Workshop 2

Tosca – Topological Optimization

Introduction

This workshop will give you an opportunity to use Tosca optimization techniques from within the Graphical User Interface (GUI) to reduce the weight of a Part. The results will be viewed in Abaqus/CAE. You will take the approximate lug design analysed during Workshop 1 and then use topological optimization to reduce the volume (weight) whilst maximizing the stiffness.

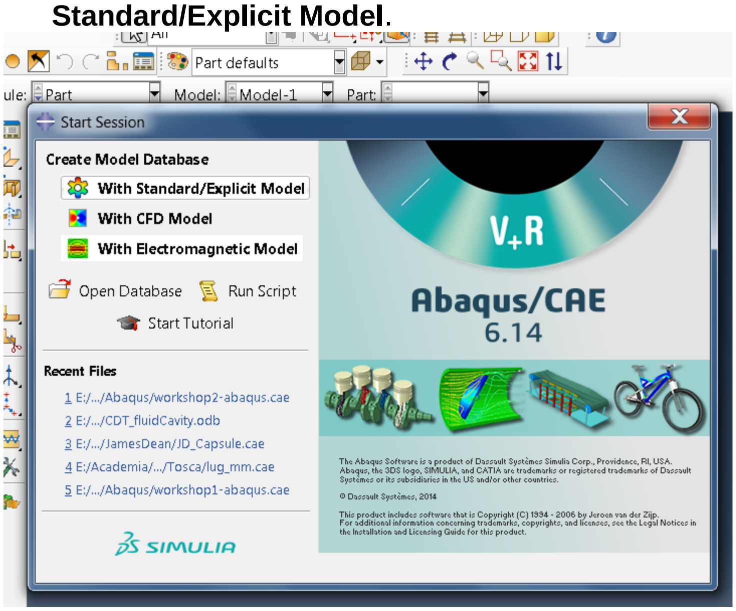
Preliminaries

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.**



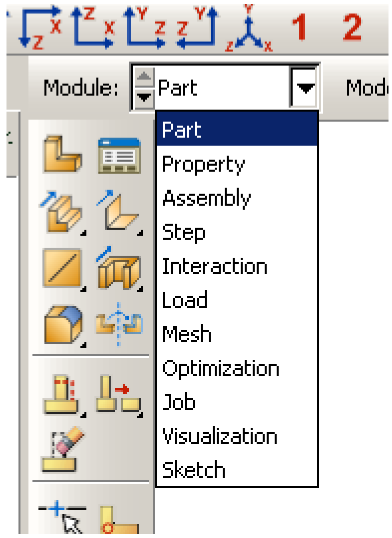
To save the model database, select **File→Save As** from the main menu bar and type the file name **LugOpt** in the **Save Model Database As** dialog box. Click **OK**.

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

Creating a part

In this section you will import the .sat file of the lug you created during the Abaqus workshop, as a three-dimensional, deformable solid body.

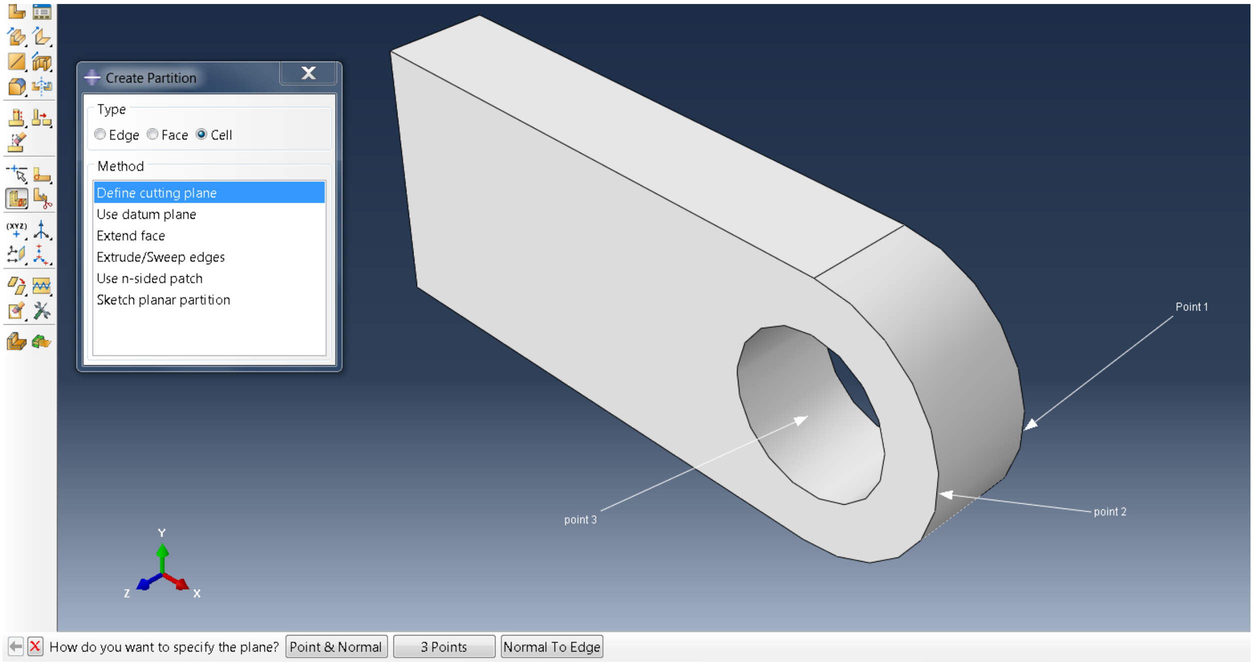
1. Abaqus/CAE automatically loads the Part module. Any other module can be accessed from the **Module** list located in the context bar.



