

## Workshop 1

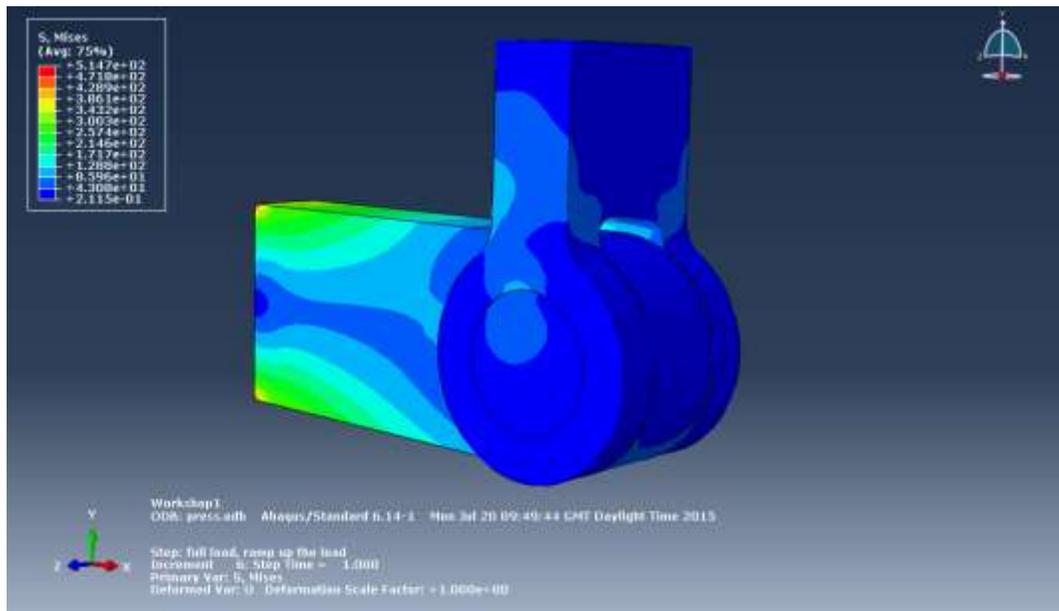
### Abaqus model build and Simulation

Estimated time to build the model: 60 – 90 mins.

Estimated time to run the model on Windows with 1 cpu: 25 mins.

#### Introduction

This workshop will give you an opportunity to generate an Abaqus model using the Graphical User Interface (GUI), Abaqus/CAE, apply loads, boundary conditions and interactions and submit and monitor an analysis. The results will be viewed in Abaqus/CAE.



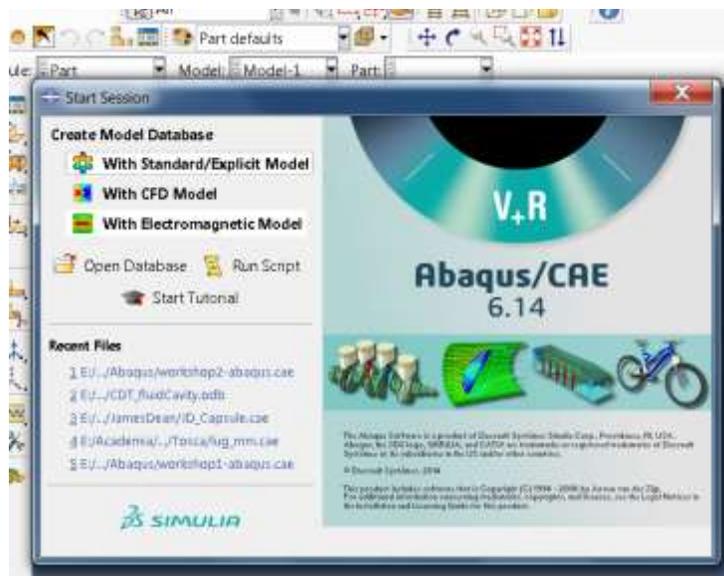
#### Preliminaries

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

***abaqus cae***

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

In the **Start Session** dialog box, underneath **Create Model Database**, click **With Standard/Explicit Model**.



To save the model database, select **File**→**Save As** from the main menu bar and type the file name **LugAssembly** 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 lug and extruding it.

1. Abaqus/CAE automatically loads the Part module. Any other module can be accessed from the **Module** list located in the context bar, as shown in Figure W1–2.

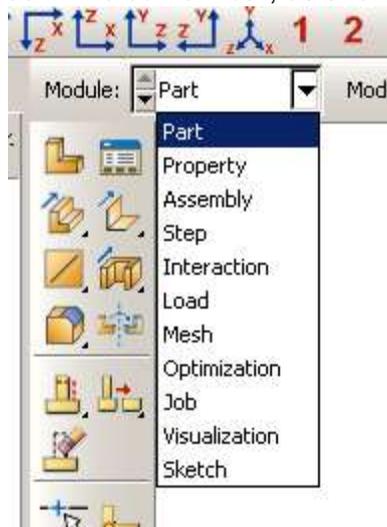


Figure W1–2. Module list

2. From the main menu bar, select **Part**→**Create** to create a new part. In the **Create Part** dialog box that appears, name the part **lug**, and specify an approximate size of **400**.

Accept the default settings of a three-dimensional, deformable body with a solid, extruded base feature. Click **Continue**.

- a. 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. At any stage of the process you can 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.

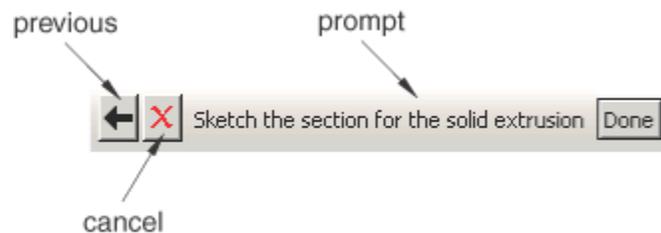


Figure W1–4 Prompt area.

- b. The Sketcher toolbox appears in the left side of the main window, and the Sketcher grid appears in the viewport. The grid is a 2D plane on which you create the base shape for your part. Abaqus/CAE overlays the sheet with a grid of invisible grid points to help you position the cursor as you draw, move, resize, or reshape objects. By default, when you move the cursor near a grid point, the cursor automatically moves, or snaps, to the point. This behavior allows you to easily position the cursor precisely on a grid point while also providing you with full control when the cursor is not close to any grid point.
3. In the viewport, sketch the rectangular body of the lug as follows:
    - a. To sketch the profile of the lug, you first need to select the 'create lines - connected' drawing tool, as shown in Figure W1–5. Create 3 lines with vertices at (100,0); (0,0); (0,50) and (100,50), then click mouse button 2 twice to complete the task.

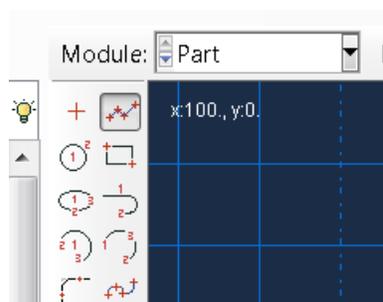


Figure W1–5 Create Lines - Connected sketch tool.

- b. Next create the radiused end of the lug using the **Create Arc; Centre and 2 end points** tool, . As shown in Figure W1-6, define the arc centre at (100,25) then pick the end points of the open box to complete the lug profile. Click Mouse button 2 to complete the operation.

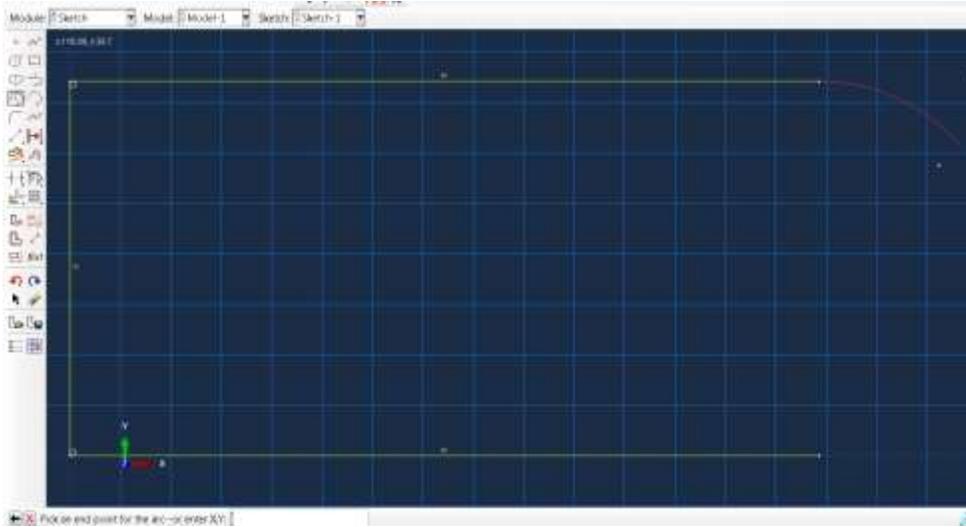


Figure W1–6 Adding semi-circular end to lug profile.

- c. Lastly, add the 30mm diameter hole for the lug using the Create Circle, Centre and Perimeter tool. Select (100,25) as the circle centre followed by (100,40) as the circle perimeter to end up with the profile shown in Figure W1-7

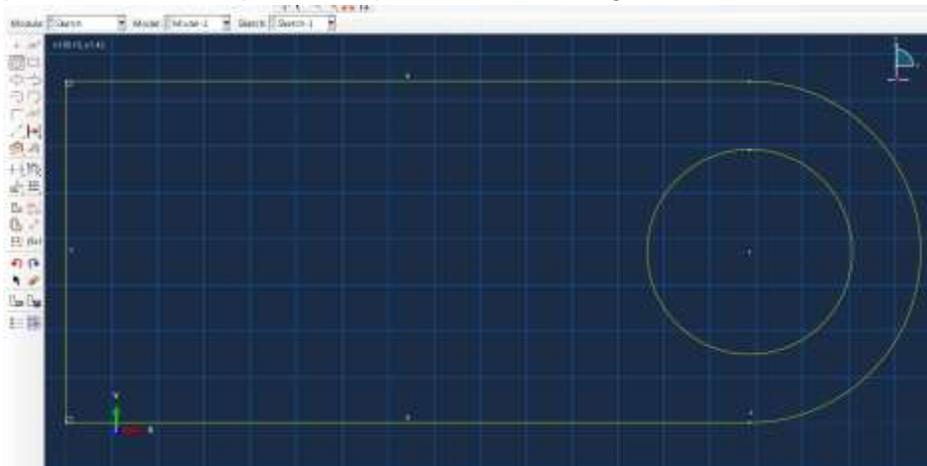


Figure W1–7 Adding Circular Hole to lug profile.

