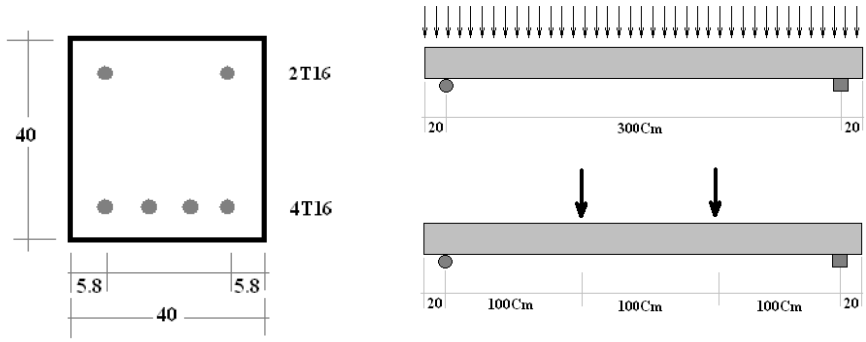
In addition, you can create an *X–Y* data object while using the table editor to create a material in the Property module. For more information, see "Entering tabular data," Section 3.2.7.

Once you have specified your *X–Y* data object, you can save it or you can display it in the form of an *X–Y* plot. Saving the *X–Y* data object allows you to subsequently plot, edit, rename, delete, or operate on it; it also allows you to copy the *X–Y* data object to an output database file for use in later Abaqus sessions. For *X–Y* data originating from sources other than output database history output, you must save your data to later produce an *X–Y* plot containing multiple data objects. Saved *X–Y* data objects are retained only for the duration of the session. For the *X–Y* data object to be persistent across sessions, you must copy it to an output database file.

If you have copied an *X–Y* data object to your output database file during an earlier session, you can access that object when you open the file in the current session. You can display the *X–Y* data object in the form of an *X–Y* plot, just as you can any other specified objects. To edit, rename, delete, or operate on a data object that was created in a previous session, you must load it to the current session.



3D Beam

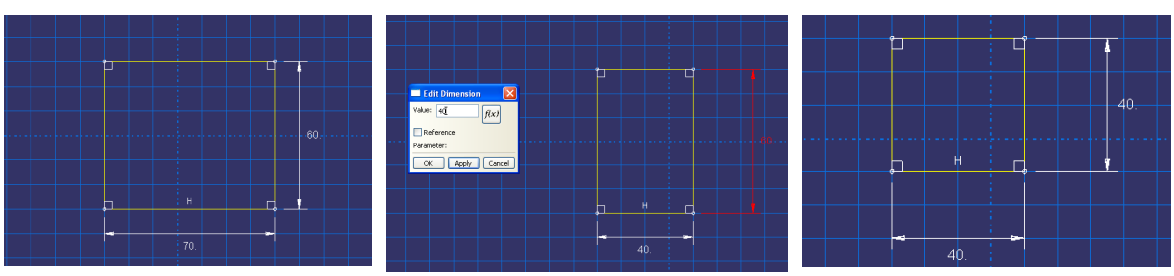


The span of reinforcement concrete beam is 3m c/c. The cross section dimension of beam is 40x40Cm with 2T16 bar on top and 4T16 on bottom. First model and analysis beam by a uniform load of 1t/m and then by a 2 concentrated load with value of 2 ton each.

Part Module

1) Part→Create part→Approximate size=400Cm→Continue

2) Create lines: Rectangle →Draw an arbitrarily rectangle →Auto Dimension → Select rectangle →Press ESC→Press Done (Fig 1a) →Edit Dimension Value →Select horizon dim→Enter 40cm in text box→ok(Fig 1b) →Select Vertical dim→Enter 40cm in text box→ok(Fig 1c).

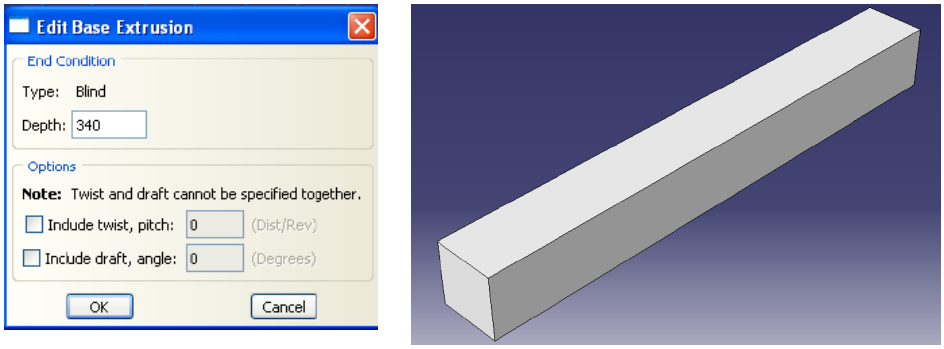


a b c

Fig. 1

3)Translate →Move→select whole section→Done→select lower left corner of the section→Select origin →x

4) Press Done→Enter 340Cm for length of beam (Fig 2a)→ok (Fig 2b).



a b

Fig. 2

4) Render model: Wireframe→Create wire: Point to point→click Disjoints wires→click Add

5.8, 5.8, 0.0, Enter, 5.8, 5.8, 340.0, Enter.
15.25, 5.8, 0.0, Enter, 15.25, 5.8, 340.0, Enter.
24.7, 5.8, 0.0, Enter, 25.7, 5.8, 340.0, Enter.
34.2, 5.8, 0.0, Enter, 34.2, 5.8, 340.0, Enter.

5.8, 34.2, 0.0, Enter, 5.8, 34.2, 340.0, Enter.
34.2, 34.2, 0.0, Enter, 34.2, 34.2, 340.0, Enter.
Ok (Fig. 3)

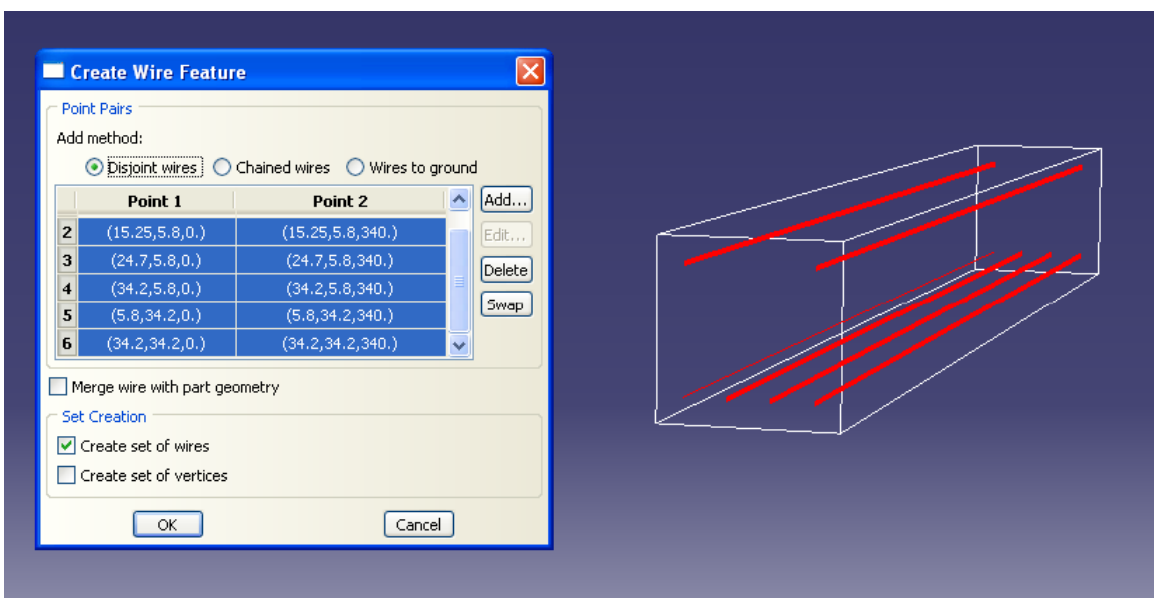


Fig. 3

Property Module

Create Material→Type concrete for Name→General→Density→Enter 0.0025→ Mechanical →Elasticity→Elastic→ Enter 218000 for Young's modulus→Enter 0.15 for Pisson's ratio→ok.

Create Material→Type steel for Name→General→Density→Enter 0.0078→ Mechanical →Elasticity→Elastic→ Enter 2100000 for Young's modulus→Enter 0.2 for Pisson's ratio→ok.

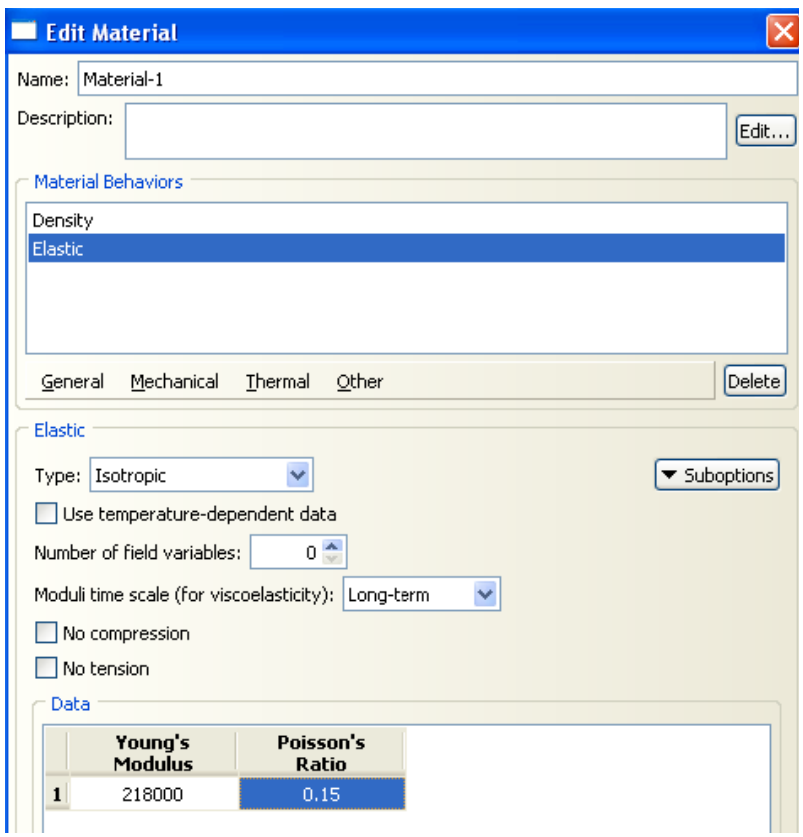
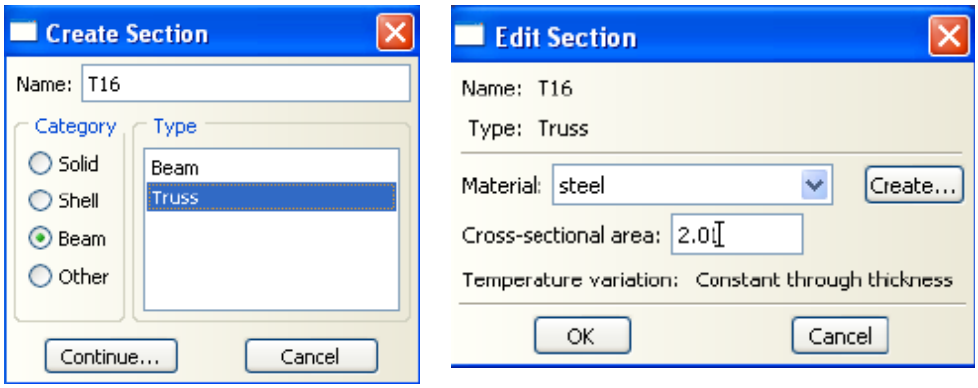


