Workshop 3

Tosca – Shape Optimization

Introduction

This workshop will give you an opportunity to use further optimization techniques from within the Graphical User Interface (GUI) to reduce the peak stresses within a Part. The results will be viewed in Abaqus/CAE.

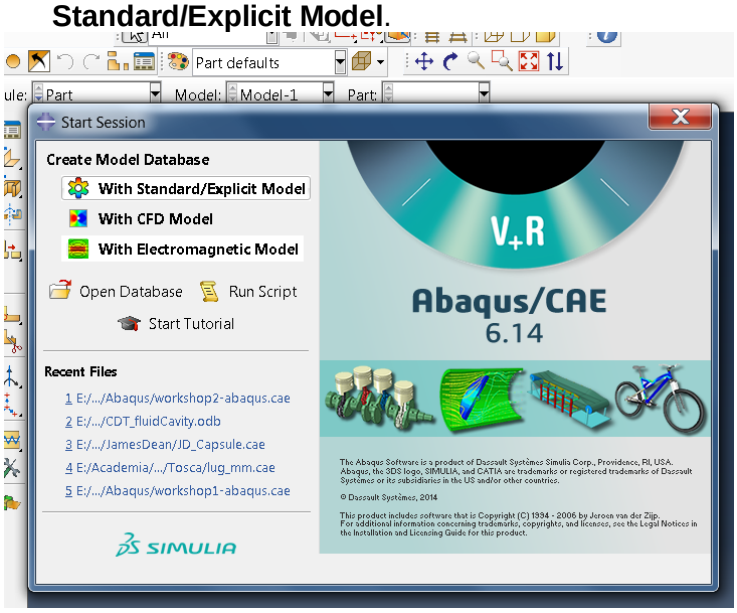
Preliminaries

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**.



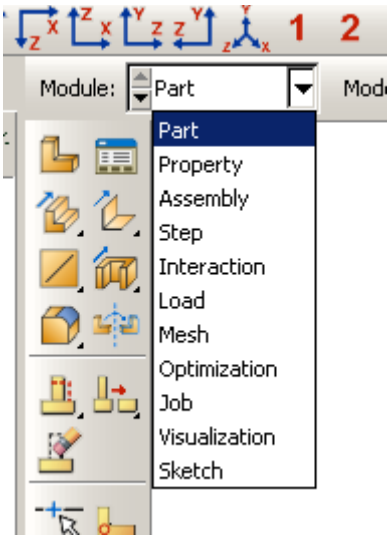
To save the model database, select **File→Save As** from the main menu bar and type the file name **LugOptShape** in the **Save Model Database As** dialog box. Click **OK**.

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

Creating a part

In this section you will import the .inp file of the lug you created during the Tosca Topology 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 > Model** to import a new model. Change the **File Filter** dropdown to **Abaqus Input File (*.inp,*.pes)**. Navigate to the directory where you saved file 'optimize_lug_mm_surface.inp' in workshop 2 and select the file. Click OK to import the file.
3. Abaqus/CAE automatically loads the new model (optimize_lug_mm_surface) into the Assembly module.

At present the part is meshed with triangular shell elements. Navigate to the Mesh module and use the query tool to query any **Element** in the model. The Element type and connectivity is displayed in the message area at the bottom of the GUI. You will now convert this surface triangular mesh to a continuum solid tetrahedral mesh.

1. First select the **Edit Mesh** icon change the category to **Mesh** and select **Convert tri to tet**. Abaqus/CAE knows that there is only one volume which is completely enclosed with shells and so simply asks if it is OK to mesh the part with tets. Answer yes and then use the query tool again to query one of the elements. This will confirm that the surface tri mesh has been converted to linear tetrahedral elements. In Abaqus this type of element (C3D4) should not be used in regions of concern, and so you will now convert them to higher order (quadrilateral) tetrahedral elements.
2. From the main menu select **Mesh > Element Type** and drag pick the entire structure for the region to be assigned element types. Click mouse button 2 or **done** to complete the pick and then select **Quadratic** on the Element Type dialogue box. Click **OK** to complete the operation. Check the new element type using the query tool.