- d. Click **Done** in the prompt area to exit the sketcher.  
 e. Enter an extrusion depth of **20** in the **Edit Base Extrusion** dialog box, and click **OK**.  
 Abaqus/CAE displays an isometric view of the new part, as shown in Figure W1–8.  
 Save the database  .

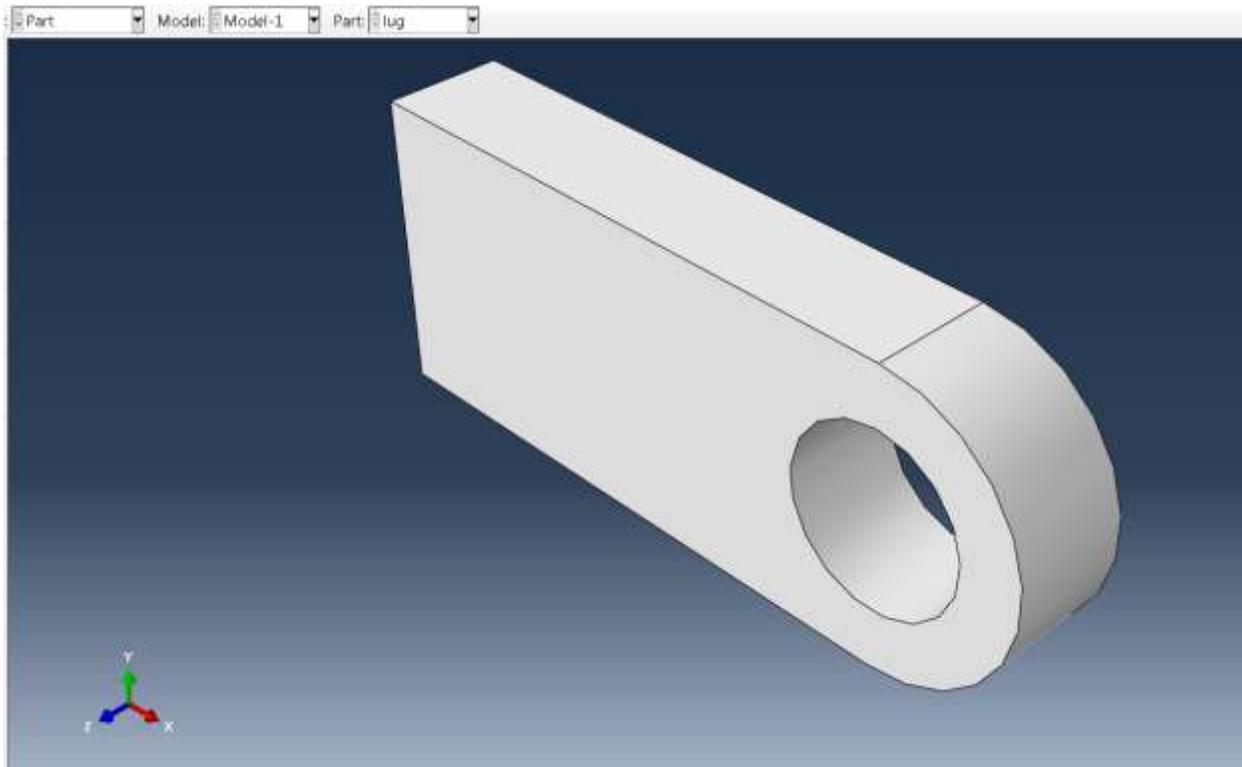
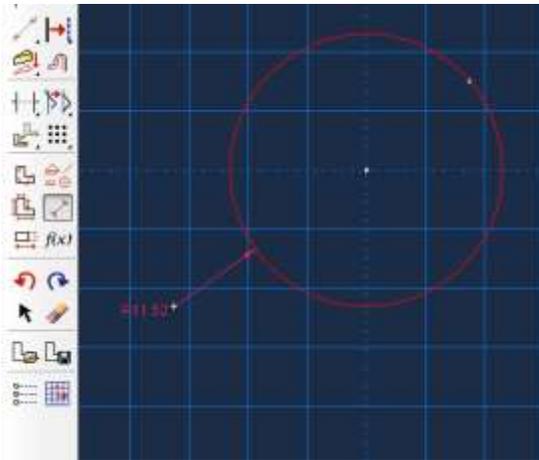


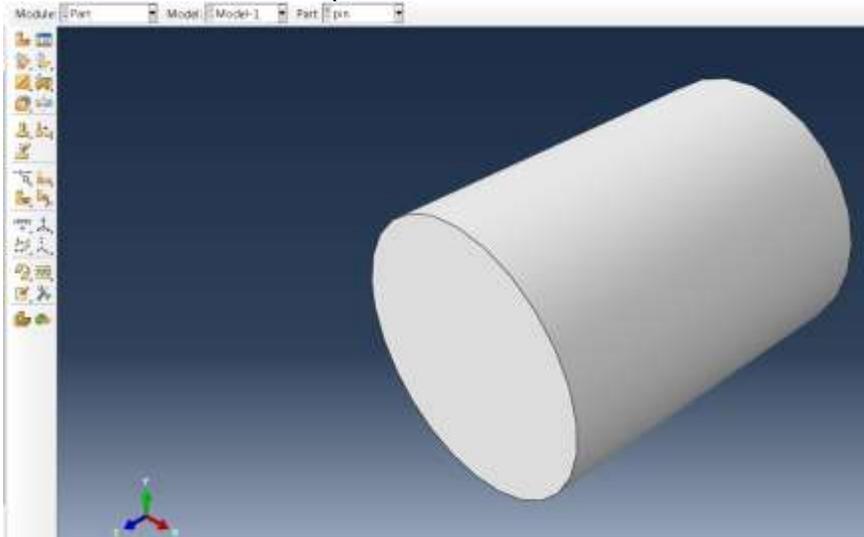
Figure W1–8 Extruded part.

4. Now create the second part – the Pin. From the main menu bar, select **Part**→**Create** to create a new part. In the **Create Part** dialog box that appears, name the part **pin**. Accept the default settings of a three-dimensional, deformable body with a solid, extruded base feature, approximate size of 200 mm. Click **Continue**
  - a) We will first create a rough approximation of the pin profile and then use constraints and dimensions to refine the sketch. Use the Create Circle, Centre and perimeter tool to generate a circle of any size. Suggest you use (0,0) as the circle centre.
  - b) Then use the Add Dimension tool  and select the circle circumference. Set the radius to 15.0 mm.

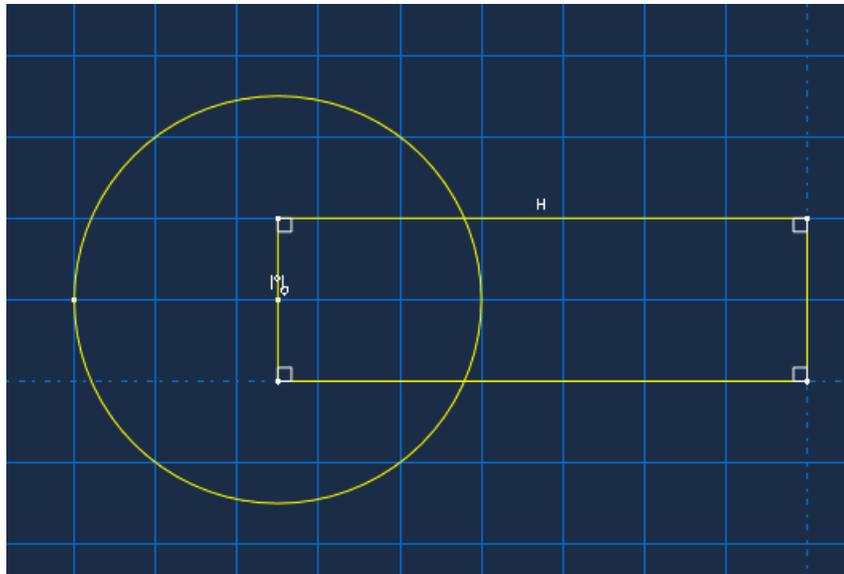


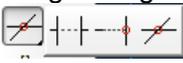
- c) Then click 'done' to accept the profile and exit the sketcher.
- d) Enter an extrusion depth of **40** in the **Edit Base Extrusion** dialog box, and click **OK**.

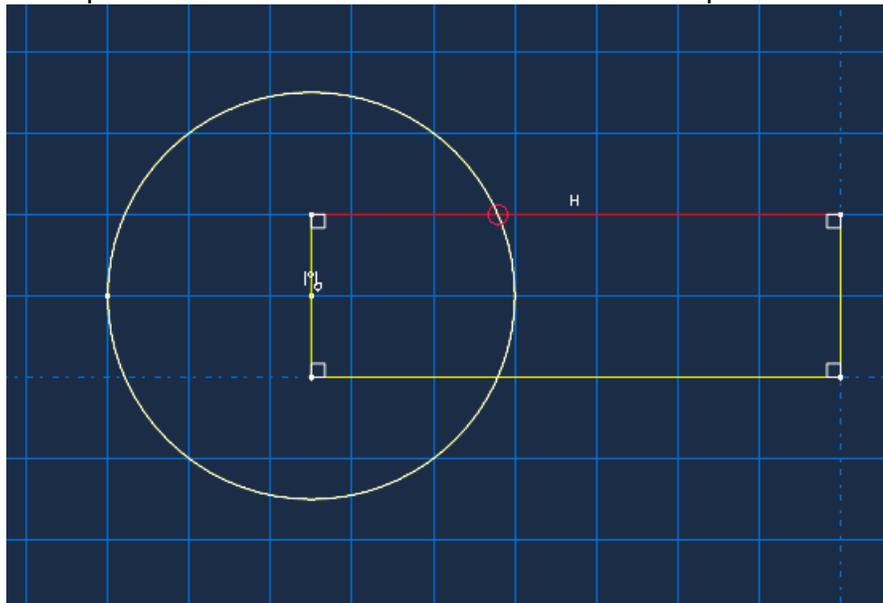
Save the Abaqus/CAE database.