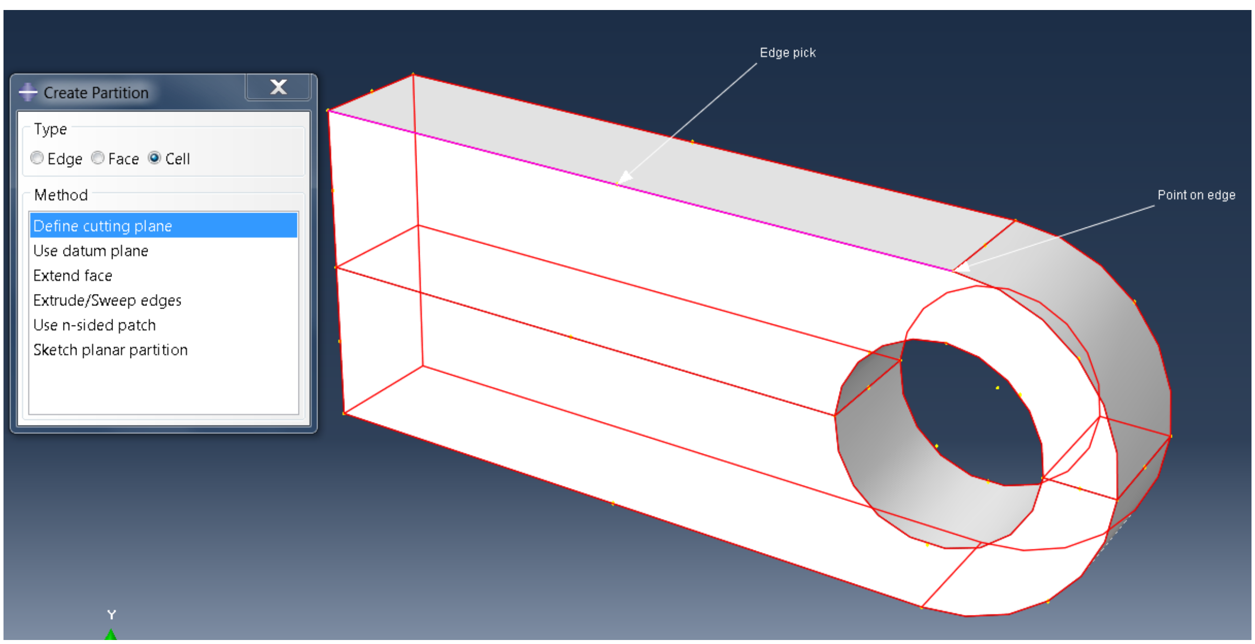
2. From the main menu bar, select **File > Import > Part** to import a new part. Navigate to the directory where you saved file 'lug.sat' in workshop 1 and select the file. Click OK to import the file
3. In the **Create Part from ACIS File** window rename the part **Part-1**, accept all other defaults and click OK to import the part into the model.

At present the part is meshable using a **sweep** technique, however, you will partition the part in order to make it meshable with a uniform mesh which is more favourable for the Optimization task.

1. First you will create a partition which will split the single cell horizontally into 2 equal cells. From the menu bar select **Tools > Partition** and then **Cell > Define cutting plane** in the create partition edit box
2. Select 3 points when asked how you want to specify the plane – then click the 3 points shown.

