ABAQUS Tutorial

3D Modeling

This exercise intends to demonstrate the steps you would follow in creating and analyzing a simple solid model using ABAQUS CAE.

Introduction

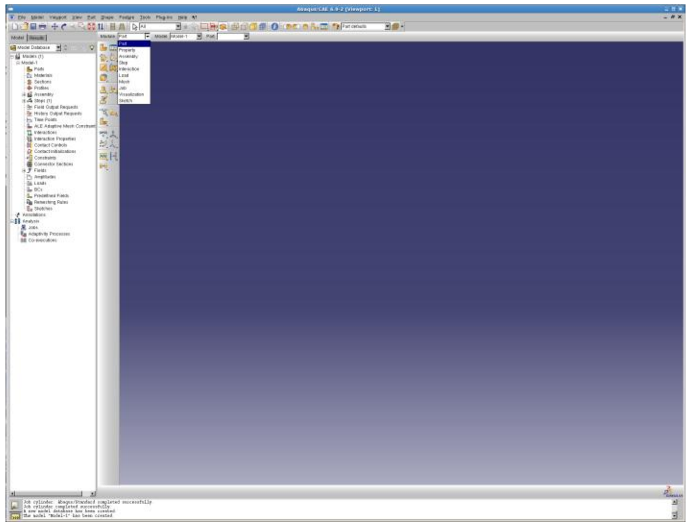
A solid undergoes thermal expansion due to the application of heat along with deformation due to applied load.

Model Definition

Consider a thin aluminum cylinder of length 1 m and inner and outer radii 0.2 m & 0.21 m respectively. The cylinder is kept fixed at one end and at the other end a tensile load of 200 kPa is applied. The fixed end of the cylinder is at 273.15 K (the ambient temperature) and the free end at 274.15 K (all other sides are insulated). The cylinder expands due to the heat flow.

The various functions within ABAQUS are organized into modules and we are going to use these modules to define the steps in our procedure.

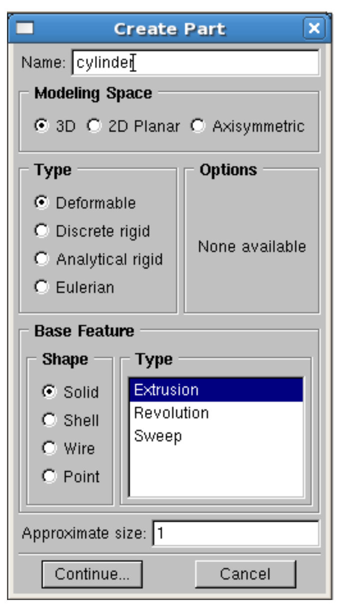
1. >module load abaqus/6.9-2
2. >abaqus cae
3. Once you start ABAQUS CAE select **Create Model Database** to create a new model.
4. The default module that opens up is the **Part Module.**



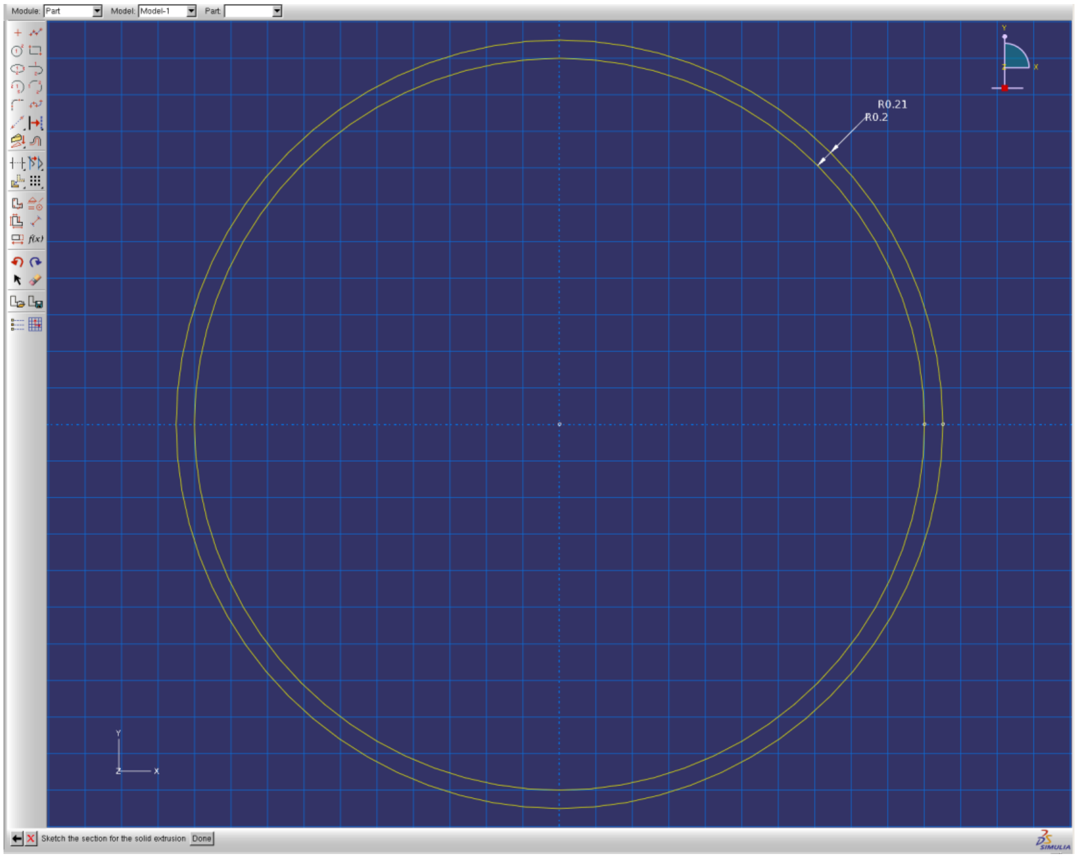
Part Module:

This module allows you to create the geometry required for the problem. To create a 3-D geometry you first create a 2-D profile and then manipulate it to obtain the solid geometry.

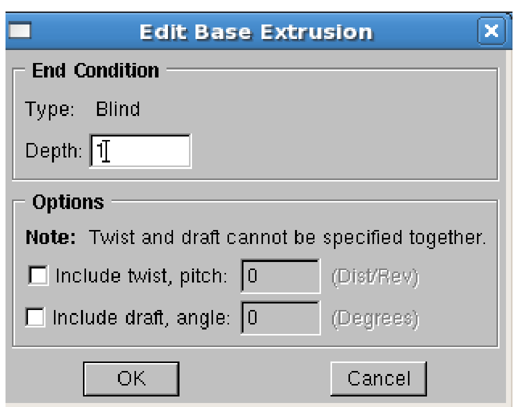
1. From the **Part Toolbox** on the left of the viewport select **Create Part**.
2. You can name the part as **cylinder** or anything else you like. We are going to create a **deformable solid** shape in the **3-D** modeling space through **extrusion** so we do not change the default selections.
3. Enter 1 as the approximate size and click **Continue**.



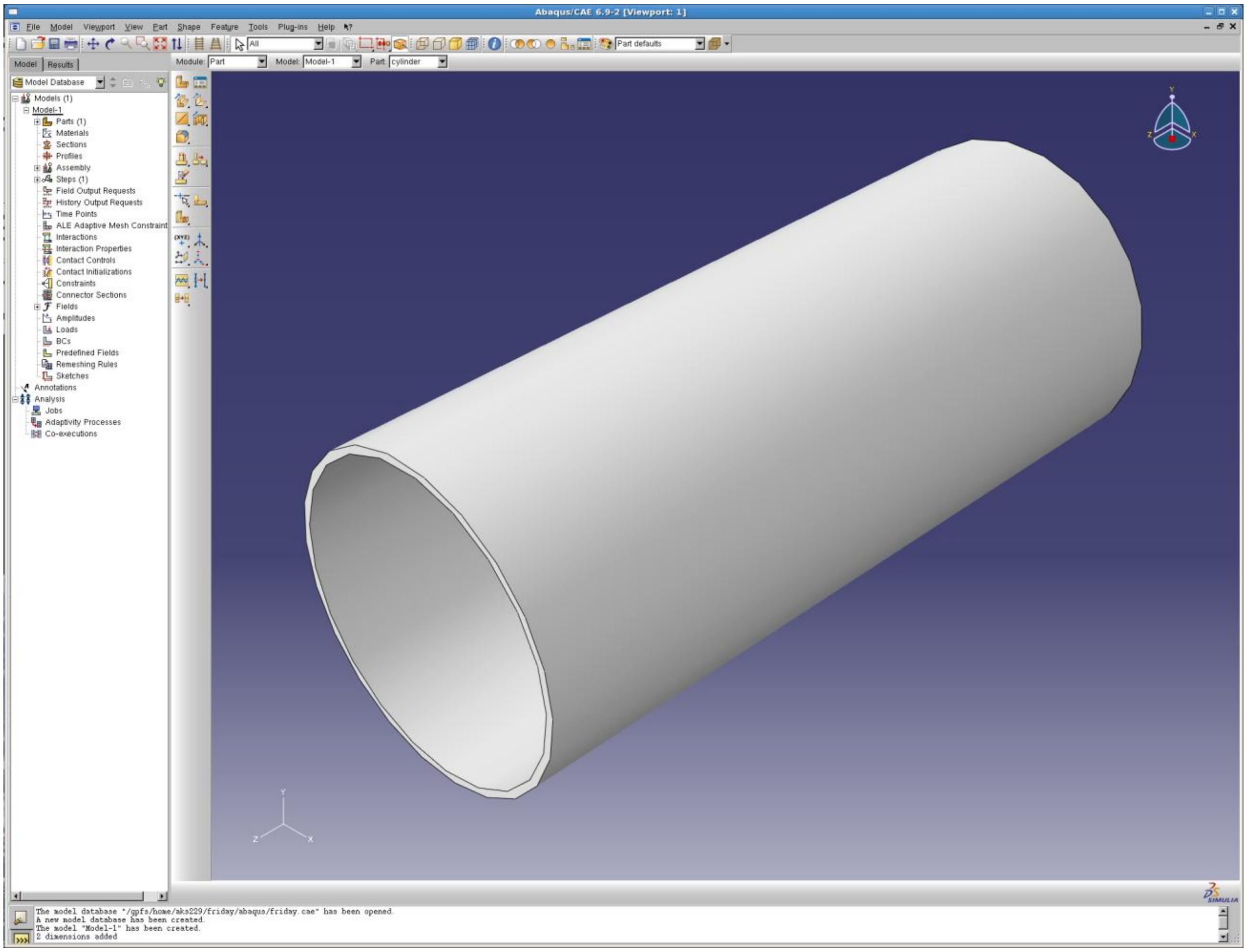
4. Click **Create Circle Center and Perimeter** on the drawing toolbox and enter 0, 0 as the center point in field below the viewport and press **Enter**. Enter the perimeter point as 0.21, 0 and press **Enter** to complete the circle. Similarly make another circle with the same center and the perimeter point as 0.2, 0. Press **Esc** to exit the circle definition and then press **Done.**