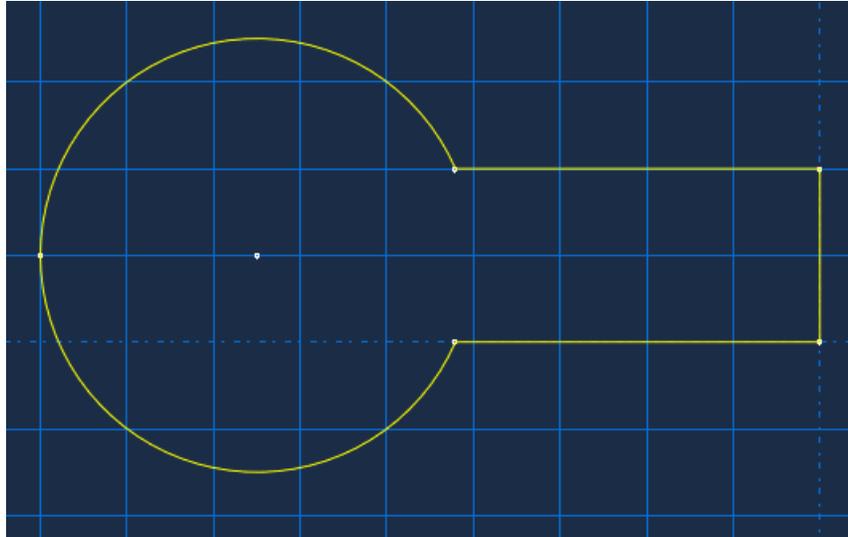
5. Now create the third part – the second lug. From the main menu bar, select **Part**→**Create** to create a new part. In the **Create Part** dialog box that appears, name the part **lug2** and specify an approximate size of **400**. Accept the default settings of a three-dimensional, deformable body with a solid, extruded base feature. Click **Continue**
  - a) Create a rectangle  of dimensions 65 x 20. . Note that in this instance the bottom right hand corner of the rectangle is at (0,0). Then superimpose a circle  of diameter 50 centred half way up the rectangle end at x = -65.



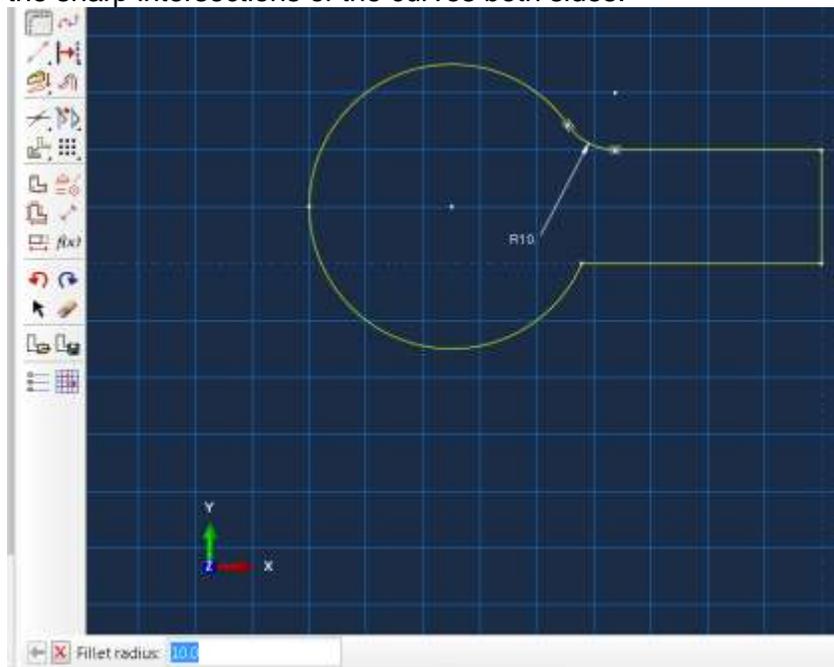
- b) Then break each of the horizontal rectangle edges and the circle at the intersection points using the 'split' button . Note that you will have to press and hold the mouse button to select this option.



- c) After breaking all 4 curves use the delete button  to remove the internal unwanted curves.

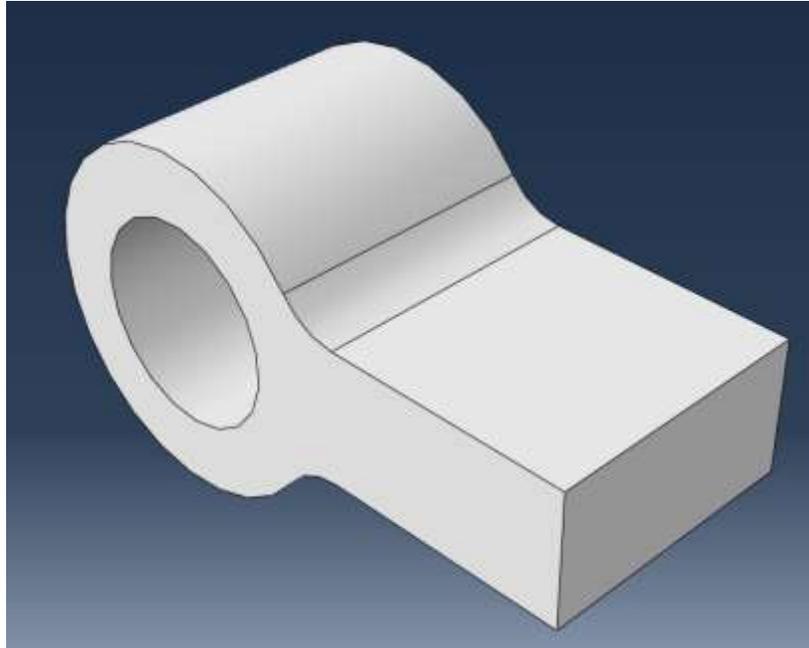


- d) Next use the fillet radius between 2 curves tool  to create 10mm fillet radii at the sharp intersections of the curves both sides.



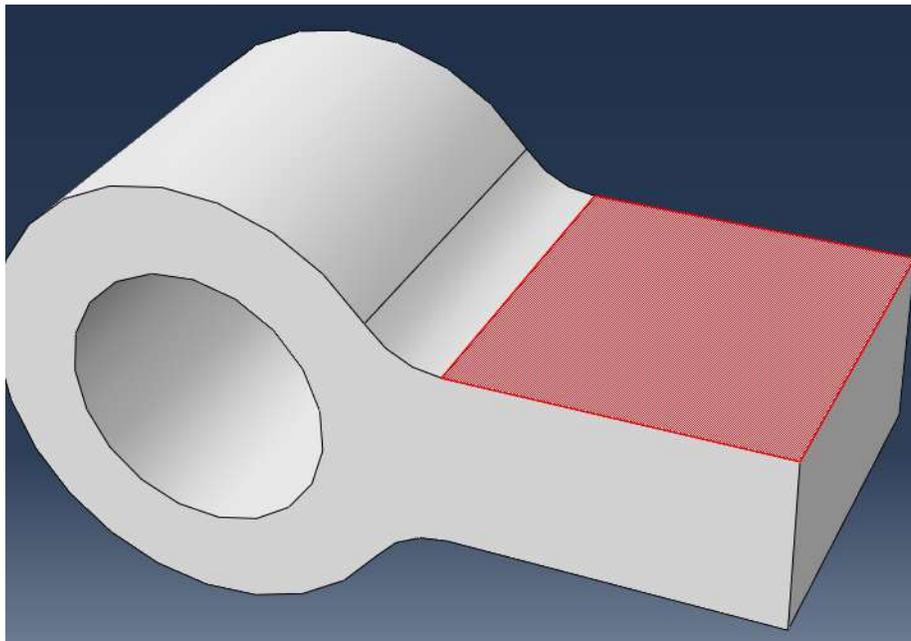
- e) Create a 30 mm diameter circle for the pin hole at the curved section centre.  
 f) Then click 'done' to accept the profile and exit the sketcher.  
 g) Enter an extrusion depth of **40** in the **Edit Base Extrusion** dialog box, and click **OK**.

Save the Abaqus/CAE database.



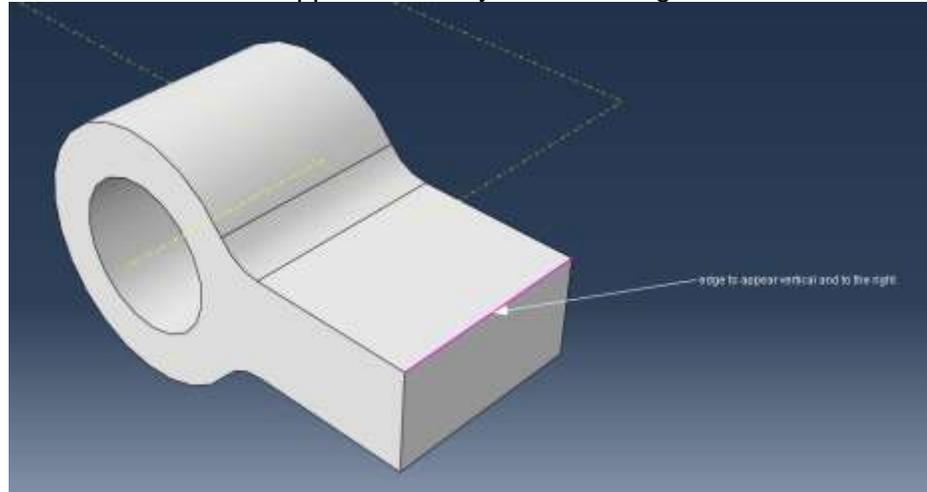
6. Now we will use the geometry editing toolset to add features to the **lug2** part. These features will be used to finalize the geometry of the lug2 part.
- a. First create a datum plane to use as the basis for the sketch, which will be used to generate the sketch for the extrusion. Use the '**Create datum plane: offset from**

**plane**' tool  and select the top plane of the lug rectangular body.

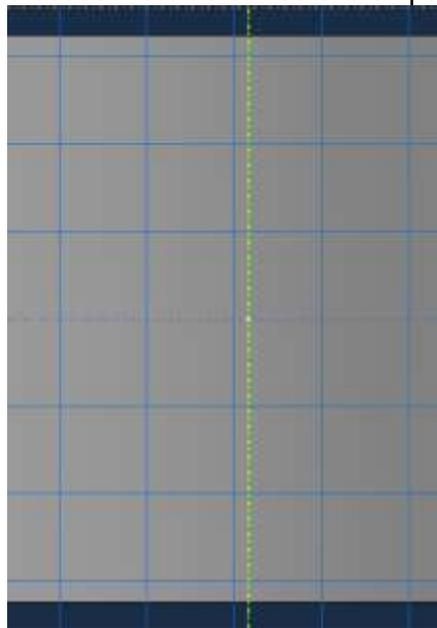


- b. Enter an offset value of 30 mm away from the face.