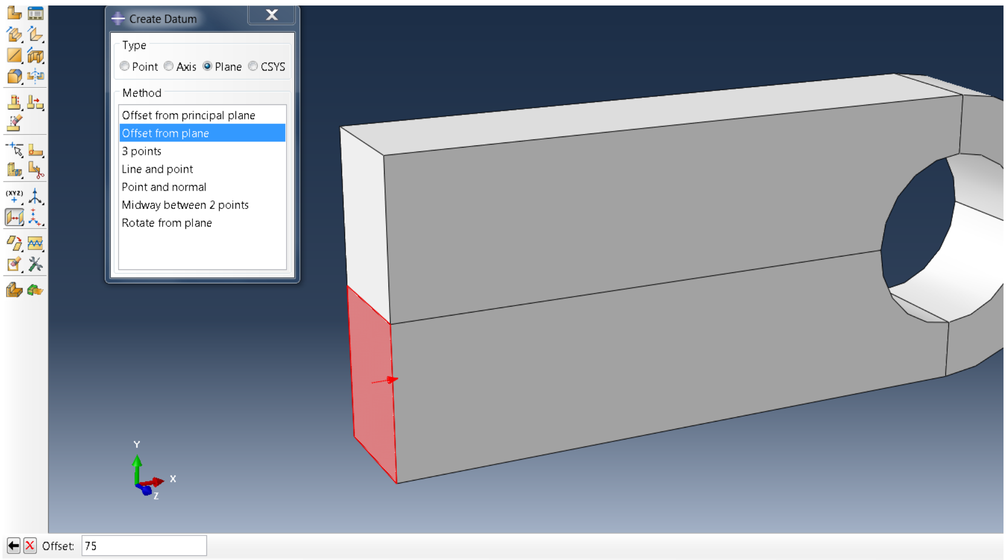


3. Click **Create partition** to split the single cell part horizontally into 2 equal cells.
4. Then to add a second vertical partition through the hole centre, select **Tools > Partition** and then **Cell > Define cutting plane** in the create partition edit box
5. Select both cells of the lug to be partitioned – using shift-click or drag pick, then click done.
6. Select **Normal to Edge** as the means to specify the plane
7. Then select the top front edge of the lug and the point on the hole centerline

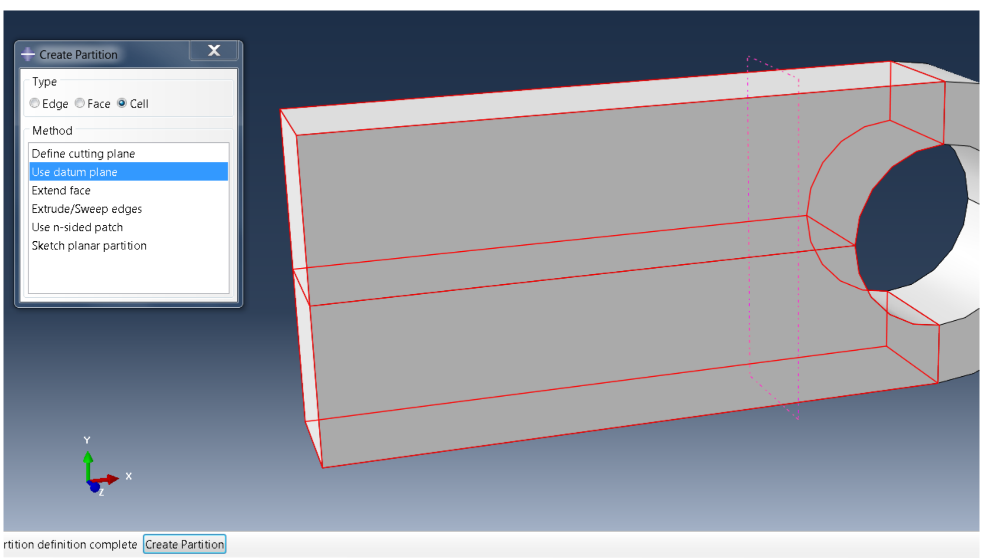


8. Click **Create Partition** and **done** to complete the action

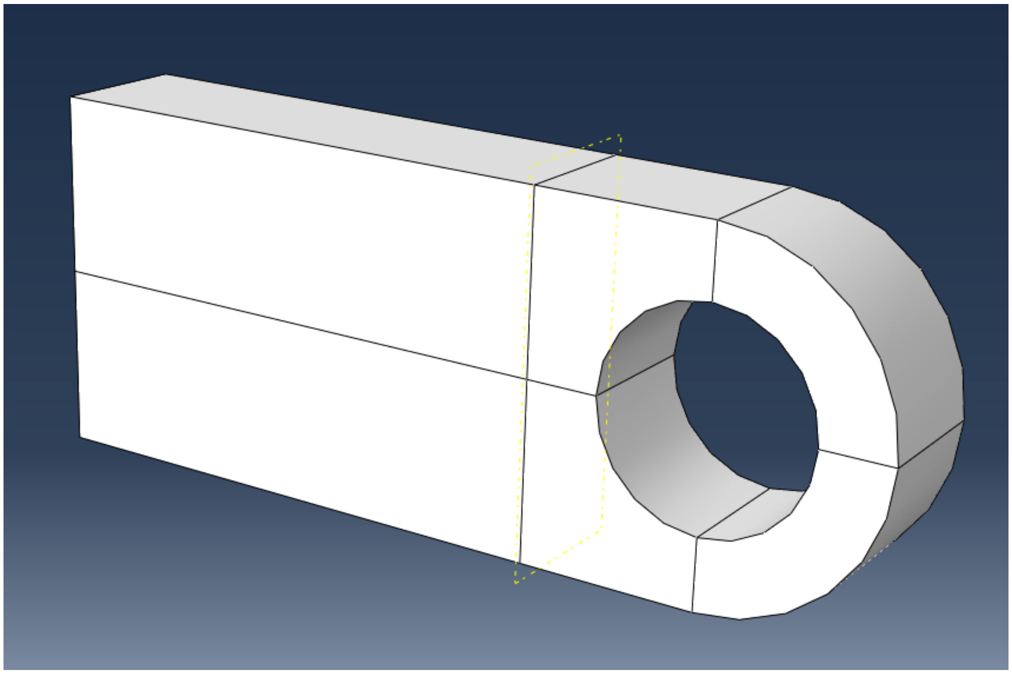
9. Finally you will add a second vertical partition offset by 75 mm from the fixed end of the lug. For this partition you will first create a datum plane to act as the partition plane – from the main menu select **Tools > Datum > Plane** and then **Offset from plane**.
10. Select one of the faces on the fixed end of the lug and **Enter Value**. You will be offsetting the end plane towards the hole of the lug so you may need to **flip** the direction arrow. Once the arrow points towards the hole click OK and enter an offset value of 75 mm.