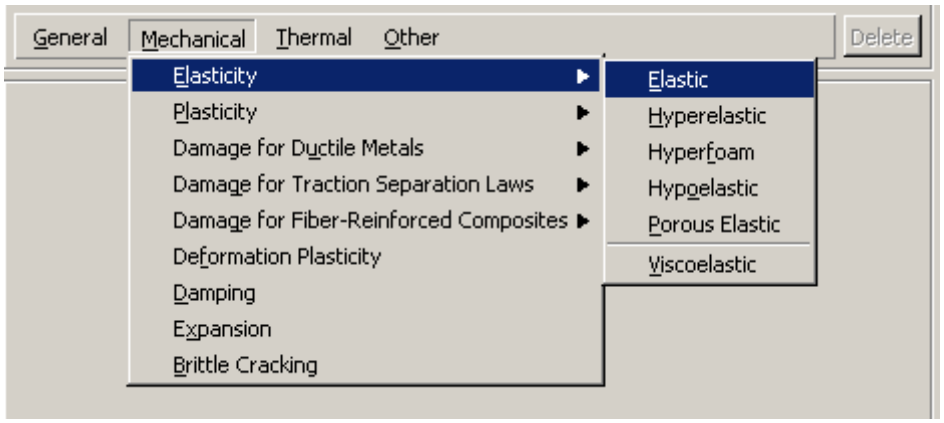
Save the Abaqus/CAE database.

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 **optimize_lug_mm_surface**.
   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.



3. From the material editor's menu bar, select **Mechanical→Elasticity→Elastic**.
   Abaqus/CAE displays the **Elastic** data form.
4. 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.
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 optimized lug part. 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 **optimize_lug_mm_surface**.
   The **Create Section** dialog box appears.
2. In the **Create Section** dialog box:
   a. Name the section **solid_section**.

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:
   d. Accept the default selection of **steel** for the **Material** associated with the section.
   e. 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.
   f. Click **OK**.

To assign the section definition to the optimized lug model:

1. In the Model Tree, expand the branch under Model **optimize_lug_mm_surface** 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 to select all elements 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 elements.

   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.

Save the Abaqus/CAE database.

Assembling the model

The assembly for this analysis was done during the import of the model.

Configuring the analysis

In this simulation we are interested in the static response of the optimized lug to the same pressure load (50 N/mm^2) 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 **optimize_lug_mm_surface**.
   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**.

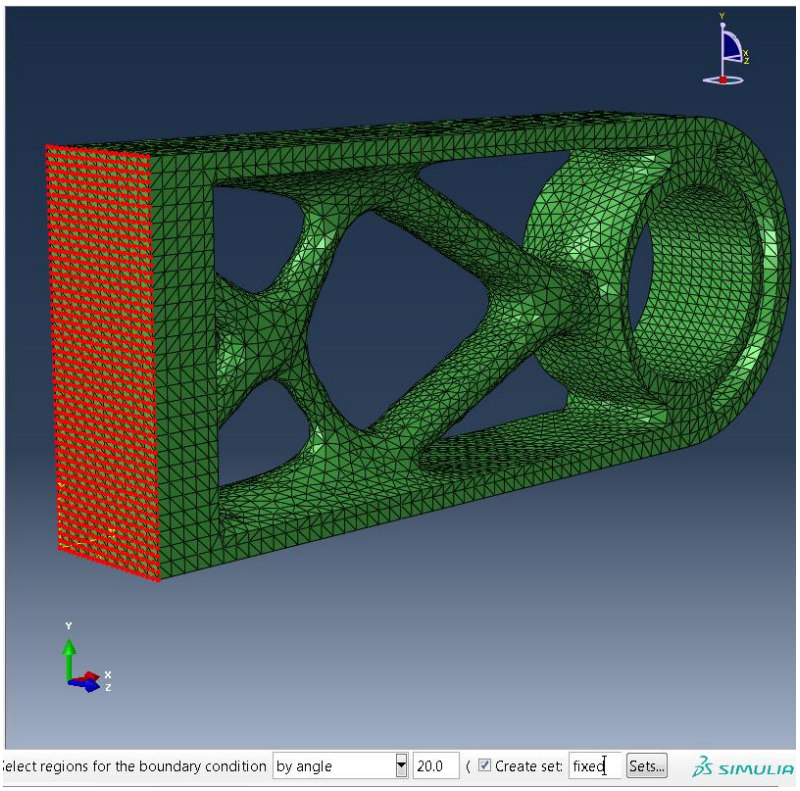
   The step editor appears.
3. In the **Edit Step** dialog box:
   a. In the **Description** field of the **Basic** tabbed page, enter **Load the new design**.
   b. Click the **Incrementation** tab, and accept the value of **1** that appears in the **Initial** text field.
   c. Click the **Other** tab 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 the rectangular end of the lug:

1. In the Model Tree, double-click **BCs** to create a new boundary condition in the model **optimize_lug_mm_surface**.
   Abaqus/CAE switches to the Load module, and the **Create Boundary Condition** dialog box appears.
2. In the **Create Boundary Condition** dialog box:
   g. Name the boundary condition **Fixed**.
   h. Select **Initial** as the step in which the boundary condition will be activated.
   i. In the **Category** list, accept the default category selection **Mechanical**.
   j. In the **Types for Selected Step** list, select **Symmetry/Antisymmetry/Encastre** as the type.
   k. Click **Continue**.
      Abaqus/CAE displays prompts in the prompt area to guide you through the procedure. The face at the rectangular end of the lug will be fixed, as before.



3. Abaqus/CAE knows that there is no geometry in the model – just nodes and elements. In the dropdown box change the **Select regions** parameter from **individually** to **by angle** and accept the default angle between adjacent faces of 20 degrees. Pick any node on the end face and the whole face will be selected as shown. 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.
   b. Click **OK** to create the boundary condition definition and to exit the editor.

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

Save the Abaqus/CAE database.

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

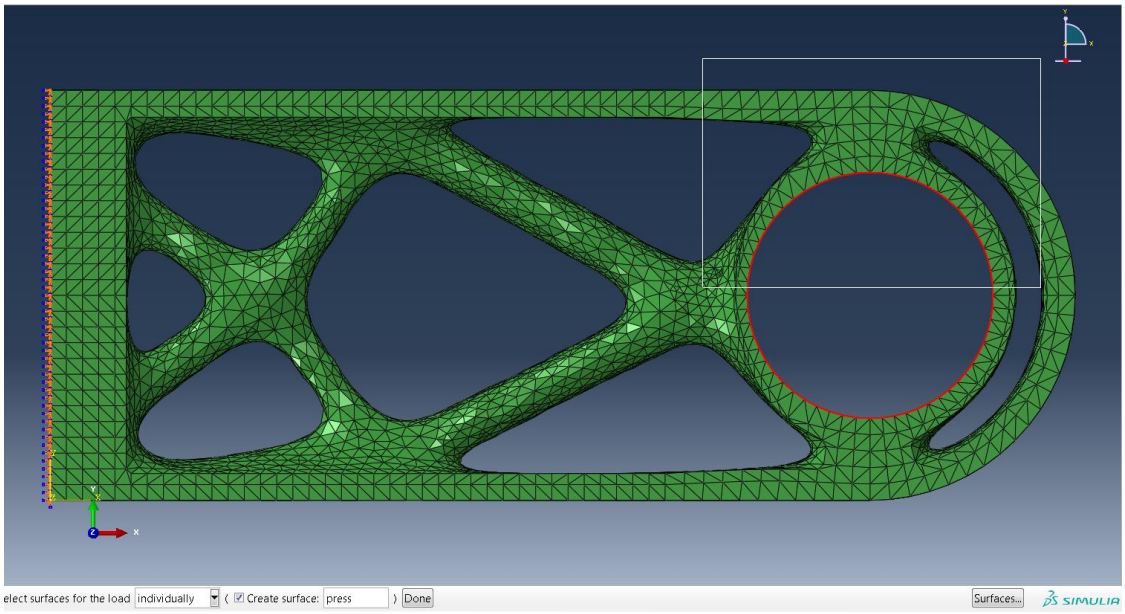
1. In the Model Tree, double-click **Loads** to create a new load in the model **optimize_lug_mm_surface**.

   The **Create Load** dialog box appears.
2. In the **Create Load** dialog box:
   c. Name the load **Pressure**.
   d. Select **Step-1** as the step in which the load will be applied.
   e. In the **Category** list, accept the default category selection **Mechanical**.

f. In the **Types for Selected Step** list, select **Pressure**.
g. Click **Continue**.

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

3. In the prompt area, select by angle to pick element faces. Pick any element in the hole and the entire inside surface of the hole will be picked. Then use the button on the view menu to change the view. Turn off the view perspective and then use 'ctrl' drag pick to remove the element faces around the top half of the hole. Note that you will have to change by angle back to individually in the prompt area to perform the drag pick operation.



