Workshop 1

Linear Static Analysis of a Cantilever Beam

Introduction

In this workshop you will become familiar with the process of creating a model interactively by using Abaqus/CAE. You will create the cantilever beam model shown in Figure W1–1.
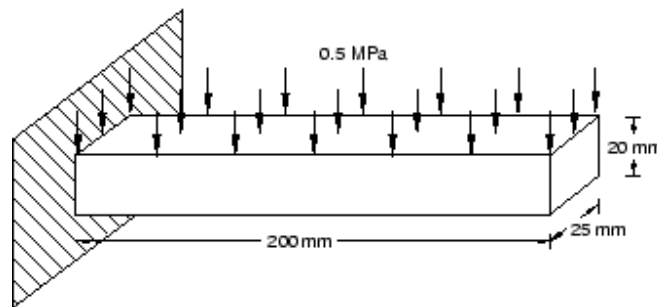

Figure W1–1 Cantilever beam.

Preliminaries

1. Enter the working directory for this workshop:

    ../IntroClass/Workshops/beam

2. Start a new session of Abaqus/CAE using the following command:

    abaqus `cae`

    where *abaqus* is the command used to run Abaqus.

3. Select **Create Model Database** from the **Start Session** dialog box.

    The Model Tree is located to the left of the toolbox area. If the Model Tree is not visible, make sure there is a check mark next to **Show Model Tree** in the **View** menu. If the Model Tree is still not visible, drag the ⁀ cursor from the left side of the Abaqus/CAE window to expand the Model Tree. The Model Tree provides a visual description of the hierarchy of items in the model database along with access to most of the model development functionality available in Abaqus/CAE. If you click mouse button 3 on an item in the tree, a menu appears listing commands associated with the item. For example, Figure W1–2 shows the menu for the **Models** container. In the **Models** menu, the **Create** menu item appears in bold because it is the default action that will be performed when you double-click the **Models** container.
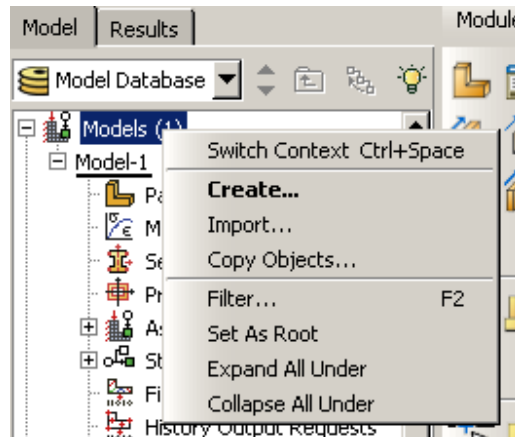
Figure W1–2 **Models** menu.

4. In the Model Tree, double-click **Models** to create a new model in the model database.

    The model attributes editor appears.

5. In the **Edit Model Attributes** dialog box that appears, enter the name **BEAM** and click **OK**.

    The model **BEAM** appears in the Model Tree, as shown in Figure W1–3. The new model is underlined to indicate that it is a current object (i.e., **BEAM** is the model you are currently working on).

    Tip: To reduce clutter in the Model Tree, collapse the container for **Model-1** by clicking the "–" sign next to the model name or set **BEAM** as the root.
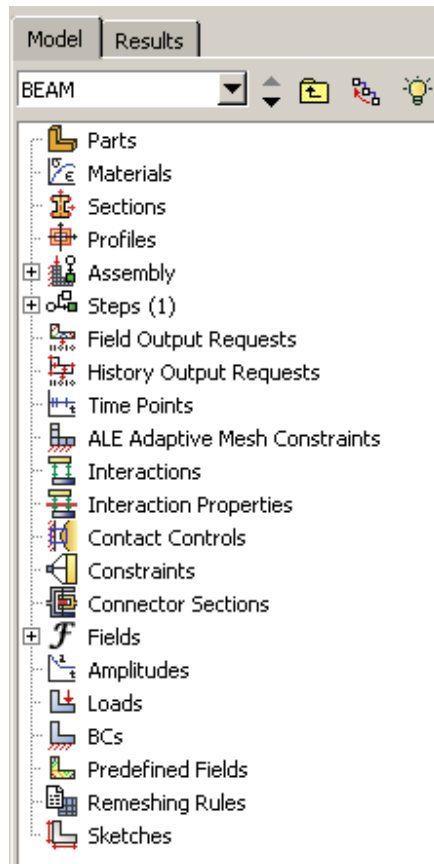
Figure W1–3 Model Tree.