11. Now select **Tools > Partition** and then **Cell > Use datum plane** in the create partition edit box
12. Select both end cells of the lug to be partitioned – using shift-click or drag pick, then click done.



13. Then select the datum plane you just defined. Click **Create Partition** and **done** to complete the partition.



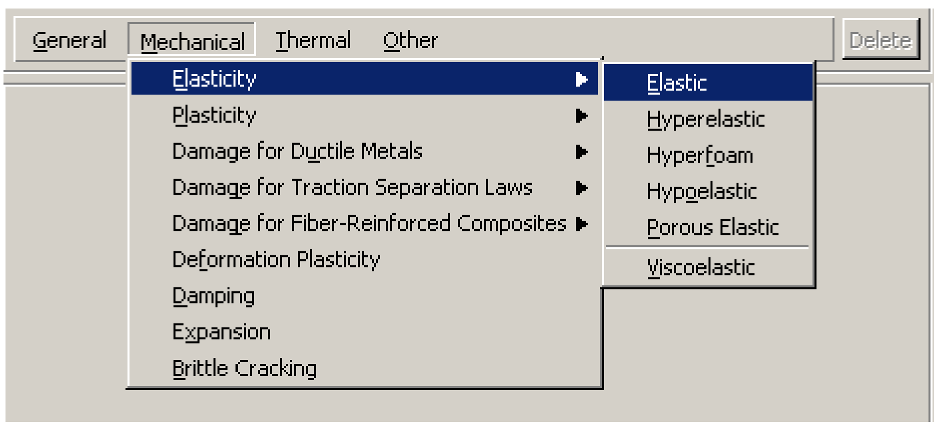
Save the Abaqus/CAE database.

Creating a material definition

You will now create a single linear elastic material with a Young's modulus of 200×10^3 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.
1. In the **Edit Material** dialog box, name the material **steel**. Notice the various options available in this dialog box.



2. From the material editor's menu bar, select **Mechanical→Elasticity→Elastic**.
   Abaqus/CAE displays the **Elastic** data form.
3. Enter a value of **200.E3** for Young's modulus and a value of **0.3** for Poisson's ratio in the respective fields. Use **[Tab]** to move between cells, or use the mouse to select a cell for data entry.
4. 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 **Model-1**.
   The **Create Section** dialog box appears.
2. In the **Create Section** dialog box:
   a. Name the section **solid_section**.
   a. Accept the default category **Solid** and the default type **Homogeneous**.
   b. Click **Continue**.

   The solid section editor appears.
3. In the **Edit Section** dialog box:
   b. Accept the default selection of **steel** for the **Material** associated with the section.
   c. 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.
   d. Click **OK**.

To assign the section definition to the cantilever beam:

1. In the Model Tree, expand the branch for the part **Part-1** (click the "+" symbol to expand the **Parts** container and then click the "+" symbol next to the part **Part-1**).
2. Double-click **Section Assignments** to assign a section to the part **Part-1**.
   Abaqus/CAE displays prompts in the prompt area to guide you through the procedure.
3. Drag pick or shift click to select all cells in 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 **solid_section** as the section definition, and click **OK**.
   Abaqus/CAE colors the lug 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 **Part-1**.

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.
   Abaqus/CAE switches to the Assembly module, and the **Create Instance** dialog box appears.
2. In the **Create Instance** dialog box, select **Part-1,** accept the Instance type by default 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 lug to a pressure load applied over one half of the hole surface. 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 lug.
- A general, static analysis step, in which you will apply a pressure load to the bottom face of the hole.

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 **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 **Step-1**.
   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 lug**.
   b. Click the **Incrementation** tab, and accept the value of **1** that appears in the **Initial** text field.
   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 Step-1.

To apply boundary conditions to one 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**.