5. Enter the extrusion depth as 1 and press **OK.**



6. Click **Auto-Fit View** in the toolbar above to zoom out and view all the points.



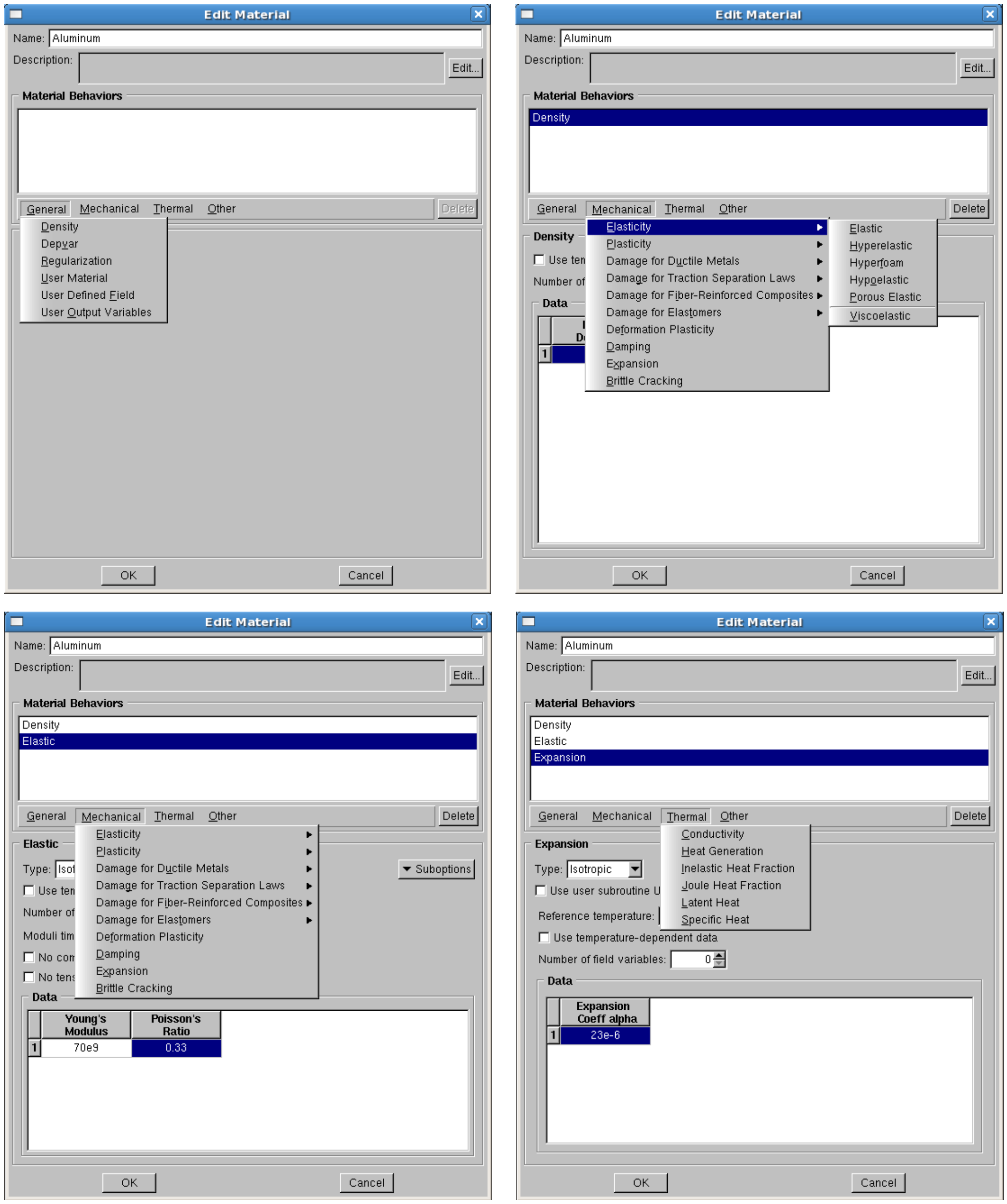
This finishes our work in the **Part** module. Select **Module: Property** from the toolbar above the viewport.

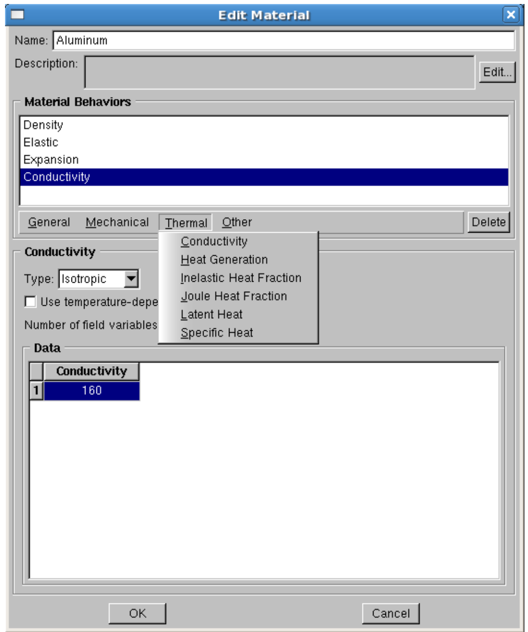
Property Module:

In this module you define the material properties for your analysis and assign those properties to the available parts.

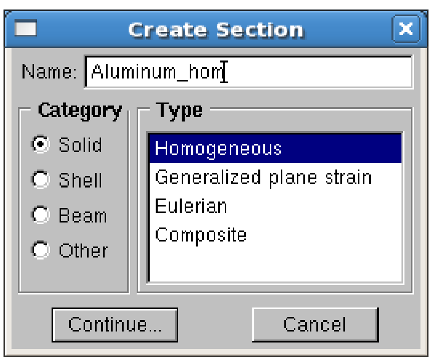
1. Select **Create Material** from the **Property Toolbox.**
2. Enter material name as **Aluminum**. Click on the **General** tab and select **Density** from the drop-down menu. Type in the mass density as 2700. Click on the **Mechanical** tab and select **Elasticity>Elastic** from the drop-down menu. Enter the Young's Modulus as 70E9 and the Poisson's Ratio as 0.33. Click on the **Mechanical** tab and select **Expansion**. Edit the reference temperature to 273.15 and the expansion coefficient to 23e-6. Click on

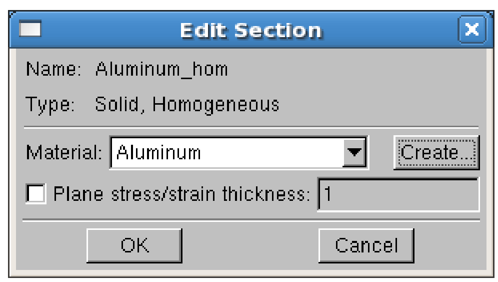
the **Thermal** tab and select **Conductivity.** Enter the thermal conductivity as 160. Click on the **Thermal** tab and select **Specific Heat**. Enter the value as 900 and click **OK**.