6. To save the model database, select **File→Save As** from the main menu bar and type the file name **BEAM** in the **Save Model Database As** dialog box. Click **OK**.

The **.cae** extension is added to the file name automatically.

Creating a part

In this section you will create a three-dimensional, deformable solid body by sketching the two-dimensional profile of the beam (a rectangle) and extruding it.

1. In the Model Tree, double-click **Parts** to create a new part in the model **BEAM**.

The **Create Part** dialog box appears.

2. In the **Create Part** dialog box, name the part **Beam**, and specify an approximate size of **600**. Accept the default settings of a three-dimensional, deformable body with a solid, extruded base feature. Click **Continue**.

Abaqus/CAE displays text in the prompt area near the bottom of the window to guide you through the procedure, as shown in Figure W1–4. Click the cancel button to cancel the current task; click the backup button to cancel the current step in the task and return to the previous step.
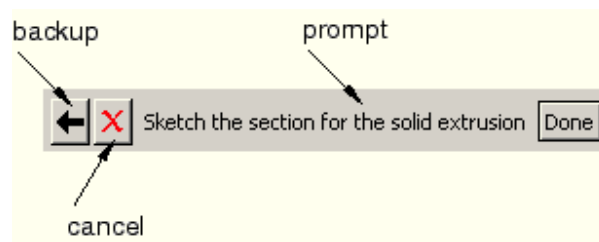
Figure W1–4 Prompt area.

The Sketcher toolbox appears in the left side of the main window, and the Sketcher grid appears in the viewport.

3. To sketch the profile of the cantilever beam, you need to select the rectangle drawing tool, as shown in Figure W1–5.
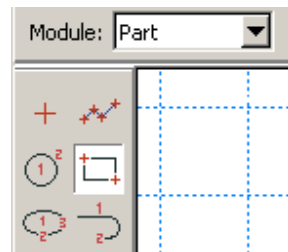

Figure W1–5 Rectangle sketch tool.

4. In the viewport, sketch the rectangle using the following steps:

    a. You will first sketch a rough approximation of the beam and then use constraints and dimensions to refine the sketch. Select any two points as the opposite corners of the rectangle.

    b. Click mouse button 2 anywhere in the viewport to exit the rectangle tool.

    c. The Sketcher automatically adds constraints to the sketch (in this case the four corners of the rectangle are assigned perpendicular constraints and one edge is designated as horizontal).

    d. Use the dimension tool to dimension the top and left edges of the rectangle. The top edge should have a horizontal dimension of **200** mm, and the left edge should have a vertical dimension of **20** mm. When dimensioning each edge, simply select the line, click mouse button 1 to position the dimension text, and then enter the new dimension in the prompt area.

    e. The final sketch is shown in Figure W1–6.
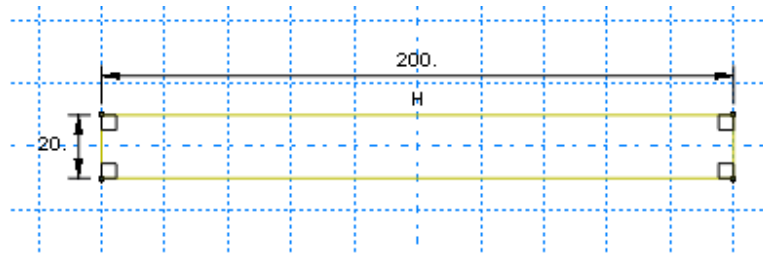
Introduction to Abaqus

Figure W1–6 Sketch of the rectangle.

