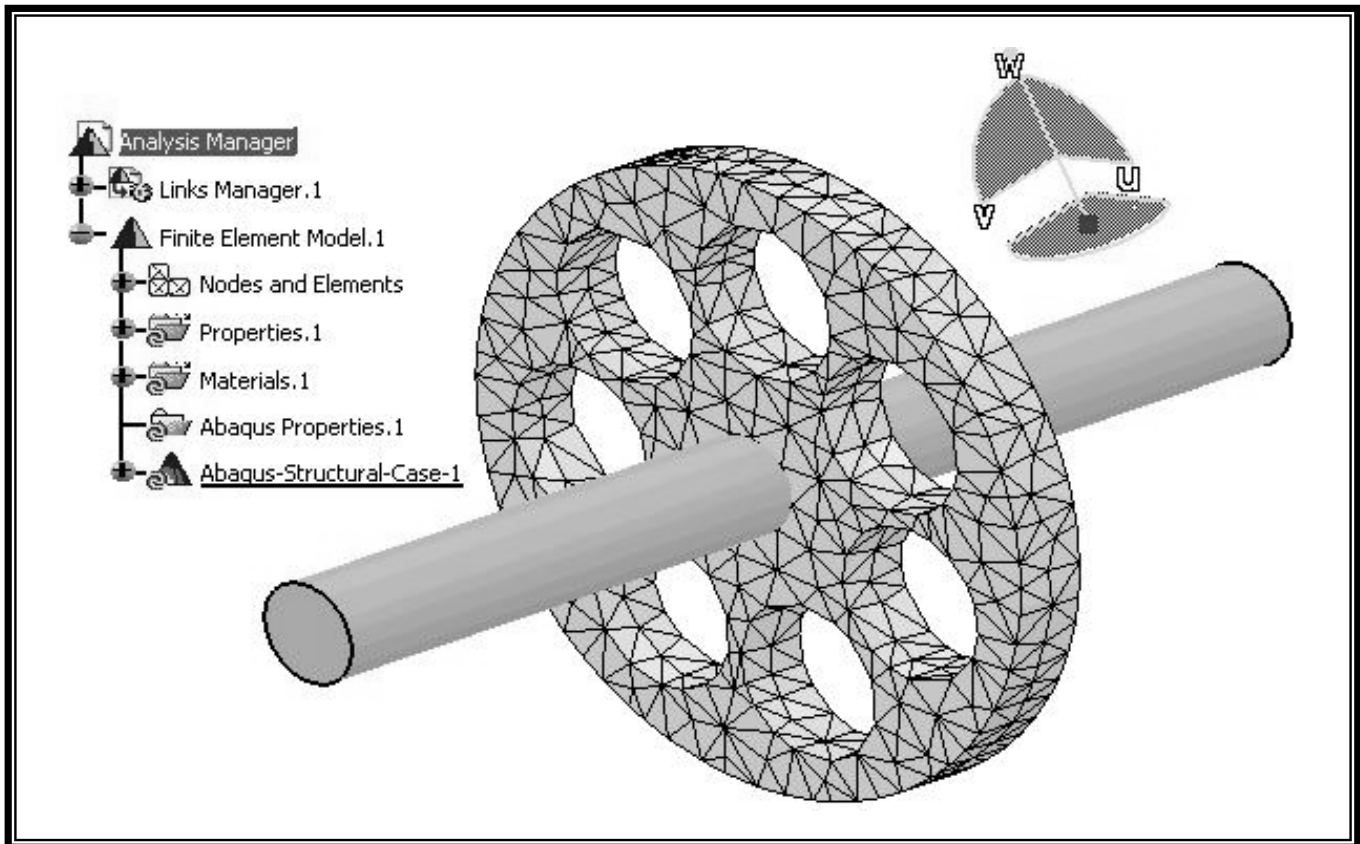


ABAQUS for CATIA V5 Tutorials

AFC V2.5



Nader G. Zamani

University of Windsor

Shuvra Das

University of Detroit Mercy

SDC
PUBLICATIONS

Schroff Development Corporation

www.schroff.com

Chapter 3

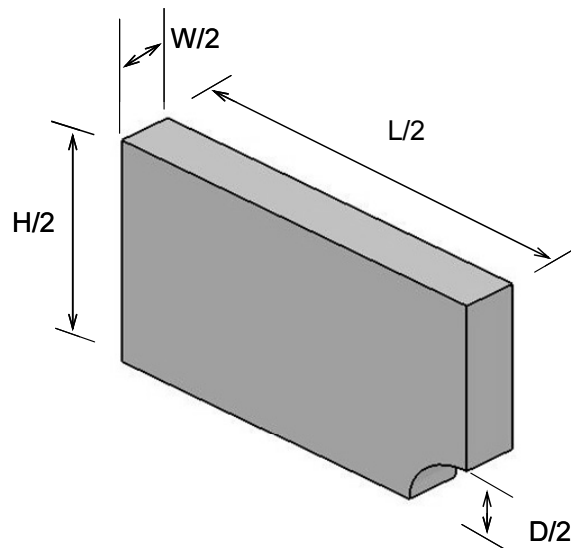
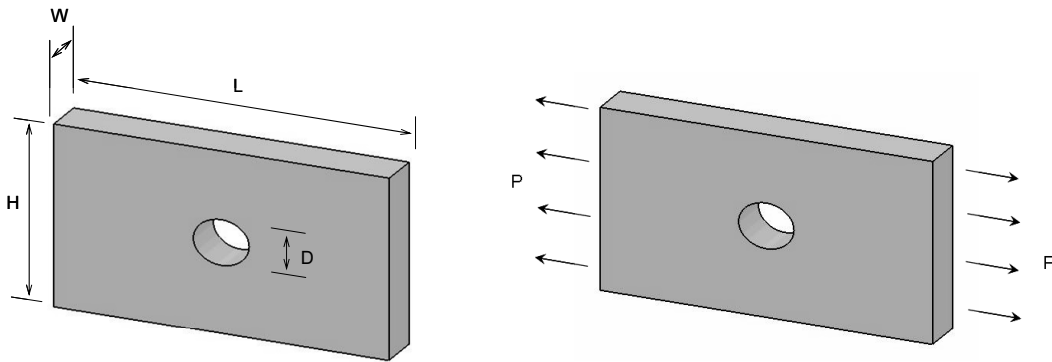
Elastic-Plastic Analysis of a Notched Plate

Introduction:

This tutorial is an extension of the problem described in chapter 2. The plate with a central hole is pulled with a high load which drives the part into the plastic range. The true stress/true strain curve is provided in tabular form for the plasticity model.


Problem Statement:

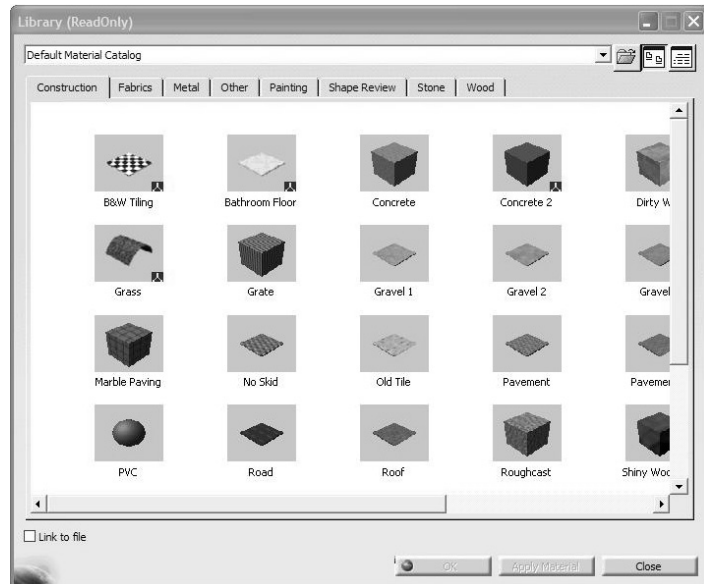
The steel plate shown below is subjected to a pressure load P at the two ends. Contrary to the earlier model in chapter 2, the three planes of symmetry are used to reduce the finite element model as shown.



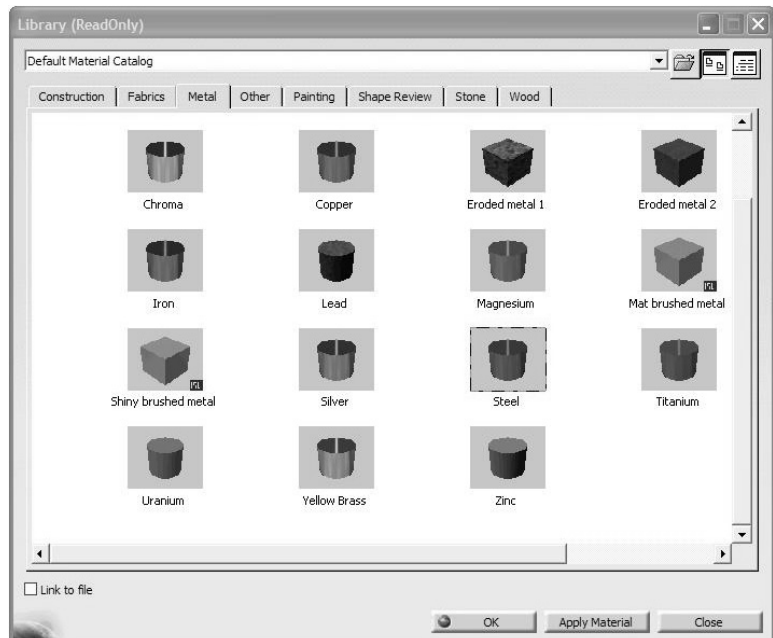
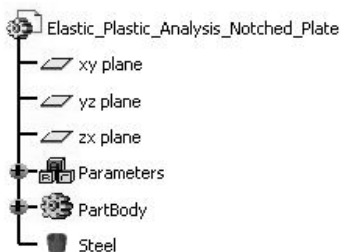
CATIA Model:

Create the CATIA model of the indicated plate with the following dimensions.
 $L = 0.15\text{m}$, $H = 0.1\text{m}$, $W = 0.02\text{m}$, and $D = 0.025\text{m}$.

Use the **Apply Material** icon  from the bottom row of toolbars. The use of this icon opens the material database box as shown next.

