Exercise 15

In this exercise we will analyze a cantilever beam shown schematically in the figure below which is loaded by a force on the tip. The results will be compared when the force is treated as a follower and a non-follower load.

Pick **File > Set Work Directory** and set the work directory to the **BeamCantilever** folder

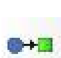
Open the model database **Beam.cae**. It will appear as shown below.

The model consists of a single part, Beam. The beam shown in above figure has a length of 0.6m and a square cross section of 20mm x 20mm. As the cross-section dimensions are considerably smaller than the axial length, it is modeled as one-dimensional line. It is assumed to be made from steel with a Young's modulus of 200GPa and a Poisson's ratio of 0.3. It is meshed with 3-node quadratic beam elements (B22). The beam is loaded with a concentrated force of 6000 N in the negative y-direction at right end node.

Defining Step

We assume that load is applied slowly such that inertia effects can be neglected. So analysis will be performed using the Static, General procedure.

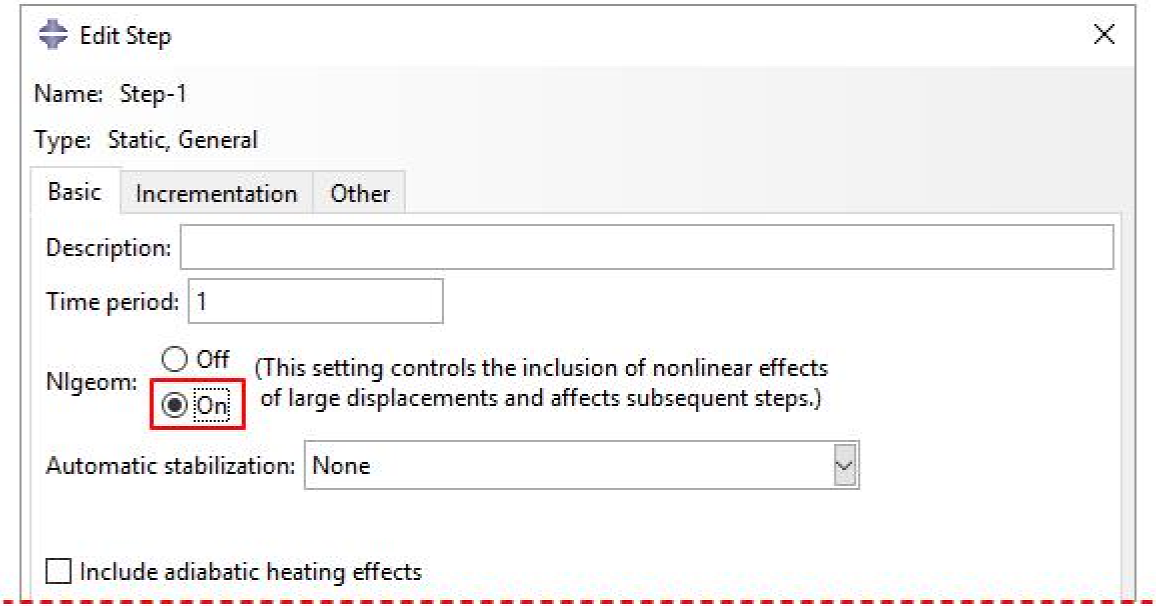
Change to **Step** module.

Pick  to create a new step and select the **Static, General** step.