Fig. 4

Create Section→Type concrete for Name→continue→select "Concrete" from material combo box→ok.
Create Section→Type T16 for Name→beam→truss continue→ "Steel" from material combo box→Enter 2.01 for cross-sectional area→ok. Fig. 5



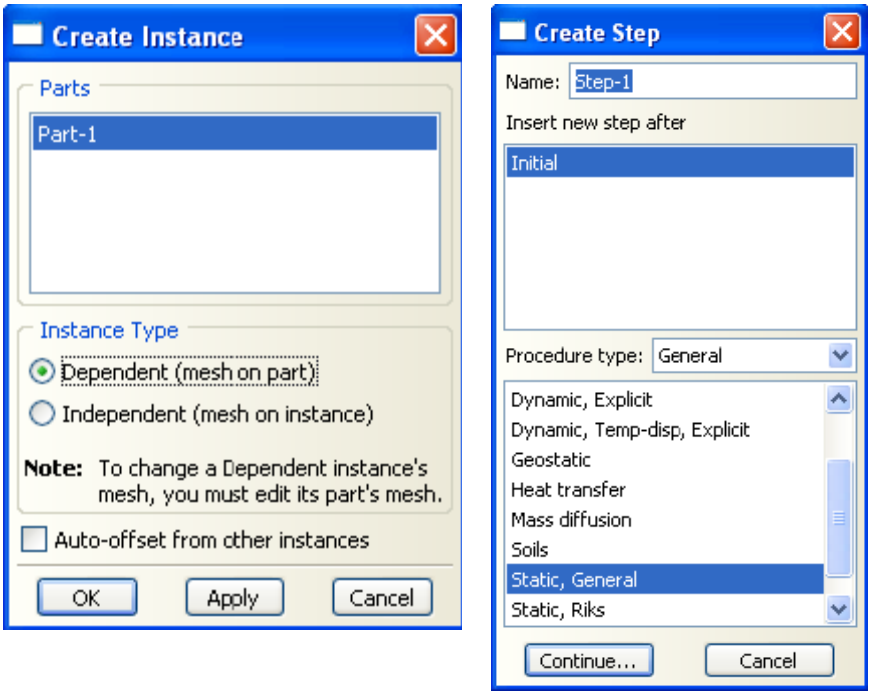
a b

Fig. 5

Assign Section→select concrete →Done →(concrete)ok →Done.
Assign Section→select All→unselect concrete → Done→(T16)ok →Done.

Assembly Module

Instant Part → ok (part-1) Fig. 6a.



a b

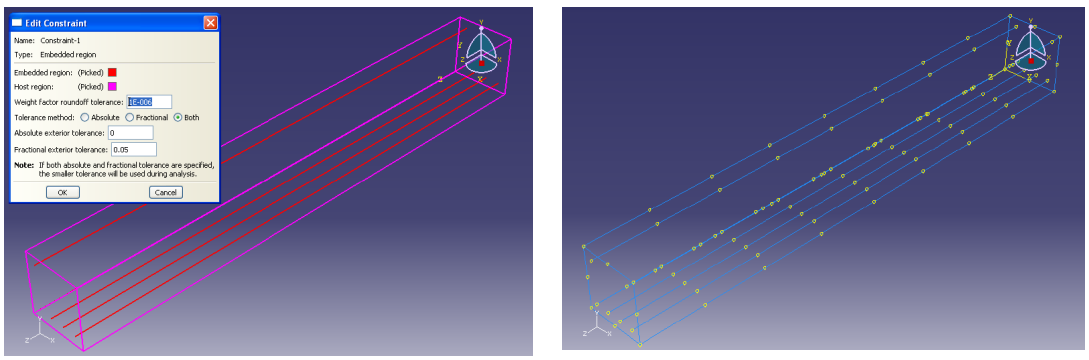
Fig. 6

Step Module

Create Step → Continue (step-1, Static, General) (Fig. 6b)→ Incrementation → Increment Size: 0.01 →ok.

Interaction Module

Create Constraint → Select "Embedded Region" (Fig. 6b)→ Continue → click: Render Model: WireFrame → Select whole model→ deselect concrete beam → Done → Select Region → Done (Fig. 7a)→ok(Fig. 7b).

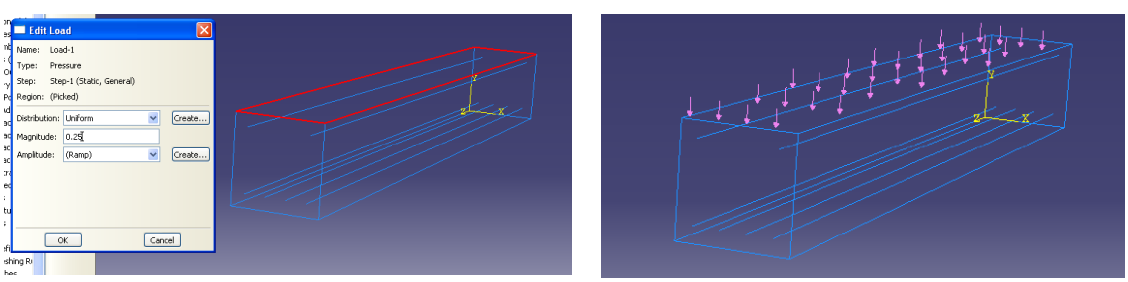


a b

Fig. 7

Load Module

Create Load → Select "Pressure" (Fig. 8a)→ Continue → Select top Surface of the beam → Done→ Type 0.25 (kg/cm2) → ok (Fig. 8b).

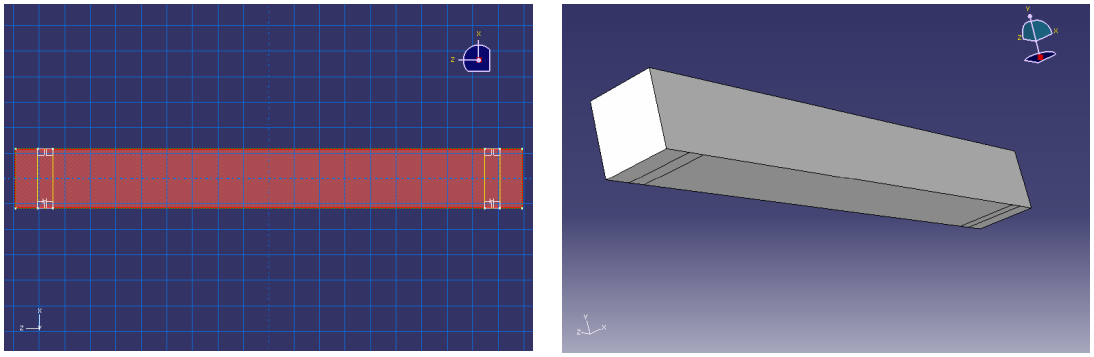


a b

Fig. 8

Go to Part Module → Render Model: Shaded → Rotate model to see the bottom → Partition Face: Sketch → Select bottom face of the beam → Done→ Select bottom right edge of the beam (now going to sketch module).

Create lines: Rectangle → Type: 145, 20, Press enter, 155,-20, Press enter → Type: -145, 20, Press enter, -155,-20, Press enter→ Esc (Fig 9a) →Done→Done (Fig. 9b).



a b

Fig. 9

Go to Load Module → Create Boundary Condition → Continue → Select partition face (press shift and select to regions) → Done→Select Pinned option→ok Fig. 10.

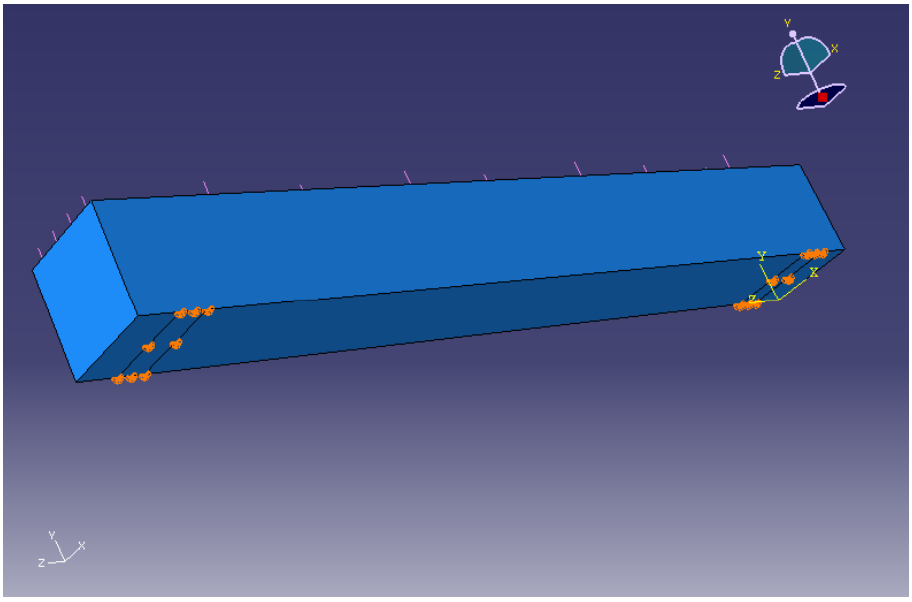
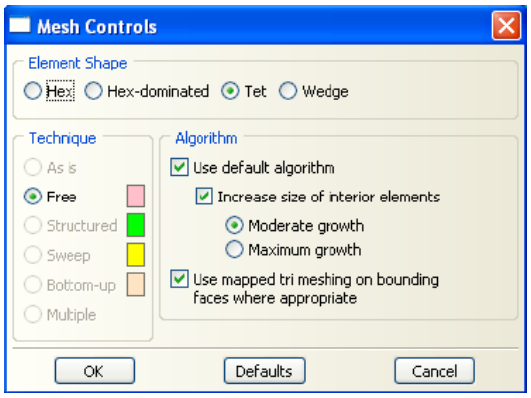


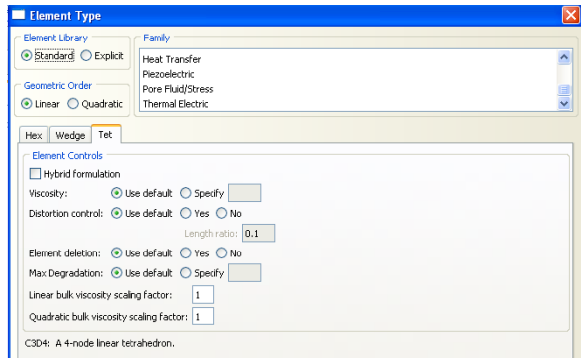
Fig. 10

Mesh Module

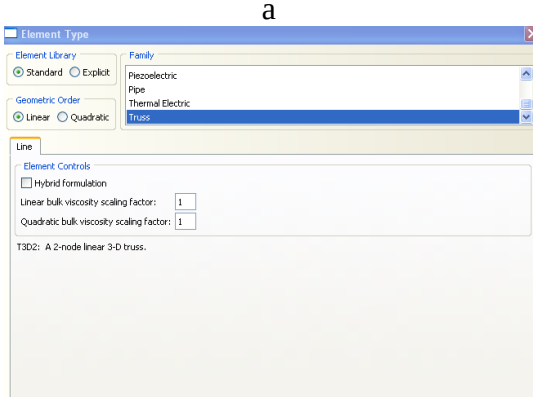
Select "Part" option → Seed Part → Type "10" for Approximate global size → ok→ Assign Mesh controls → Select whole model → Done→ Select "Tet" Option (Fig. 11a)→ ok → Done → Mesh Part → Yes → Assign Element Type→ Select concrete → Done (Fig. 11b)→ok → Select whole model→ Unselect concrete→Done→ Truss(Fig. 11c) → ok → Done (Fig. 11d).