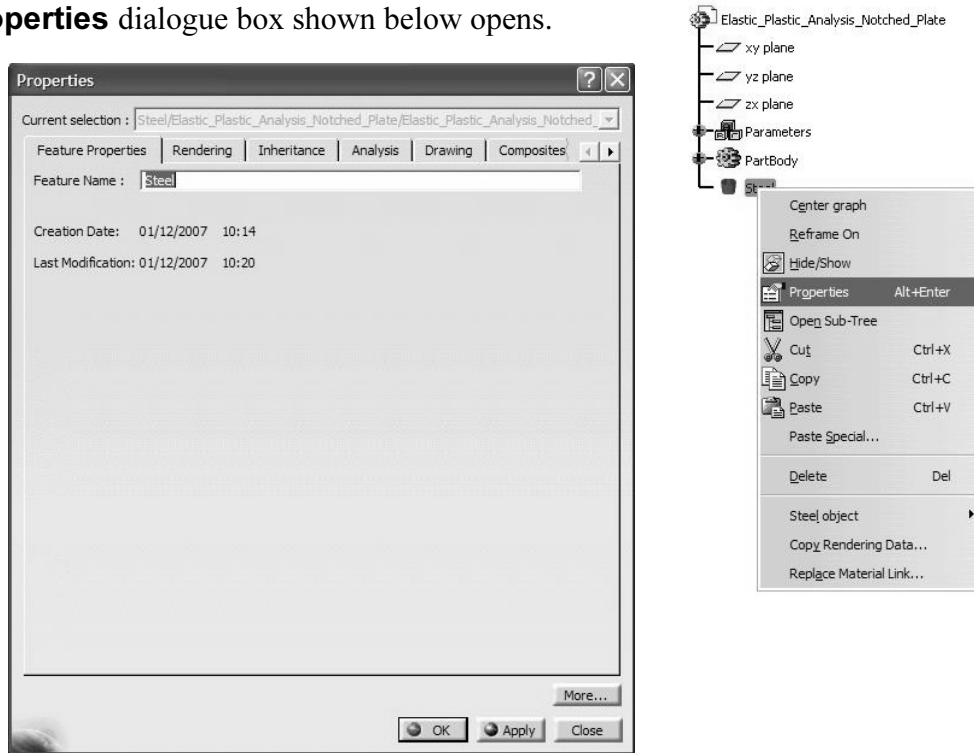


Choose the **Metal** tab on the top; select **Steel**. Use your cursor to pick the part on the screen at which time the **OK** and **Apply Material** buttons can be selected. Close the box. The material property is now reflected in the tree.

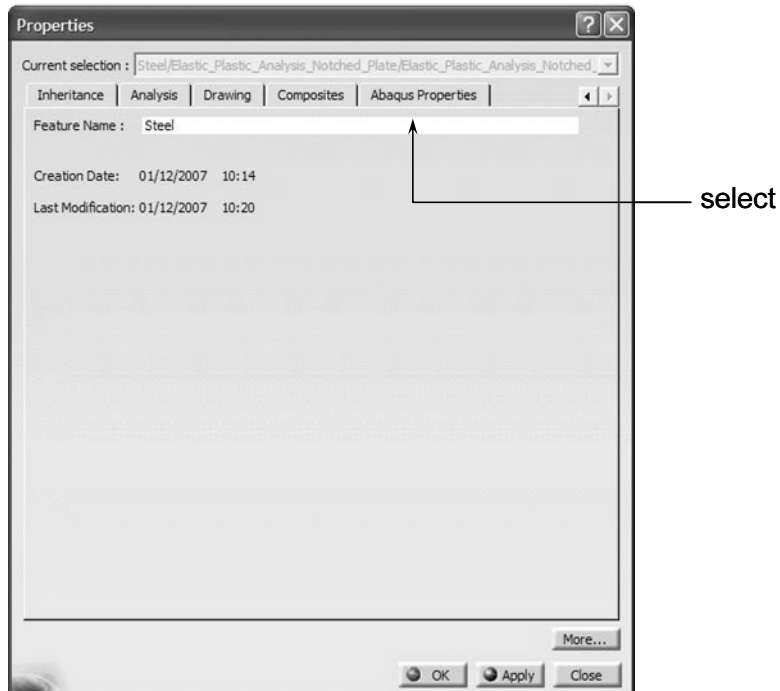


Point the cursor to the **Steel** branch just created in the tree, right-click, and select the properties from the contextual menu as shown.

The **Properties** dialogue box shown below opens.

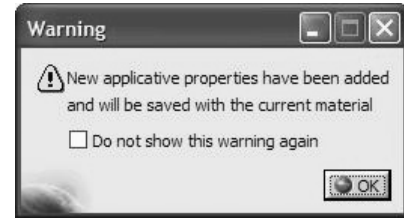
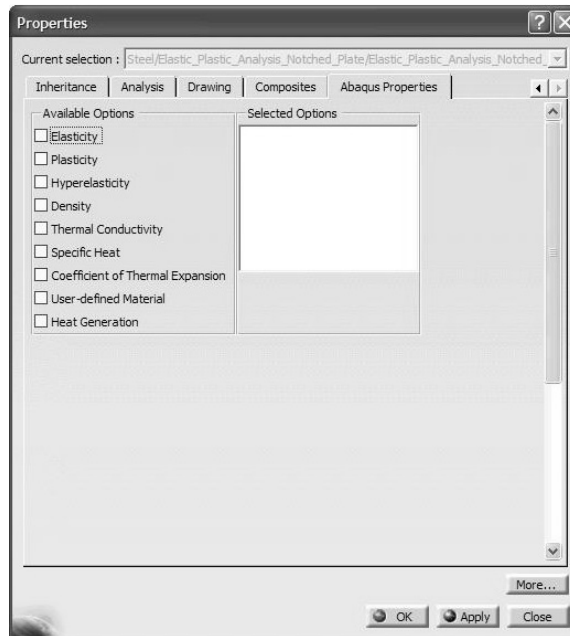


Select the **ABAQUS Properties** tab.

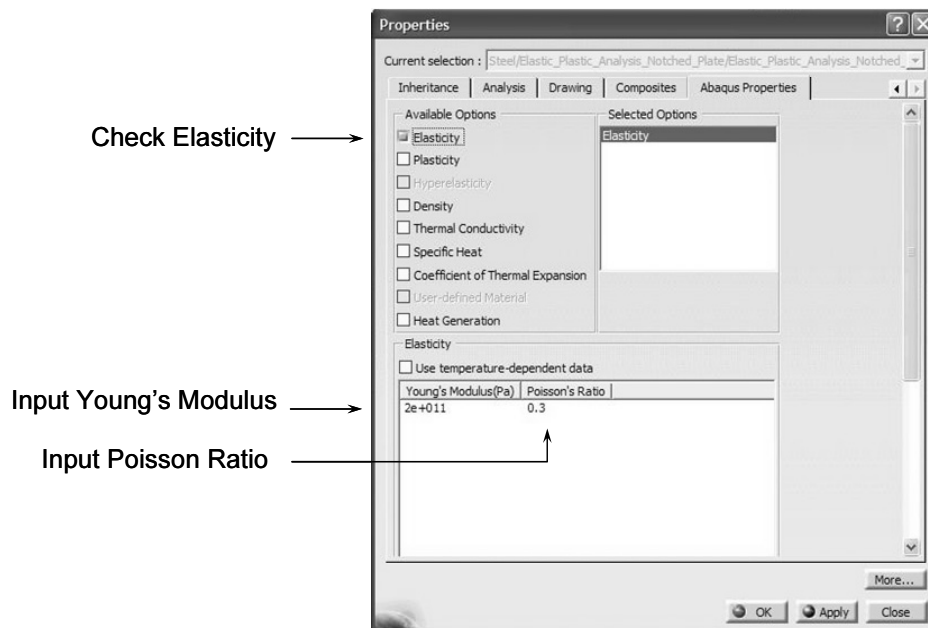


A **Warning** box appears the first time that the **ABAQUS Properties** are loaded. Ignore the warning by pressing **OK** to close the box.

The initial (unfilled) window is shown below.



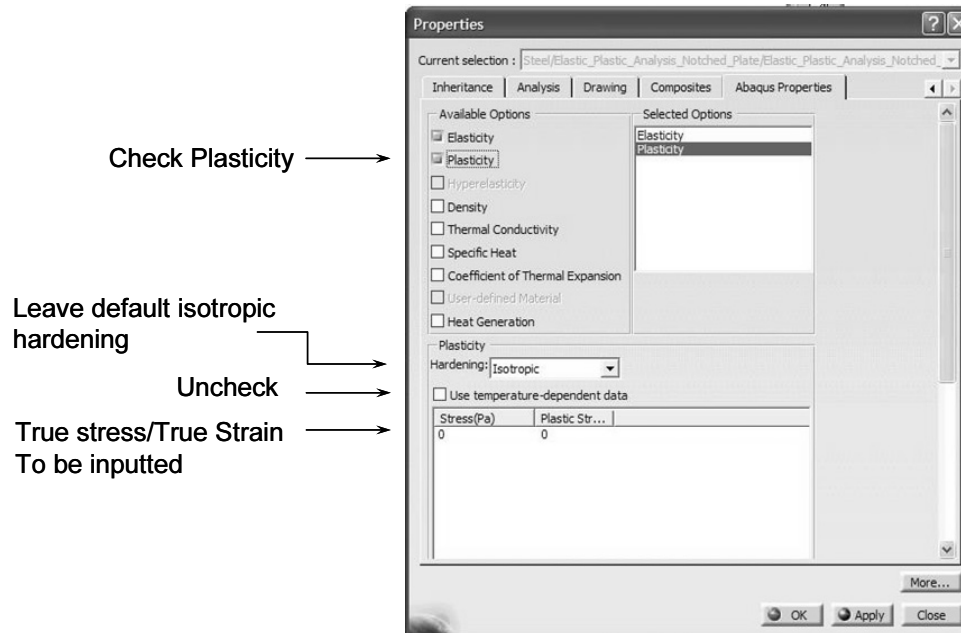
Once again, all the data supplied here overrules the data provided by the CATIA materials database. Select the **Elasticity** box. Since the material data in the present problem is independent of temperature, uncheck the box shown below. Finally, input the Young's modulus and the Poisson's ratio. This will be the data that will appear in the **ABAQUS INPUT** file.



The next several steps are new and were not required in chapter 2, which was primarily dealing with linear elastic analysis.

Check the **Plasticity** box Plasticity and select the default **Isotropic Hardening** rule. Since the material data in the problem under consideration is assumed to be at a fixed temperature, uncheck the appropriate box.

You are now in the position to input the true stress/true strain data.