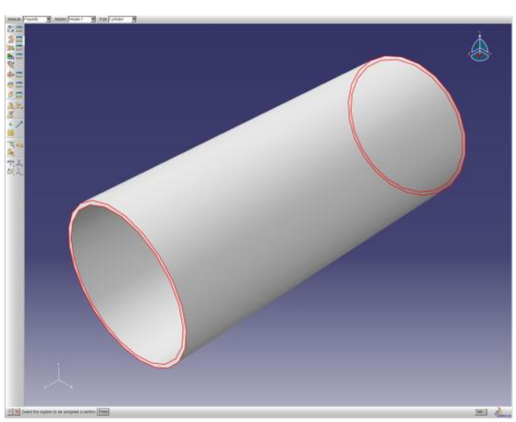


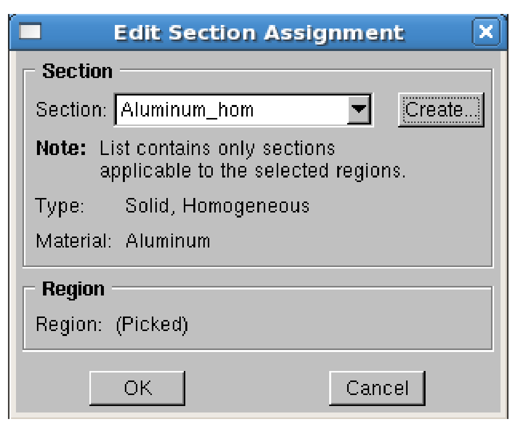
3. Select **Create Section** from the property toolbox. Name the section as you like. We need a **solid homogeneous** section for our problem. Click **Continue**. Select the material as Aluminum and click **OK.**



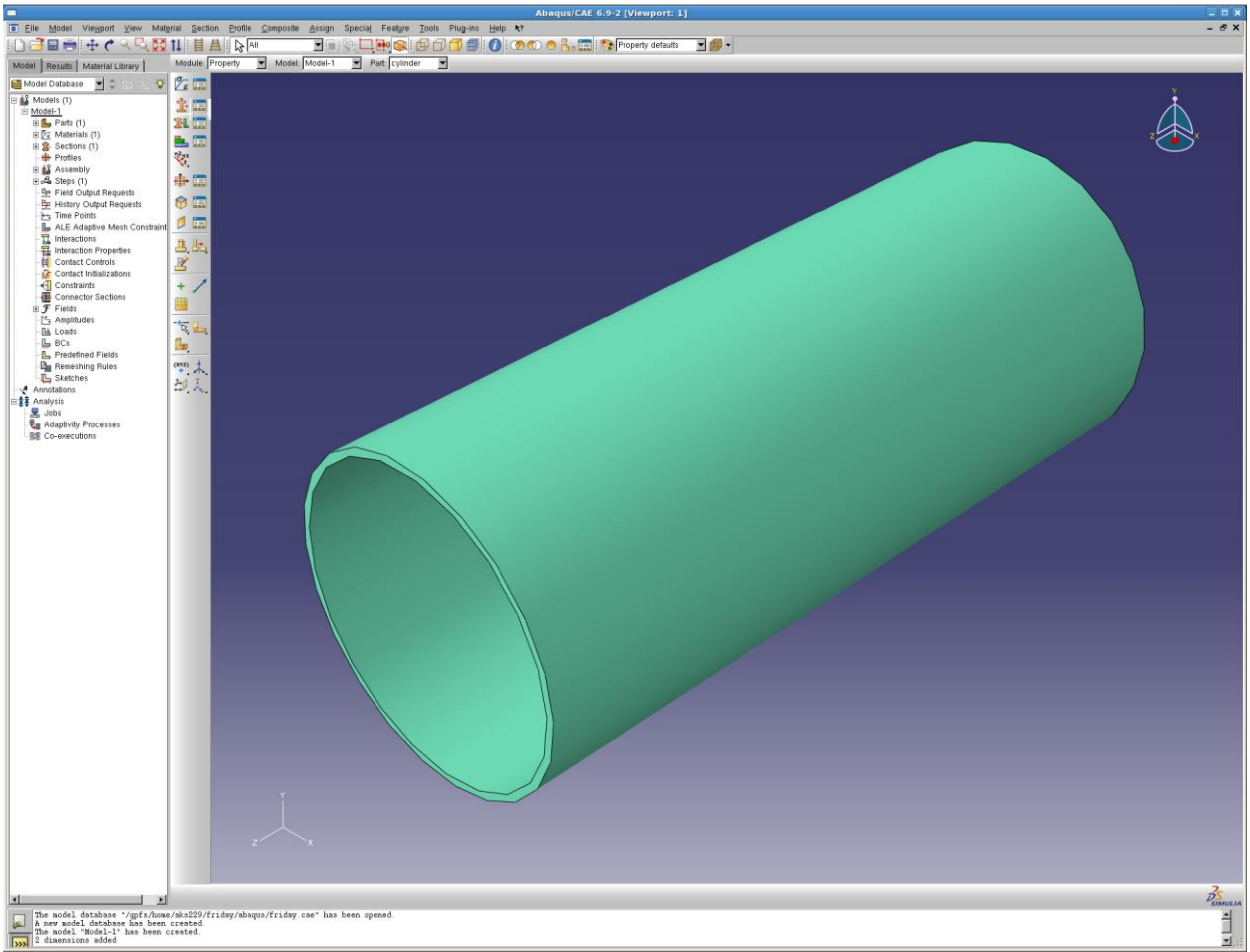


4. Click **Assign Section** on the property toolbox and select the part from the viewport. Click **Done** below. Select the section you had created and click **OK**.





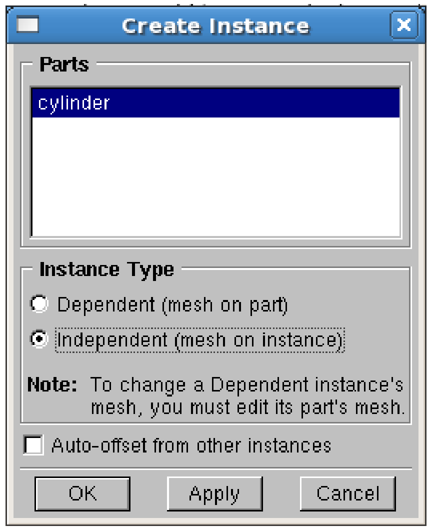
Our work in the Property module is done and we select the **Assembly Module** from the toolbar above the viewport.



Assembly Module:

This module allows you to assemble together parts that you have created. Even if you have a single part you need to include it in your assembly.

1. Select **Instance Part** from the **Assembly Toolbox.**
2. Select the part you have created from the parts list and then select **Instance type: Independent**. Click **OK.**

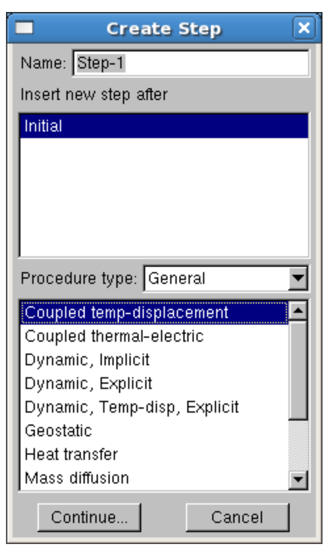


Select **Module: Step** from the toolbar above.

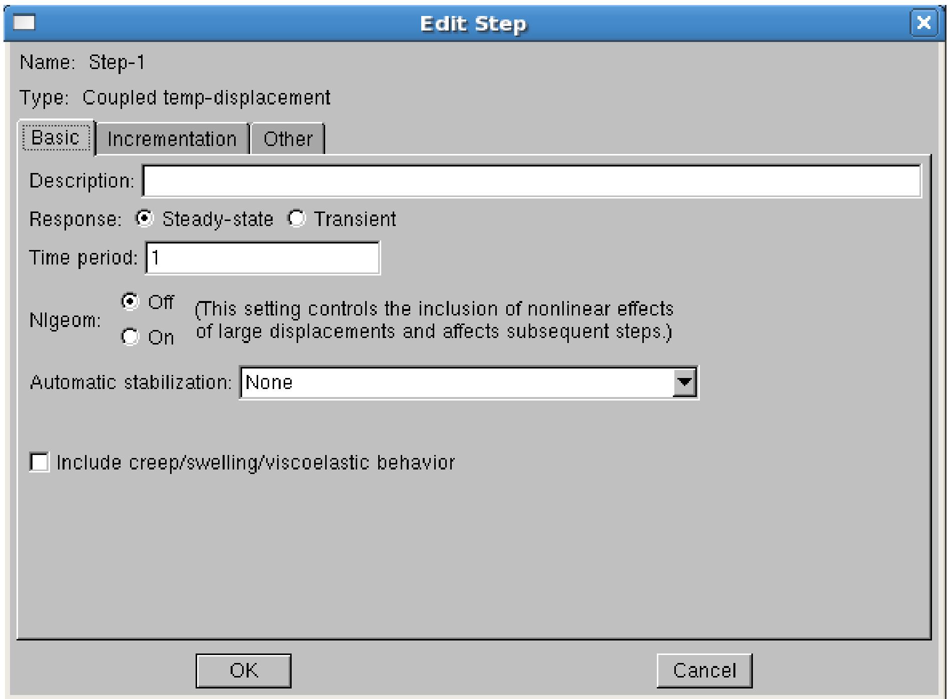
Step Module:

This module allows you to select the kind of analysis you want to perform on your model and define the parameters associated with it. You can also select which variables you want to included in the output files in this modules. You apply loads over a step. To apply a sequence of loads create several steps and define the loads for each of them.