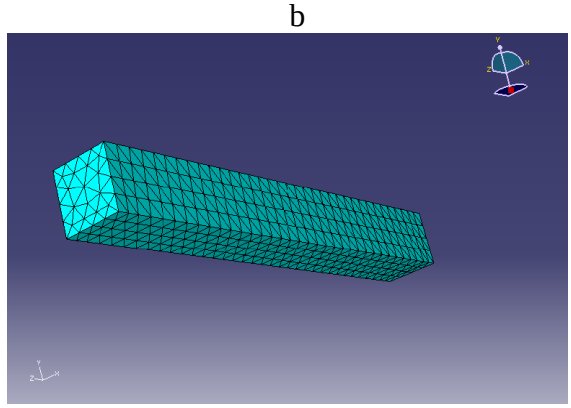
a



b



c

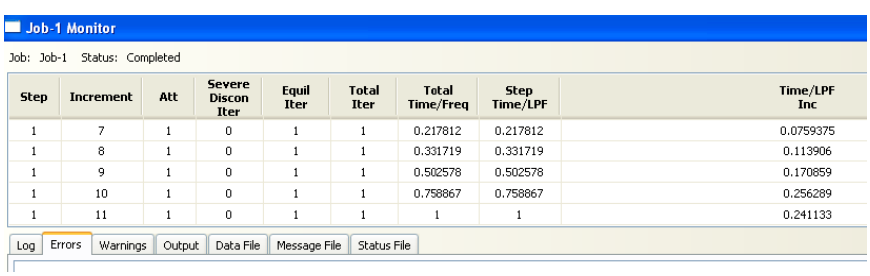


d

Fig. 11

Job Module

Job Create → Continue → ok → Job Manager → Submit → Monitor.



Job-1 Monitor

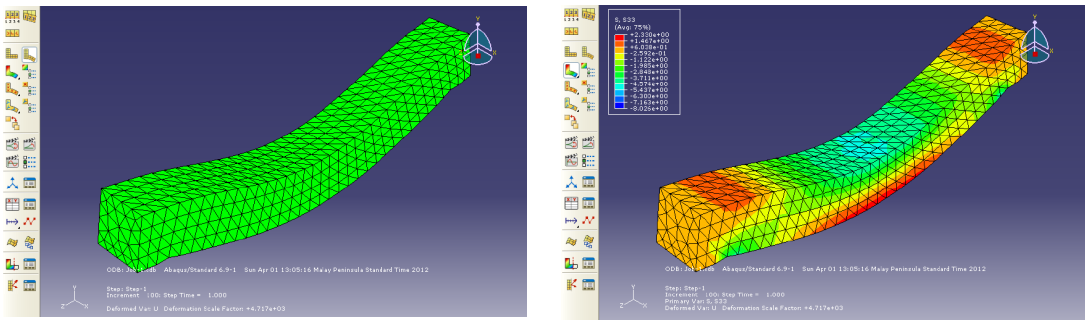
Job: Job-1 Status: Completed

Step	Increment	Att	Severe Discon Iter	Equil Iter	Total Iter	Total Time/Freq	Step Time/LPF	Time/LPF Inc
1	7	1	0	1	1	0.217812	0.217812	0.0759375
1	8	1	0	1	1	0.331719	0.331719	0.113906
1	9	1	0	1	1	0.502578	0.502578	0.170859
1	10	1	0	1	1	0.758867	0.758867	0.256289
1	11	1	0	1	1	1	1	0.241133

Log | Errors | Warnings | Output | Data File | Message File | Status File

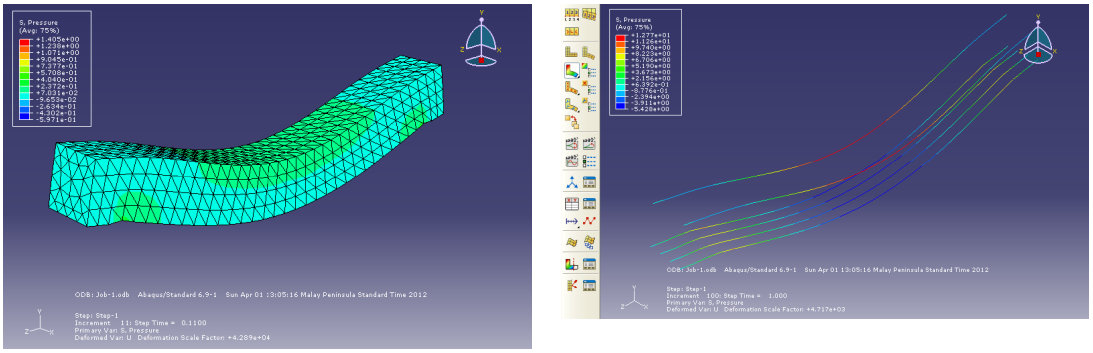
Results Module

Plot Deformed Shape (Fig. 12a). Plot Contours on deformed shape→ Select S33 (Fig. 12b). Select Pressure (Fig. 13a). Create Display Group→ Elements→ Material Assignment → steel → Replace → Dismiss (Fig. 13b). Animation: Time History.



a b

Fig. 12



a b

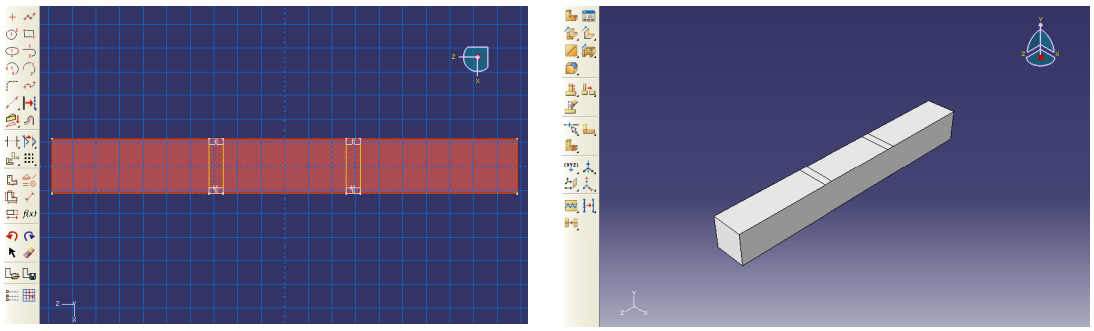
Fig. 13

Apply two concentrate Loads

Part Module

Go to Part Module → Render Model: Shaded → Rotate model to see the Top Face → Partition Face: Sketch → Select Top face of the beam → ok → Done→ Select Top right edge of the beam (now going to sketch module).

Create lines: Rectangle → Type: 45, 20, Press enter, 55,-20, Press enter → Type: -45, 20, Press enter, -55,-20, Press enter→ Esc (Fig 14a) →Done →Done (Fig. 14b).

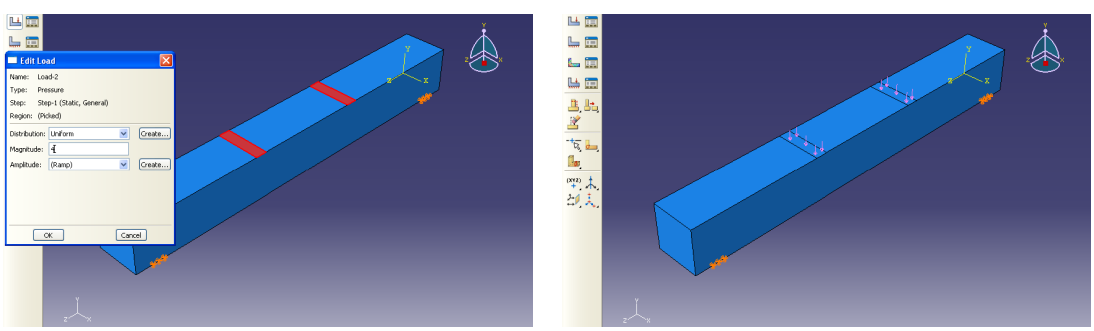


a b

Fig. 14

Load Module

Create Load → Select "Pressure" (Fig. 15a)→ Continue → Select partition Area on the top of the beam → Done→ Type 4 (kg/cm2) → ok (Fig. 15b).



a b

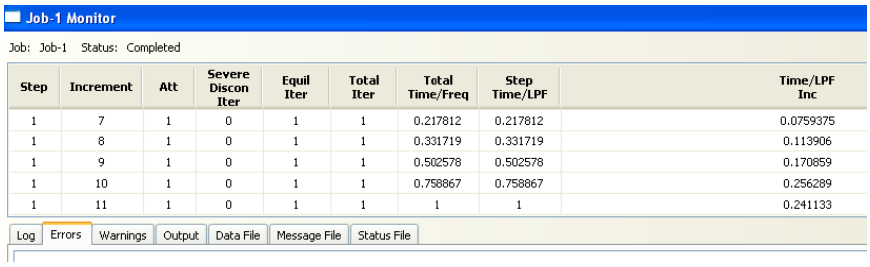
Fig. 15

Mesh Module

Select "Part" option → Seed Part → Type "10" for Approximate global size → ok→ Mesh Part → Yes

Job Module

Job Create → Continue → ok → Job Manager → Submit → Monitor.



Job-1 Monitor

Job: Job-1 Status: Completed

Step	Increment	Att	Severe Discon Iter	Equil Iter	Total Iter	Total Time/Freq	Step Time/LPF	Time/LPF Inc
1	7	1	0	1	1	0.217812	0.217812	0.0759375
1	8	1	0	1	1	0.331719	0.331719	0.113906
1	9	1	0	1	1	0.502578	0.502578	0.170859
1	10	1	0	1	1	0.758867	0.758867	0.256289
1	11	1	0	1	1	1	1	0.241133