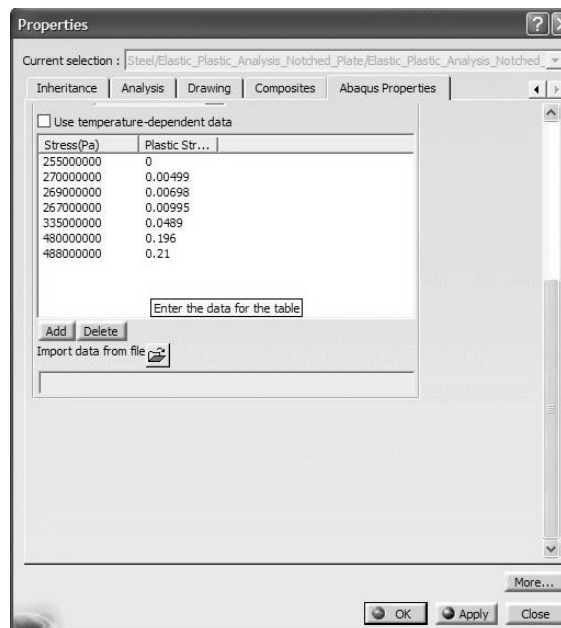


The data shown below is for AISI 1020 hot-rolled steel. Note that the first entry in the table corresponds to the yield strength at the plastic strain of zero.

NOTE: In order to add a new line of data you have to press the **Add** button. To reach the **Add** button you will have to scroll down with the side bar.

Stress(Pa)	Strain
265000000	0
270000000	0.00499
269000000	0.00698
267000000	0.00995
335000000	0.0489
480000000	0.196
488000000	0.21


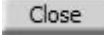
The first line of the table must be the yield strength at zero plastic strain.



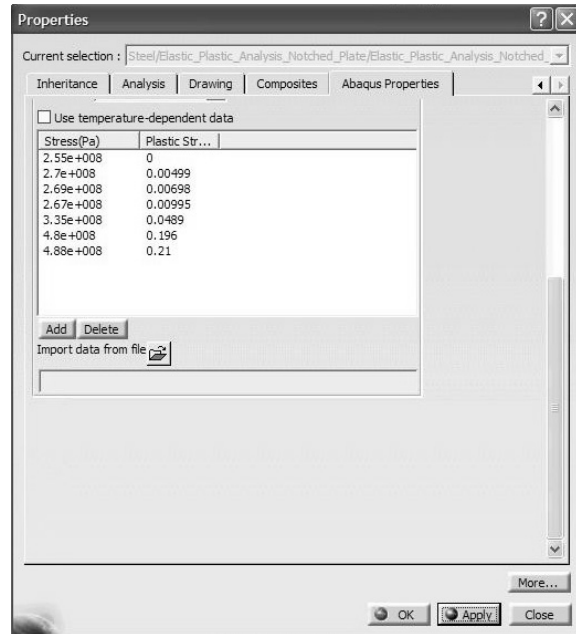
Recall that the engineering stress/engineering strain data can be converted to the true stress/true strain data through the following formulas which are valid up to the necking point.


$$\epsilon_{true} = \ln(1 + \epsilon_{eng})$$

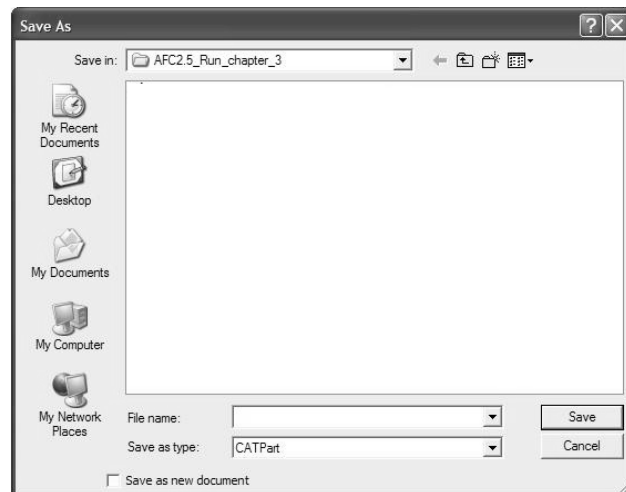
$$\sigma_{true} = \sigma_{eng}(1 + \epsilon_{eng})$$

Upon the completion of the data entry, press the **Apply**  button which displays the data in a different format. Close the box by pressing .

Stress(Pa)	Strain
2.65e+008	0
2.7e+008	0.005
2.69e+008	0.007
2.67e+008	0.01
3.35e+008	0.049
4.8e+008	0.196
4.88e+008	0.21



Click on the **Save** icon  and save the part file in a directory of your choice.



If the part is still “gray”, one can change the rendering style. From the **View** toolbar

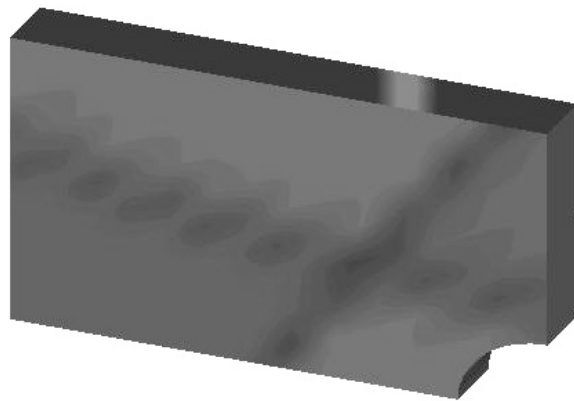


, select the **View mode** toolbar



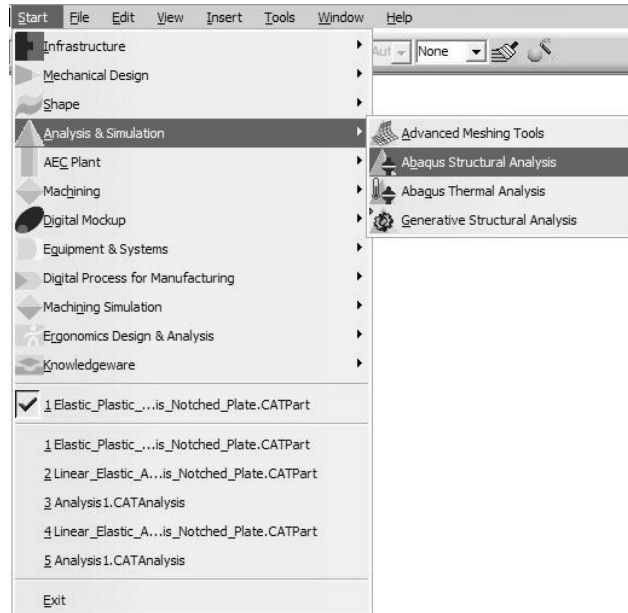
Next choose the **Shading with Material** icon .

The part now appears shaded as shown below.

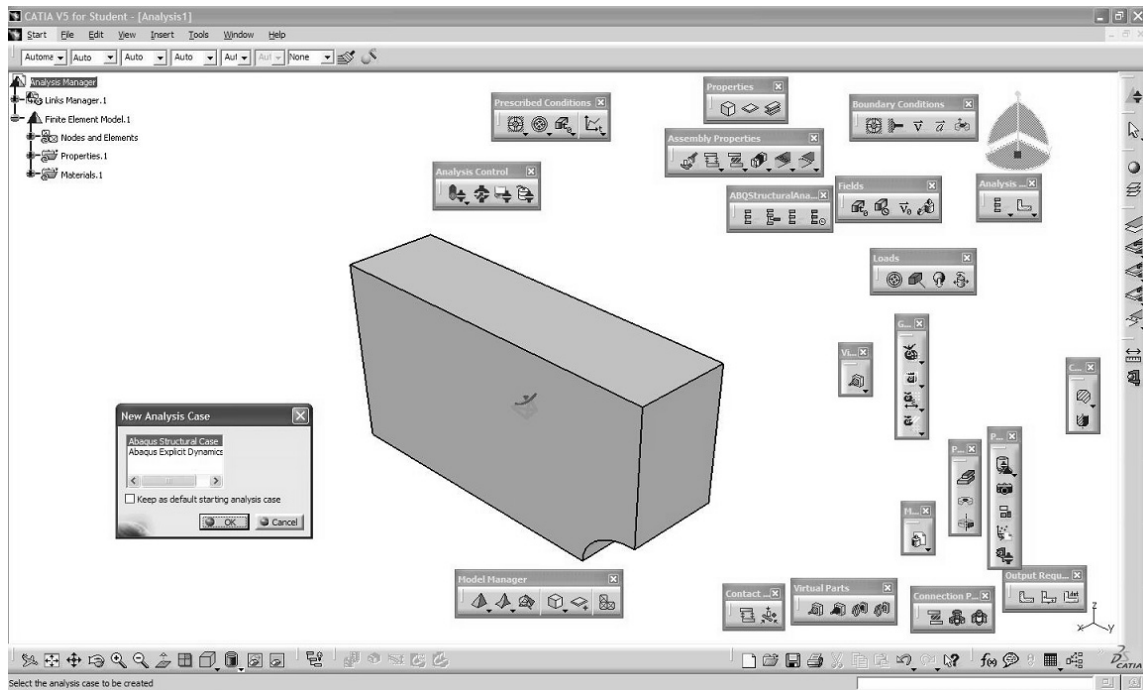


Entering the ABAQUS Structural Analysis Workbench:

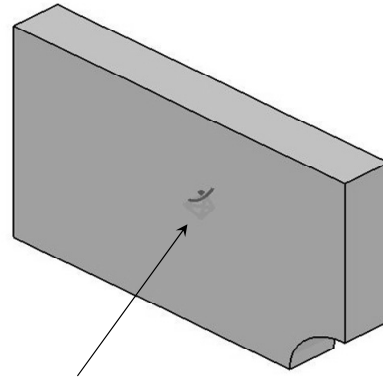
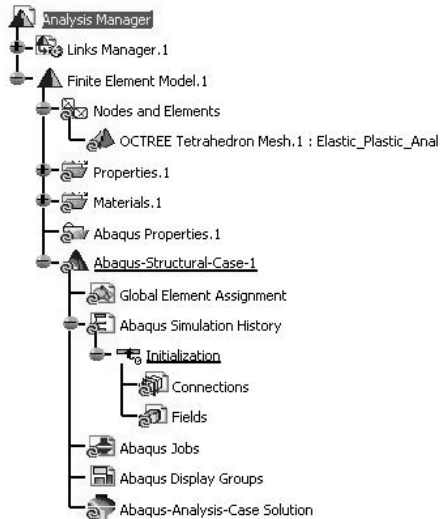
Select **Start** from the top menu, scroll down and select **Analysis & Simulation**, followed by **ABAQUS Structural Analysis**.




You will then find yourself in the new workbench with FEA icons and toolbars. The full interface with all toolbars in the default position is displayed below.