1. Select **Create Step** from the **Step Toolbox.**
2. Name the step as you want and select **Coupled temp-displacement** as the procedure. Click **Continue.**



3. The edit step dialog box lets you choose the solution technique, the solver type and define the time stepping strategy.
4. Under **Basic** change the Response to **Steady-state** and click **OK.**

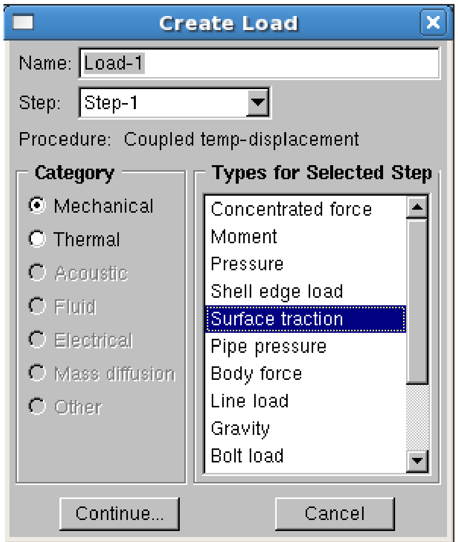


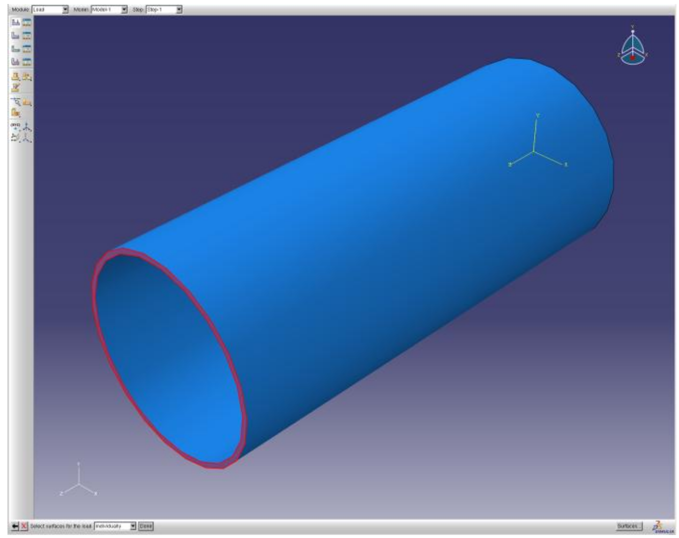
The **Interaction Module** allows you to set up interactions (contact, film), constraints, connectors, fasteners and wire feature between parts. Our problem does not involve any of these features but it will be a good idea to explore this module on your own at a later time. Select **Module: Load** from the toolbar above.

Load Module:

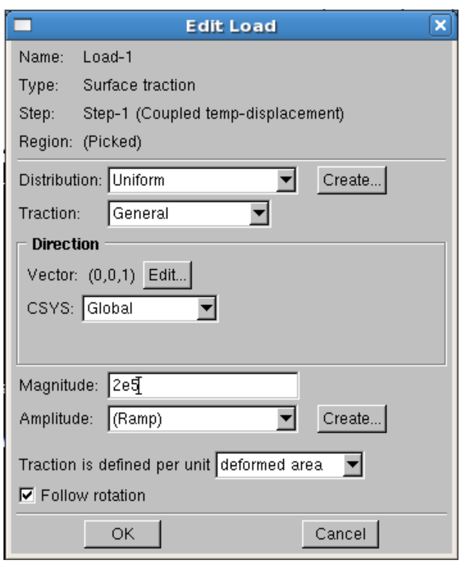
The Load Module is where you define the loads and boundary conditions for your model for a particular step (indicated in the toolbar above). You can even define loads and boundary conditions as fields like electric potential, acoustic pressure, etc.

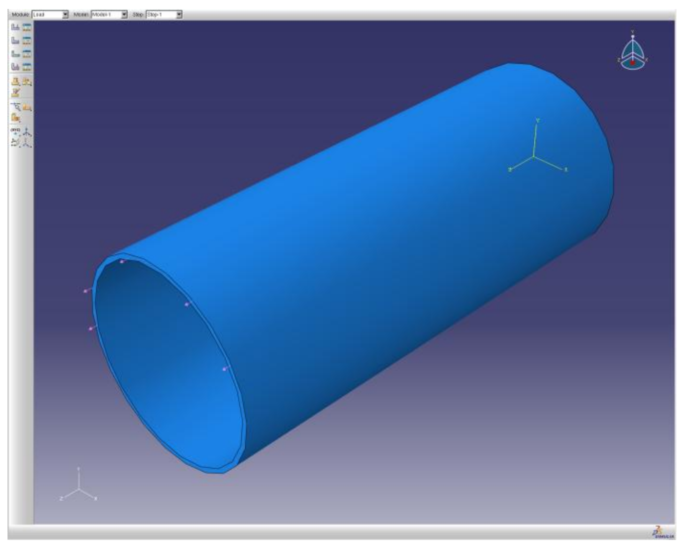
1. Select **Create Load** from the **Load Toolbox**. Select **Surface Traction** and click **Continue.** Select the top face of the cylinder (z=1) (it gets highlighted in red) and click **Done**.



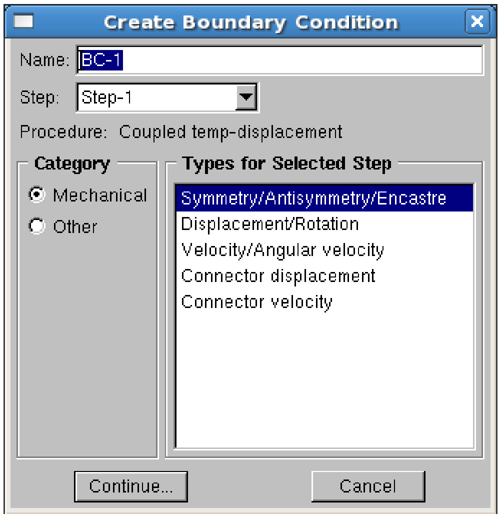


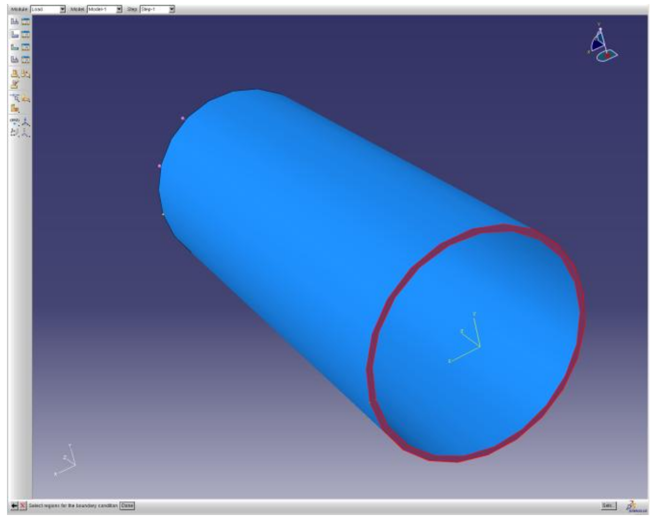
2. Change the **Traction** type to **General**. Click on the **Edit** tab under **Direction** in the dialog box. Enter the starting point of the direction vector as (0, 0, 0) and the end point as (0,0, 1). Enter the **Magnitude** as 2e5 and click **OK.**

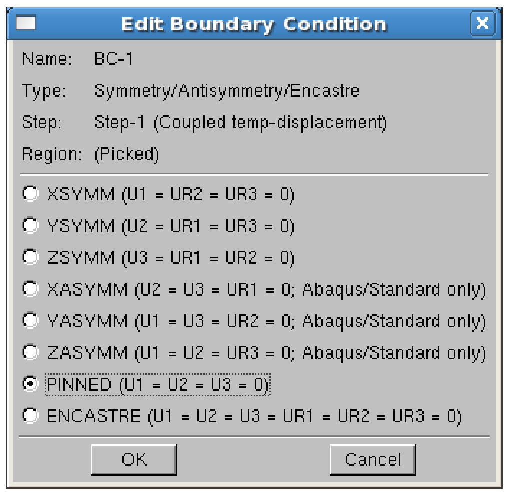




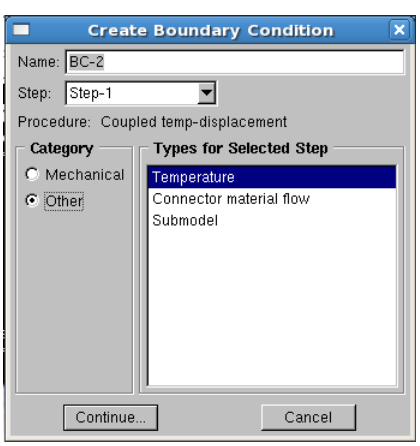
3. Select **Create Boundary Condition** from the **Load Toolbox**. Select **Symmetry / Antisymmetry / Encastre** and click **Continue.** Select the bottom face (z=0) of the cylinder and click **Done.** Select **Pinned** (U1=0, U2=0, U3=0) and click **OK.**