5. Click **Done** in the prompt area to exit the sketcher.
6. Enter an extrusion depth of **25** in the **Edit Base Extrusion** dialog box, and click **OK**.

   Abaqus/CAE displays an isometric view of the new part, as shown in Figure W1–7. The **Parts** icon in the Model Tree changes to indicate that the model contains one part, .
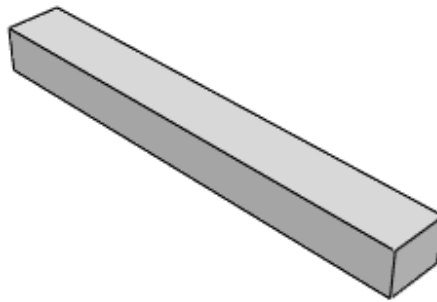

Figure W1–7 Extruded part.

Creating a material definition

You will now create a single linear elastic material with a Young's modulus of $209{\times}10^3$ MPa and Poisson's ratio of 0.3.

To define a material:

1. In the Model Tree, double-click **Materials** to create a new material in the model **BEAM**.

   Abaqus/CAE switches to the Property module, and the material editor appears.

2. In the **Edit Material** dialog box, name the material **Steel**. Notice the various options available in this dialog box.
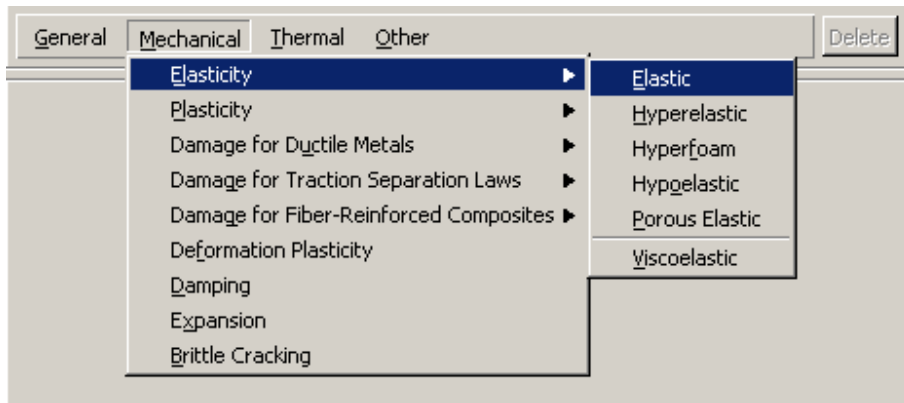
Figure W1–8 Pull–down menu.

3. From the material editor's menu bar, select **Mechanical→Elasticity→Elastic,** as shown in Figure W1–8.

   Abaqus/CAE displays the **Elastic** data form.

4. Enter a value of **209.E3** for Young's modulus and a value of **0.3** for Poisson's ratio in the respective fields, as shown in Figure W1–9. Use **[Tab]** to move between cells, or use the mouse to select a cell for data entry.

Figure W1–9 Material editor.

5. Click **OK** to exit the material editor.

Defining and assigning section properties

Next, you will create a homogeneous solid section and assign it to the beam. The section will refer to the material **Steel** that you just created.

To define the homogeneous solid section:

1. In the Model Tree, double-click **Sections** to create a new section in the model **BEAM**.

   The **Create Section** dialog box appears.

2. In the **Create Section** dialog box:

   a. Name the section **BeamSection**.

   b. Accept the default category **Solid** and the default type **Homogeneous**.

   c. Click **Continue**.

   The solid section editor appears.