Log | Errors | Warnings | Output | Data File | Message File | Status File

Fig. 16

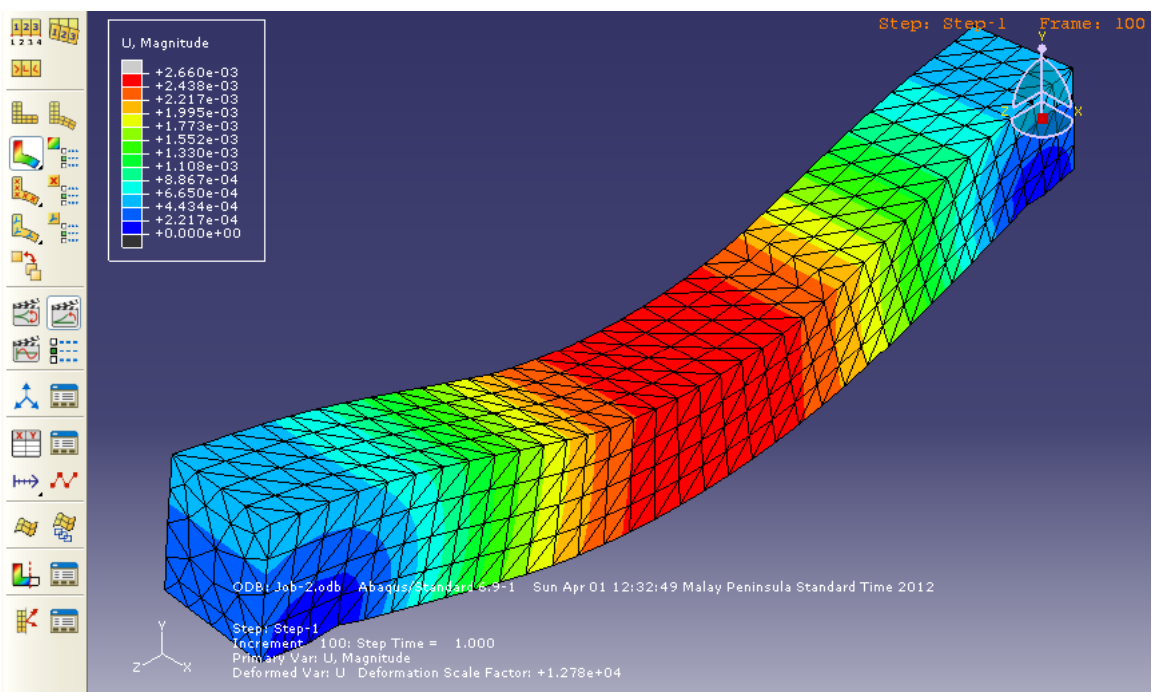
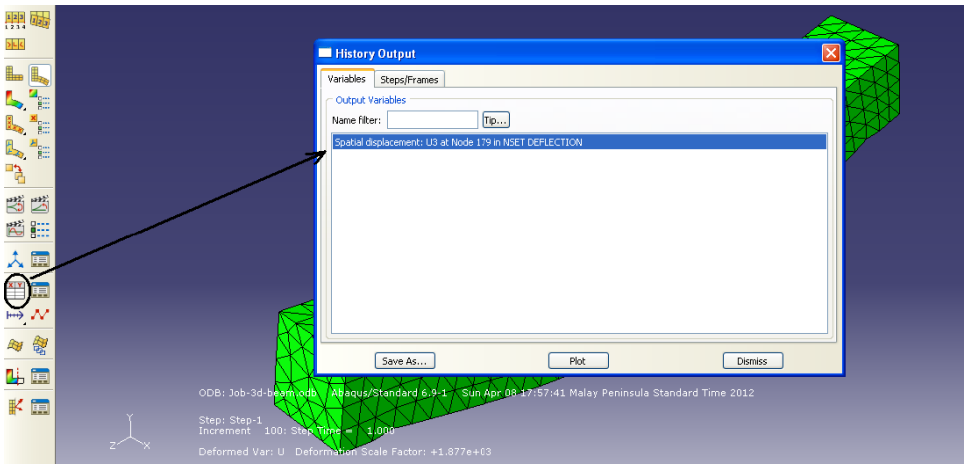
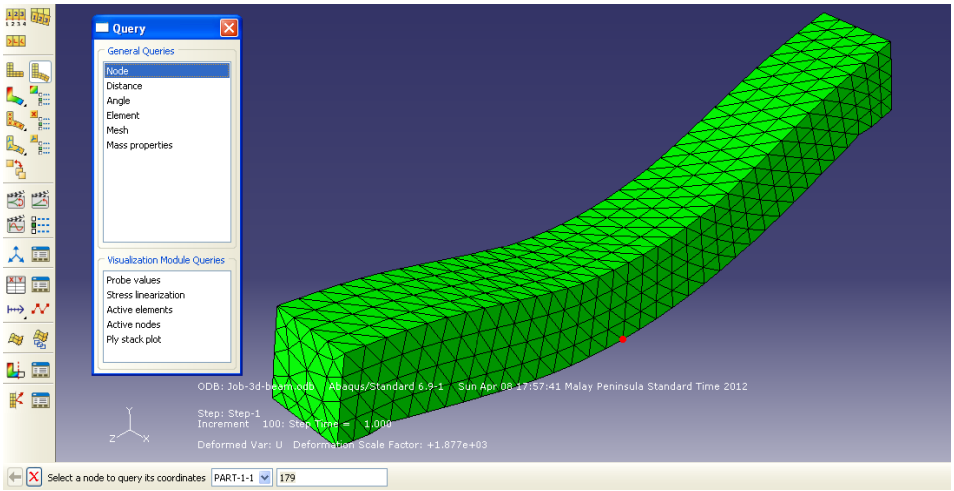
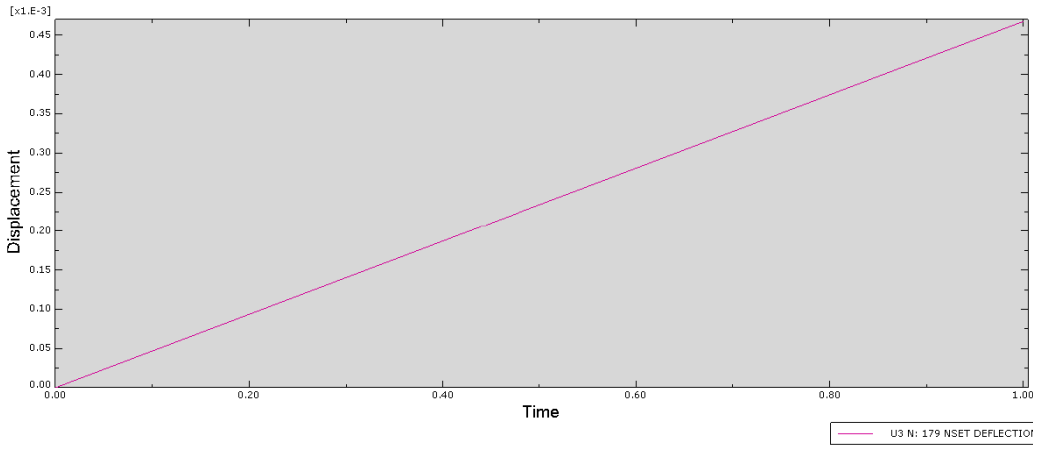


Fig. 17

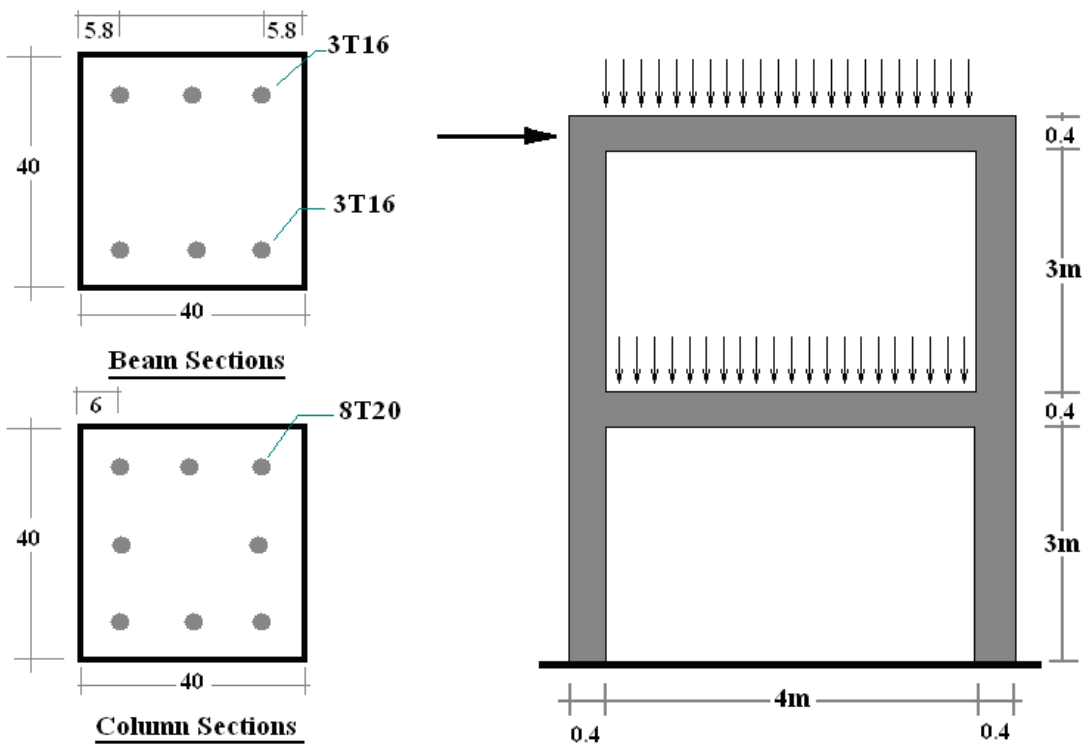

History Output
Variables
Steps/Frames
Output Variables
Name filter:
Tip...
Spatial displacement: U3 at Node 179 in NSET DEFLECTION
Save As...
Plot
Dismiss
ODB: Job-3d-beam.odb Abaqus/Standard 6.9-1 Sun Apr 08 17:57:41 Malay Peninsula Standard Time 2012
Step: Step-1
Increment 100: Step Time = 1.000
Deformed Var: U Deformation Scale Factor: +1.877e+03


Query
General Queries
Node
Distance
Angle
Element
Mesh
Mass properties
Visualization Module Queries
Probe values
Stress linearization
Active elements
Active nodes
Ply stack plot
ODB: Job-3d-beam.odb Abaqus/Standard 6.9-1 Sun Apr 08 17:57:41 Malay Peninsula Standard Time 2012
Step: Step-1
Increment 100: Step Time = 1.000
Deformed Var: U Deformation Scale Factor: +1.877e+03
Select a node to query its coordinates
PART-1-1
179


[x1.E-3]
0.45
0.40
0.35
0.30
0.25
0.20
0.15
0.10
0.05
0.00
Displacement
0.00
0.20
0.40
0.60
0.80
1.00
Time
U3 N: 179 NSET DEFLECTIO

3D Frame

The span of reinforcement concrete Frame is 4m c/c and height of each stories is 3m C/C. The cross section dimension of beam and columns are 40x40Cm. The gravity load on beams is 4t/m and lateral load is 10t on the top of the frame as shown.