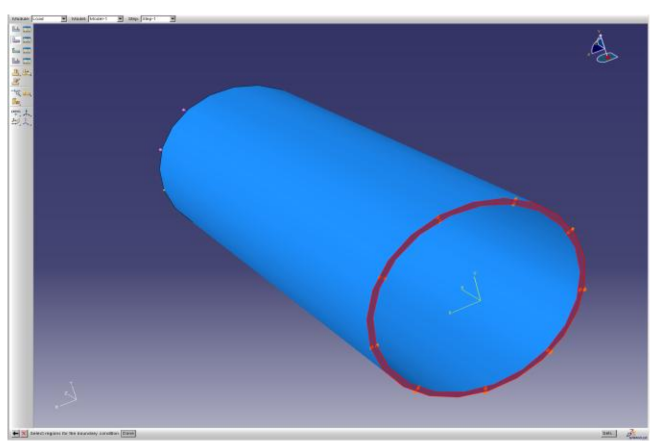


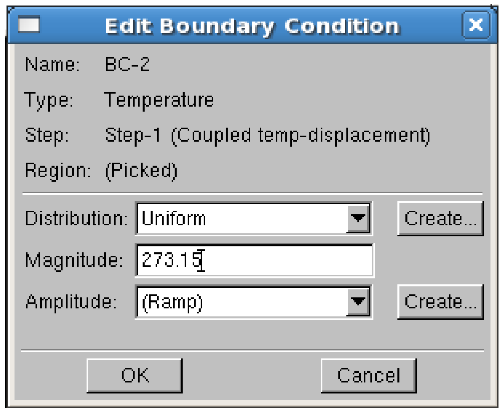


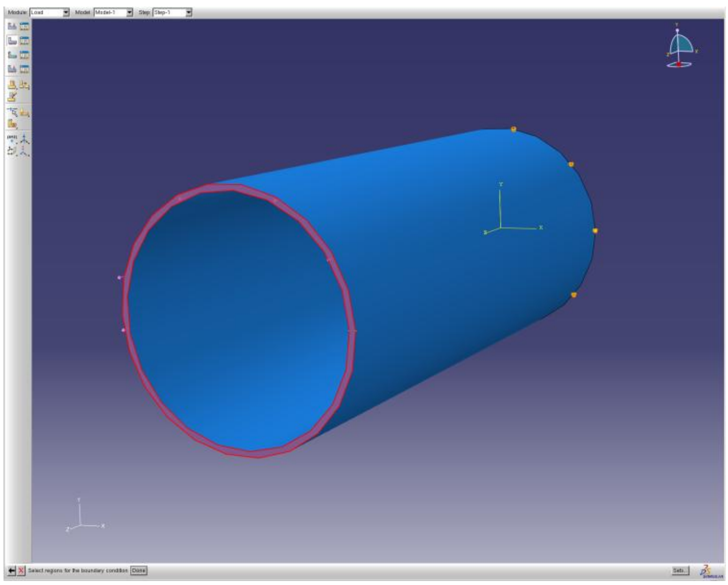


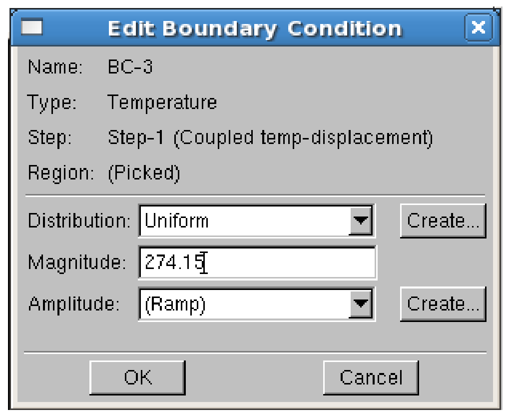
4. Again select **Create Boundary Condition** from the **Load Toolbox.** Switch Category to **Other** and select **Temperature** and click **Continue**. Select the bottom face of the cylinder and press **Done**. Enter the magnitude as 273.15 and click **OK**. Similarly put the top face at 274.15.









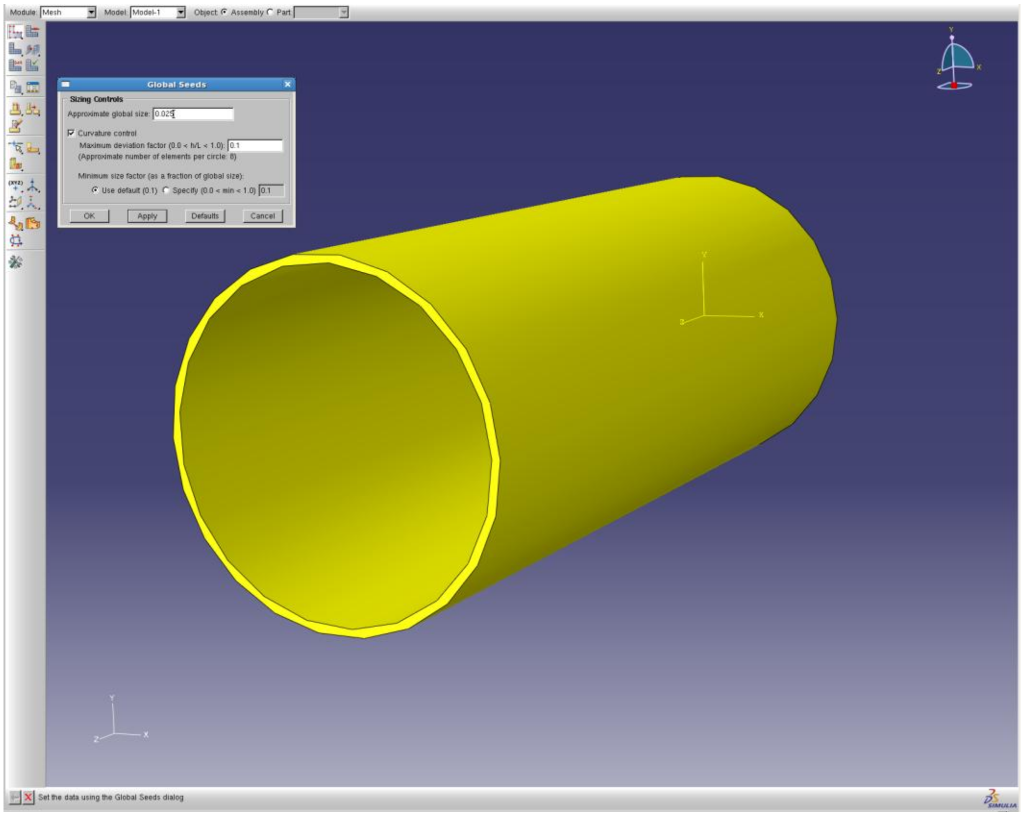


Now that we have defined the loads and the boundary conditions we move on to mesh the geometry. Select **Module: Mesh** from the toolbar above the viewport.

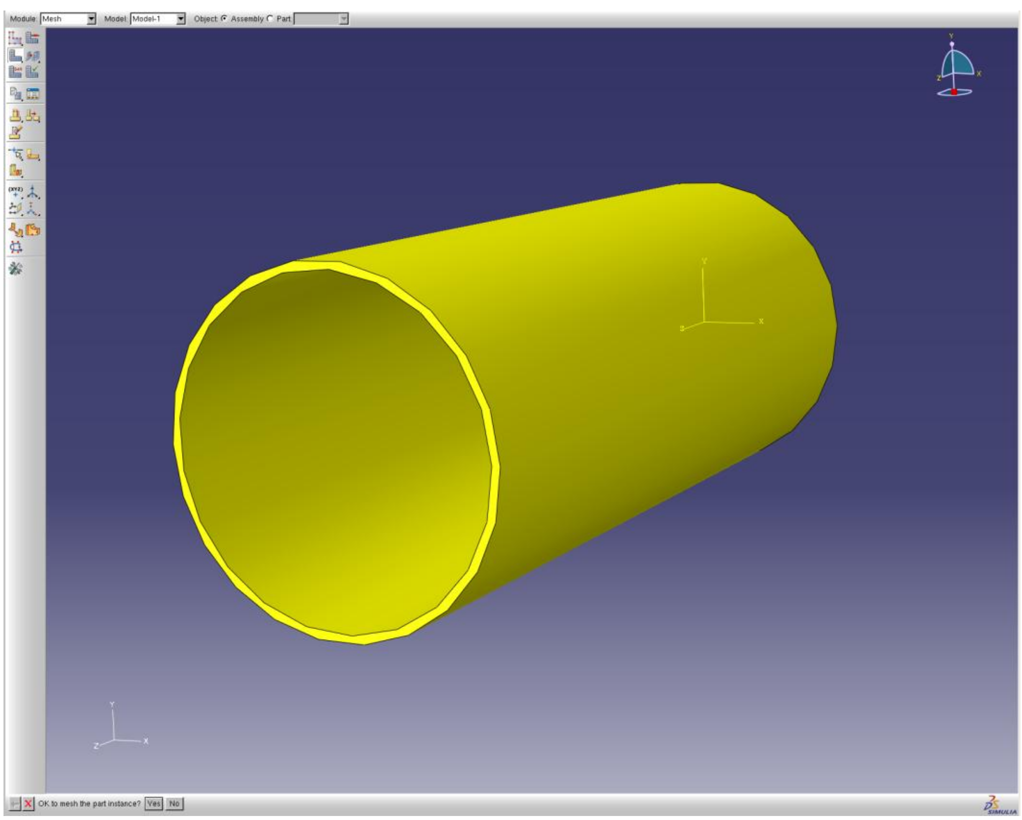
Mesh Module:

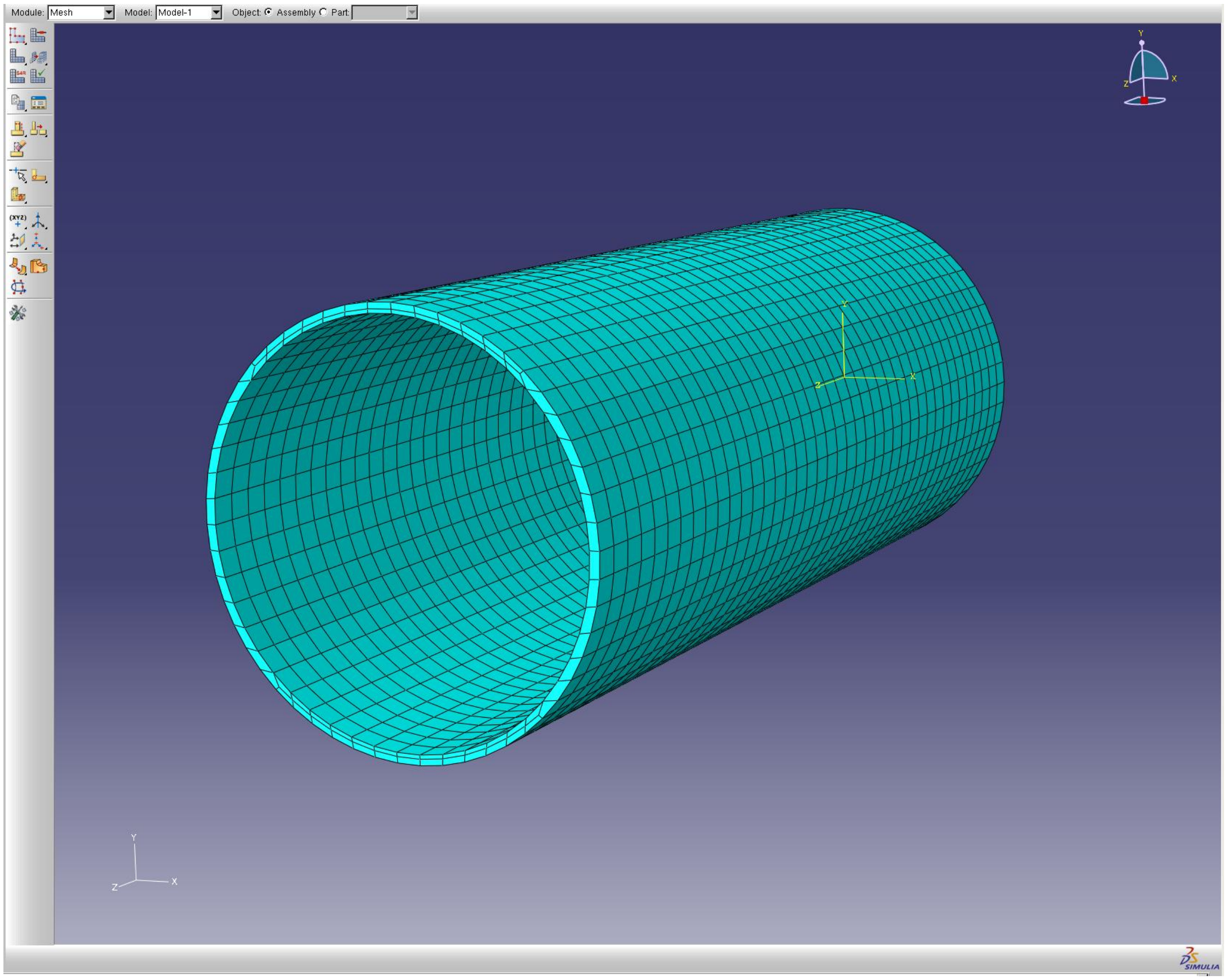
The mesh model controls how you mesh your model – the type of element, their size etc.

1. Select **Seed Part Instance** from the mesh toolbox. Enter the approximate global size as 0.025.

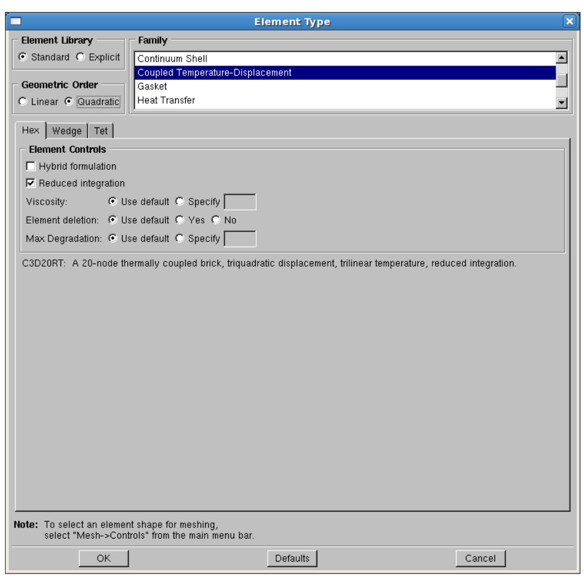


2. Click on **Mesh Part Instance** and then on **Yes** to mesh the model.





3. Select **Assign Element Type** from the mesh toolbox. Under **Family** select Coupled Temperature-Displacement and switch **Geometric Order** to Quadratic. Click **OK**.

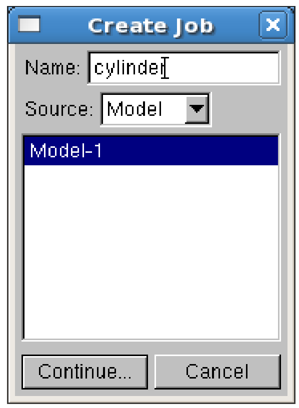


When finished select **Module: Job** from the toolbar above.

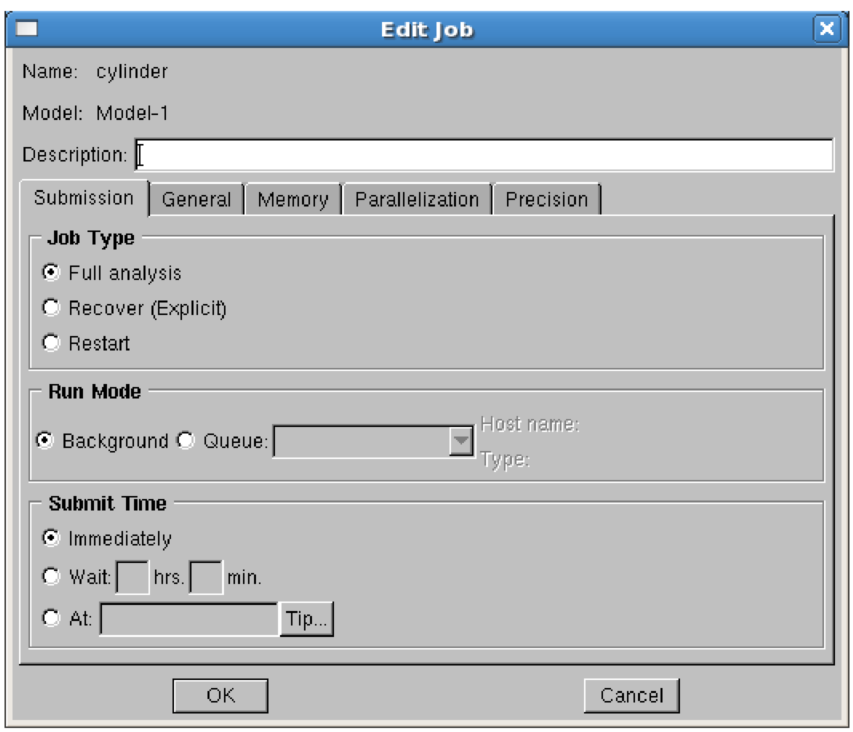
Job Module:

This module allows you to submit your model for analysis.

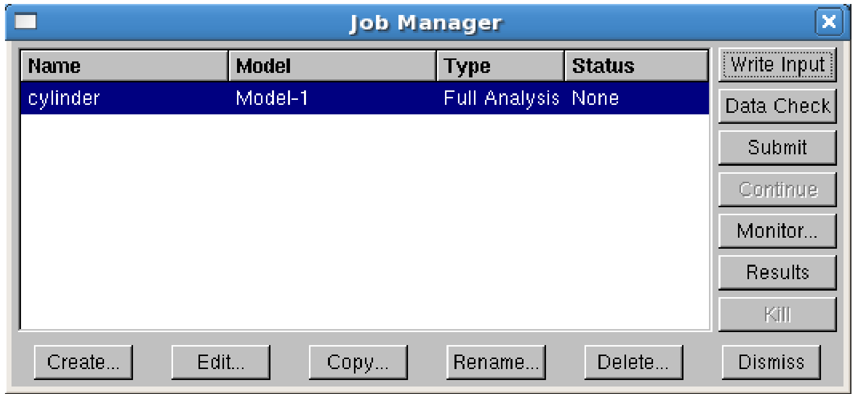
1. Select **Create Job** from the **Job Toolbox**. Name the job as you like. Select your model and click **Continue**.