3. In the **Edit Section** dialog box:

   a. Accept the default selection of **Steel** for the **Material** associated with the section.

   b. Accept the default value of **1** for **Plane stress/strain thickness**.

   Note: For three-dimensional solid geometry, this value is not used. It is only relevant for two-dimensional geometry.

   c. Click **OK**.

To assign the section definition to the cantilever beam:

1. In the Model Tree, expand the branch for the part **Beam** (click the "+" symbol to expand the **Parts** container and then click the "+" symbol next to the part **Beam**).

2. Double-click **Section Assignments** to assign a section to the part **Beam**.

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

3. Click anywhere on the beam to select the entire part as the region to which the section will be assigned.

4. Click mouse button 2 in the viewport or click **Done** in the prompt area to accept the selected geometry.

   The section assignment editor appears.

5. In the **Edit Section Assignment** dialog box, accept the default selection of **BeamSection** as the section definition, and click **OK**.

   Abaqus/CAE colors the beam green to indicate that the section has been assigned.

Assembling the model

The assembly for this analysis consists of a single instance of the part `Beam`.

To assemble the model:

1. In the Model Tree, expand the branch for the **Assembly** of the model **BEAM** and double-click **Instances** to create a new part instance.

   Abaqus/CAE switches to the Assembly module, and the **Create Instance** dialog box appears.

2. In the **Create Instance** dialog box, select `Beam` and click **OK**.

   Abaqus/CAE displays the new part instance in the viewport.

Configuring the analysis

In this simulation we are interested in the static response of the cantilever beam to a pressure load applied over the beam top. This is a single event, so only a single analysis step is needed for the simulation. Consequently, this model will consist of two steps:

- An initial step, in which you will apply a boundary condition that constrains one end of the cantilever beam.

- A general, static analysis step, in which you will apply a pressure load to the top face of the beam.

Abaqus/CAE generates the initial step automatically, but you must create the analysis step yourself.

To create a general, static analysis step:

1. In the Model Tree, double-click **Steps** to create a new step in the model **BEAM**.

   Abaqus/CAE switches to the Step module, and the **Create Step** dialog box appears.

2. In the **Create Step** dialog box:

   a. Name the step `BeamLoad`.

   b. From the list of available general procedures in the **Create Step** dialog box, select **Static, General** if it is not already selected.

   c. Click **Continue**.

   The step editor appears.

3. In the **Edit Step** dialog box:

   a. In the **Description** field of the **Basic** tabbed page, enter `Load the top of the beam`.

   b. Click the **Incrementation** tab, and delete the value of `1` that appears in the **Initial** text field. Type a value of `0.1` for the initial increment size.

   c. Click the **Other** tab to see its contents; you can accept the default values provided for the step.

   d. Click **OK** to create the step and to exit the step editor.

Applying a boundary condition and a load to the model

Next, you will define the boundary condition and loading that will be active during the **BeamLoad** step.

To apply boundary conditions to one end of the cantilever beam:

1. In the Model Tree, double-click **BCs** to create a new boundary condition in the model **BEAM**.

    Abaqus/CAE switches to the Load module, and the **Create Boundary Condition** dialog box appears.

2. In the **Create Boundary Condition** dialog box:

    a. Name the boundary condition **Fixed**.

    b. Select **Initial** as the step in which the boundary condition will be activated.

    c. In the **Category** list, accept the default category selection **Mechanical**.

    d. In the **Types for Selected Step** list, select **Displacement/Rotation** as the type.

    e. Click **Continue**.

        Abaqus/CAE displays prompts in the prompt area to guide you through the procedure. The face at the end of the cantilever beam will be fixed.