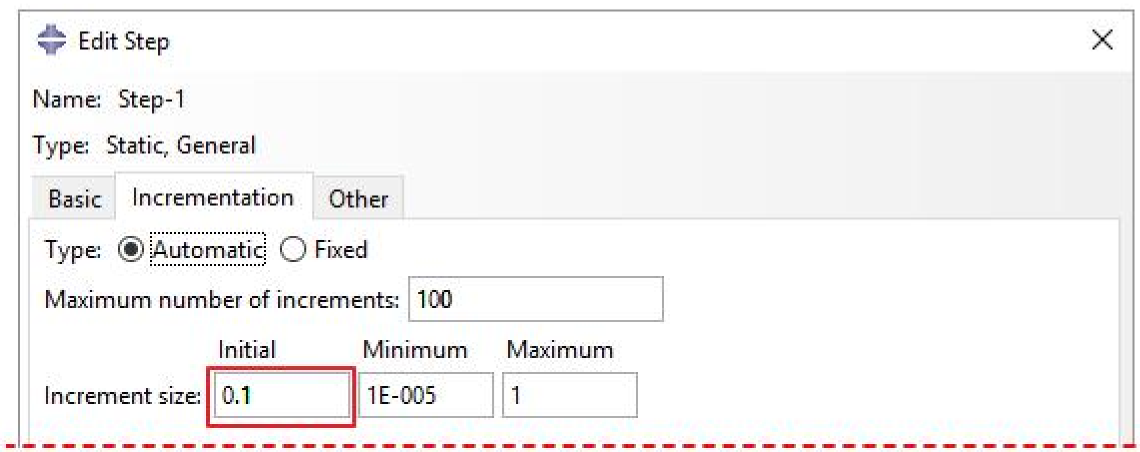
Pick **Continue** and Edit Step dialog box will appear.

We will perform a geometrically nonlinear analysis as large deformations are expected.

So turn on the Nlgeom option as shown below.



Under the Incrementation tab, set the initial increment size to **0.1**

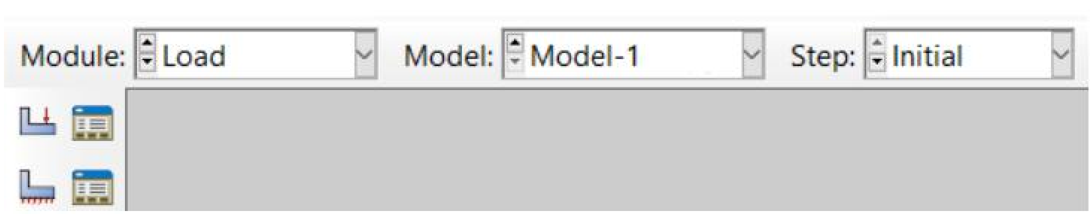


Pick **OK** to complete the definition of step.

Defining Load

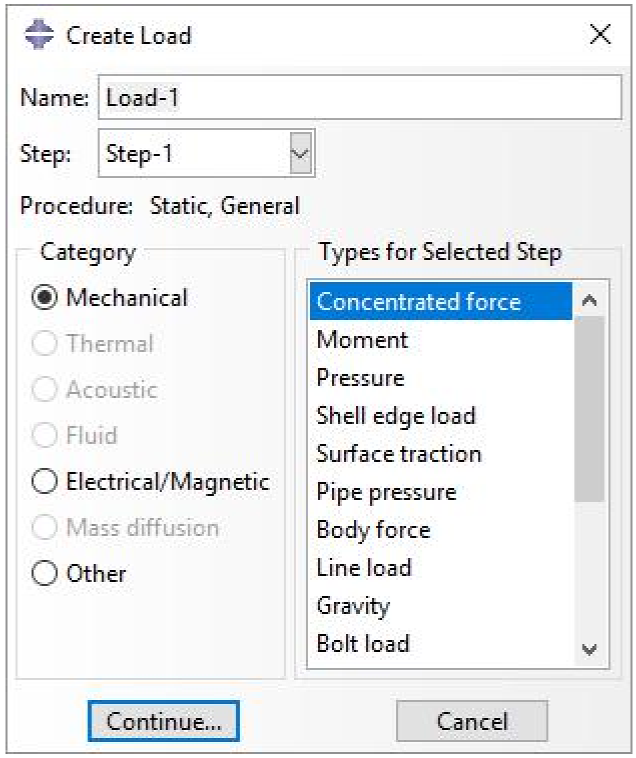
A concentrated force will be applied on the right end node of the beam.

To define load, change to **Load** module.