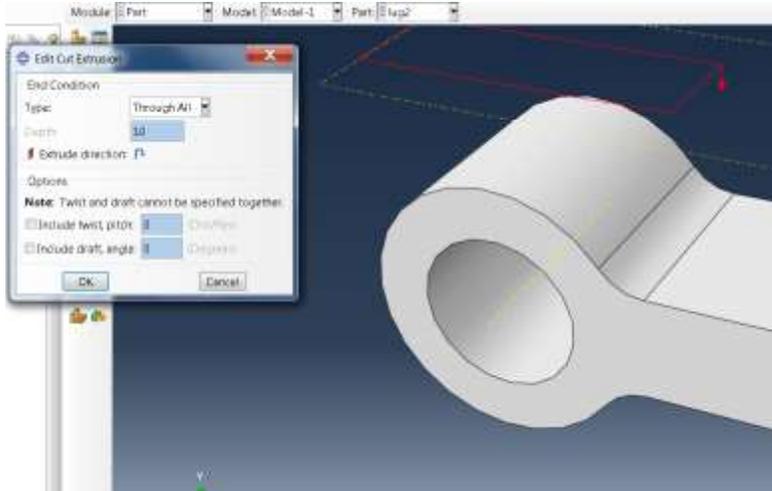
- c. From the main menu select **Tools > Datum > Axis > Axis of cylinder** and select the hole surface to be the cylindrical face.
- d. Next use the Create Cut: Extrude tool  to cut away the central part of the lug body. Select the newly created datum plane as the sketch plane, and the back edge of the lug to be the axis that will appear vertically and to the right.



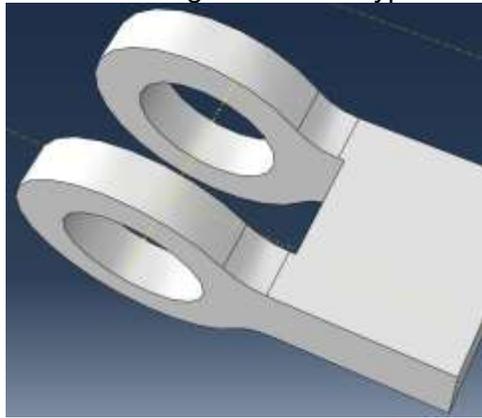
- e. You will now use the **sketcher options** tool  to reposition the sketch on the grid. On the General tab click **Origin** to **Align grid**. Select the point where your yellow dotted datum axis crosses the blue dotted sketch pad y-axis.



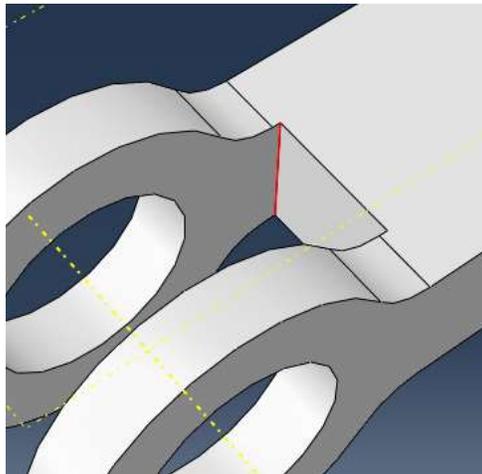
On the General tab change the Grid Sheet size to 200 and the Grid spacing to 5 to set up a regular grid, then create a rectangle  with 2 corners (30, 10.5) and (-30, -10.5) Note that you will have to enter these coordinates in the text prompt box, then **done** to exit the sketcher.



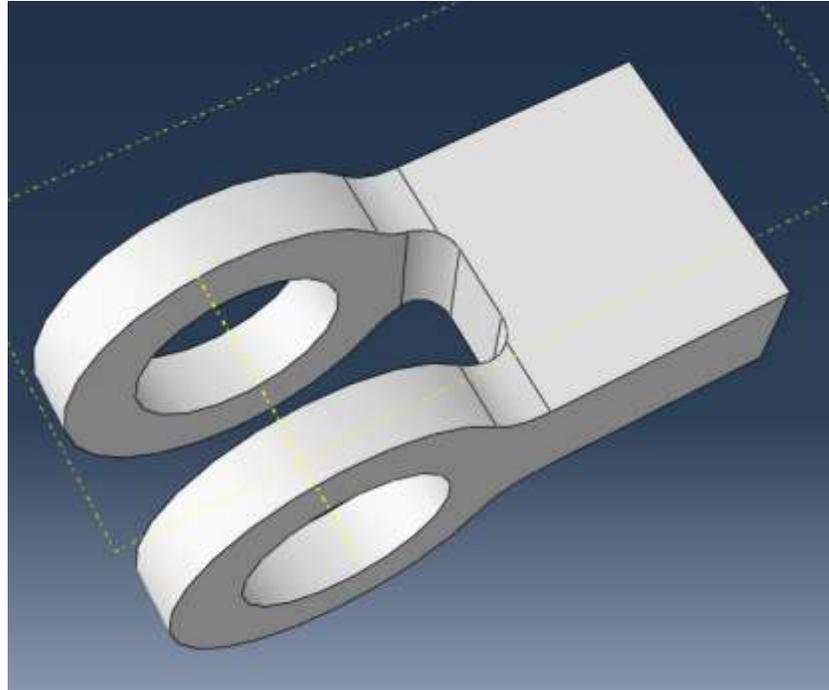
- f. Accept the default 'Through All' for the type of cut extrusion to finish the cut.



- g. Now use the 'Create round or fillet' tool  to fillet the sharp corners left by the cut. Select the corner on one side of the cut



and define the fillet radius as 5 mm. Do the same operations on the other side of the cut to finish with the part as shown. Then save your Abaqus/CAE database.



## 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 **Model-1**.  
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.

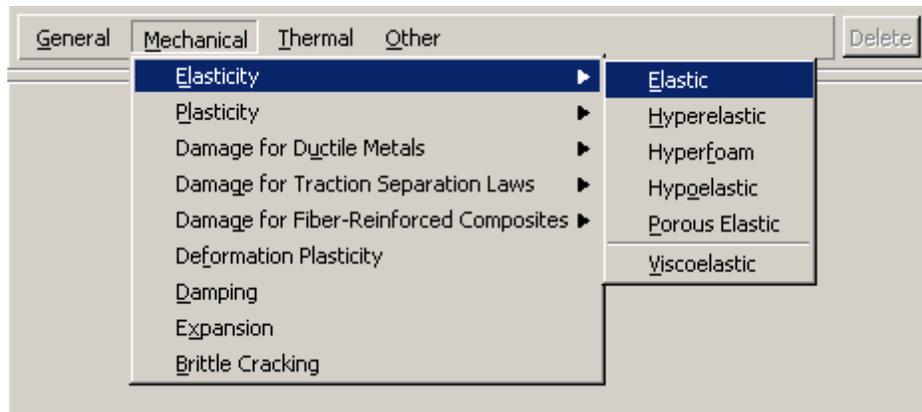


Figure W1–8 Pull–down menu.

- From the material editor's menu bar, select **Mechanical**→**Elasticity**→**Elastic**, as shown in Figure W1–8.  
Abaqus/CAE displays the **Elastic** data form.
- 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.

Data		
	Young's Modulus	Poisson's Ratio
1	209.E3	0.3

Figure W1–9 Material editor.

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

## Defining and assigning section properties

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

### To define the homogeneous solid section:

- In the Model Tree, double-click **Sections** to create a new section in the model **Model-1**. The **Create Section** dialog box appears.
- In the **Create Section** dialog box:
  - Name the section **IugSection**.
  - Accept the default category **Solid** and the default type **Homogeneous**.
  - 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 lug:

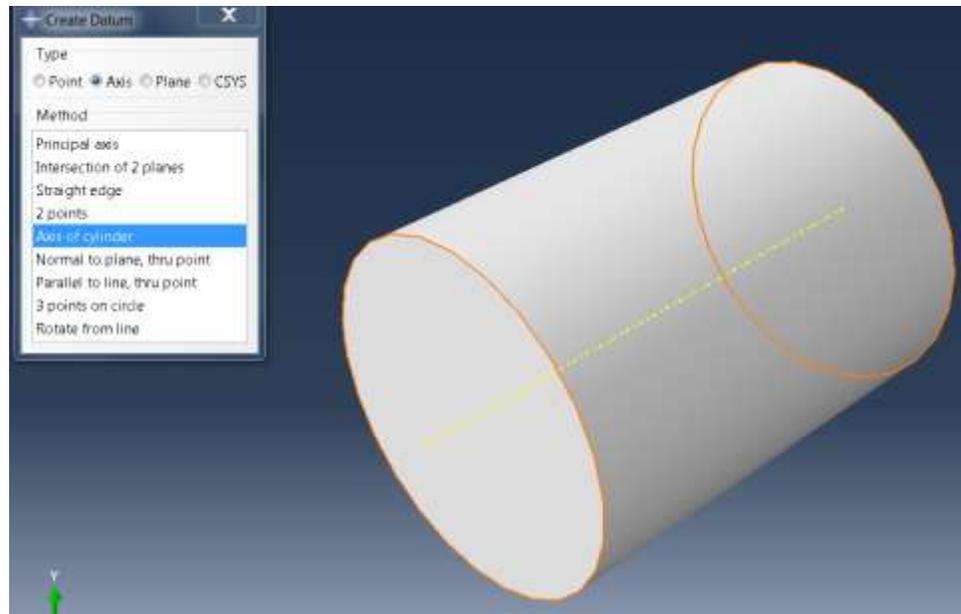
1. In the Model Tree, expand the branch for the part **lug** (click the “+” symbol to expand the **Parts** container and then click the “+” symbol next to the part **lug**).
2. Double-click **Section Assignments** to assign a section to the part **lug**.  
Abaqus/CAE displays prompts in the prompt area 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 **LugSection** as the section definition, and click **OK**.  
Abaqus/CAE colors the lug green to indicate that the section has been assigned.