3. Click the **Show/Hide Selection Options** icon [icon] in the prompt area to open the

    Options dialog box. Deselect **Select the Entity Closest to the Screen** [icon] in the **Options** dialog box so that you can pick the desired face, as shown in Figure W1–10.
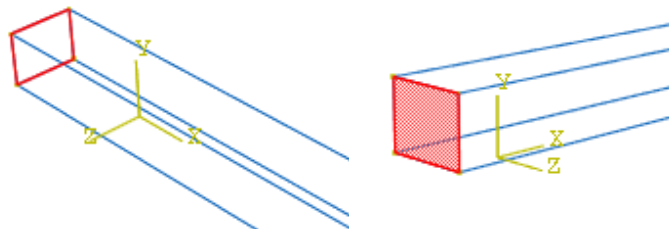

Figure W1–10 Fixed end.

If you click when the selection is ambiguous, Abaqus/CAE will select the face closest to you and display buttons in the prompt area that allow you to choose the required face. To cycle through the valid selections, do the following:

    a. From the prompt area, click **Next** or **Previous** until the desired face is selected.

    b. Click **OK** to accept the current selection.

4. Click mouse button 2 in the viewport or click **Done** in the prompt area to accept the selected geometry.

The **Edit Boundary Condition** dialog box appears. When you are defining a boundary condition in the initial step, all six degrees of freedom are unconstrained by default.

5. In the **Edit Boundary Condition** dialog box:

    a. Toggle on **U1**, **U2**, and **U3**, since only the translational degrees of freedom need to be constrained (the beam will be meshed with solid elements later).

    b. Click **OK** to create the boundary condition definition and to exit the editor.

Abaqus/CAE displays arrows at each corner and midpoint on the selected face to indicate the constrained degrees of freedom, as shown in Figure W1–11.

To apply a load to the top of the cantilever beam:

1. In the Model Tree, double-click **Loads** to create a new load in the model **BEAM**.

    The **Create Load** dialog box appears.

2. In the **Create Load** dialog box:

    a. Name the load **Pressure**.

    b. Select **BeamLoad** as the step in which the load will be applied.

    c. In the **Category** list, accept the default category selection **Mechanical**.

    d. In the **Types for Selected Step** list, select **Pressure**.

    e. Click **Continue**.

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

3. In the viewport, select the top face of the beam as the surface to which the load will be applied. The desired face is highlighted in Figure W1–11.
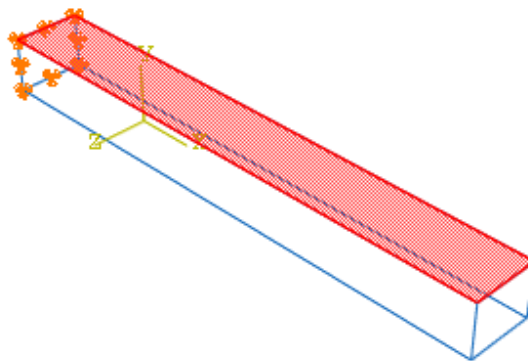

Figure W1–11 Fixed boundary conditions; face for distributed load.

4. Click mouse button 2 in the viewport or click **Done** in the prompt area to indicate that you have finished selecting regions.

5. In the **Edit Load** dialog box:

    a. Enter a magnitude of **0.5** for the load.

b. Accept the default **Amplitude** selection (`Ramp`) and the default **Distribution** (`Uniform`).

c. Click **OK** to create the load definition and to exit the editor.

Abaqus/CAE displays downward–pointing arrows along the top face of the beam to indicate the load is applied in the negative 2–direction.

Meshing the model

You use the Mesh module to generate the finite element mesh. You can choose the meshing technique that Abaqus/CAE will use to create the mesh, the element shape, and the element type. Abaqus/CAE uses a number of different meshing techniques. The default meshing technique assigned to the model is indicated by the color of the model when you enter the Mesh module; if Abaqus/CAE displays the model in orange, it cannot be meshed without assistance from the user.

To assign the mesh controls:

1. In the Model Tree, double-click **Mesh** in the branch for the part **Beam**.

    Abaqus/CAE switches to the Mesh module and displays the part **Beam**.