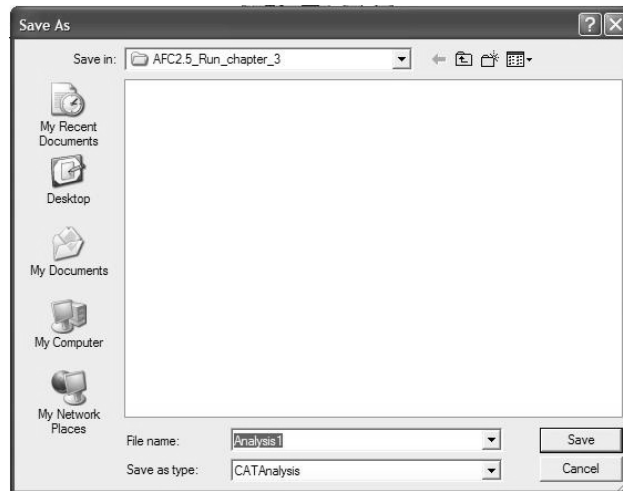


Close the **New Analysis Case** by pressing **OK**.
The representative element “Size” and “Sag” appears on your model. We have also shown the fully expanded tree for your information.

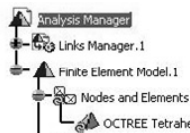


Representative Element “Size” and “Sag”

Click on the **Save** icon  and save the **Analysis.1** file in a directory of your choice.




In order to change the size and sag, double-click on the branch **OCTREE Tetrahedron Mesh.1**. The data can be edited. Keep the default values and close the window. Make sure **Parabolic** is selected.



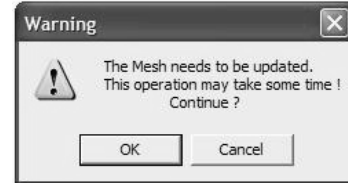
Double click on
this branch



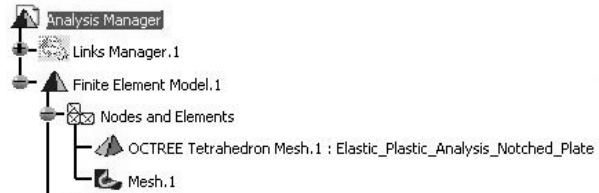
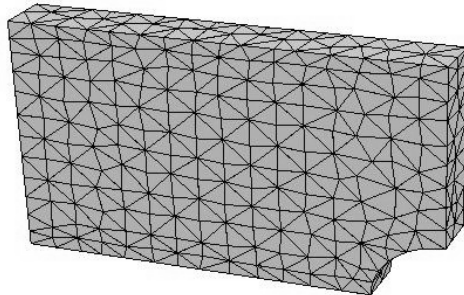
Select the **Mesh Visualization** icon  from the **Model Manager** toolbar




From the screen, select the block. The warning message below appears that you can ignore by pressing **OK**.



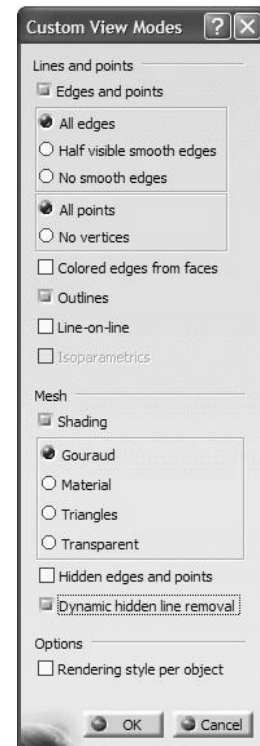
The finite element mesh appears on the screen. Note that the display of the mesh also gets recorded in the tree by the addition of the **Mesh.1** branch.



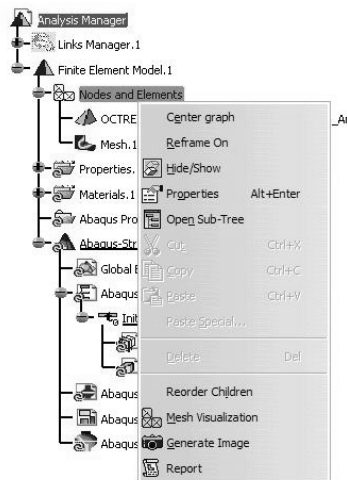
By selecting the **Customize** icon  from the **View mode** toolbar



you can open the **Custom View Modes** dialogue box below and select different options for display purposes. The above mesh was displayed by checking the **Dynamic hidden line removal** option.




An alternative way to display the mesh is to point the cursor to the **Nodes and Elements** branch in the tree, right-click, and select **Mesh Visualization**.



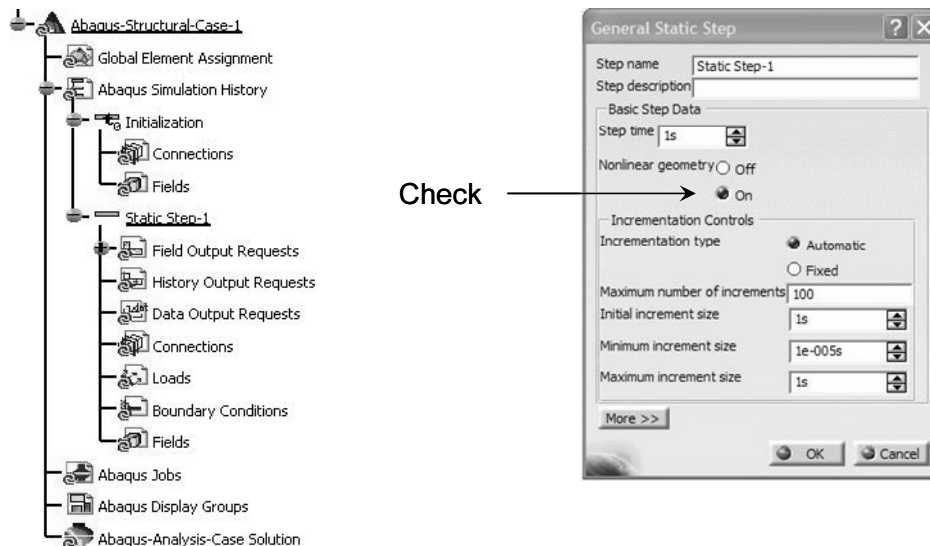
Creating an ABAQUS Step:

The **Analysis ...** toolbar  has a sub-toolbar labeled **Structur...**



Select the **General Static Step** icon . The dialogue box shown below opens. Inspect the different options but keep the default entities. Since large plastic deformation is anticipated, check the **Nonlinear geometry** to be turned **On**.

A **Static Step-1** branch appears in the tree. Clearly this step belongs to **ABAQUS-Analysis-Case-1**.

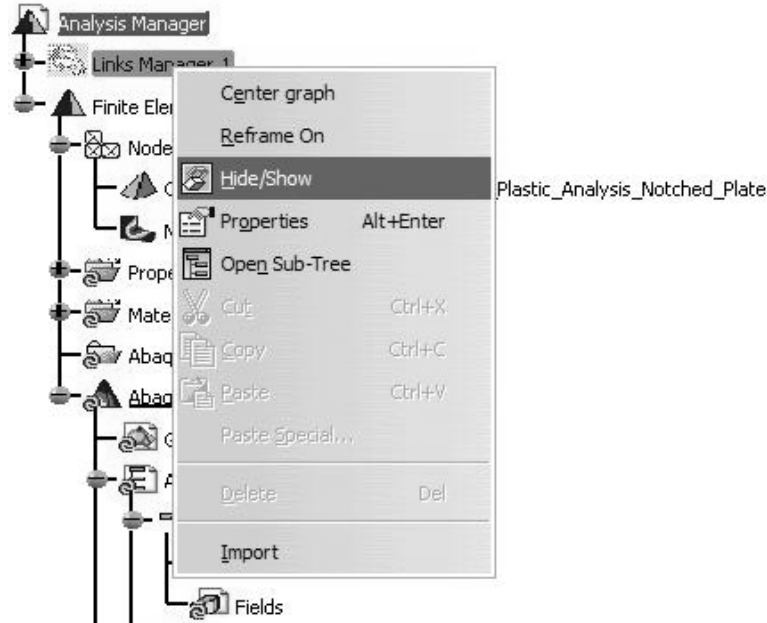


Press on **OK** to close the **General Static Step** dialogue box.



Applying the Pressure Load:

CATIA and ABAQUS for CATIA V5 (AFC) are geometrically based packages. This means that the restraints and loads have to be applied on the geometrical entities associated with the part. For example, these entities can be a vertex, an edge, a face, or the entire body. One cannot apply restraints and loads on nodes or elements. The mesh that appears on the screen must therefore be replaced with the actual part.

Point the cursor to the branch of the tree labeled **Links Manager.1**, right click, and select **Hide/Show**. The part then appears on the screen. The part is superimposed on the mesh. If you want to see the part only, point the cursor to the **Nodes and Elements** branch, right click, and select **Hide/Show**

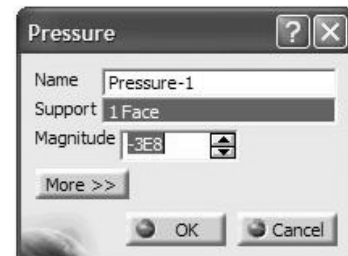
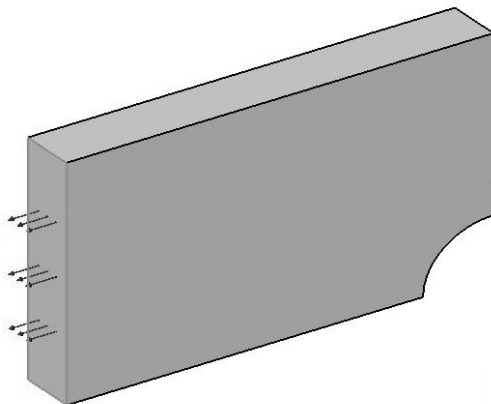


The **Prescribed Conditions** toolbar  has a sub-toolbar called

Loads . Select the **Pressure Load** icon .

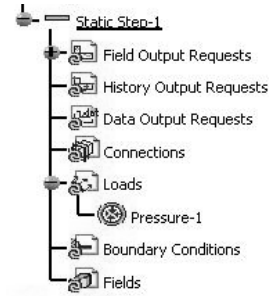
This action opens the **Pressure** dialogue box shown below.

Select the end face of the plate and enter **-3E+8Pa** for the **Magnitude**. The completed box is displayed below.




The negative pressure indicates that the selected surface is under a tensile load.


The tree reflects the presence of the pressure loading.