Now repeat these operations to create sections called **pinsection** and **lug2section** using material ‘Steel’ and assign them to parts **pin** and **lug2** respectively

## Assembling the model

The assembly for this analysis consists of an instance of the part **lug**, an instance of the part **pin** and an instance of the part **lug2**. In order to make the assembly of these parts a little easier we will first create datum axes in the part module for each part.

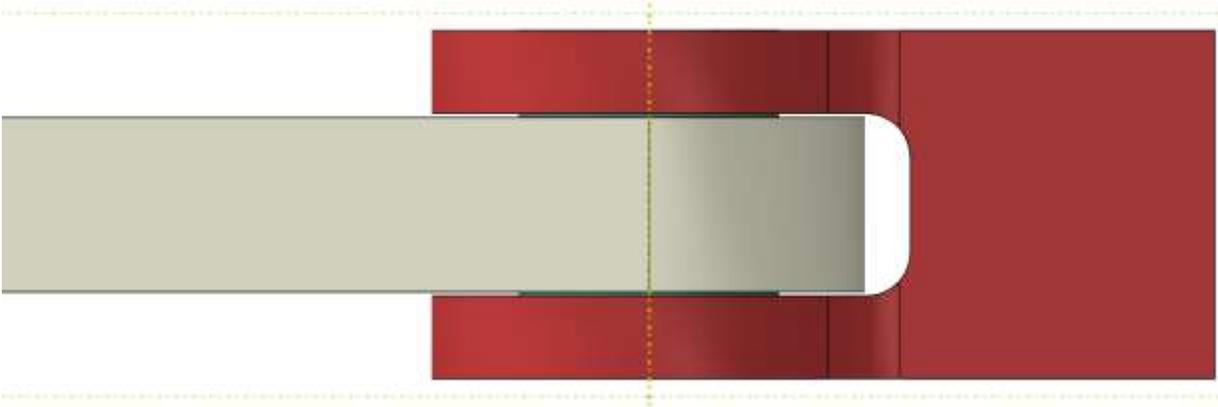
Switch to the Part module and from the menu bar select Tools > Datum > Axis > Axis of Cylinder. You have already defined the axis for part **lug2**, so for **lug** and **pin** define a datum axis using the cylindrical face.



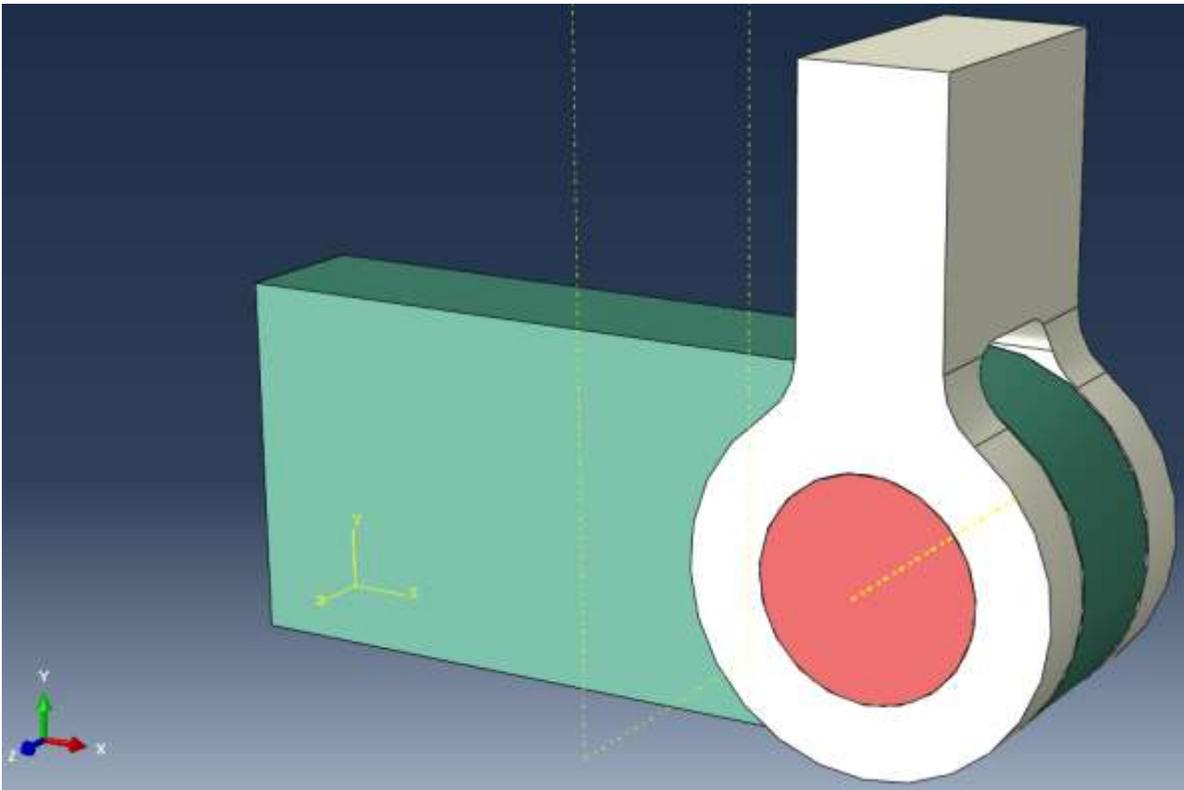
### To assemble the model:

1. In the Model Tree, expand the branch for the **Assembly** of the model **Model-1** and double-click **Instances** to create a new part instance.
  - a. Abaqus/CAE switches to the Assembly module, and the **Create Instance** dialog box appears.
2. In the **Create Instance** dialog box, select all 3 parts, i.e. **lug**, **pin** and **lug2**, accept the Instance type to be **Dependent (mesh on part)** and click **OK**.
  - a. Abaqus/CAE displays the new part instances in the viewport. These now need to be assembled correctly.
3. From the top menu bar select **Constraint > Edge to Edge**. Pick the datum axis of the pin on the movable instance; then pick the datum axis of the first lug on the fixed instance. Check the alignment of the arrows and then OK to move the instance. Note that the coaxial constraint could also have been used here.
4. Repeat this operation using the datum axis of the second lug on the movable instance and the datum axis of the first lug on the fixed instance. (Note: use display group  to select the desired instance to facilitate the selection of the corresponding datum axis).
5. The components should now have a common centerline on their datum axes.
6. Use assembly operations to further position instances correctly: There are two ways of doing this:
  - a. Use the **Instance > translate** dropdown to move **Lug2 and Pin** instances correctly in the assembly. For translation vector and enter point (0.0,0.0,0.0) as the start point of the vector and enter (0.0,0.0,-10.0) as the end point of the vector. Note that the translation vector you need may be different from what is mentioned here, depending on how you built the model.
  - b. Alternatively, you can use face to face constraint with 0.5 distance between the faces to correctly position the instances.

You can use the colour palette    to change from default to parts or part instances to make visualization clearer. You should have the following assembly at the end of this operation.



7. Lastly rotate **lug2** using **Instance > Rotate** through 90 degrees around any two points on the datum axis so that lug2 points up the y-axis as shown. NB Abaqus/CAE rotate uses a Right Hand Rule.



# Configuring the Interactions

In this simulation we have 3 components which will interact with each other. In this case we will be using contact interactions and specifying General Contact between the part instances.

## To create the general contact:

1. In the Model Tree, double-click **Interactions** to create a new interaction in the model **Model-1**.  
Abaqus/CAE switches to the Interaction module, and the **Create Interaction** dialog box appears.
2. In the **Create Interaction** dialog box:
  - a. Name the interaction **contact**.
  - b. Select General contact (Standard) – which is only available in the Initial step.
  - c. Click **Continue**.
3. Accept the default contact domain **All \*with self**. This means that all external faces of all part instances may come into contact with each other – or themselves.
4. We now need to create an Interaction property to assign to this contact condition:-
  - a. Click the **Create Interaction Property** button  on the contact properties tab.
  - b. Name the Property **friction** and accept the default type = contact, then continue
  - c. Create a **Mechanical > Tangential** property with friction formulation = penalty and a coefficient of friction = 0.2, then click **OK**
  - d. In the Edit Interaction box the Global Property Assignment dropdown has been populated with our new interaction property, **friction**, so select this and then click OK to complete the contact interaction. Then save the Abaqus/CAE database.

# Configuring the analysis

In this simulation we are interested in the static response of the first lug to a pressure applied over the flat surface of the second lug. Although this is a single event we will apply the total displacement over two load steps following the initial step so, consequently, this model will consist of three steps:

- An initial step, in which you will apply boundary conditions to the assembly
- A general, static analysis step in which you will apply a very small pressure to 'settle' the contact.
- A general, static analysis step in which you will apply a pressure of 37.5 N/mm<sup>2</sup> normal to the end face of **lug2**.