2. From the main menu bar, select **Mesh→Controls**.

3. In the **Mesh Controls** dialog box, accept **Hex** as the default **Element Shape** selection.

4. Accept **Structured** as the default **Technique** selection.

5. Click **OK** to assign the mesh controls and to close the dialog box.

To assign an Abaqus element type:

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

2. In the **Element Type** dialog box, accept the following default selections that control the elements that are available for selection:

    • **Standard** is the default **Element Library** selection.

    • **Linear** is the default **Geometric Order**.

    • **3D Stress** is the default **Family** of elements.

3. In the lower portion of the dialog box, examine the element shape options. A brief description of the default element selection is available at the bottom of each tabbed page.

4. In the **Hex** tabbed page, select **Incompatible modes**.

    A description of the element type C3D8I appears at the bottom of the dialog box. Abaqus/CAE will now mesh the part with C3D8I elements.

5. Click **OK** to assign the element type and to close the dialog box.

To mesh the model:

1. From the main menu bar, select **Seed→Part** to seed the part.

The **Global Seeds** dialog box appears. The default global element size is based on the size of the part.

2. In the **Global Seeds** dialog box, enter an approximate global size of **10** and click **OK**.

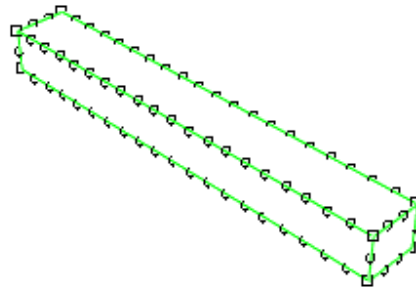Abaqus/CAE applies the seeds to the part, as shown in Figure W1–12.

Figure W1–12 Seeded part instance.

3. From the main menu bar, select **Mesh→Part** to mesh the part.

4. Click **Yes** in the prompt area or click mouse button 2 in the viewport to confirm that you want to mesh the part instance.

Abaqus/CAE meshes the part instance and displays the resulting mesh, as shown in Figure W1–13.
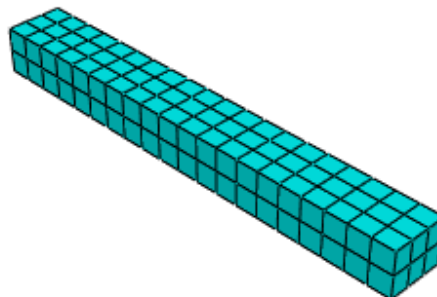
Figure W1–13 Part instance showing the resulting mesh.

Creating and submitting an analysis job

The definition of the model **BEAM** is now complete. Next, you will create and submit an analysis job to analyze the model.

To create and submit an analysis job:

1. In the Model Tree, double-click **Jobs** to create a new analysis job.

Abaqus/CAE switches to the Job module, and the **Create Job** dialog box appears.

2. In the **Create Job** dialog box, name the job **Deform** and select the model **BEAM**. Click **Continue**.

The job editor appears.

3. In the **Description** field of the **Edit Job** dialog box, enter `Workshop 1`.

4. Click the tabs to see the contents of each folder of the job editor and to review the default settings. Click **OK** to accept the default job settings.

5. In the Model Tree, expand the **Jobs** container and click mouse button 3 on the job **Deform**.

   The menu for the job **Deform** appears.

6. From the job menu, select **Submit**.

   The icon for the job will change to indicate the status of the job in parenthesis after the job name. As the job runs the status **Running** will be shown in the Model Tree. When the job completes successfully, the status will change to **Completed**, as shown in Figure W1–14.
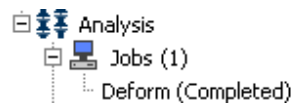
Figure W1–14 Job status in the Model Tree.

Viewing the analysis results

You are now ready to view the results of the analysis in the Visualization module.