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:
   h. Enter a magnitude of **50** for the load.
   i. Accept the default **Amplitude** selection (**Ramp**) and the default **Distribution** (**Uniform**).
   j. Click **OK** to create the load definition and to exit the editor.

   Abaqus/CAE displays downward–pointing arrows along the element faces in the hole to indicate the load is applied onto the element face. NB a negative pressure would be applied away from the element face – as a traction.

Save the Abaqus/CAE database.

Creating and submitting an analysis job

The definition of the model **optimize_lug_mm_surface** 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.

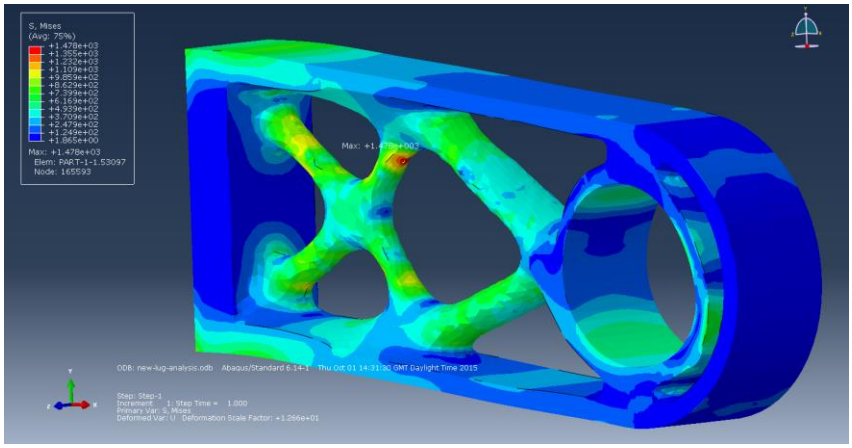
To create and submit an analysis job:

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 **new -lug-analysis** and select the model **optimize_lug_mm_surface**. Click **Continue**.
   The job editor appears.
3. In the **Description** field of the **Edit Job** dialog box, enter **Workshop 3**.
4. Click the **Parallelization** tab and check the box to **Use multiple processors**. Use the up arrow to set the number of processors to **4**.
5. 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.
6. From the menu toolbar select **Job > Manager**.
   The menu for the job manager appears.
7. From the job manager, select the job **new -lug-analysis** then select **Submit**.
8. 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 **new-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 (**new-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. Check the button **Free edges** to turn off the element edges and make the plot a little clearer. Note the deformation scale factor has been automatically set.

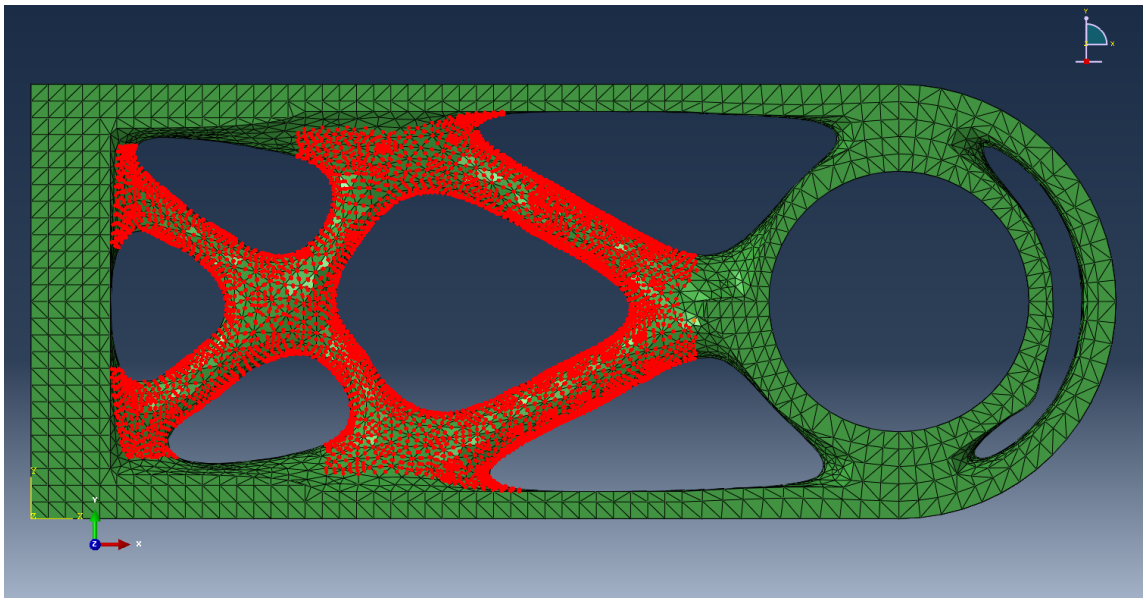


Shape Optimization

Now you will perform a shape optimization of the lug. The aim will be to reduce the local stresses in the cross members. Switch back to the Optimization module.