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

## To create a general, static analysis step:

1. In the Model Tree, double-click **Steps** to create a new step in the model **Model-1**.  
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 **settle**.
  - 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 **settle contact**.
  - b. Check the button to turn Nlgeom on.
  - c. 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.
  - d. Click the **Other** tab to see its contents; you can accept the default values provided for the step.
  - e. Click **OK** to create the step and to exit the step editor.
4. Repeat these operations to create a third step called **full-load**. Note that for this step Nlgeom is turned on by default.

## Applying the boundary conditions and loads to the model

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

### To apply boundary conditions to the fixed end of the lug:

1. In the Model Tree, double-click **BCs** to create a new boundary condition in the model **Model-1**.  
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 first lug will be fixed.
3. In the select options box  deselect **Select the Entity Closest to the Screen**  so that you can pick the desired face, as shown in Figure W1–10.

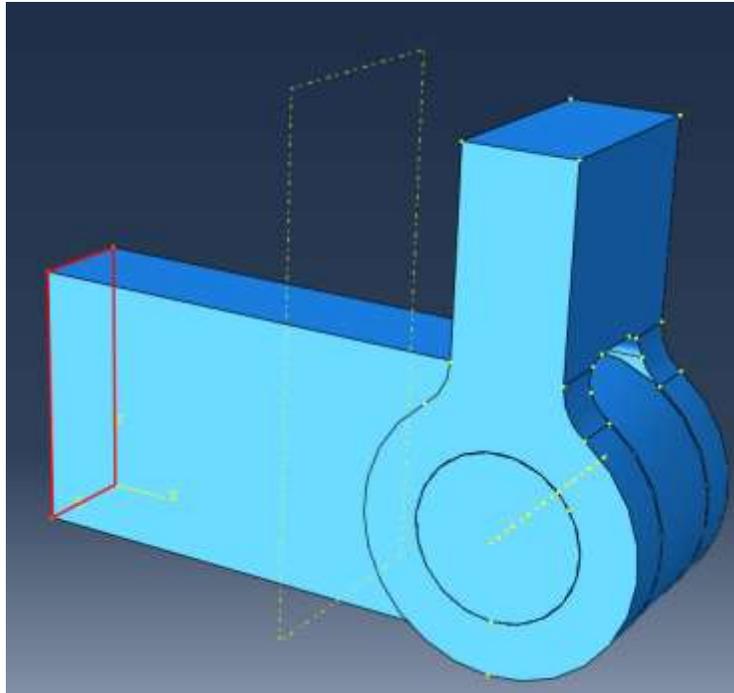


Figure W1–10 Correct face selected.

When you move the cursor over this face, Abaqus/CAE will change the cursor, adding ellipsis marks (...) to the right of the arrow, to indicate that more than one selection object is available. 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 lug 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 faces to indicate the constrained degrees of freedom, as shown in Figure W1–11.

Use the same procedure to apply a boundary condition called **guide** to the free end of **lug2** in the initial step and in degrees of freedom 1 and 3 only (we will be loading lug2 in degree of freedom 2, the Y-axis).

### To apply a load to the top of lug2:

1. In the Model Tree, double-click **Loads** to create a new load in the model **Model-1**. The **Create Load** dialog box appears.

2. In the **Create Load** dialog box:
  - a. Name the load **small-pressure**.
  - b. Select **settle** 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 lug2 as the surface to which the load will be applied. The desired face is highlighted in Figure W1–11.

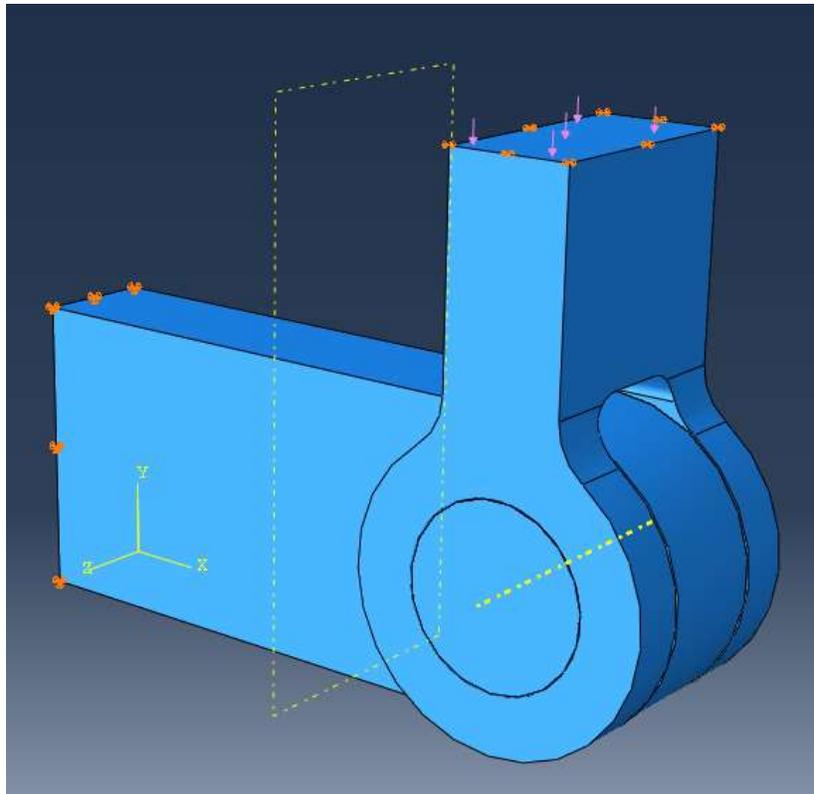


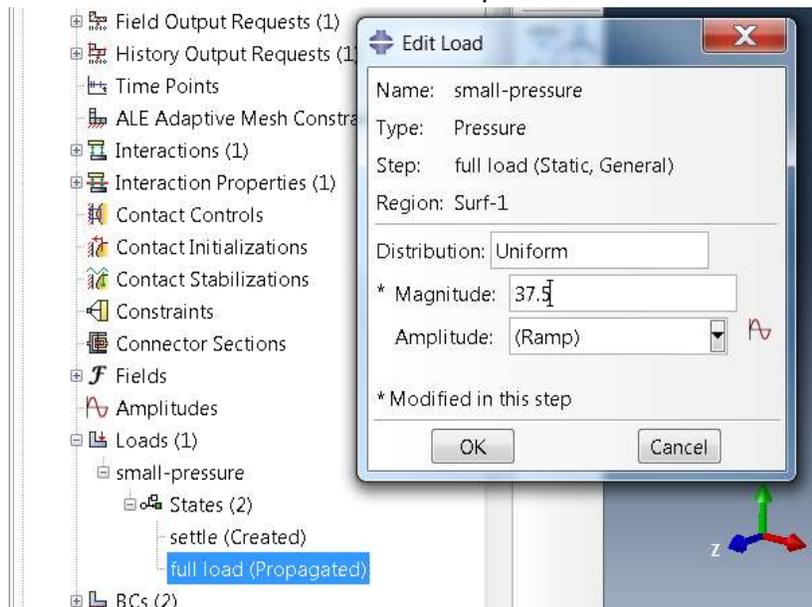
Figure W1–11 Boundary conditions; and distributed load.

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 **0.001** 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.

The **small-pressure** load is intended to settle the contact between lugs and pin – the full load will be applied in the third **full-load** step. In order to do this expand Loads in the model tree,

then expand load **small-pressure** and states and double-click '**full-load (Propagated)**' to edit the value of the load in this step. Apply the full load of  $37.5 \text{ N/mm}^2$ . Then click OK to complete the modification. Then save the Abaqus/CAE database



## 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 for each of the part instances. 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 **Lug**. Abaqus/CAE switches to the Mesh module and displays the part **Lug**.
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 **Sweep** as the default **Technique** selection.
5. Change the Algorithm to **Medial axis**.
6. Click **OK** to assign the mesh controls and to close the dialog box.

### To assign an Abaqus element type:

7. From the main menu bar, select **Mesh**→**Element Type**, select the region if required.
8. 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.
- **3D Stress** is the default **Family** of elements.

Change the **Geometric Order** from **Linear** to **Quadratic**

9. In the middle portion of the dialog box, examine the element shape options (Hex, Wedge, Tet). A brief description of the default element selection is available at the bottom of each tabbed page.
10. In the **Hex** tabbed page, accept the default **Reduced integration**.  
A description of the element type C3D20R appears at the bottom of the dialog box. Abaqus/CAE will now use C3D20R elements when we apply the mesh.
11. Click **OK** to assign the element type and to close the dialog box.

### To mesh the model:

12. 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.
13. In the **Global Seeds** dialog box, enter an approximate global size of **4** and click **OK**.  
Abaqus/CAE applies the seeds to the part, as shown in Figure W1–12.