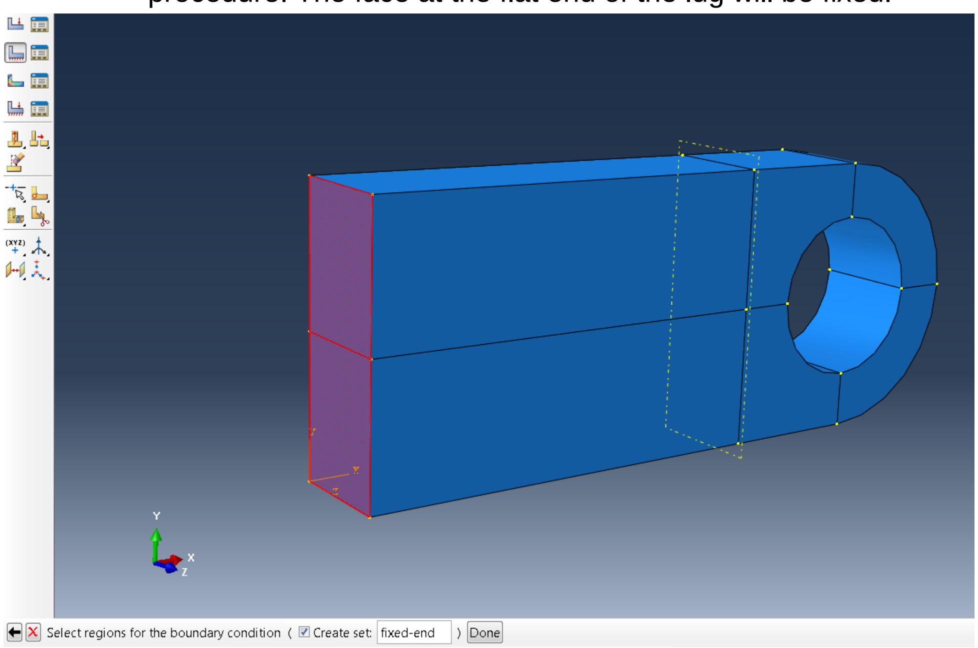
      e. Select **Initial** as the step in which the boundary condition will be activated.

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

      g. In the **Types for Selected Step** list, select **Symmetry/Antisymmetry/Encastre** as the type.

      h. Click **Continue**.

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



3. Select both faces of the rectangular end of the lug and create a set named **fixed-end** in the dialogue box. 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.
4. In the **Edit Boundary Condition** dialog box:
   a. Toggle on ENCASTRE to fully fix the end of the lug. Abaqus will simply ignore the rotational degrees of freedom BCs since they do not exist for solid continuum elements.

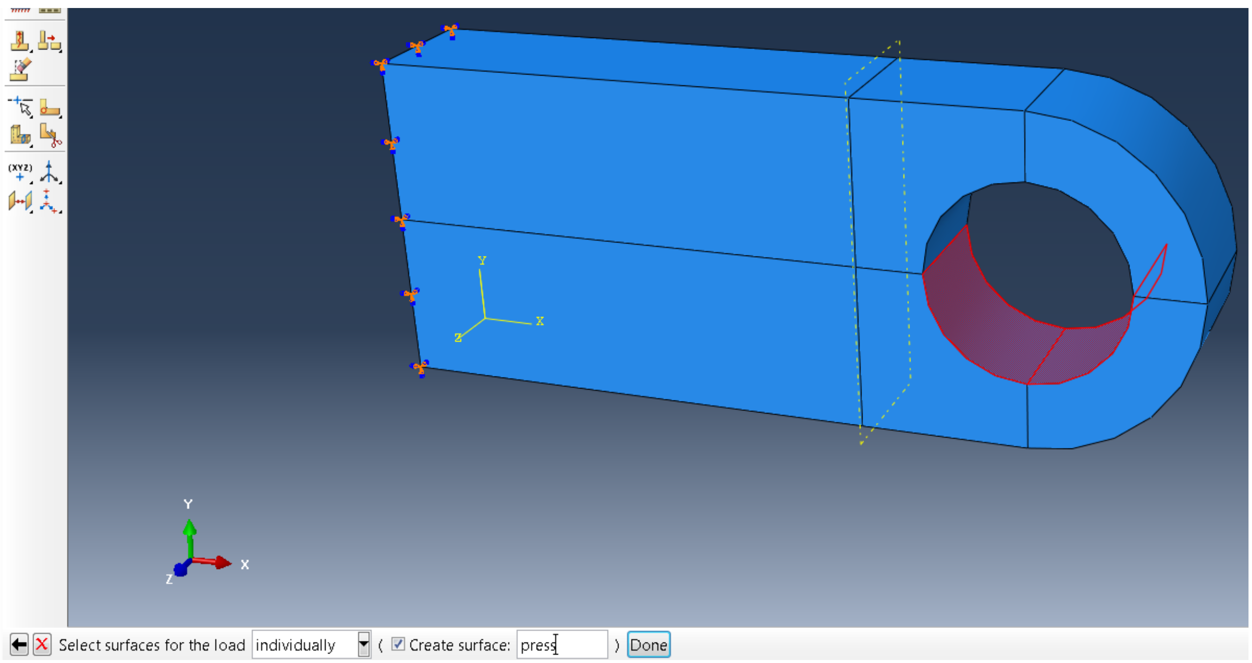
      i. Click **OK** to create the boundary condition definition and to exit the editor.

Abaqus/CAE displays arrows at each corner and midpoint on the selected face to indicate the constrained degrees of freedom.