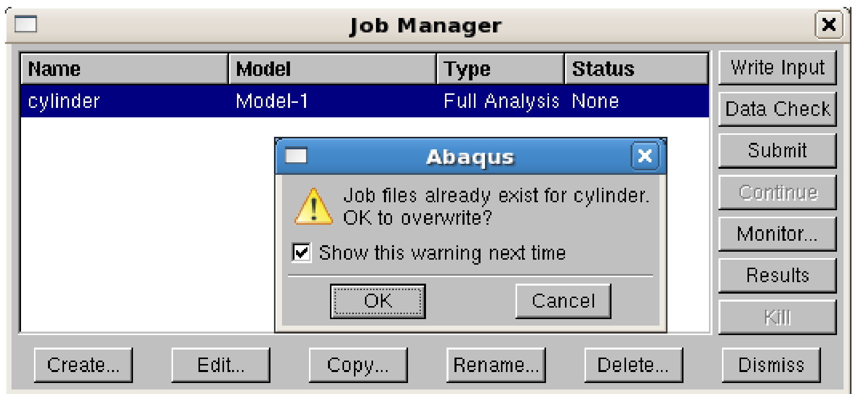
2. You can add a description to the job, allocate memory, allot multiple processors and select precision. Use the default values and click **OK.**



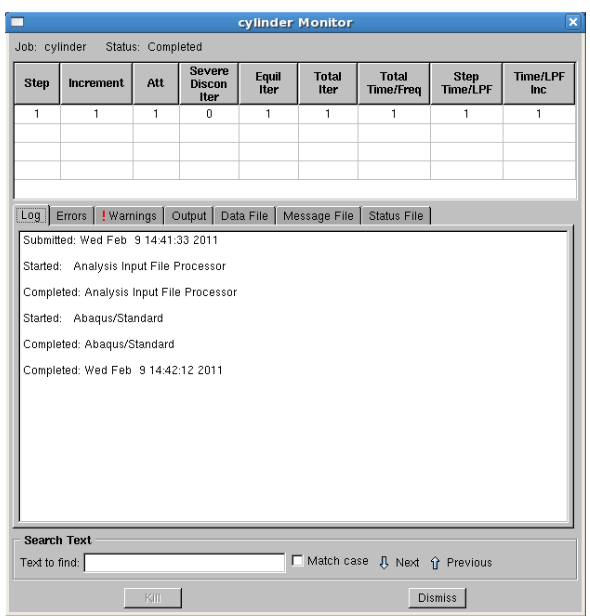
3. Select the **Job Manager** from the toolbox and click on the **Write Input** tab.



4. If you are running the job for the first time it is advisable to run **Data Check** to check the input file for errors. Click OK to overwrite the job files.



5. Once the data check is completed **Submit** the job for analysis. Click OK to overwrite the job files. You can click **Monitor** to observe the progress of the solution process. You can see the errors, warnings, data and message file.

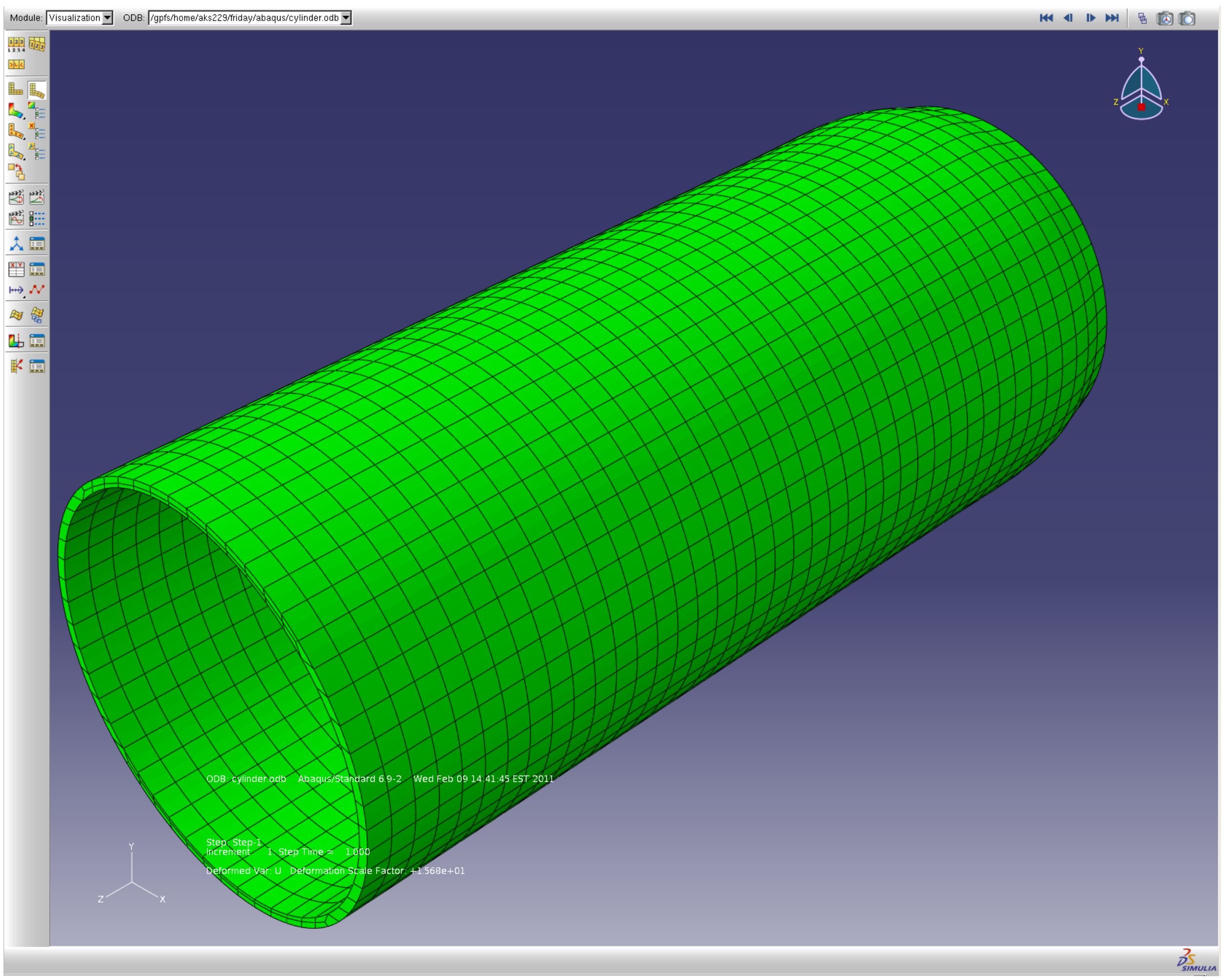


6. Once the job is completed click on the **Results** tab on the job manager. This opens the **Visualization Module** for postprocessing.

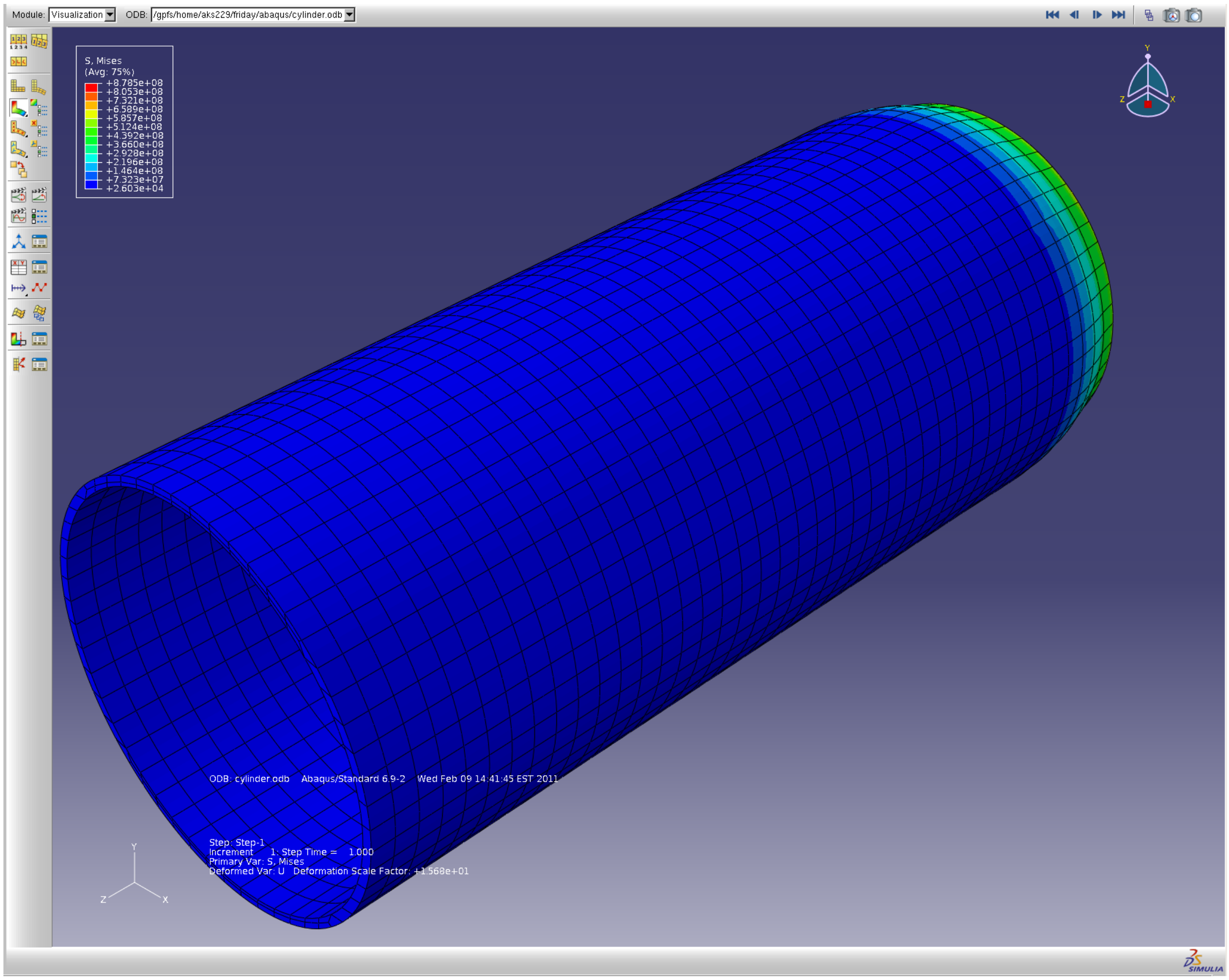
Visualization Module:

This model allows you to look at your model after deformation. You can also plot values of stress, displacement, reaction forces, etc. as contours on your model surface or as vectors or tensors.