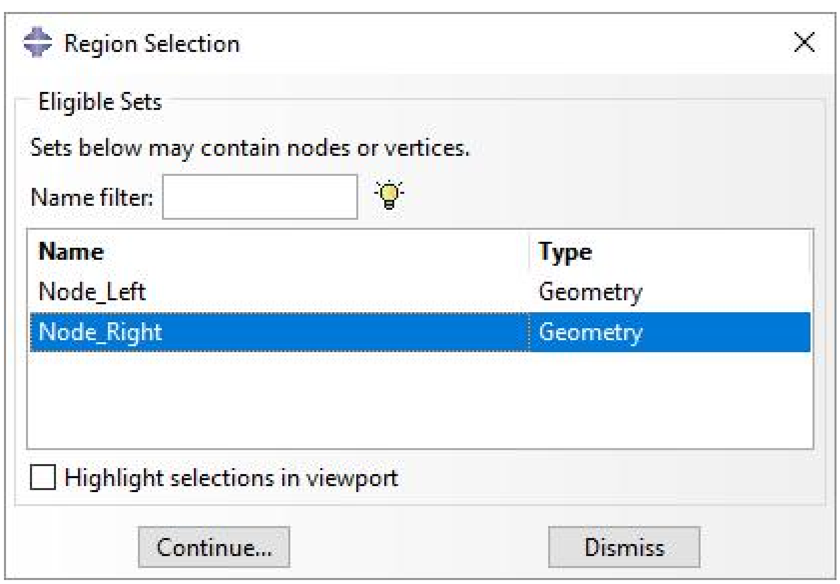


Pick  to create a load and select Step-1 in the Step field.

Select **Concentrated** in the Types field and pick **Continue.**

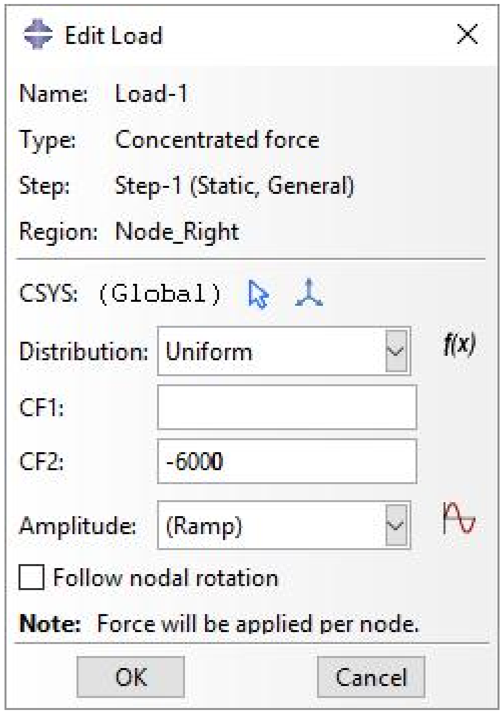


Now the system will ask to select points for the load. We have already defined a set for that purpose, so pick **Sets** and select the Node_Right.



Pick **Continue** and Edit Load dialog box will appear.

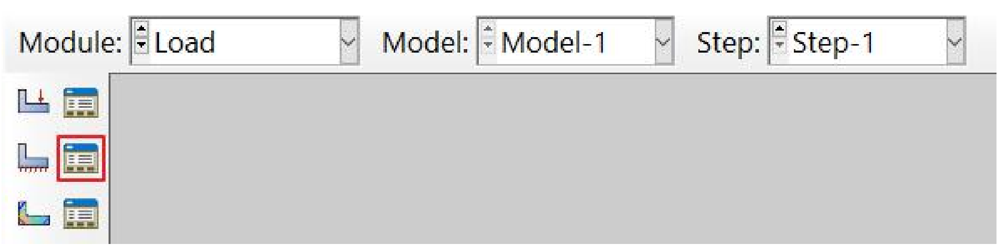
Enter **-6000** as the force magnitude in y-direction (CF2).



Pick **OK** and it completes the definition of load.

Boundary Conditions

Boundary conditions required for the analysis have already been defined. To review all the boundary conditions pick  while the Load module is active as shown in the figure below.



The boundary condition "BC-Left" constrains translational and rotational degrees of freedom of left node of the beam.

Pick **Dismiss** to close the manager.