Create the Set containing the region to be optimized

1. From the main menu select Tools > Set > Create. Name the set **design_nodes** and check the type is set to Node. Click **continue**.
2. Use the **by angle** and **individually** settings and **Shift click** to add and **Ctrl Click** to remove to select the nodes on the cross members and knuckle joint as shown. Remove all nodes that sit on the angle between the sides and the inside surface. Click **done** to create the set.



Create the Optimization Task

3. In the model tree under the model **optimize_lug_mm_surface** double click **Optimization Tasks**. Abaqus/CAE will automatically open the **Create Optimization Task** window. Select **Shape optimization** and click **continue**.
4. Select the set **design_nodes** for the optimization region and click continue.
5. Check the box to **Freeze boundary condition regions**.
6. Under **Mesh Smoothing Region** select **Specify first layer: (Not Picked)** and set the number of layers to smooth to **1**. Then use the button to specify the set '**design_nodes**' for the first layer.
7. Specify the Number of node layers adjoining the task region to remain free to **1**.
8. Switch to the **Mesh Smoothing Quality** tab and change the target mesh quality to **Medium**.

Create the Design Responses

9. 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_VM_stress**. Accept the default **Single Term** and click **continue**.

10. Select **Point (nodes)** as the design response region type – and the set **design_nodes.**
11. Select **Stress** as the variable, and **Mises hypothesis**. Change the Operator on values in region to **Maximum**.
12. Switch to the **Steps** tab and **Specify** the **Source of values**. Click the + button to add a model and choose **optimize_lug_mm_surface**. Click in the Step and Load Case box to add **Step-1** from the dropdown
13. .Set the Operator on values across steps and load cases to **Maximum value**. Then Click **OK** to complete the response.
14. Create a second Design response called **DRESP_Volume**. Select the design response region type to **Body (elements)** – then select in viewport all elements in the model.
15. Choose **Volume** as the variable and click **OK** to complete the definition.

Create the Objective Function

16. 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 **minimize_stress** and click **continue**.
17. Set the target response to **Minimize the maximum design response values**.
18. Click in the Name box and select the design response **DRESP_VM_stress**.
19. Accept all other defaults and click **OK**.

Create the optimization Constraints

20. 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 **volume_constraint** and click **continue**.
21. Use the dropdown to populate the design response name box with **DRESP_Volume**.
22. Change the constraint response to **A fraction of the initial value** and set this to 1.
23. Click **OK** to complete the constraint.

Create the Geometric Restrictions

24. 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 **Growth** and click **continue**.
25. Select the **Sets** button in the bottom right of the graphics pane, then select **design_nodes** and **continue**.
26. For the Optimization Displacement, set both **Maximum in shrink direction** and **Maximum in growth direction** to **1**
27. Click **OK** to complete the restriction.

Save the Abaqus/CAE database.

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. Switch to the Job module and from the main menu select **Optimization > Create**.

2. In the dialog box, name the job **ShapeOpt** and select the model **optimize_lug_mm_surface** and the Task **Task-1**.
3. In the **Description** field of the **Edit Job** dialog box, enter **Workshop 3 – shape 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 > ShapeOpt**.