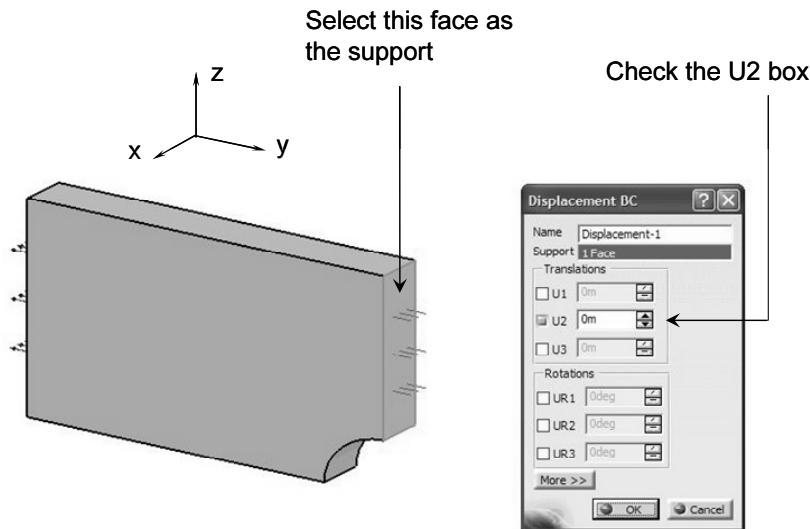



Applying the Restraints:

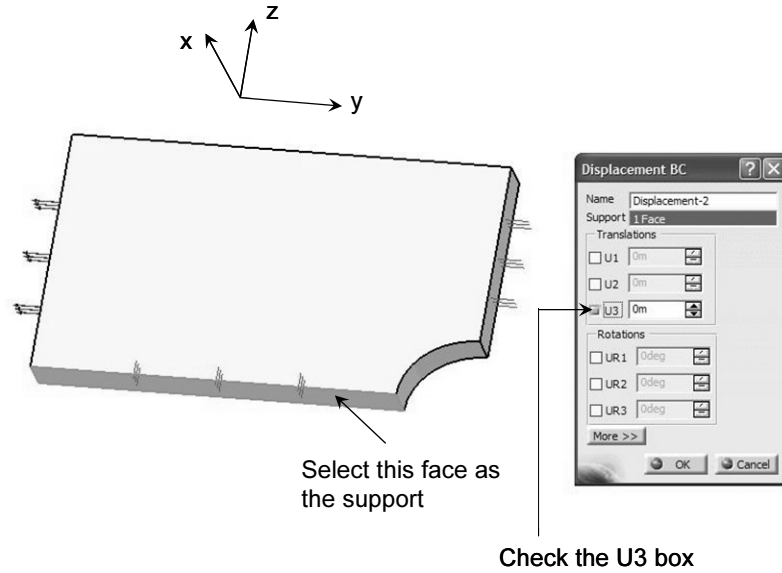
The **Boundary Conditions** toolbar  is a sub-toolbar of the


Prescribed Conditions toolbar .

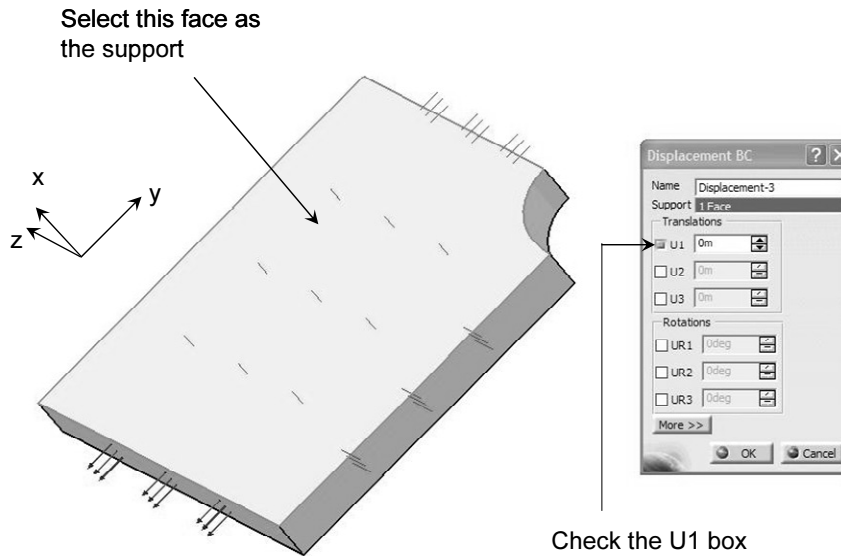
Select the **Displacement Boundary Condition** icon . In the resulting dialogue box, select the face shown as **Support**. Assuming the xyz coordinate system is as indicated below, check the **U2** box. If your part is oriented differently with respect to the xyz coordinate system, check the appropriate U1, U2, or U3 box. The displacement normal to this face must be zero regardless.



Once again, select the **Displacement Boundary Condition** icon . In the resulting dialogue box, select the face shown as **Support**. Assuming the xyz coordinate system is as indicated below, check the **U3** box. If your part is oriented differently with respect to the xyz coordinate system, check the appropriate U1, U2, or U3 box. The displacement normal to this face must be zero regardless.

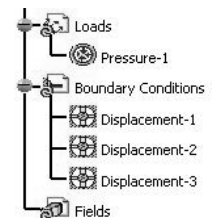


Select the **Displacement Boundary Condition** icon . In the resulting dialogue box, select the face shown as **Support**. Assuming the xyz coordinate system is as indicated below, check the **U1** box. If your part is oriented differently with respect to the **xyx** coordinate system, check the appropriate U1, U2, or U3 box. The displacement normal to this face must be zero regardless.



The three prescribed displacements imposed appear in the history tree.

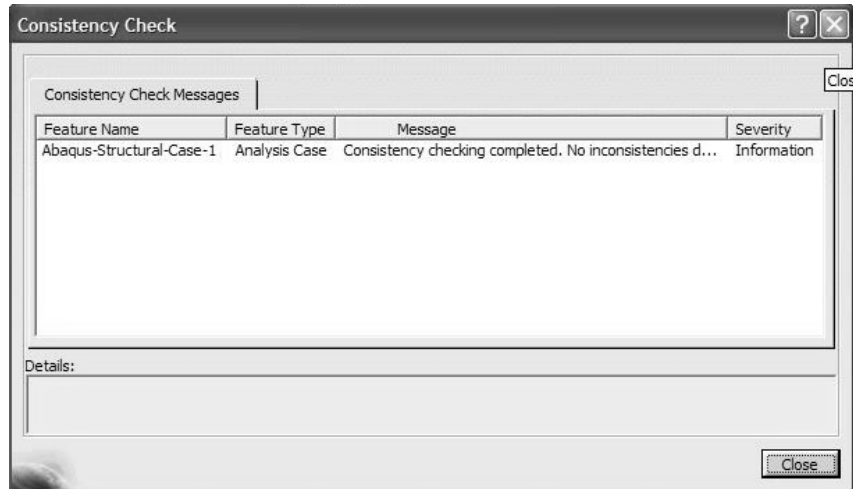
Once again, it is emphasized that if your xyz orientation is different from what is displayed, you have to check the appropriate U1, U2, and U3 boxes consistent with your xyz directions.



Select the **ABAQUS Consistency Checker** icon  from the **Analysis Control**



toolbar. The resulting dialogue box indicates that no inconsistencies were detected.



Close the **Consistency Check** box.

Creating an ABAQUS Job:

Select the **Create Job** icon  from the **Analysis Control**

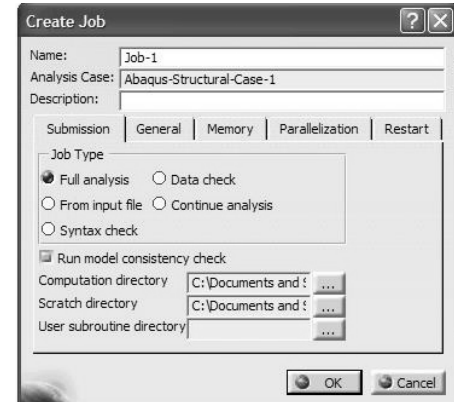
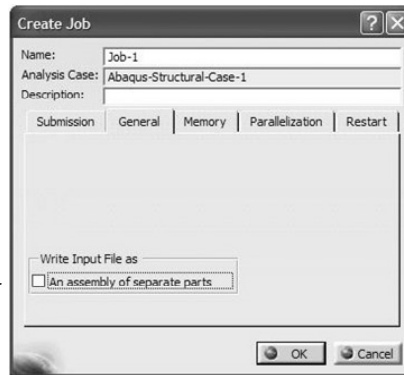
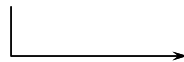


toolbar. The dialogue box shown below appears.

Keep all the default settings. Note that the location of the **Computation directory** and the **Scratch directory** can be set (changed) in this box.

Select the **General** tab and uncheck the box shown.

Very important:
Uncheck this box



If you forget to uncheck the **An assembly of separate parts** box, you will not be able to attach the results after the run is completed. In that case, to view the results, you need to open a dummy analysis file and import the ODB file.

Close the **Create Job** box by pressing **OK**.

A **Job-1** branch is created in the tree.




Submission Using the Job Manager:

Select the **Job Manager** icon  in the **Analysis Control** toolbar

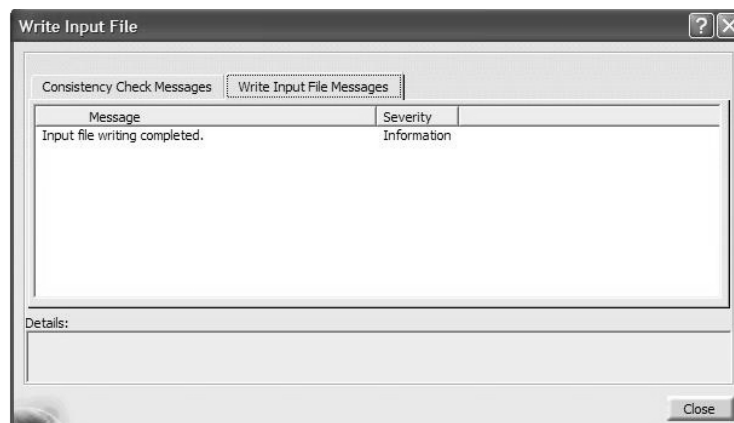
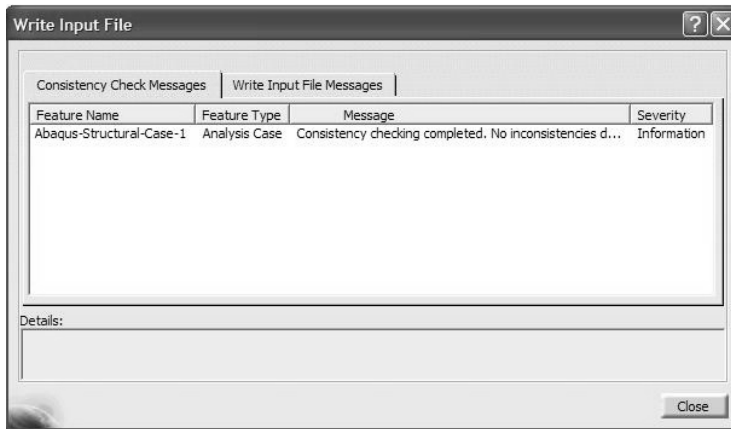


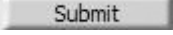
The **Job Manager** dialogue box appears on the screen.