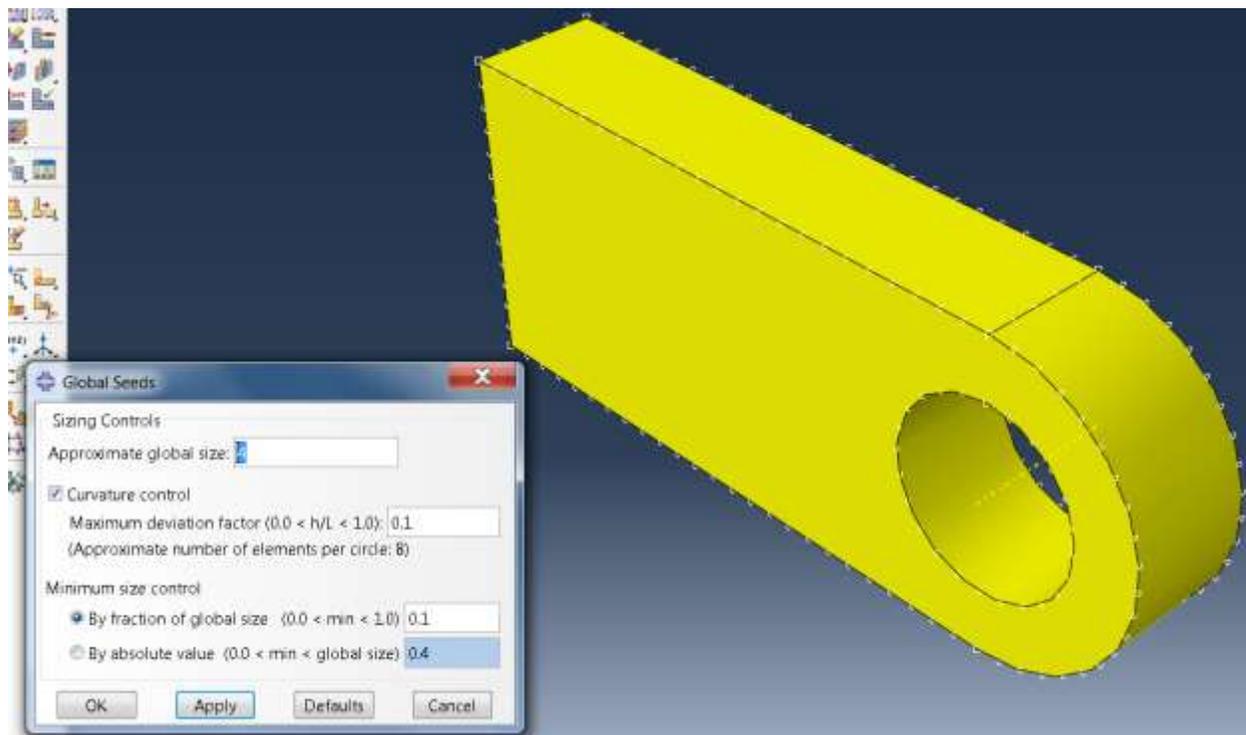


Figure W1–12 Seeded part instance.

14. From the main menu bar, select **Mesh**→**Part** to mesh the part.
15. 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.

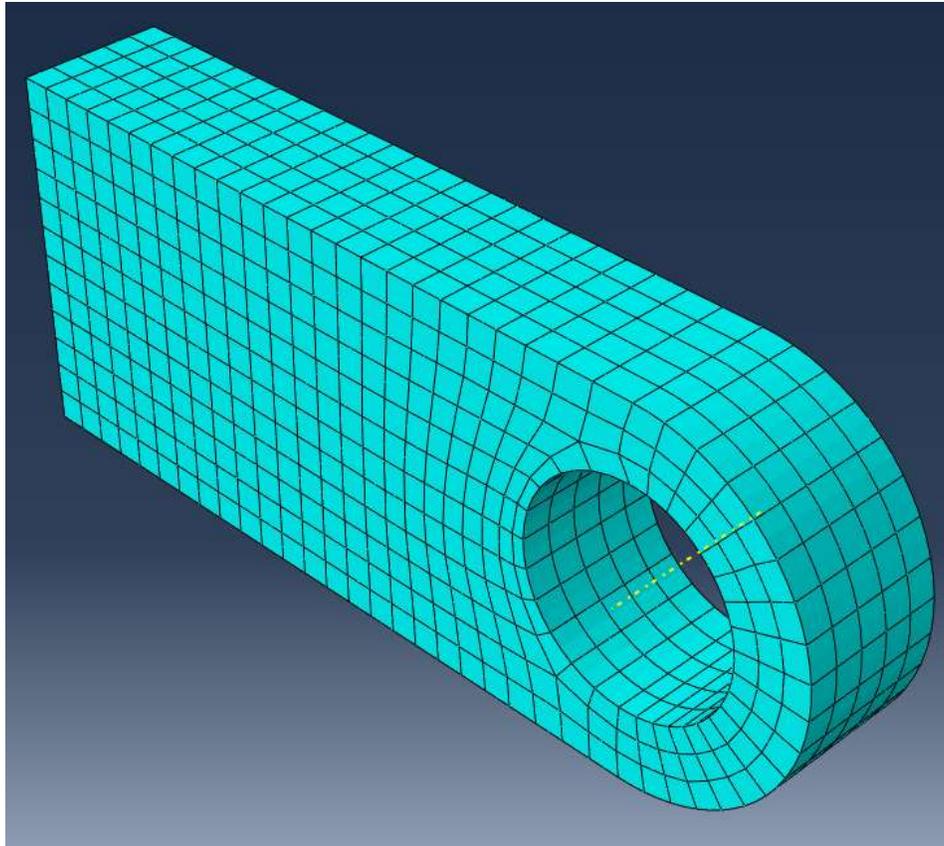


Figure W1–13 Part instance showing the resulting mesh.

## Mesh the other 2 parts

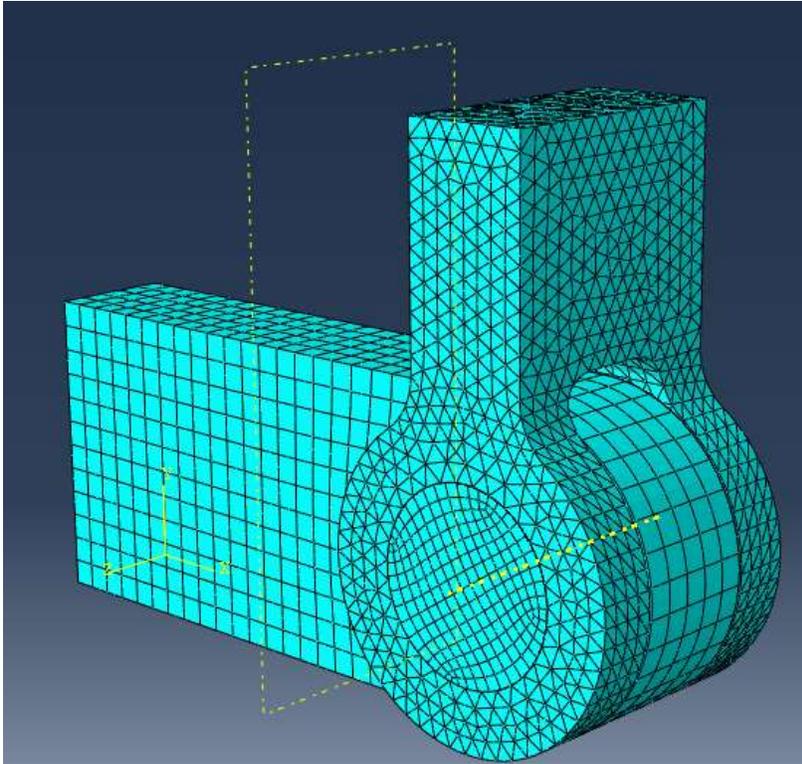
Now mesh the other 2 parts using the same procedure.

You will need to apply mesh controls to the part **pin** to define a **sweep** mesh with **medial axis**, the part **pin** should have a mesh seed of 2 mm and an element type of C3D20R.

The part **lug2** will not mesh using hex elements (without significant extra effort from the user) and so is coloured orange when you open the **mesh controls** dialogue box. So you will change the **Element Shape** to **Tet** (tetrahedral) and then accept the default settings. Note how the colour of the part **lug2** has now changed to pink to indicate that it is meshable. Apply a global mesh seed of 3 mm and continue to mesh the part **lug2** using an element type of C3D10HS (Note: this element type was called C3D10I in versions earlier than Abaqus2017 and is a **quadratic** tetrahedral element with **improved surface stress formulation**).

Now in the top menu bar click on Assembly

Module:  Model:  Object:  Assembly  Part:  and check that **Show Native Mesh**  is toggled on. Your meshed assembly now should look like this:-



Then save the Abaqus/CAE database.

## Creating and submitting an analysis job

The definition of the model **Model-1** 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 **Press** and select the model **Model-1**. 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. On the Parallelization tab check the **Use multiple processors** box and set the number of cpus according to what is available for your classroom. Click **OK** to accept the other default job settings.
5. From the menu toolbar select **Job > Manager**.  
The menu for the job manager appears.
6. From the job manager, select the job **press** then select **Submit**.  
As the job runs the status will be shown in the job manager window. Select **Monitor** to follow the status of the job as it runs. Click on each of the tabs to see what is reported during/after analysis.

press Monitor

Job: press Status: Running

Step	Increment	Att	Severe Discon Iter	Equil Iter	Total Iter	Total Time/Freq	Step Time/LPF	Time/LPF Inc