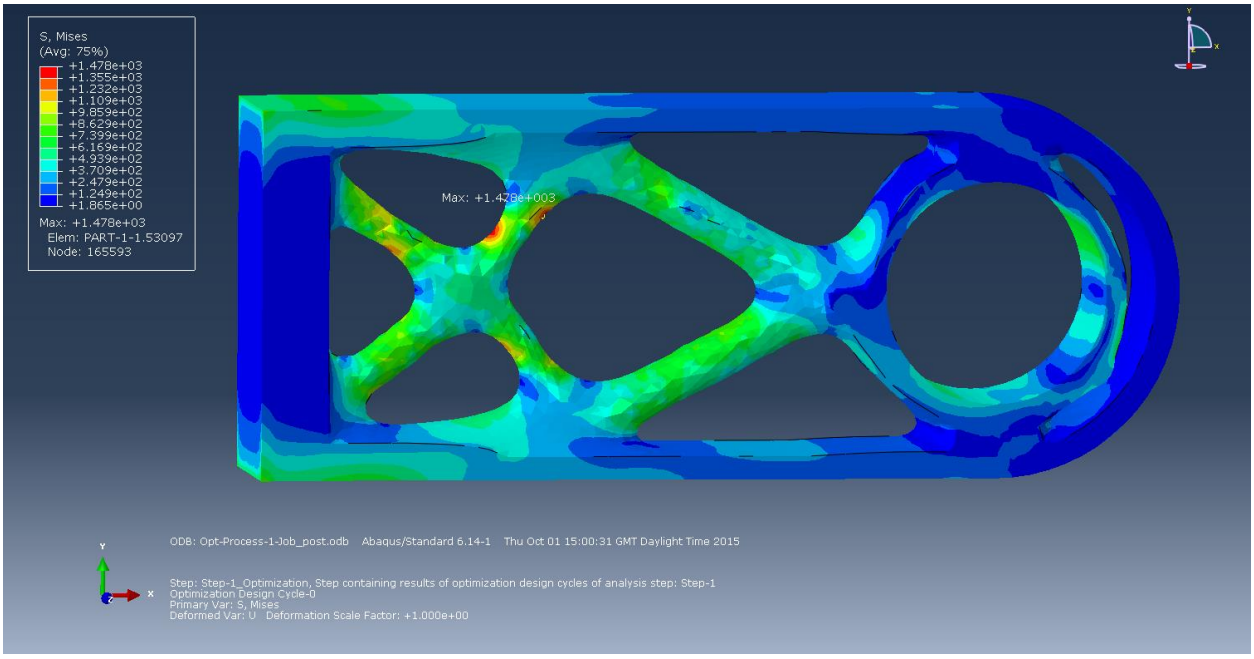
   The optimization process begins.
6. From the main menu select **Optimization > Monitor > ShapeOpt**. to review progress with the optimization process. As the job runs the status will be shown in the monitor window.

Viewing the optimization results

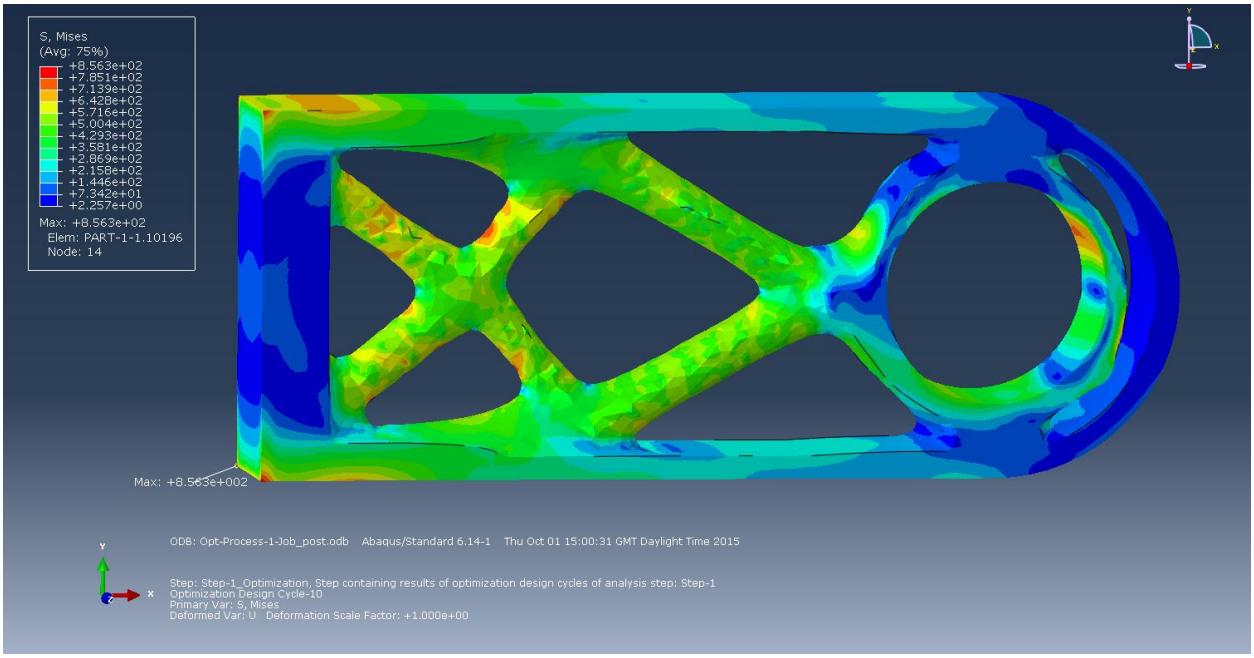
You are now ready to view the results of the optimization in the Visualization module.