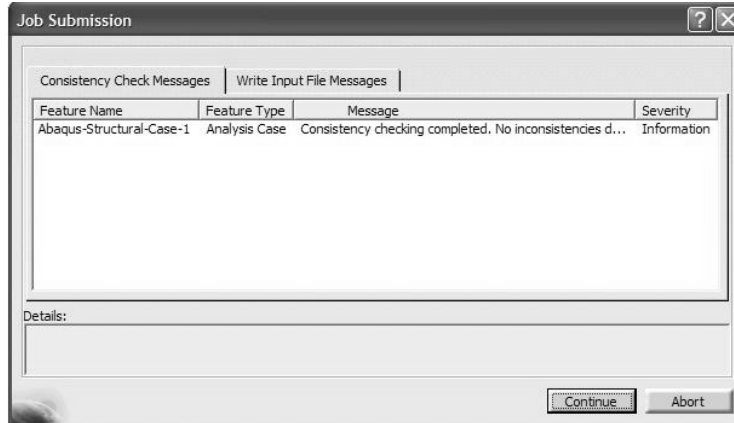


Select the **Write Input**  button in the dialogue box. This leads to the **Write Input File** window shown below. By selecting the **Write Input File Messages** tab, you get confirmation that the inp file has been written.

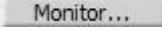
Close the window by pressing on **Close**.



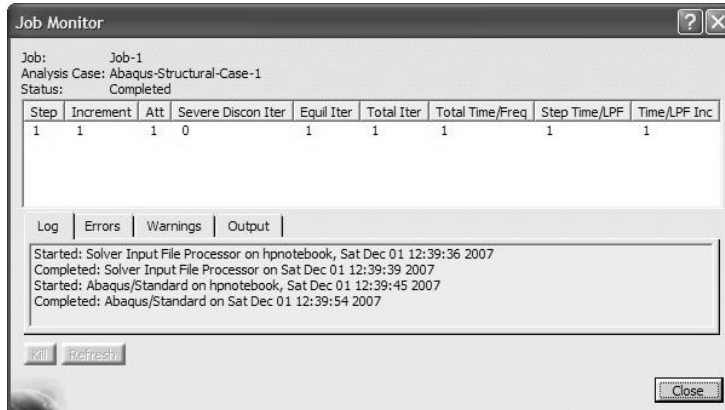
Select the **Submit**  button in the **Job Manager** window. The action results in the **Job Submission** window displayed next.



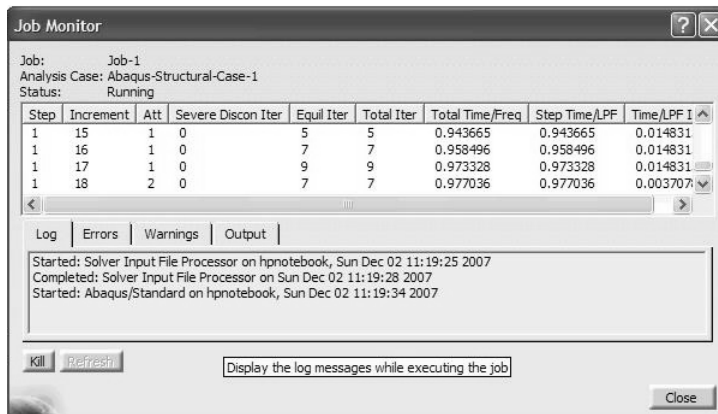
Select the **Continue**  button.

The **Status None** in the **Job Manager** dialogue box changes to **Status Run**. Select the **Monitor**  button.

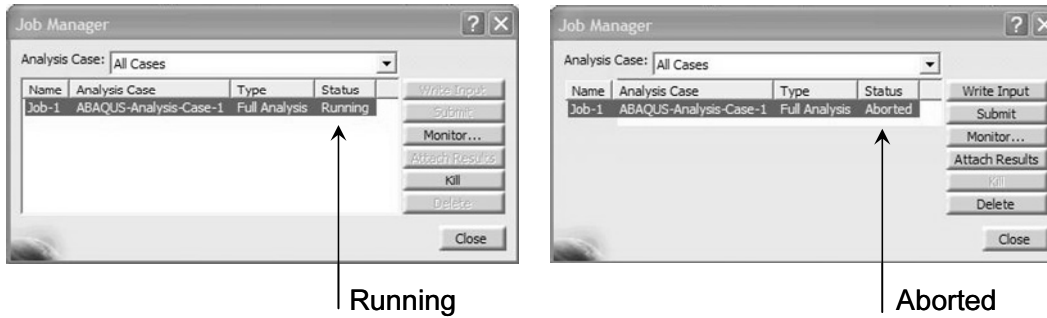
The **Job Monitor** window keeps updating the progress of the ABAQUS job.



As the run progresses, different steps are recorded in the **Job Monitor** window as shown below.

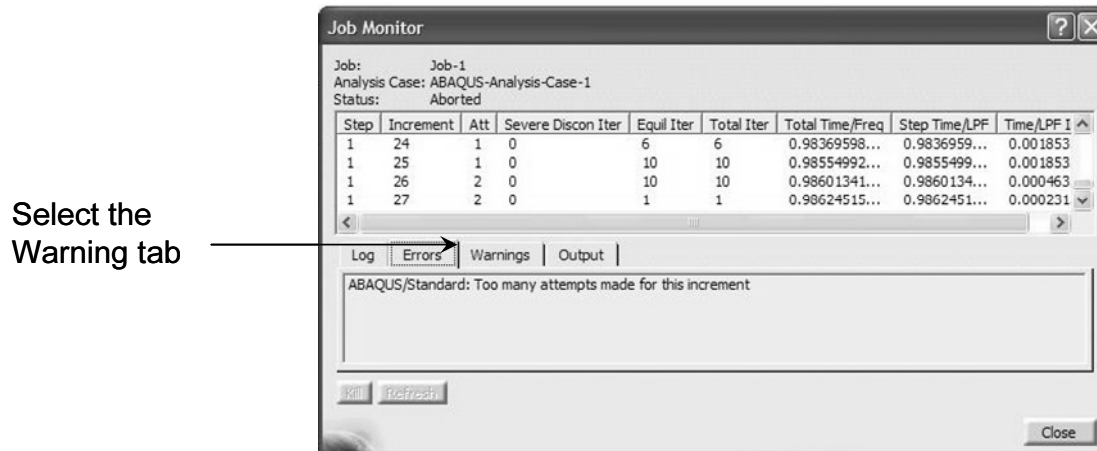
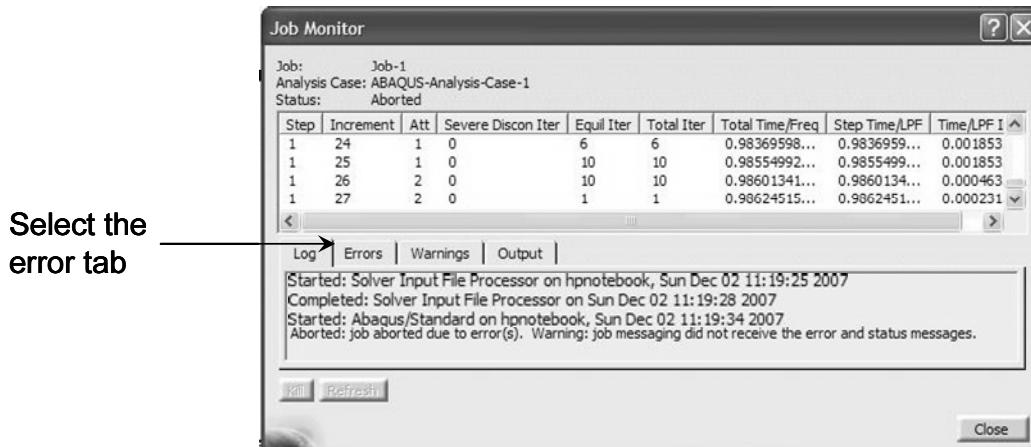


This particular job aborts after 27 iterations as displayed in the **Job Monitor** dialogue box.

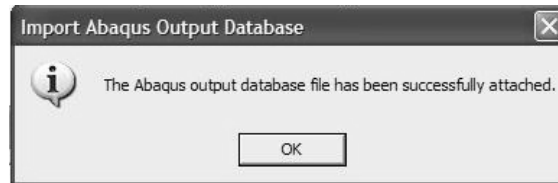


In order to view the reason behind the Abort status, you can select the **Warnings** and **Errors** tabs in the “big” **Job Monitor** window.

In this run, the reason for the **Abort** status is that, at the step 28, the number of iterations to establish equilibrium has exceeded the default value set in ABAQUS.



Close the **Job Monitor** window and select **Attach Results**  in the **Job Manager** dialogue box.



Close the above window by pressing **OK**. Also close the **Job Manager** window. The results of **Static Step-1** are attached and reflected in the tree.

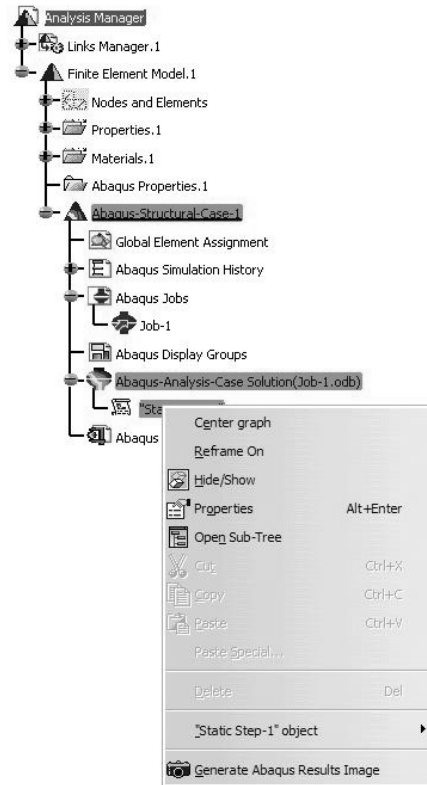


Postprocessing:

You are now in a position to conduct postprocessing.

Point the cursor to the **“Static Step-1”** branch at the bottom of the tree, right click, and select **Generate ABAQUS Results Image**.