Job Submission

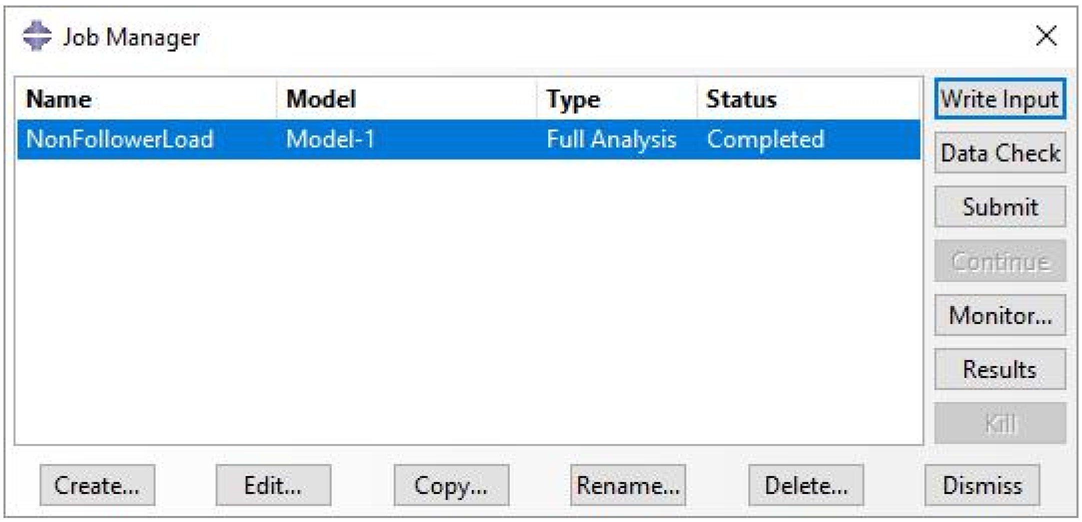
All the information required for analysis has been set up in the model. Now we can submit the job for analysis.

So change to **Job** module and pick  to open the job manager.

Pick **Create** and create a job named NonFollowerLoad or whatever name you would like.

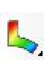
Pick **Continue** and then **OK**.

Pick **Submit** to submit the job for analysis.



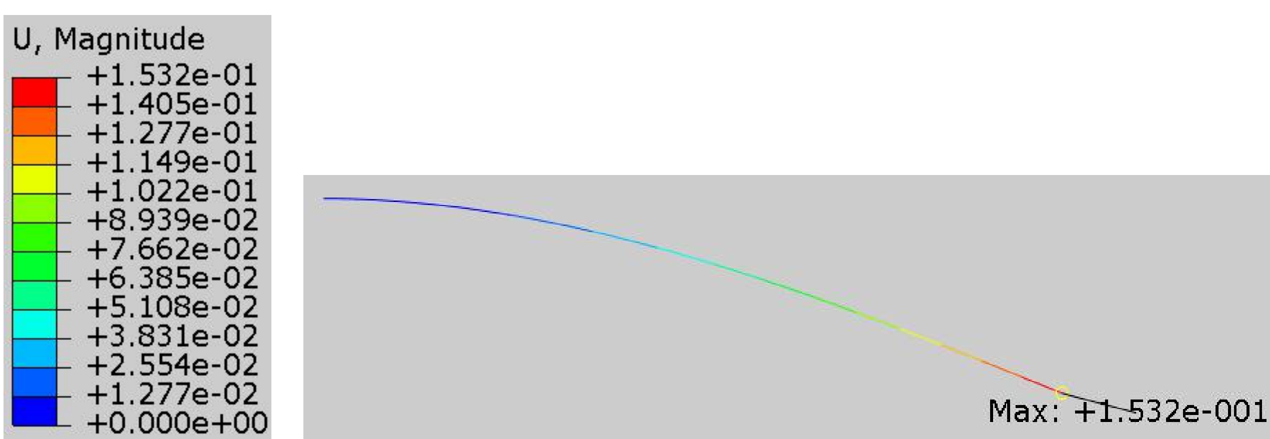
Pick **Results** to view the results in Visualization module.