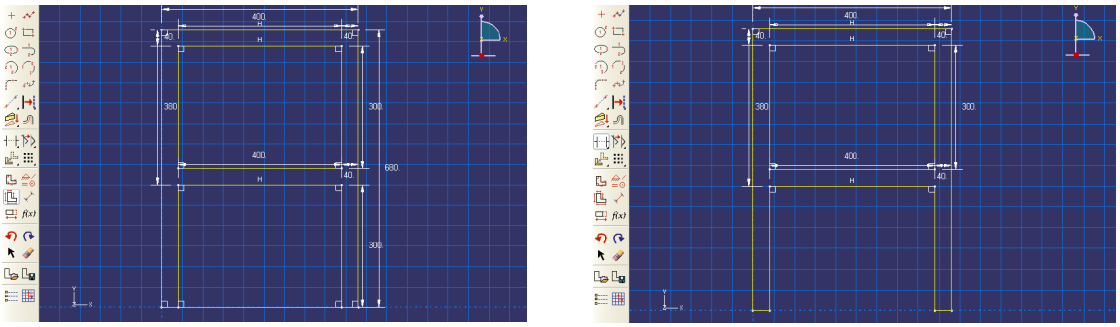


Part Module (Define Frame)

1) Part→Create part→Type "Frame" for Name →Approximate size=2000Cm→Continue
2) Create lines: Rectangle →

(0, 0) Enter (480, 680) Enter
(40, 0) Enter (440, 300) Enter
(40, 340) Enter (440, 640) Enter

Esc→Auto Dimension → Done (Fig 1a). Auto Trim →Trim base lines (Fig 1b) →Esc.



a b

Fig 1

Done → type 40 for Depth → ok

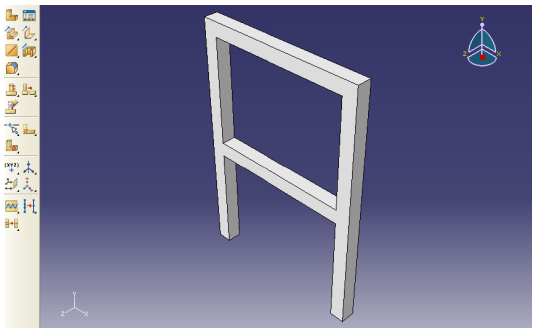
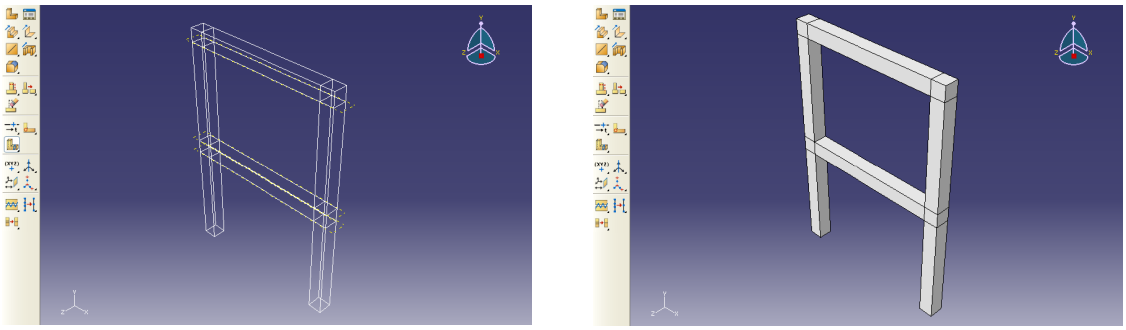


Fig 2

Tools→ Datum → Plane → Offset from principal plane → ZX plane → 300 enter → 340 enter→ 640 enter Fig 3a.
Tools→ Partition → Cell → Define Cutting Plane → 3Points → define cutting plane in two ends of all beams and columns Fig 3b.



a b

Fig 3

Part Module (Define Reinforcements)

Create Part→ Name: Columnbar → Wire→ Continue→ Create lines : Connected → 0, 0 enter → 0, 674 enter→ Esc → Done.

Create Part→ Name: Beambar → Wire→ Continue→ Create lines: Connected → 0, 0 enter → 0, 468 enter→ Esc → Done.

Create Part→ Name: Sectionbar → Wire→ Continue→ Create lines: Rectangle → 0, 0 enter → 28, 28 enter→ Esc → Done.

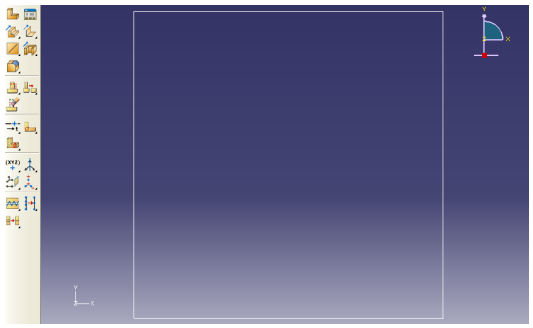


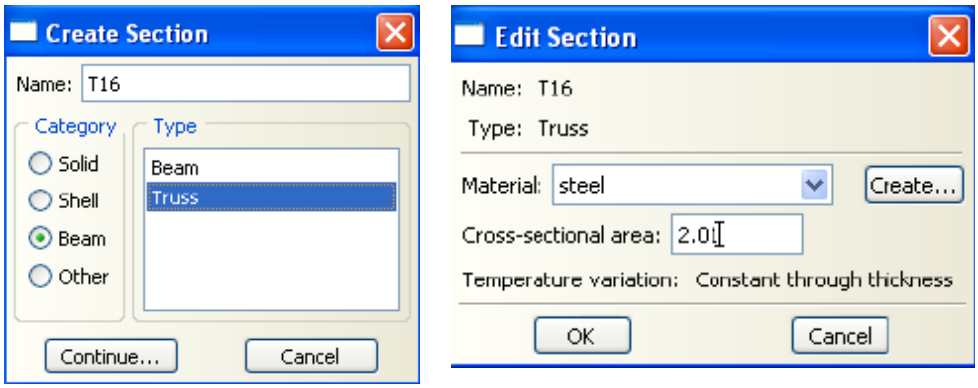
Fig 4

Property Module

Create Material→Type concrete for Name→General→Density→Enter 0.0025→ Mechanical →Elasticity→Elastic→ Enter 218000 for Young's modulus→Enter 0.15 for Pisson's ratio→ok.

Create Material→Type steel for Name→General→Density→Enter 0.0078→ Mechanical →Elasticity→Elastic→ Enter 2100000 for Young's modulus→Enter 0.2 for Pisson's ratio→ok.

Create Section→Type concrete for Name→continue→select "Concrete" from material combo box→ok.
Create Section→Type T20 for Name→beam→truss continue→ "Steel" from material combo box→Enter 3.14159 for cross-sectional area→ok
Create Section→Type T16 for Name→beam→truss continue→ "Steel" from material combo box→Enter 2.01 for cross-sectional area→ok. Fig. 5
Create Section→Type T10 for Name→beam→truss continue→ "Steel" from material combo box→Enter 0.748 for cross-sectional area→ok.



a b

Fig. 5

Part: Part-1→Assign Section→select concrete →Done →(concrete)ok →Done.
Part: Beambar→Assign Section→select part →Done →T16→ok →Done.
Part: Columnbar→Assign Section→select part →Done →T20→ok →Done.
Part: Sectionbar→Assign Section→select part →Done →T10→ok →Done.

Assembly Module

(Assembly Frame)
Instant Part → part-1 → ok Fig. 6a.

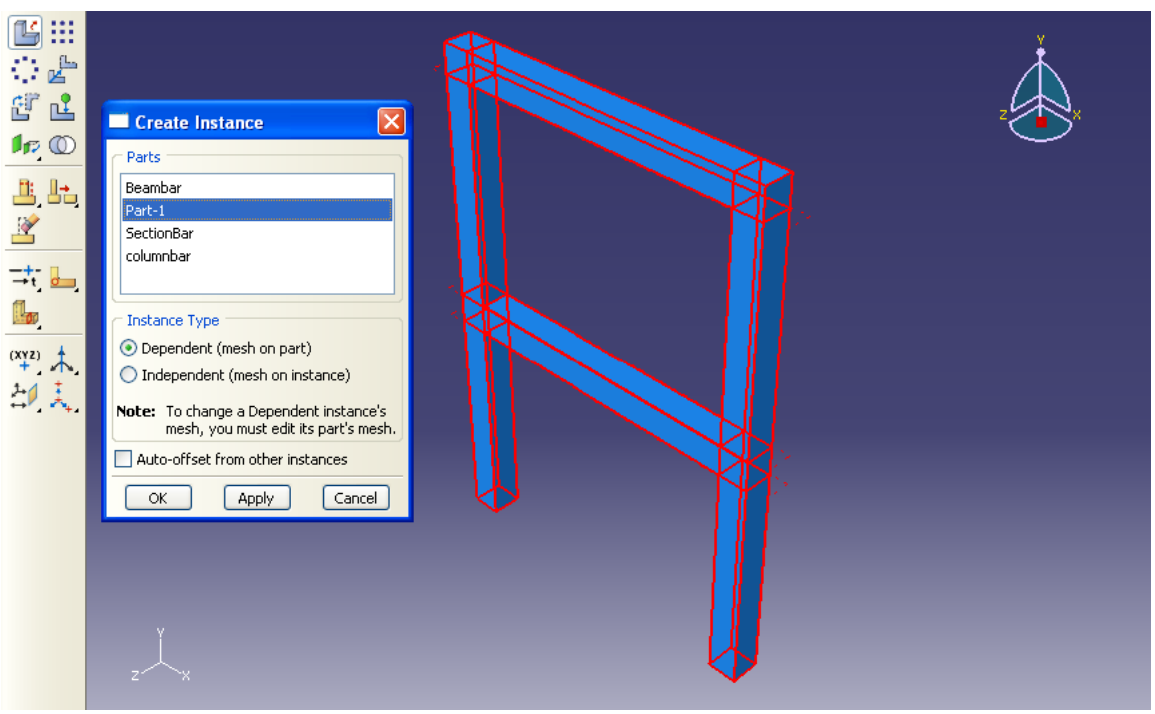
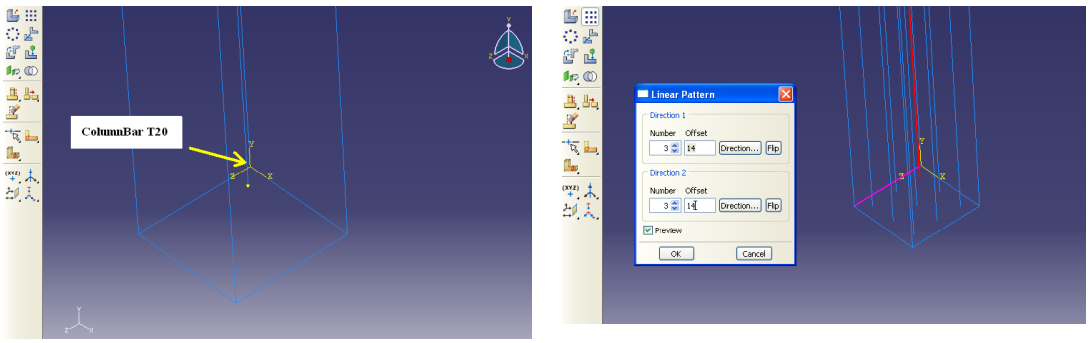


Fig. 7

(Assembly ColumnBars)

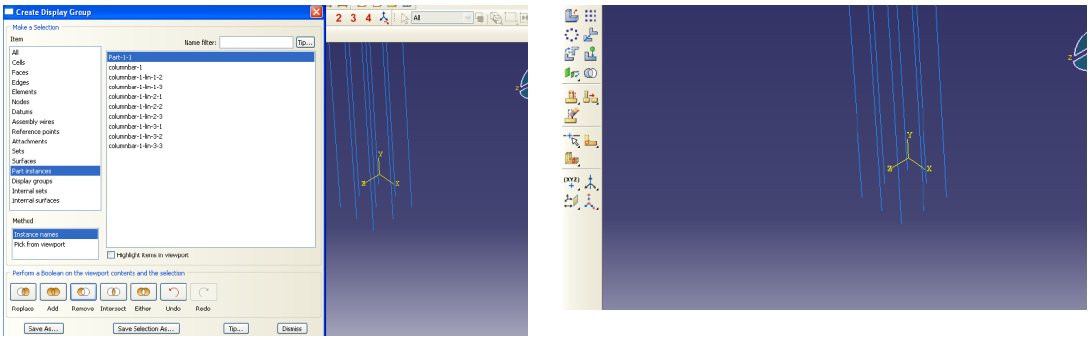
Instant Part → Columnbar → ok
Translate Instant → Select just "Columnbar" part → Done→ 0,0,0 enter → 6,0,6 enter→ ok (fig. 8a).
Linear Pattern → Select just "Columnbar" part → Done→ select 3 for Number in Direction 1 and type 14 for offset → click "direction" button in Direction 2 frame → select a line parallel Z axis for direction 2 → select 3 for Number in Direction 2 and type 14 for offset → ok (fig. 8b).



a b

Fig. 8

Create Display Group → Select "Part Instances" → Select "Part-1-1" → press Remove button (fig. 9a).→ Delete Feature → Select middle bar of the column → yes → Esc (fig. 9b).



a b

Fig. 9

Linear Pattern → Select 8 "Columnbar" part → Done→ select 2 for Number in Direction 1 and type 440 for offset → select 1 for Number in Direction 2 → ok (fig. 10).