To apply a load to the bottom face of the hole:

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 **Pressure**.
      j. Select **Step-1** as the step in which the load will be applied.
      k. In the **Category** list, accept the default category selection **Mechanical**.
      l. In the **Types for Selected Step** list, select **Pressure**.
      m. Click **Continue**.

   Abaqus/CAE displays prompts in the prompt area to guide you through the procedure.
3. In the viewport, select the two inside lower faces of the hole in the lug as the surface to which the load will be applied.



4. Name the pressure surface **press** and 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:
   c. Enter a magnitude of **50** for the load.
      n. Accept the default **Amplitude** selection (**Ramp**) and the default **Distribution** (**Uniform**).
      o. Click **OK** to create the load definition and to exit the editor.

   Abaqus/CAE displays downward–pointing arrows on the lower hole faces to indicate the load application.

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. 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 **Part-1**.
   Abaqus/CAE switches to the Mesh module and displays the part **Part-1**.
2. From the main menu bar, select **Mesh→Controls**.
3. From the prompt area select the Regions on which you wish to assign mesh controls.
4. In the **Mesh Controls** dialog box, accept **Hex** as the default **Element Shape** selection.
5. Accept **Structured** as the default **Technique** selection.
6. Click **OK** to assign the mesh controls and to close the dialog box.

To assign an Abaqus element type:

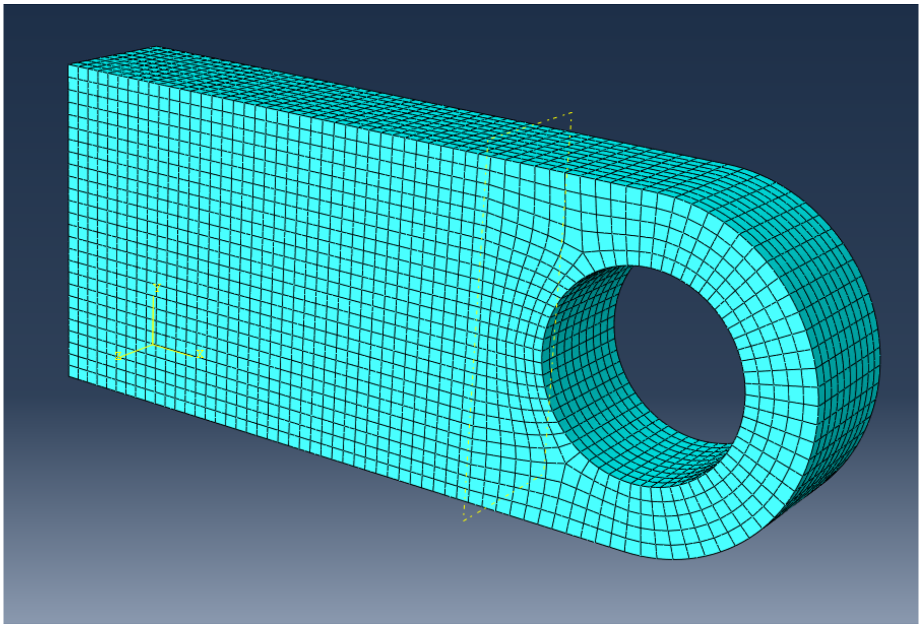
1. From the main menu bar, select **Mesh→Element Type**.
2. In the **Element Type** dialog box, accept the following default selections that control the elements that are available for selection:
   - **Standard** is the default **Element Library** selection.
   - **3D Stress** is the default **Family** of elements.

   Change the **Geometric Order** from **Linear** to **Quadratic**
3. In the middle 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, 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.
5. Click **OK** to assign the element type and to close the dialog box.

To mesh the model:

6. 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.
7. In the **Global Seeds** dialog box, enter an approximate global size of **2** and click **OK**. Abaqus/CAE applies the seeds to the part.
8. From the main menu bar, select **Mesh→Part** to mesh the part.
9. Click **Yes** in the prompt area or click mouse button 2 in the viewport to confirm that you want to mesh the part instance.
10. Abaqus/CAE meshes the part instance and displays the resulting mesh.

Save the Abaqus/CAE database.



Topology Optimization

Now you will perform a topology optimization of the lug. The aim will be to reduce the weight (volume) of the part whilst maximizing the stiffness, taking into account manufacturing limitations.

Create the Optimization Task

1. In the model tree double click **Optimization Tasks**. Abaqus/CAE will automatically switch to the Optimization Module and open the **Create Optimization Task** window. Accept the default **Topology optimization** and click **continue**.
2. Clear the **Create Set** box and click **done** to select the whole part as the optimization region.
3. On the **Advanced** tab select **Condition-based optimization**.
4. Accept all other defaults and click **OK** to complete the task.

Create the Design Responses

5. In the model tree open the **Optimization Tasks,** then **Task-1** by clicking on the + button. Then double click **Design Responses**. Abaqus/CAE will automatically open the **Create Design Response** window. Name the response **DRESP_Strain_Energy**. Accept the default **Single Term** and click **continue**.
6. Select **Whole Model** as the design response region type.
7. Select **Strain energy** as the variable, accept all other defaults and click **OK**.
8. Repeat these operations to add a second design response named DRESP_Volume choosing volume as the variable.