Postprocessing

Pick  and select **U** as output variable in the Field Output toolbar.

Pick  to open Contour Plot Options dialog box and check the **Show location** field for Max limit.

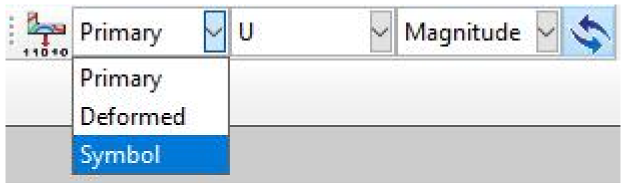
The simulation results will appear as shown below (Scale factor is set to 1).



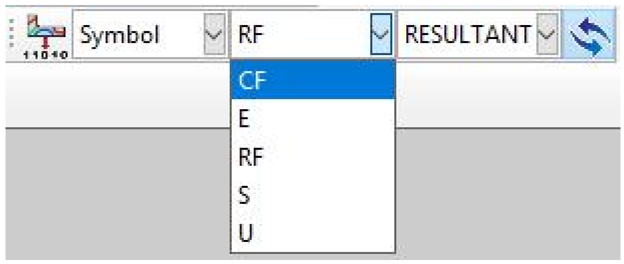
It can be seen that tip deflection is approximately 153.2mm.

Now we will generate symbol plot of the applied force vector to visulaize the changes during the step.

Select **Symbol** as category in the Field Output toolbar.



Select **CF** as output variable and make sure that **RESULTANT** is selected in component field.



The simulation results will appear as shown below (Scale factor is set to 1).

It can be seen that direction of the applied force stays parallel to the y-axis. Remember that the force was applied in the negative y-direction. This constant-direction load is called a non-follower load.

By default, direction of a concentrated force does not rotate with the node to which it is applied even when NLgeom option has been toggled on.

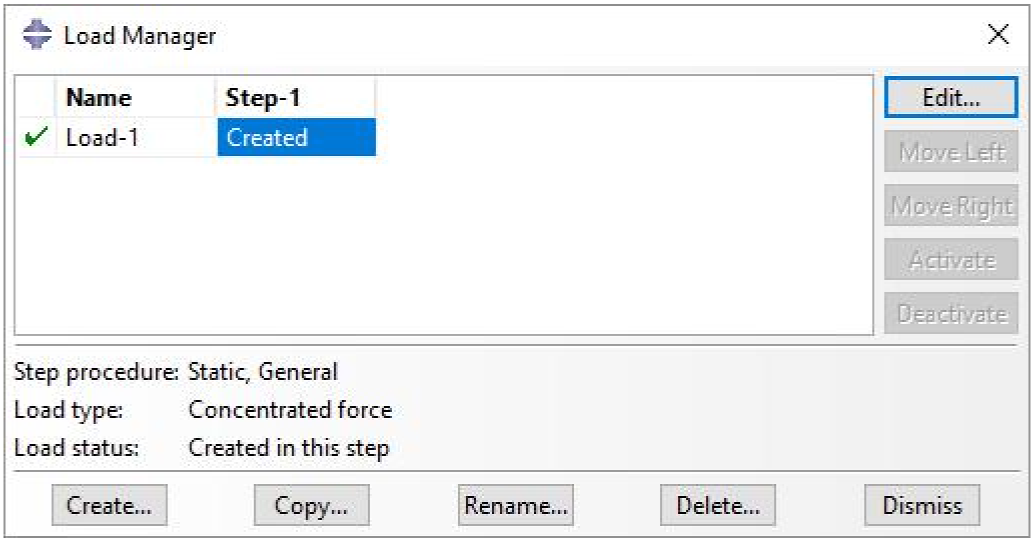
Now we will perform another analysis and specify that the applied force rotates with the node to which it is applied.