1. In the Model Tree, click mouse button 3 on the optimization process **ShapeOpt** and select **Combine** from the menu that appears. 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 **ShapeOpt** and select **Results** from the menu that appears. Abaqus/CAE switches to the Visualization module and displays the optimized topology of the lug

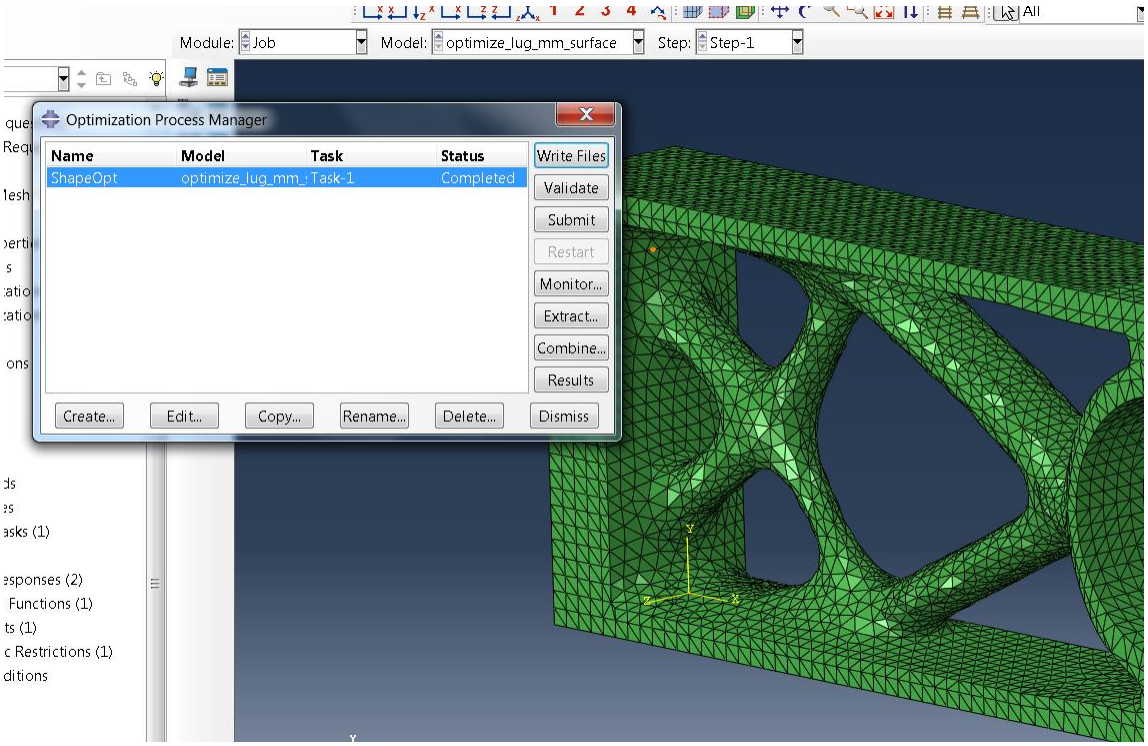
Toggle between Design Cycle 0



and Design Cycle 10 to review the increase in thickness of the cross members with the subsequent decrease in vonMises stress.



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



1. Switch back to the Job module and select **Optimization > Manager** from the main menu. Highlight the **ShapeOpt** process and select **Extract.**
2. Abaqus/CAE displays the **Extract Surface Mesh Options** dialogue box. Name the output **ShapeOpt_surface**; select **Abaqus input file** for format and use the output from the last design cycle (10). Then click **Extract** to generate the input deck

(**ShapeOpt_surface.inp**) of the surface. Note the location of the file in the message area.

You have now completed Workshop 3.