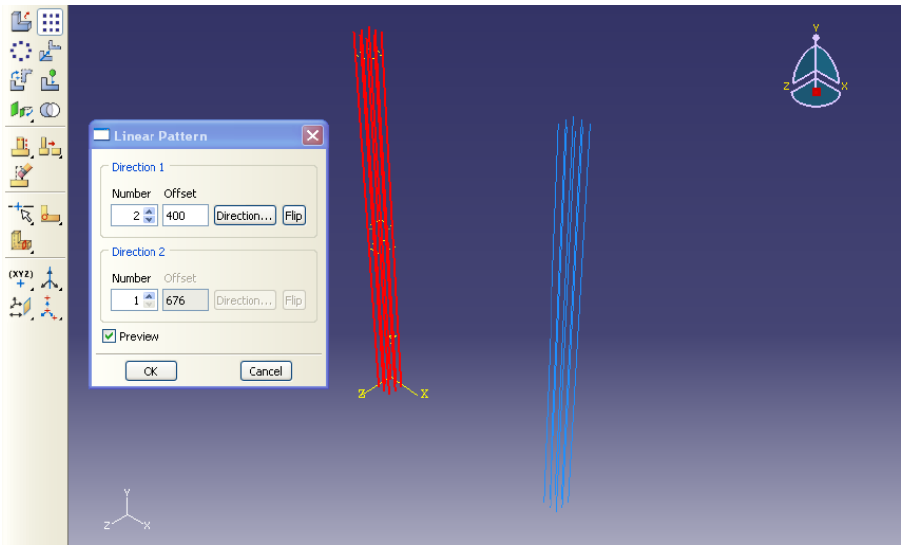
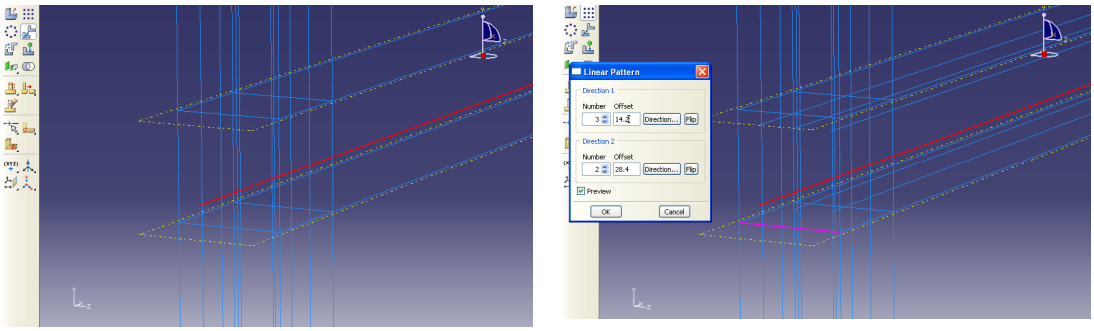


Fig. 10

(Assembly BeamBars)

Replace All → Instant Part → Beambar → ok
Translate Instant → Select just "Beambar" part → Done→ 0,0,0 enter → 4,305.8,5.8 enter→ ok (fig. 11a).

Linear Pattern → Select just "Beambar" part → Done→ select 3 for Number in Direction 1 and type 14.2 for offset → click "direction" button in Direction 1 frame → select a line parallel Z axis for direction 1 → select 2 for Number in Direction 2 and type 28.4 for offset → ok (fig. 11b).



a b

Fig. 11

Linear Pattern → Select All “Beambar” part → Done→ select 1 for Number in Direction 1 → select 2 for Number in Direction 2 and type 340 for offset → ok (fig. 12).

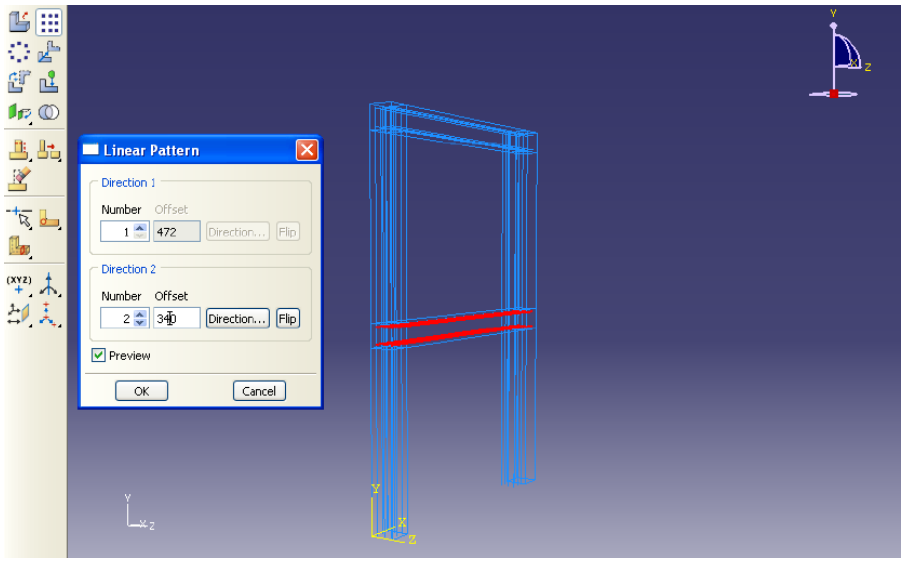


Fig. 12

(Assembly SectionBars)

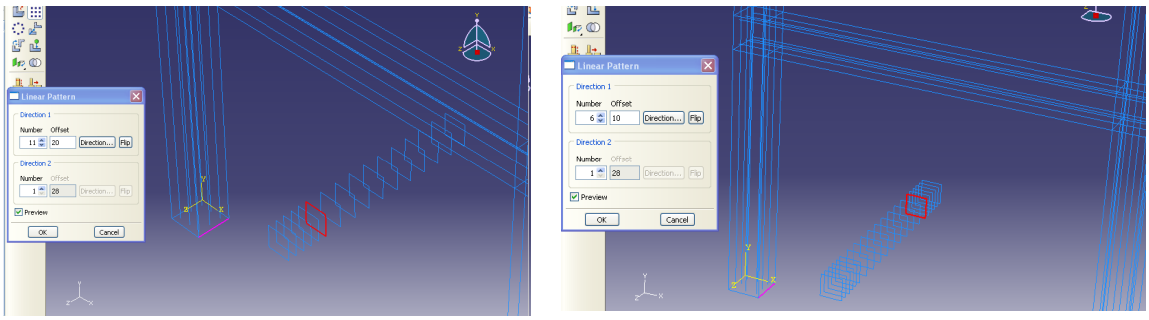
Create Display Group → Select “Part Instances” → Select “Part-1-1” → press Remove button.→ Dismiss.

Instant Part → Sectionbar → ok
Translate Instance → Select just “Sectionbar” part → Done→ 0,0,0 enter → 100,0,0 enter→ ok.

Replace All → Linear Pattern → Select just “Sectionbar” part → Done→ Direction button in Direction 1 frame → select an edge parallel Z → select 6 for Number in Direction 1 and type 10 for offset → select 1 for Number in Direction 2 → ok.

Linear Pattern → Select just last “Sectionbar” part → Done→ Direction button in Direction 1 frame → select an edge parallel Z → select 11 for Number in Direction 1 and type 20 for offset → select 1 for Number in Direction 2 → ok (fig 13a).

Linear Pattern → Select just “Sectionbar” part → Done→ Direction button in Direction 1 frame → select an edge parallel Z → select 6 for Number in Direction 1 and type 10 for offset → select 1 for Number in Direction 2 → ok.

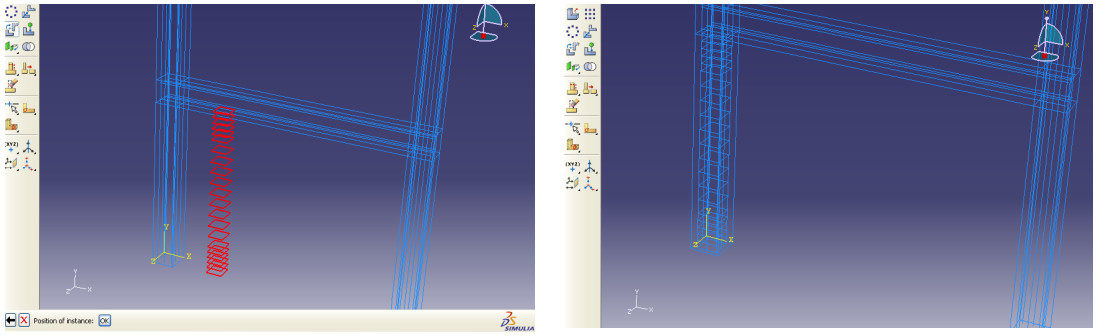


a b

Fig. 13

Rotate Instant → Select All “Sectionbar” part → Done→ 0,0,0 enter → 1,0,0 enter→ 90 → ok (fig. 14a).

Translate Instance → Select All “Sectionbar” part → Done→ Select corner point of first Sectionbar enter → Select base point of related columnbar enter→ ok (fig. 14b).



a b

Fig. 14

Linear Pattern → Select All “Sectionbar” part → Done→ select 2 for Number in Direction 1 and type 440 for offset → select 2 for Number in Direction 2 and type 340 for offset → ok (fig. 15a).

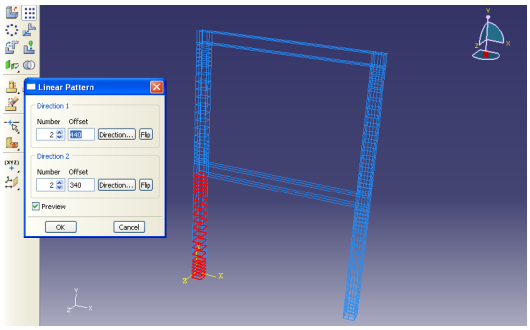
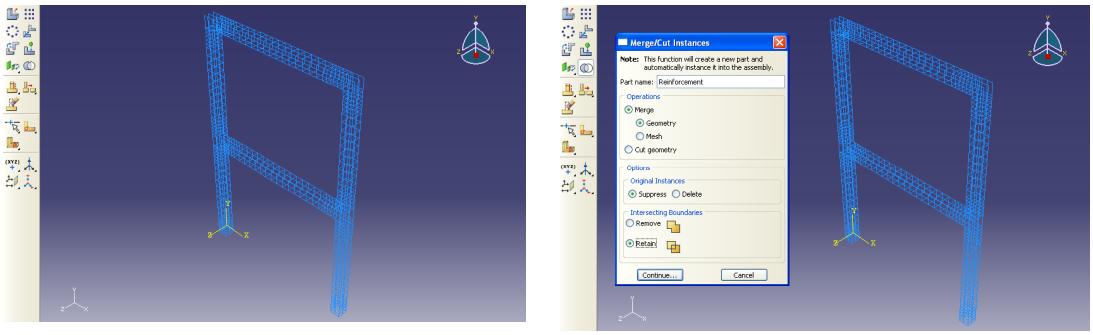


Fig. 15

Do the same process for beams section bars (fig 16a).



a b

Fig 16

Merge all reinforcement:
Instance → Merge/Cut → Type "Reinforcement" → select Retain option → Continue → select all reinforcement → Done(fig 16b).