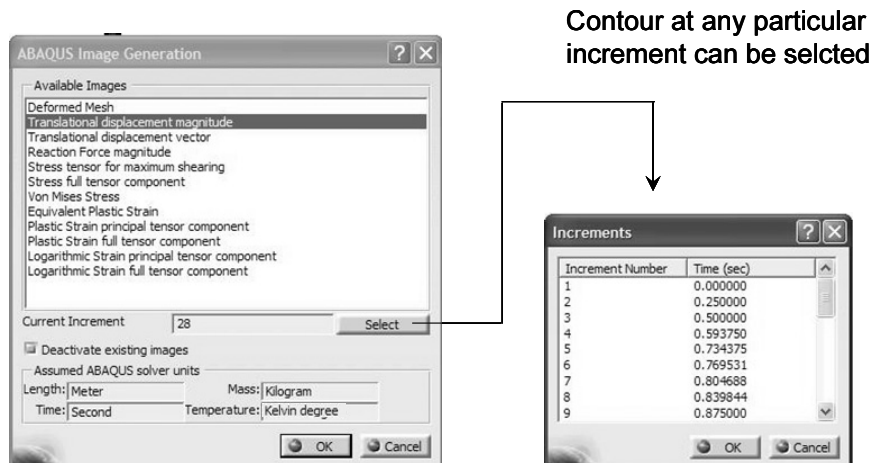
You are presented with the **ABAQUS Image Generation** dialogue box where items to be plotted can be selected.



This action can also be achieved by selecting the **Generate ABAQUS Results Image**

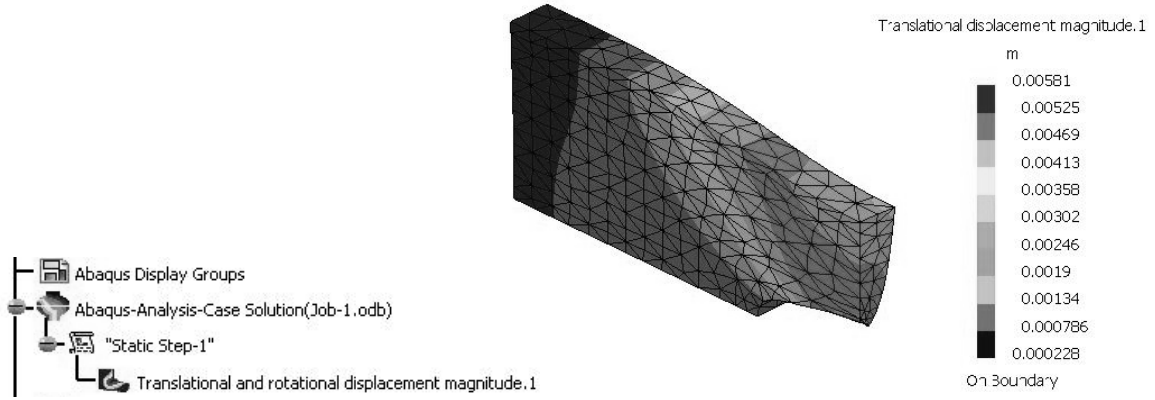


Select the **Translational displacement magnitude** from the list and press **OK**. The contour plot at any particular increment can be requested by using the **Select** button.


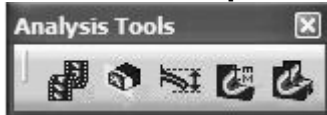


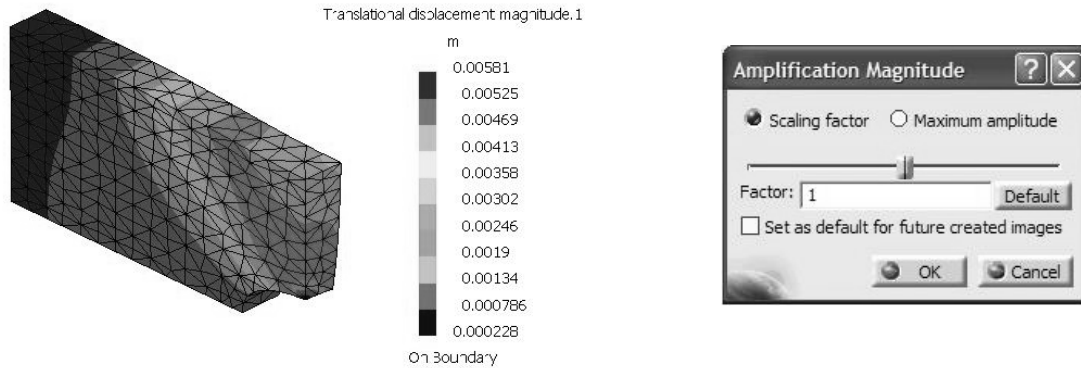
In the event that the deformed and undeformed meshes superimpose, point the cursor to the **OCTREE Tetrahedron Mesh.1**, right click and select **Hide/Show**.

The requested contour plot is displayed. The plot is also recorded in the tree.




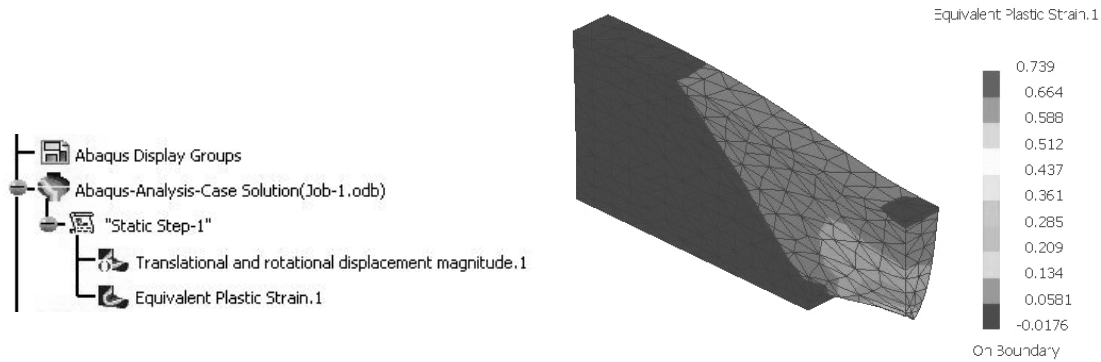
The contour plot is drawn on the deformed shape which has been magnified for viewing purposes. If you prefer the non-magnified scale, select the **Amplification Magnitude**

icon  from the **Analysis Tools** toolbar . In the resulting dialogue box, set the **Factor** to 1.

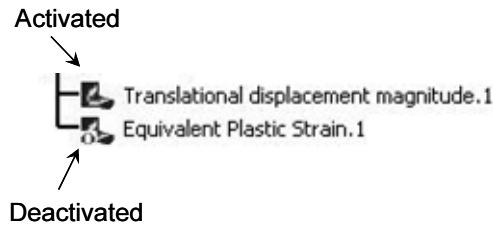



Select the **Generate ABAQUS Results Image** icon  from the **Postprocessing**

toolbar . From the **ABAQUS Image Generation** dialogue box select **Equivalent Plastic Strain**.



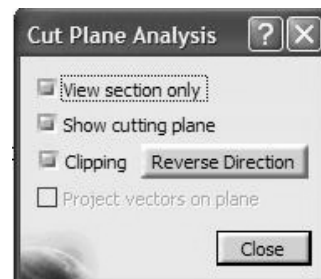
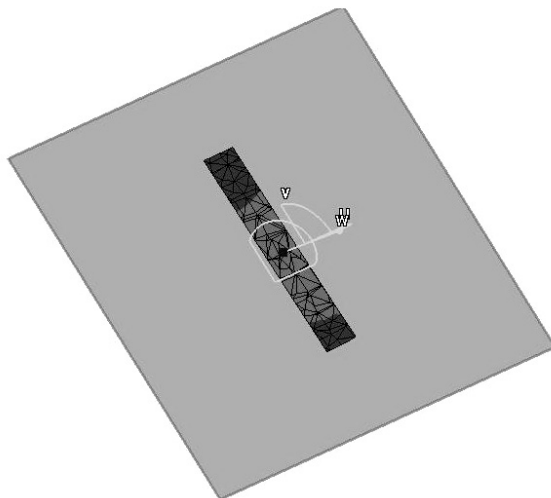
Point the cursor to the branch of the tree labeled **Equivalent Plastic Strain.1**, right click, and select **Activate/Deactivate**. This will deactivate the contour and places the part on the screen. Note that both contours plotted are deactivated now. This time activate the first contour namely, **Translational displacement magnitude.1**. Obviously, every time a contour is plotted, by default the previous contour is deactivated.



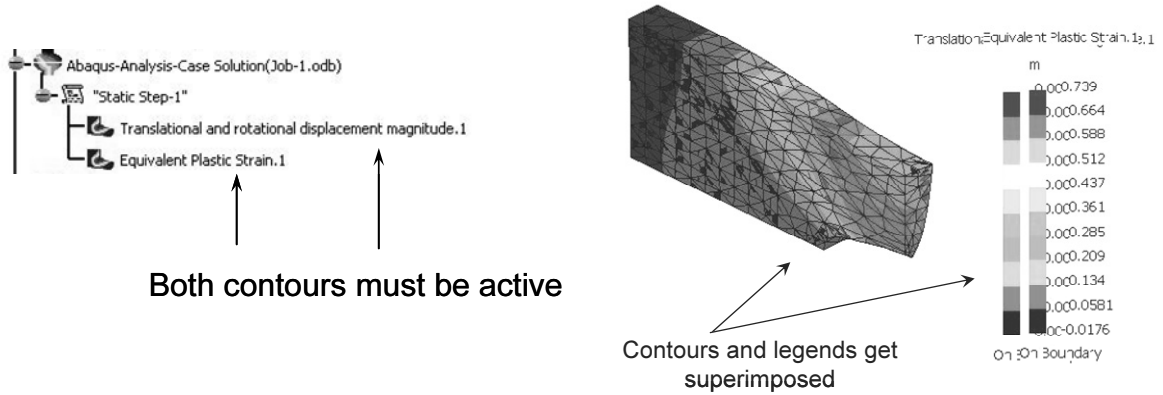
Select the **Cut Plane Analysis** icon  the **Analysis Tools** toolbar



The part is cut with a plane and the contour entity (in this case **Translational displacement**) can be viewed in the cut part. Note that the plane can be manipulated by the compass; for example, it can be translated and rotated as desired. This feature is useful to view the variable on interest inside of the part.



You may be interested in placing the two contours on the screen side-by-side. This is very simple. First you have to make both contours active. This is achieved by placing the cursor on the two contour plots in the tree, right clicking, and selecting **Activate/Deactivate**.



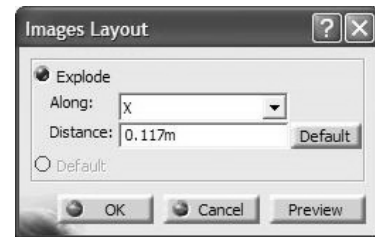
When the two contours are activated, they get superimposed above each other. The contour legends are also superimposed as indicated above.