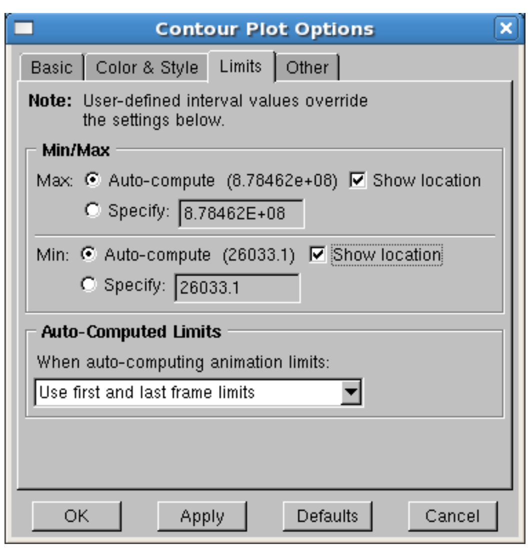
1. Select **Plot Deformed Shape** from the Visualization toolbox.

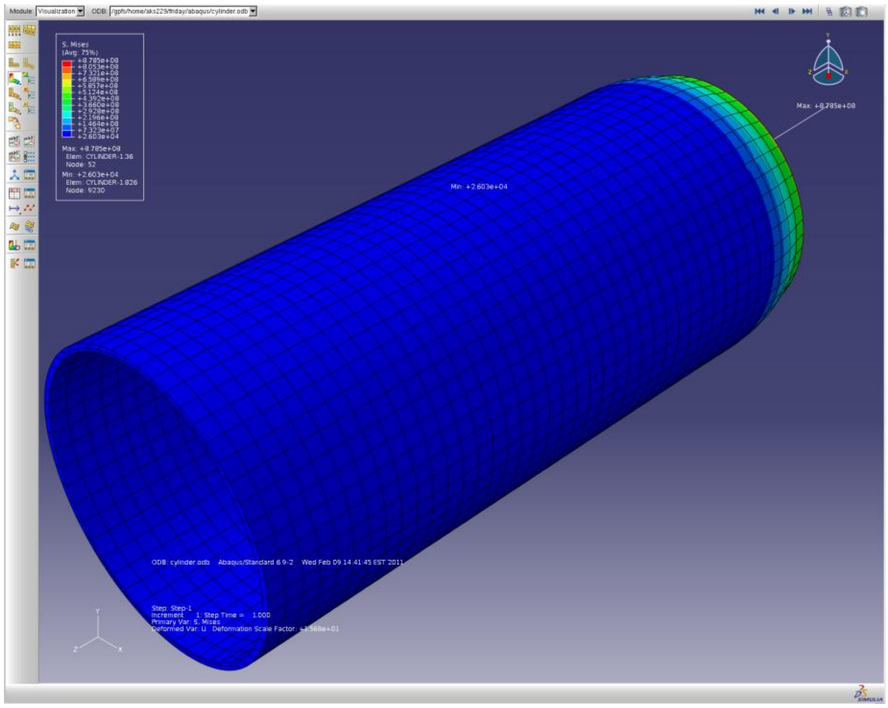


2. Select **Plot Contours on Deformed Shape** to plot stress contours on the model surface.

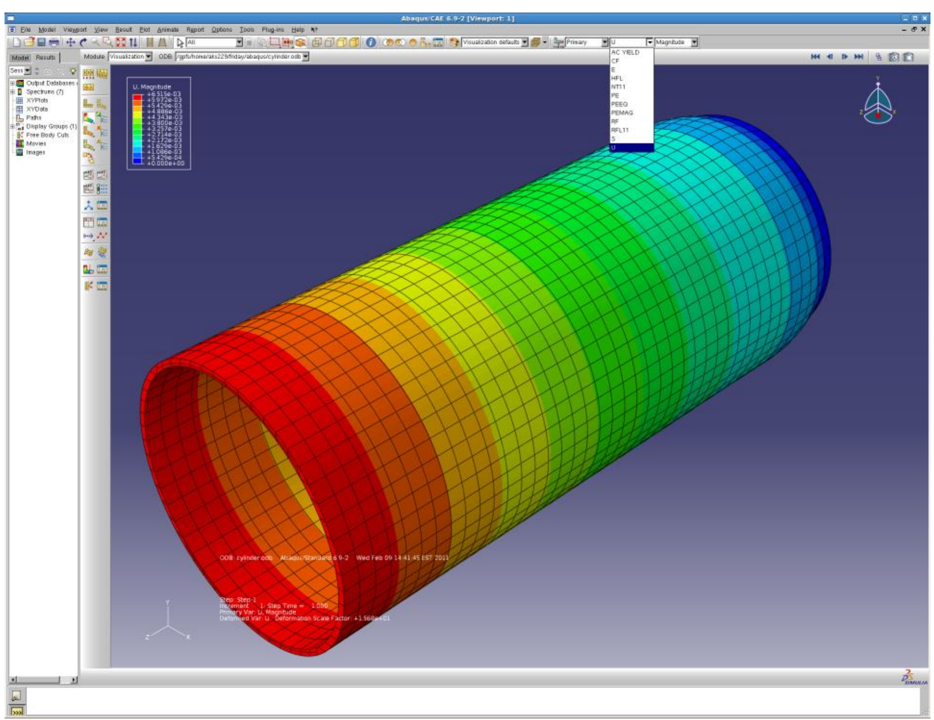


3. You can see the location of the maximum & minimum stresses by selecting **Contour Options>Limits>Show Location.**

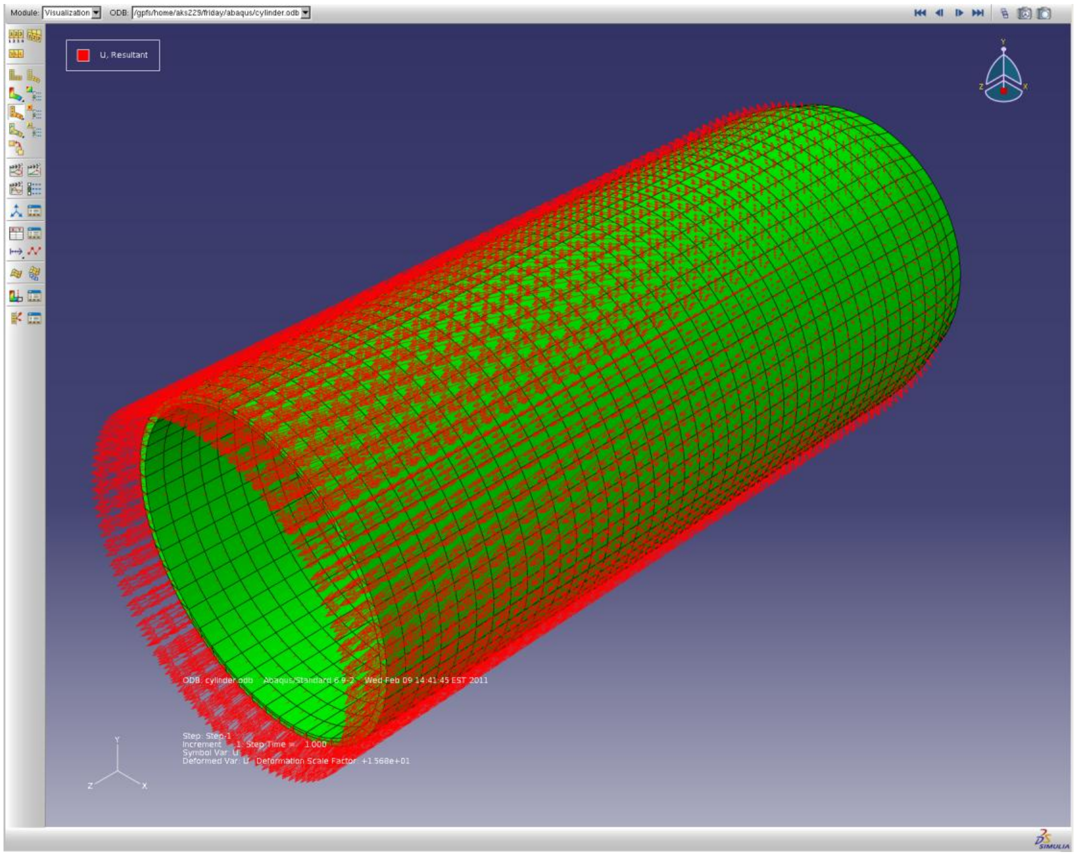




4. Select **Results>Field Output** from the main menu. This opens a dialog box that allows you to select the variable you want to plot in the viewport.
5. Select **U** (Spatial Displacement at nodes)**>Magnitude>OK** to plot the displacement contours on the model.



6. To plot displacement vectors click on **Plot Symbols on Deformed Shape** on the toolbox**.**



7. You can now animate this plot by selecting **Animate Harmonic.**

Mouse Gestures:

Ctrl+Alt+Left Click (MB1): Rotate View
Ctrl+Alt+Middle Click (MB2): Pan View
Ctrl+Alt+Right Click (MB3): Magnify View
Use Shift key to select multiple objects.

Note on System of Units:
ABAQUS has no built-in system of units. Specify all unit data in consistent units. Some common systems of consistent units:
SI: m, N, kg, s, Pa, J, kg/m^3
SI (mm): mm, N, tonne (1000 kg), s, MPa, mJ, $tonne/mm^3$
US Unit (ft): ft, lbf, slug, s, lbf/ft^2, ft lbf, $slug/ft^3$