Log Errors !Warnings Output Data File Message File Status File

```

1
Abaqus 6.14-1                               Date 20-Jul-2015   Time 09:49:50
For use at Warrington_ DS Simulia UK Ltd under license from Dassault Systemes or its s
Workshop1
settle contact
STEP      1      INCREMENT      1      STEP TIME      0.00

                S T E P      1      S T A T I C      A N A L Y S I S

settle contact

AUTOMATIC TIME CONTROL WITH -
  A SUGGESTED INITIAL TIME INCREMENT OF          0.100
  AND A TOTAL TIME PERIOD OF                     1.00
  THE MINIMUM TIME INCREMENT ALLOWED IS         1.000E-05
  THE MAXIMUM TIME INCREMENT ALLOWED IS         1.00

LINEAR EQUATION SOLVER TYPE          DIRECT SPARSE

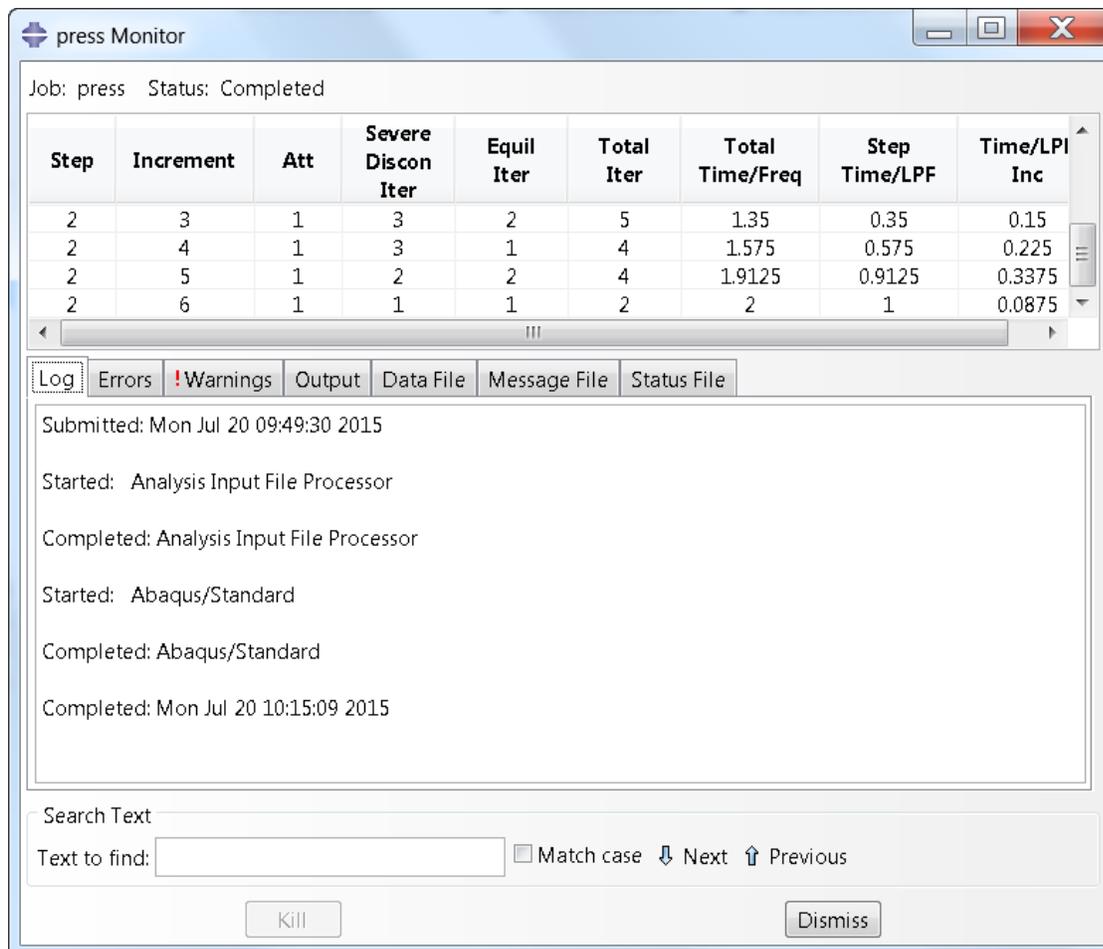
CONVERGENCE TOLERANCE PARAMETERS FOR FORCE
CRITERION FOR RESIDUAL FORCE          FOR A NONLINEAR PROBLEM          5.000E-03
CRITERION FOR DISP. CORRECTION IN A NONLINEAR PROBLEM          1.000E-02
INITIAL VALUE OF TIME AVERAGE FORCE          1.000E-02
AVERAGE FORCE          IS TIME AVERAGE FORCE
ALTERNATE CRIT. FOR RESIDUAL FORCE          FOR A NONLINEAR PROBLEM          2.000E-02
CRITERION FOR ZERO FORCE          RELATIVE TO TIME AVGC. FORCE          1.000E-05

```

Search Text

Text to find:   Match case

On successful completion of the job the log tab looks as below:-



## 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 **press** and select **Results** from the menu that appears.

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

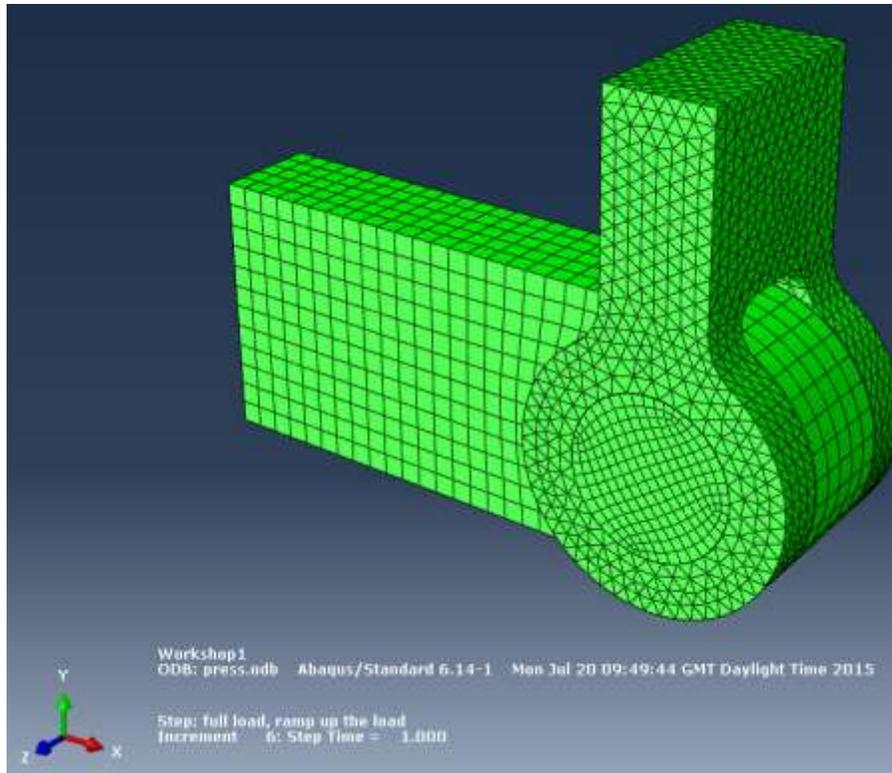


Figure W1–15 Undeformed model shape.

2. From the main menu bar, select **Plot→Deformed Shape** to view a deformed model shape plot. The deformation scale factor can be changed by selecting **Options > Common** on the main menu then specifying a Uniform or Nonuniform scale factor on the Basic tab.
3. From the main menu bar, select **Plot→Contours→on Deformed Shape** to view a contour plot of the von Mises stress, as shown in Figure W1–17.

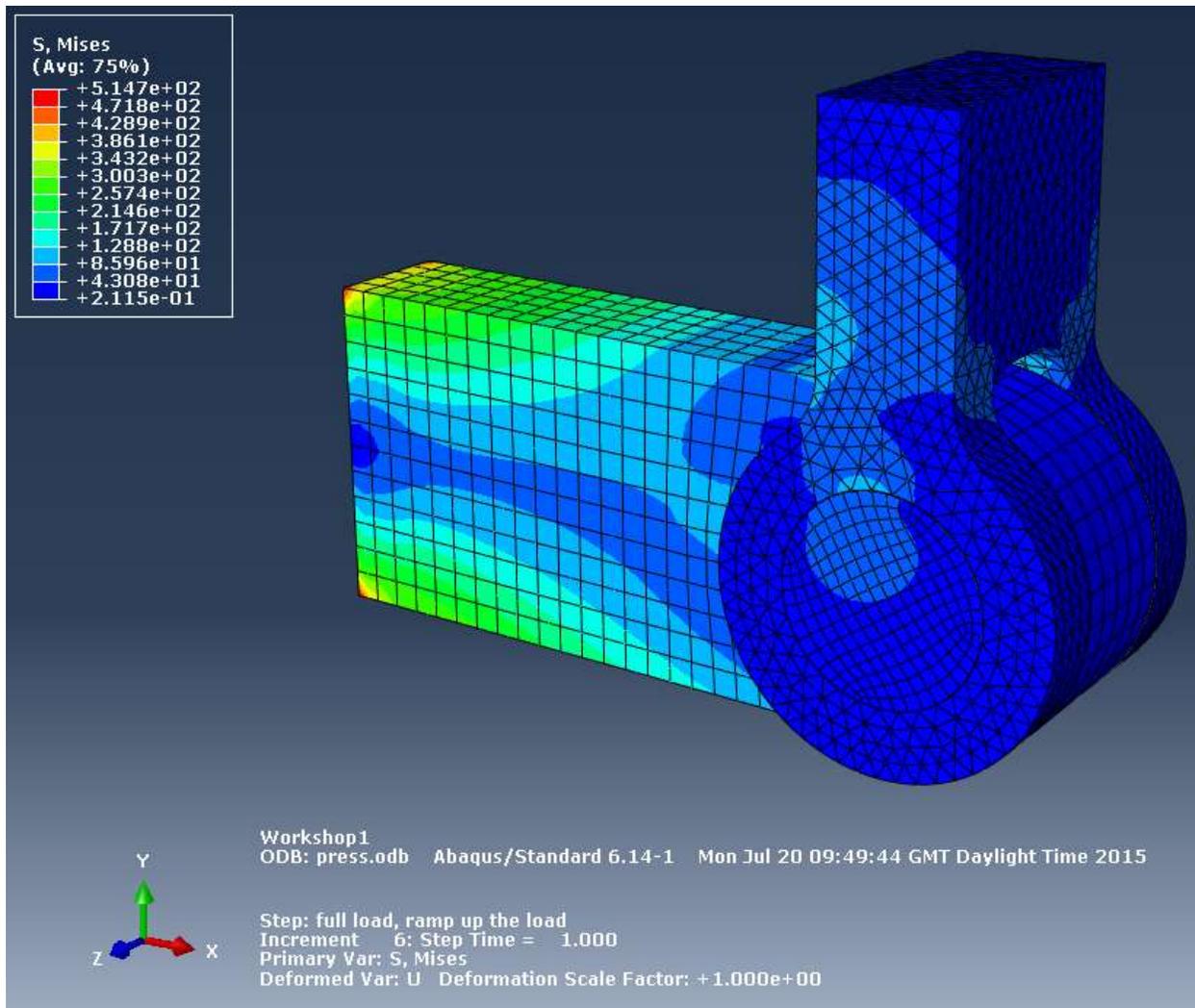


Figure W1-17 Mises stress contour plot.

4. From the main menu bar, select **File**  **Save** to save your model in a model database file.

## Export a part for use in Workshop 2

Just before we leave the session you need to export the part lug for use in the next workshop:-

1. Select the **Part** module from the dropdown
2. Select the part **lug**
3. Select **File > Export > Part** from the main menu bar
4. Name the part lug and select ACIS Version R23. Click OK

Check the Message area to check to which directory file **lug.sat** has been written.

Close the Abaqus/CAE session by selecting **File**  **Exit** from the main menu bar.