Click the **Image Layout** icon  from the **Analysis Tools** toolbar

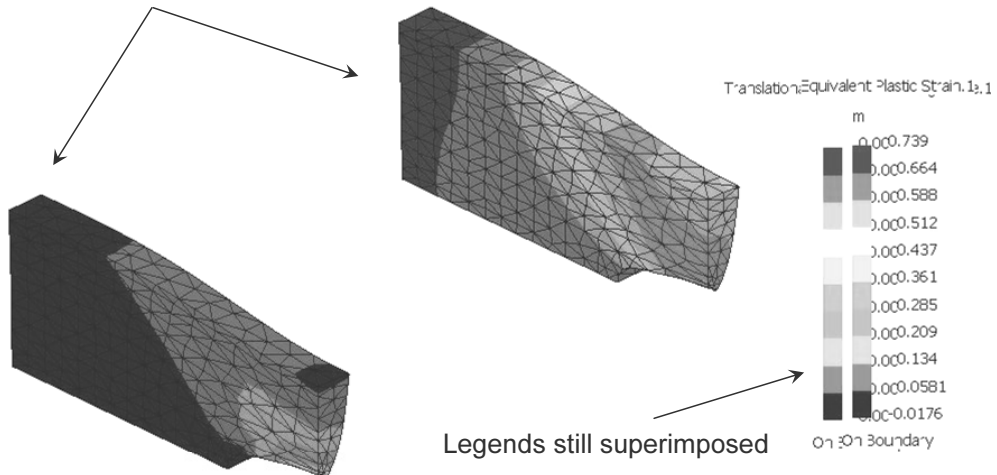


The **Images Layout** box, shown to the right, asks you to specify the direction along which the two plots are expected to be aligned.

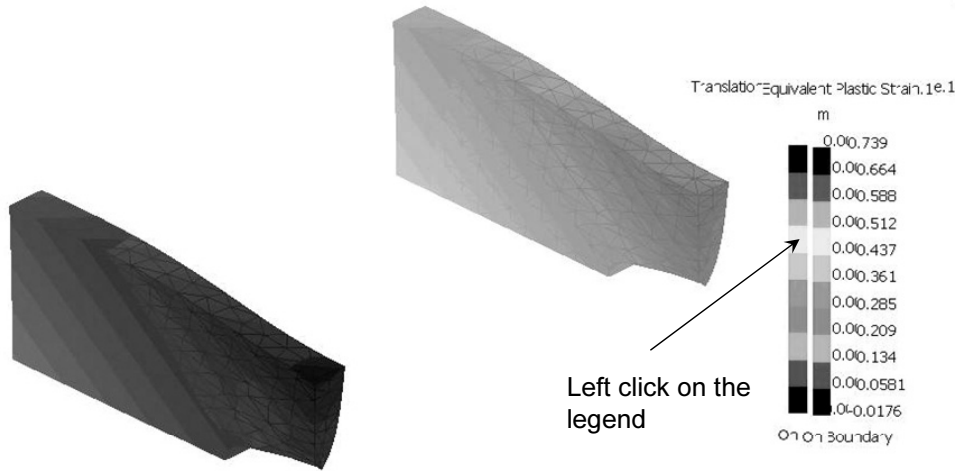
The outcome is side-by-side plots shown below.




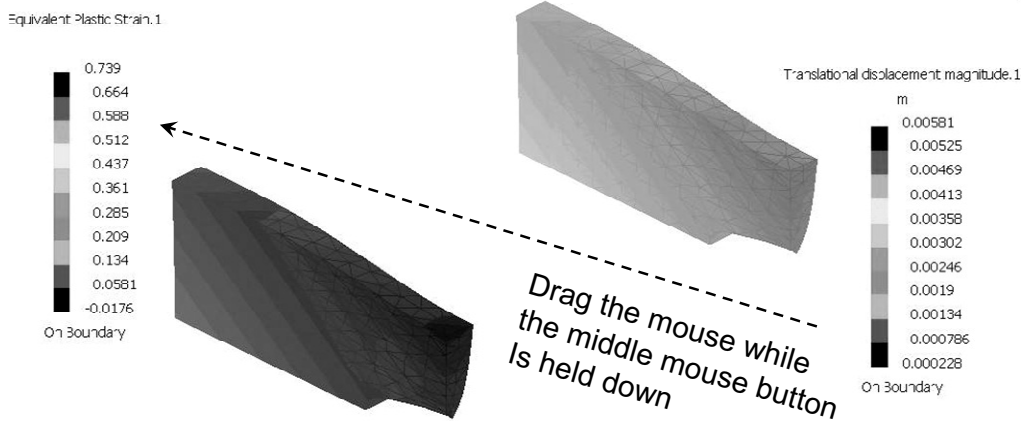
Contours side-by-side



At this point, the contours are side-by-side but the legends are still superimposed. To separate them, select the legend with the left mouse button which in turn “dims” the contour as shown below.

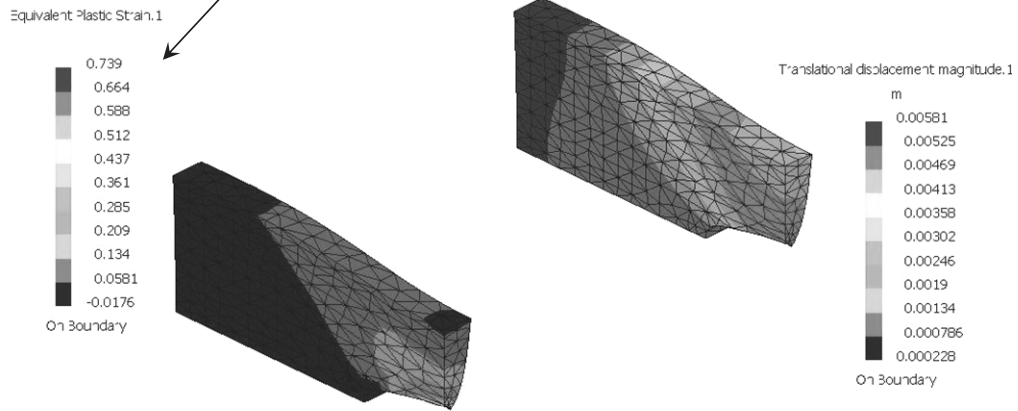


Now press the middle mouse button (and keep it pressed) and drag the mouse. In case of a two mouse button, after the legend has been selected, click on the **Pan** icon . Place the selected legend at the desired location.

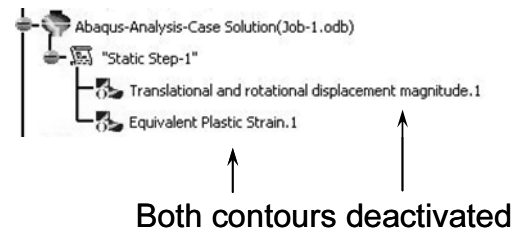


Finally, select either one of the two legends for the second time. This takes the actual contour plots out of the “dim” mode.

Select the legend again to take the contour out of the "dim" mode.



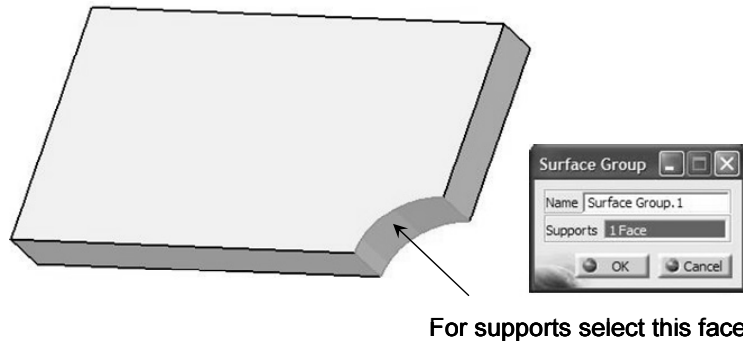
Once again, deactivate both contour plots. This places the part on the screen.



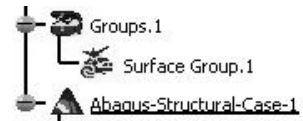
Select the **Surface Group** icon  from the **Typed Groups** toolbar



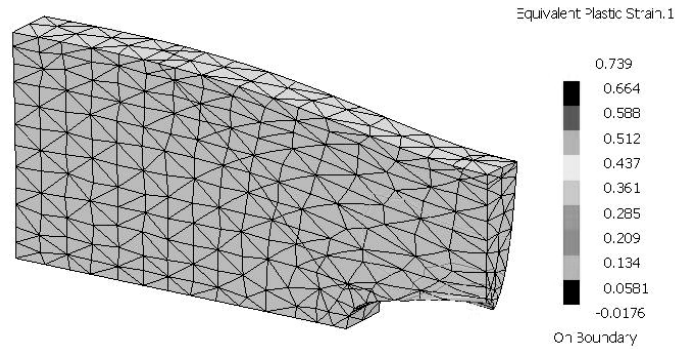
. The resulting dialogue box allows you to select the desired surface.




A surface group is recorded in the tree as shown.



Now activate the Equivalent Plastic Strain.1 contour and double-click anywhere on the contour.

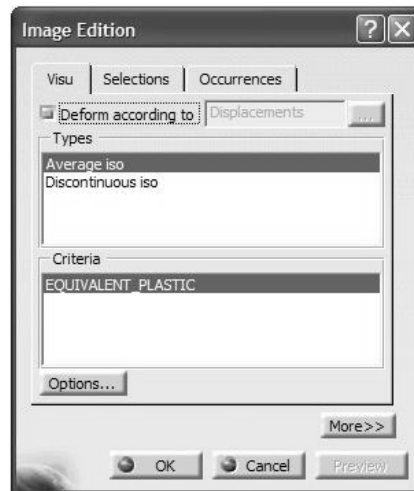


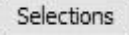

If you are not getting the shaded mode, use the **Shading with Material** icon  from

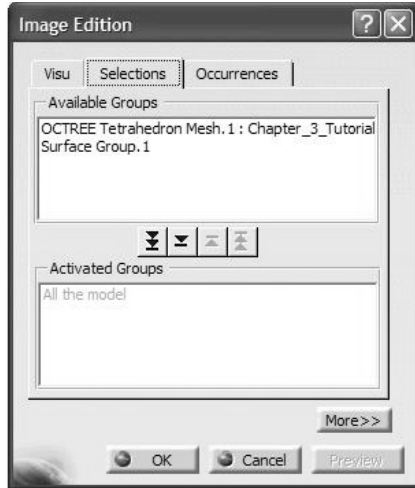
the **View Mode** toolbar



Once you double-click on the contour, the **Image Edition** dialogue box below appears.



Pick the **Selections**  tab. Using the cursor, choose the **Surface Group.1** from the list and click on the button .



The final outcome is the contour of the Equivalent Plastic Strain on the surface group created.