1. In the Model Tree, click mouse button 3 on the job **Deform** and select **Results** from the menu that appears.

   Abaqus/CAE switches to the Visualization module, opens the output database created by the job (`Deform.odb`), and displays the undeformed shape of the model, as shown in Figure W1–15.
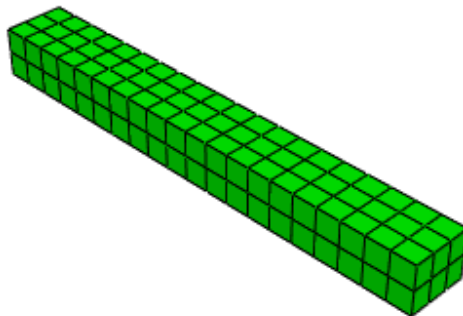

Figure W1–15 Undeformed model shape.

2. In the toolbox, click (or select **Plot→Deformed Shape** from the main menu bar) to view a deformed model shape plot, as shown in Figure W1–16.

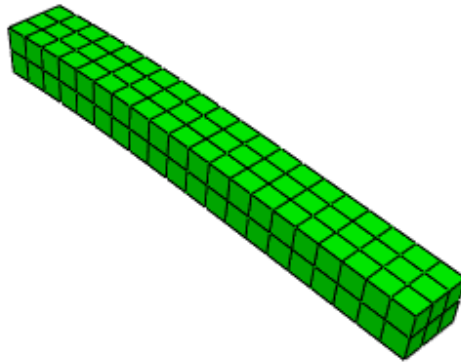Figure W1−16 Deformed model shape.

You may need to use the **Auto-Fit View** tool to rescale the figure in the viewport.

3. In the toolbox, click (or select **Plot→Contours→on Deformed Shape** from the main menu bar) to view a contour plot of the von Mises stress, as shown in Figure W1−17. In this figure, the font size for the legend text has been increased to 10 (**Viewport→ Viewport Annotation Options**).
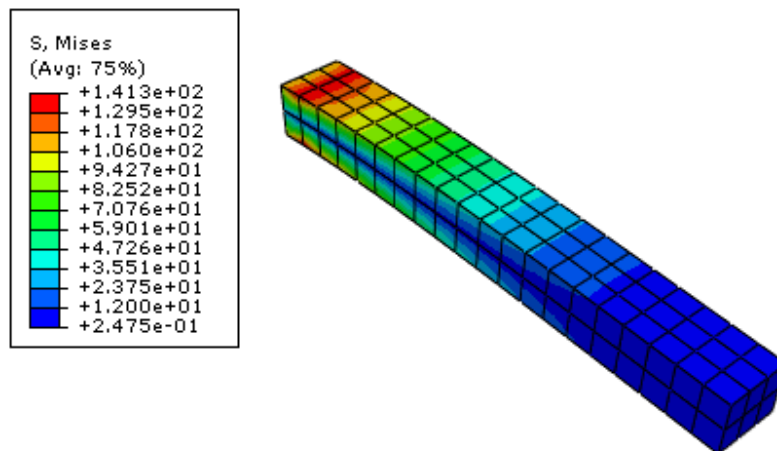

Figure W1−17 Mises stress contour plot.

4. In the toolbar, click (or select **File□Save** from the main menu bar) to save your model in a model database file. Close the Abaqus/CAE session by selecting **File□Exit** from the main menu bar.

5. Rename the **abaqus.rpy** replay file **BEAM.rpy**. Start a new Abaqus/CAE session, and execute the replay file using the command

        abaqus cae replay=BEAM.rpy

Introduction to Abaqus

Commands issued during the previous Abaqus/CAE session are executed automatically through the replay Python script. Close the Abaqus/CAE session by selecting **File→Exit** from the main menu bar.

6. Start a new Abaqus/CAE session, and rebuild the **BEAM.cae** model database using the command

```
abaqus cae recover=BEAM.jnl
```

Only the commands necessary to rebuild the model database **BEAM.cae** are executed.