Step Module

Create Step → Continue (step-1, Static, General)→ Incrementation → Increment Size: 0.01 →ok.

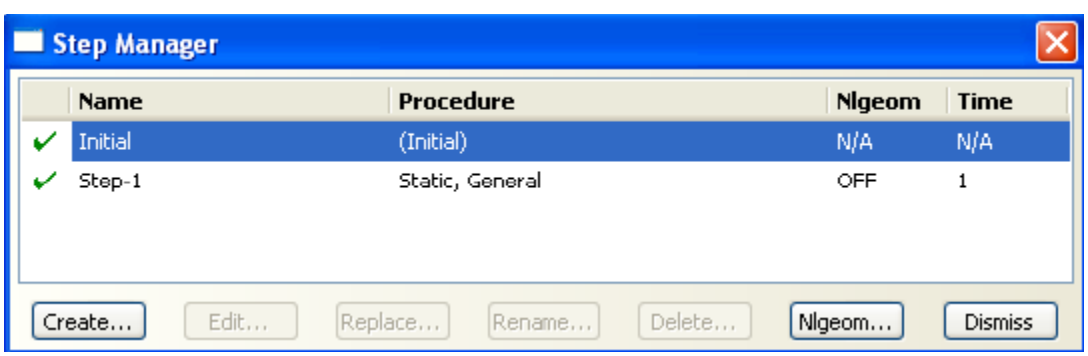
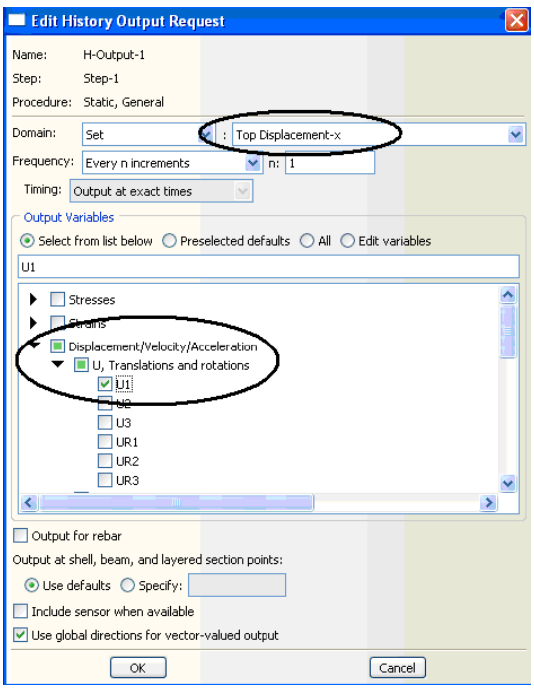


Fig 17

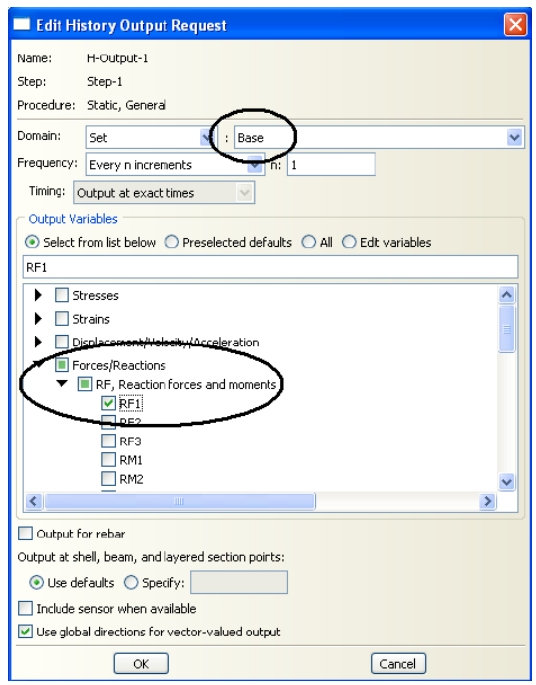
Tools → Set→ Create → type "Top Displacement-x" for Name →Done.
Tools → Set→ Create → type "Base" for Name →Done.

Create History Output → Continue → Select “Set” in Domain combo box → Select “Top Displacement-x” from right combo box → select just “U1” for this set in Output Variable section. → ok (fig. 18a).

Again:
Create History Output → Continue → Select “Set” in Domain combo box → Select “Base” from right combo box → select just “RF1” for this set in Output Variable section. → ok(fig. 18b).



a



b

Fig. 18

Interaction Module

Create Constraint → Select “Embedded Region”→ Continue →
Create Display Group → Part Instances → Select “Part-1-1” → Remove → Dismiss→ Select All Reinforcement → Done → press “Whole model” → ok.
Replace All.

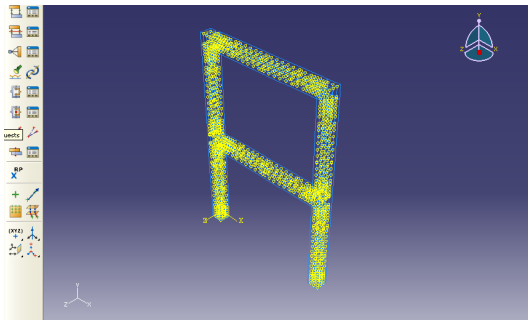


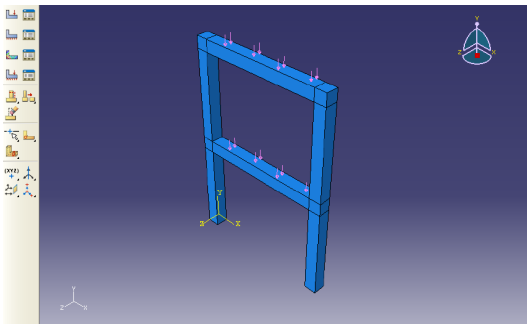
Fig. 19

Load Module

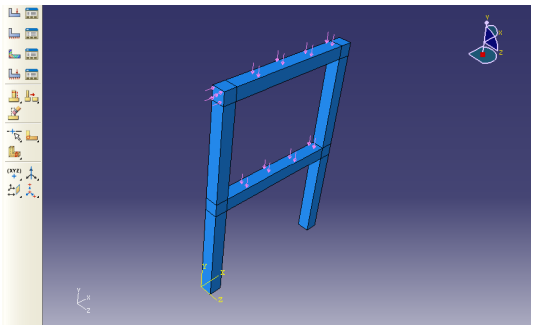
Create Load → Select "Pressure"→ Continue → Select top Surface of the beams → Done→ Type 1 (kg/cm2) → ok (Fig. 20a).

Create Load → Select "Pressure"→ Continue → Select top left Surface of frame → Done→ Type 6.25 (kg/cm2) → ok (Fig. 20a).

Create Boundary Condition → Continue → Select bottom surface of columns → Done→ Select Pinned option→ok



a



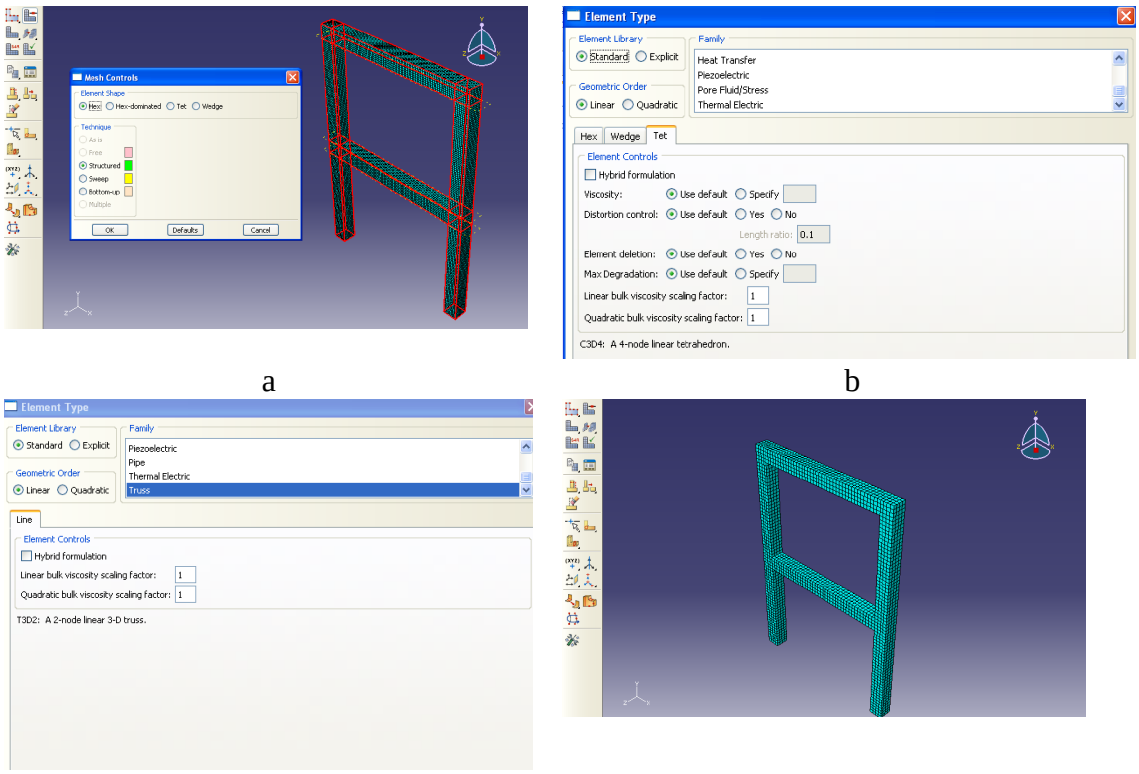
b

Fig. 20

Mesh Module

Select "Part" option → Seed Part → Type "10" for Approximate global size → ok→ Assign Mesh controls → Select whole part (frame) → Done→ ok→ Done (Fig. 21a).
Mesh Part → Yes → Assign Element Type→ Select Frame → Done → Done. →ok → Select whole model→ Unselect concrete→Done→ Truss → ok → Done → Done (Fig. 21b).

Select “Reinforcement” option → Seed Part → Type “10” for Approximate global size →Done →ok→ Assign Mesh controls → Select whole part(reinforcement) → Done→ ok→ Done (Fig. 21c).
Mesh Part → Yes → Assign Element Type→ Select Frame → Done → Done. →ok → Select whole model→ Unselect concrete→Done→ Truss→ ok → Done → Done (Fig. 21d).



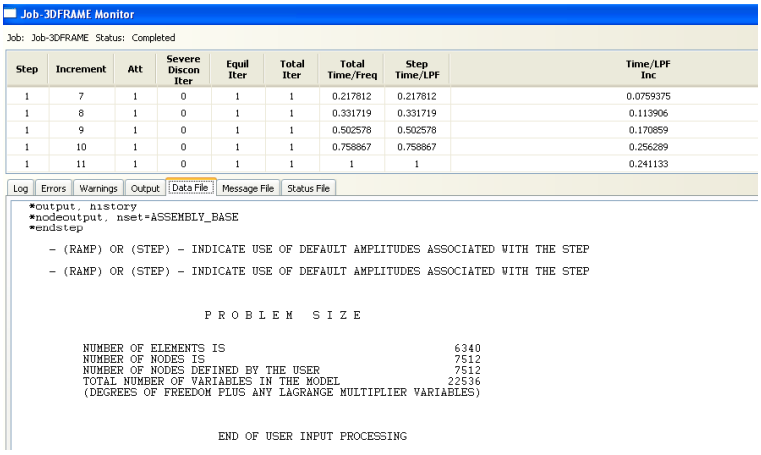
a b

c d

Fig. 21

Job Module

Job Create → Continue → ok → Job Manager → Submit → Monitor.