Create the Objective Function

9. In the model tree open the **Optimization Tasks,** then **Task-1** by clicking on the + button. Then double click **Objective Functions**. Abaqus/CAE will automatically open the **Create Objective Function** window. Name the response **obj_max_stiffness** and click **continue**.
10. Accept the **Minimize design response values** target.

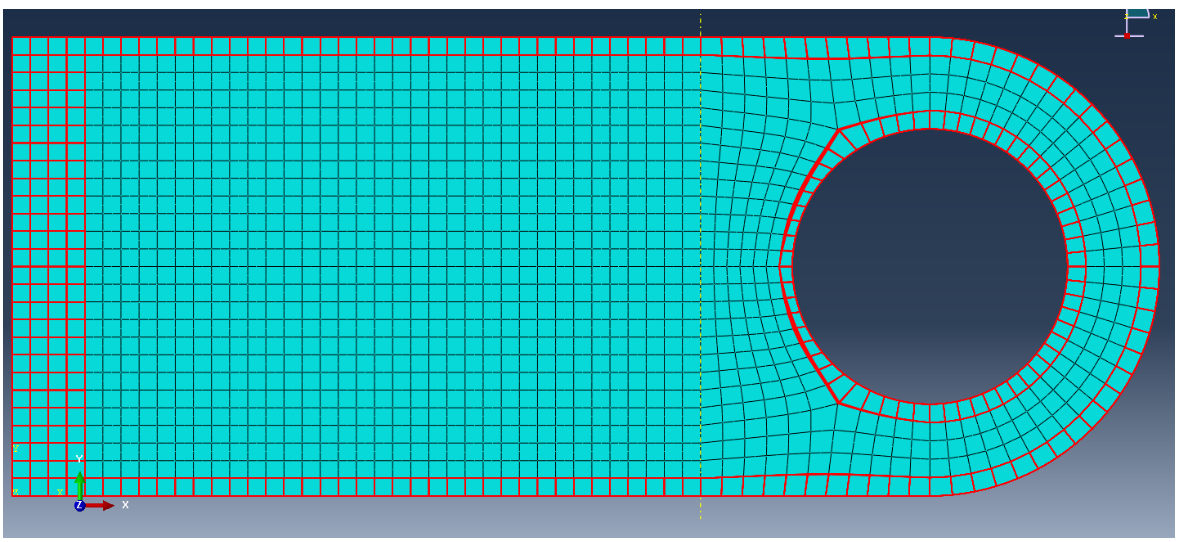
11. Click in the Name box and select the design response DRESP_Strain_Energy.
12. Accept all other defaults and click **OK**.

Create the optimization Constraints

13. In the model tree open the **Optimization Tasks,** then **Task-1** by clicking on the + button. Then double click **Constraints**. Abaqus/CAE will automatically open the **Create Constraint** window. Name the constraint **constraint_volume** and click **continue**.
14. Use the dropdown to populate the design response name box with **DRESP_Volume**.
15. Change the constraint response to **A fraction of the initial value** and set this to 0.3.
16. Click **OK** to complete the constraint.

Create the Geometric Restrictions

17. First you need to create an element set to apply the restriction to. Select from the views toolbar and then remove the perspective by clicking .
18. From the main menu select **Tools > Set > Create**. Name the set **frozen_elems** and set the type to **Element**. Click **continue**, Abaqus/CAE will automatically populate the screen with elements. Use shift click to select all perimeter elements, 4 deep at the fixed end, to add to the set. Ctrl click will remove elements selected in error. Note you can also try **select by topology** rather than **Individually**.



19. In the model tree open the **Optimization Tasks,** then **Task-1** by clicking on the + button. Then double click **Geometric Restrictions**. Abaqus/CAE will automatically open the **Create Geometric Restriction** window. Select Type **Frozen area** and click **continue**.
20. Select the **Sets** button in the bottom right of the graphics pane, then select **frozen_elems** and **continue**.
21. Click **OK** to complete the first restriction.
22. Create a second Geometric Restriction of Type **Demold control (Topology)**, select **done** to include the whole model. Accept the default **Demolding with a central plane** and for the **Pull Direction** select the vector definition and use the button to define a vector with start point (0,0,0) and end point (0,0,1). Click **OK** to complete the Geometric Restriction.

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, and then an Optimization job to perform the optimization.

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 **lug-analysis** 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 2**.
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. From the menu toolbar select **Job > Manager**.
   The menu for the job manager appears.
6. From the job manager, select the job **lug-analysis** then select **Submit**.
7. 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.

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 **lug-analysis** and select **Results** from the menu that appears. Abaqus/CAE switches to the Visualization module, opens the output database created by the job (**lug-analysis.odb**), and displays the undeformed shape of the model.
2. Click on the **Plot Contours on Deformed Shape** button  to see a contour plot of the field variables.
3. Select **Options > Common** from the main menu to change the style of the plot and/or the deformation scale factor of the plot.