Job-3DFRAME Monitor

Job: Job-3DFRAME Status: Completed

Step	Increment	Att	Severe Discon Iter	Equil Iter	Total Iter	Total Time/Freq	Step Time/LPF	Time/LPF Inc
1	7	1	0	1	1	0.217812	0.217812	0.0759375
1	8	1	0	1	1	0.331719	0.331719	0.113906
1	9	1	0	1	1	0.502578	0.502578	0.170859
1	10	1	0	1	1	0.758867	0.758867	0.256289
1	11	1	0	1	1	1	1	0.241133

Log | Errors | Warnings | Output | Data File | Message File | Status File

```
*output, history
*nodeoutput, nset=ASSEMBLY_BASE
*endstep

  - (RAMP) OR (STEP) - INDICATE USE OF DEFAULT AMPLITUDES ASSOCIATED WITH THE STEP

  - (RAMP) OR (STEP) - INDICATE USE OF DEFAULT AMPLITUDES ASSOCIATED WITH THE STEP


                    P R O B L E M   S I Z E

      NUMBER OF ELEMENTS IS                                6340
      NUMBER OF NODES IS                                   7512
      NUMBER OF NODES DEFINED BY THE USER                  7512
      TOTAL NUMBER OF VARIABLES IN THE MODEL              22536
      (DEGREES OF FREEDOM PLUS ANY LAGRANGE MULTIPLIER VARIABLES)


                    END OF USER INPUT PROCESSING
```

Fig. 22

Visualization Module

Plot → Deformed Shape (Fig 23).

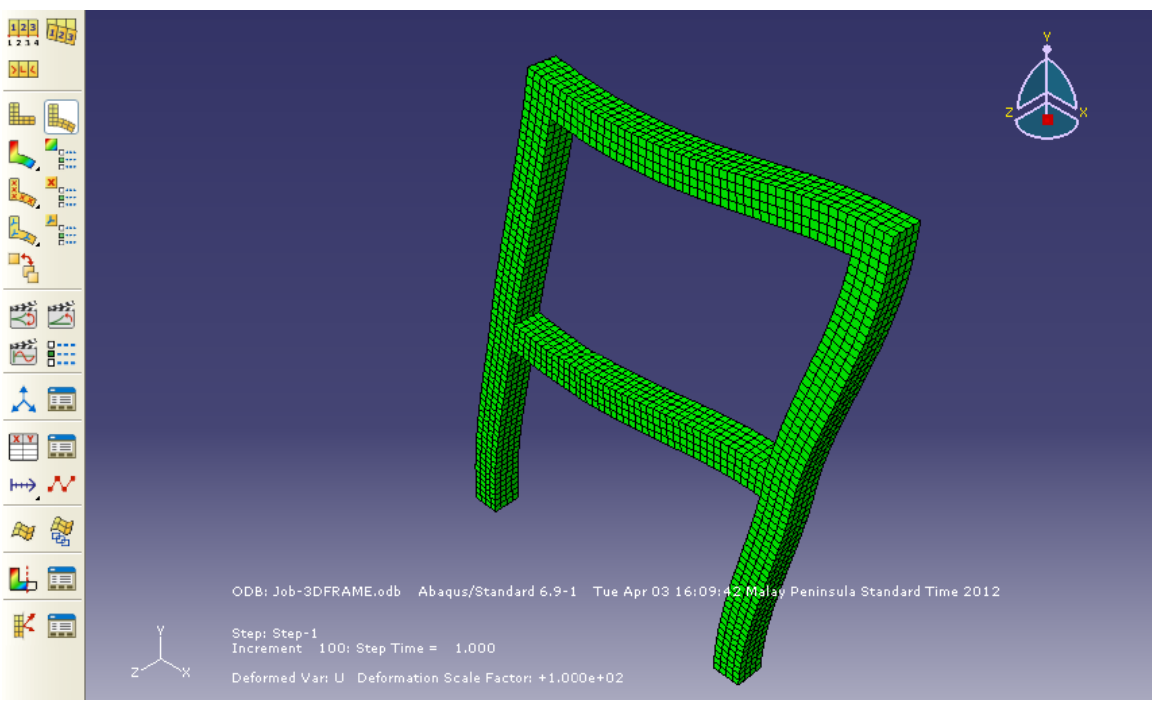


Fig. 23

Plot → Contours → On Deformed Shape → Select: Primary, U, Magnitude. (Fig. 24)

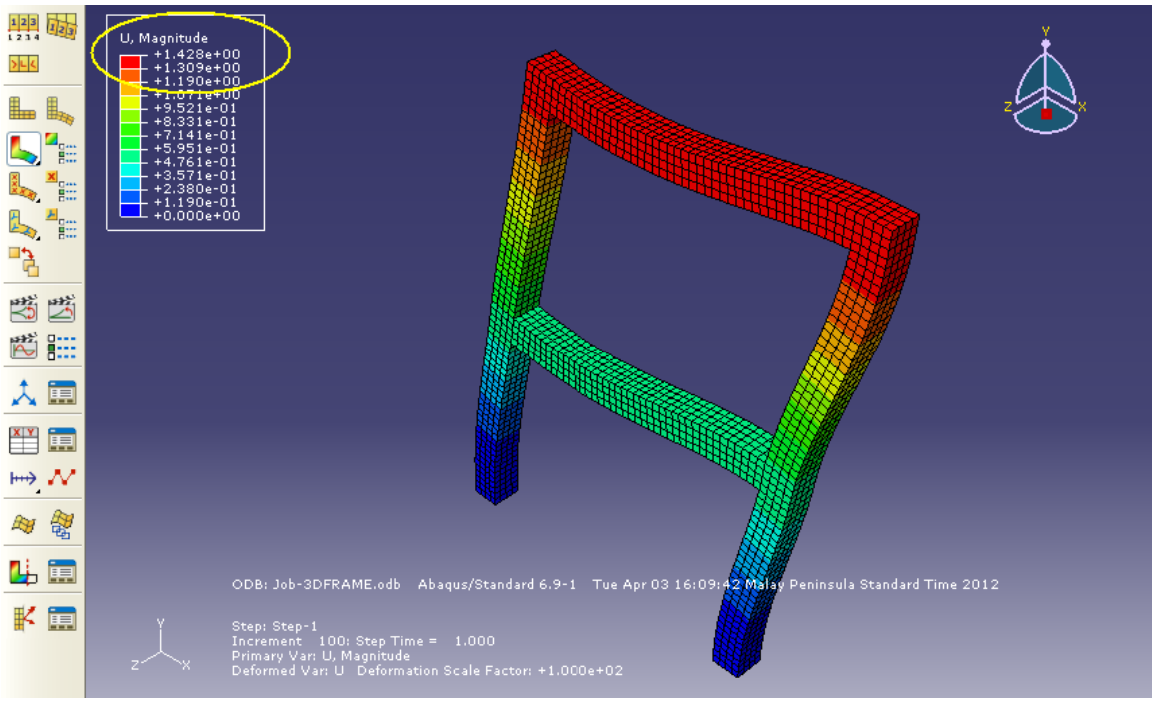

U, Magnitude
+1.428e+00
+1.309e+00
+1.190e+00
+1.071e+00
+9.521e-01
+8.331e-01
+7.141e-01
+5.951e-01
+4.761e-01
+3.571e-01
+2.380e-01
+1.190e-01
+0.000e+00
ODB: Job-3DFRAME.odb Abaqus/Standard 6.9-1 Tue Apr 03 16:09:42 Malay Peninsula Standard Time 2012
Step: Step-1
Increment 100: Step Time = 1.000
Primary Var: U, Magnitude
Deformed Var: U Deformation Scale Factor: +1.000e+02

Fig. 24

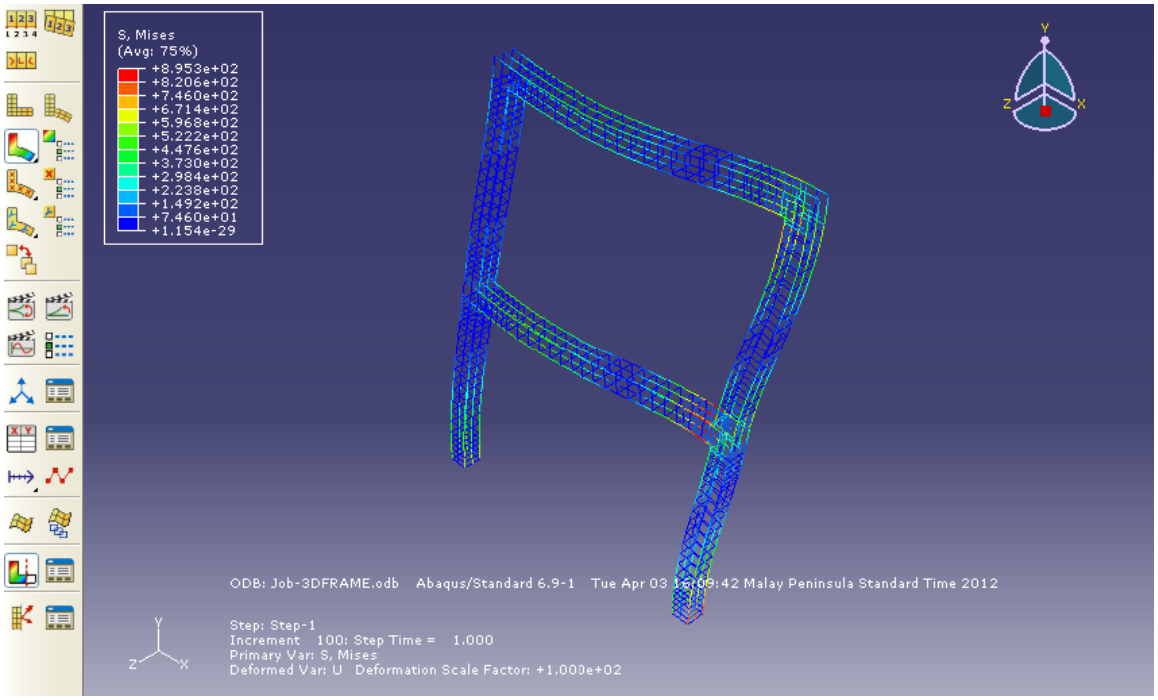

S, Mises
(Avg: 75%)
+8.953e+02
+8.206e+02
+7.460e+02
+6.714e+02
+5.968e+02
+5.222e+02
+4.476e+02
+3.730e+02
+2.984e+02
+2.238e+02
+1.492e+02
+7.460e+01
+1.154e-29
ODB: Job-3DFRAME.odb Abaqus/Standard 6.9-1 Tue Apr 03 16:09:42 Malay Peninsula Standard Time 2012
Step: Step-1
Increment 100: Step Time = 1.000
Primary Var: S, Mises
Deformed Var: U Deformation Scale Factor: +1.000e+02

Fig. 25

This document was created with Win2PDF available at http://www.win2pdf.com.
The unregistered version of Win2PDF is for evaluation or non-commercial use only.
This page will not be added after purchasing Win2PDF.