Creating and submitting an Optimization job

Next, you will create and submit an Optimization job to perform the optimization.

To create and submit an analysis job:

1. In the Model Tree, double-click **Optimization Processes** to create a new optimization job.

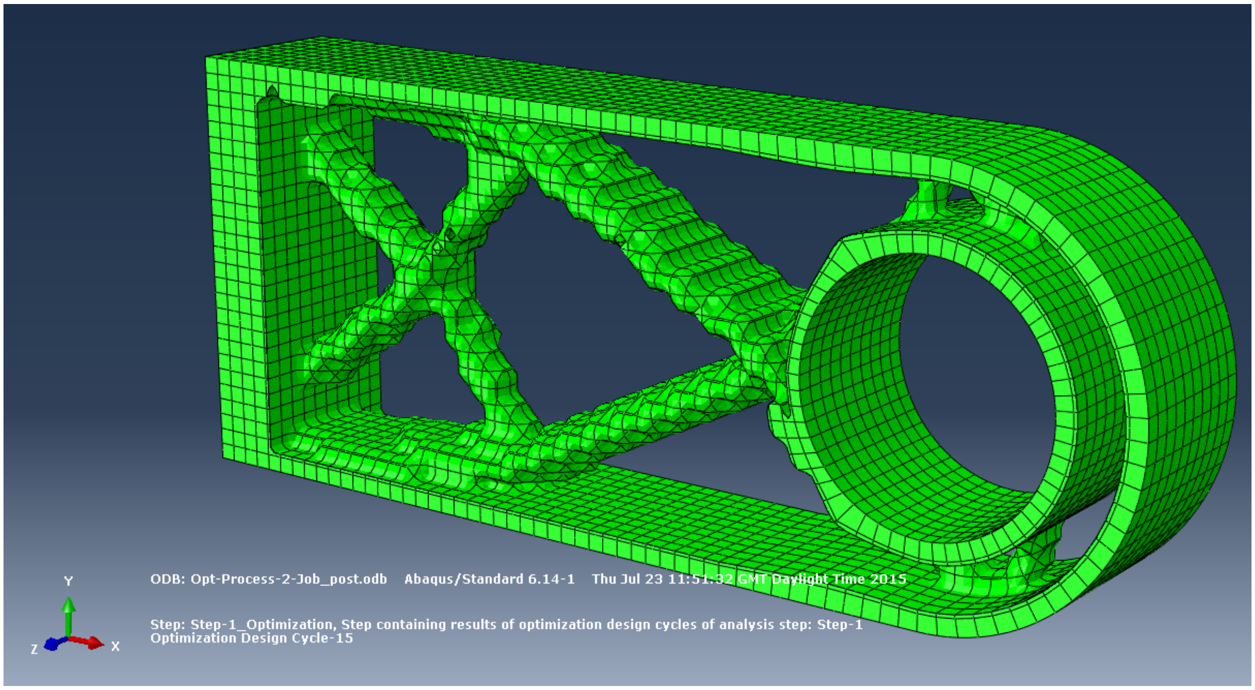
Abaqus/CAE switches to the Job module, and the **Edit Optimization Process** dialog box appears.

2. In the dialog box, name the job **optimize_lug_mm** and select the model **Model-1** and the Task **Task-1**.
3. In the **Description** field of the **Edit Job** dialog box, enter **Workshop 2 – topological optimization**.
4. Click the **Parallelization** tab to use multiple processors and set the number to 4. Click OK to complete the process definition.
5. From the menu toolbar select **Optimization > Submit > optimize_lug_mm**.
   The optimization process begins.
6. From the main menu select **Optimization > Monitor > optimize_lug_mm**. to review progress with the optimization process. As the job runs the status will be shown in the monitor window.

Viewing the optimization results

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

1. In the Model Tree, click mouse button 3 on the optimization process **optimize_lug_mm** and select **Combine** from the menu that appears to generate a unique results file (.odb). Abaqus/CAE opens the **Combine Optimization Results** window. Note the location of the results directory and note and accept all other default settings. Click **submit** to combine the results of all the individual optimization analyses. Then **Close** the window.
2. In the Model Tree, click mouse button 3 on the optimization process **optimize_lug_mm** and select **Results** from the menu that appears. Abaqus/CAE switches to the Visualization module and displays the optimized topology of the lug



These raw results from the topology optimization process are generally not useable – if you contour stresses you will notice a marked increase in von Mises stress over the original Part.

So you will extract a smoothed surface from this optimization to work with in a new Abaqus/CAE database.

1. Switch back to the Optimization module and expand the **Optimization Processes** on the model tree. Click mouse button 3 on the Optimization Process **optimize_lug_mm** and select **Extract.**
2. Abaqus/CAE displays the **Extract Surface Mesh Options** dialogue box. Name the output **optimize_lug_mm_surface**; select **Abaqus input file** for format and use the output from the last design cycle (15). Change the **Iso value (0-1)** to 0.1. Then click **Extract** to generate the input deck (optimize_lug_mm_surface.inp) of the surface.