Specifying Follower Force

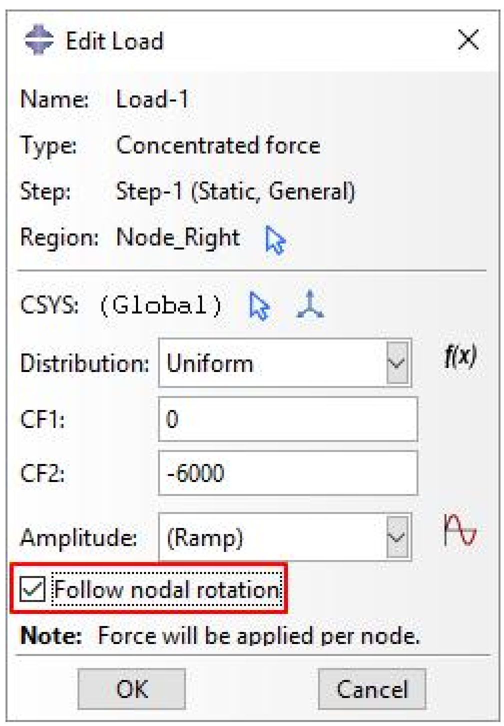
To edit the load, change to **Load** module.

Open the Load Manager by picking  .

Pick the Load-1 and select **Edit**.



Check the **Follow nodal rotation** checkbox as shown below.



When activated, this makes sure that the direction of a concentrated force rotates with the node to which it is applied. This can be used only when NLgeom option has been toggled on and can be used only at nodes with active rotational degrees of freedom (such as the nodes of beam and shell elements).

Pick **OK** to apply the changes and exit the dialog box.

Job Submission

All the information required for analysis has been set up in the model. Now we can submit the job for analysis.

So change to **Job** module and pick  to open the job manager.

Pick **Create** and create a new job named FollowerLoad or whatever name you would like.

Pick **Continue** and then **OK**.

Pick **Submit** to submit the job for analysis.

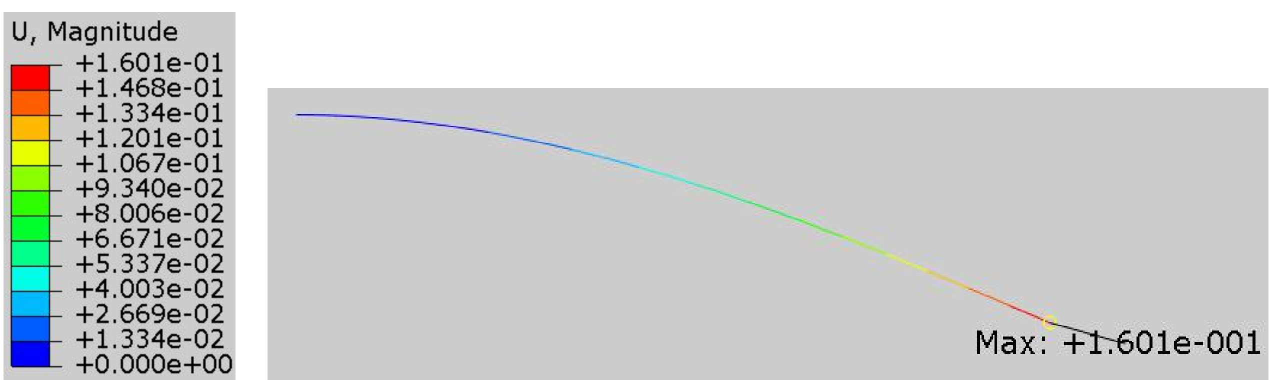
Pick **Results** to view the results in Visualization module.

Postprocessing

Pick  and select **U** as output variable in the Field Output toolbar.

Pick  to open Contour Plot Options dialog box and check the **Show location** field for Max limit.

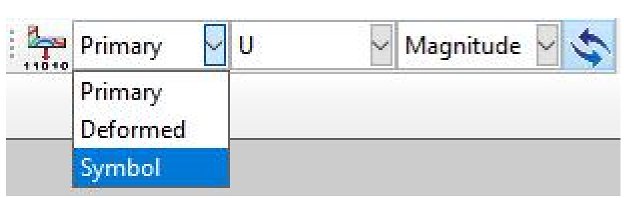
The simulation results will appear as shown below (Scale factor is set to 1).



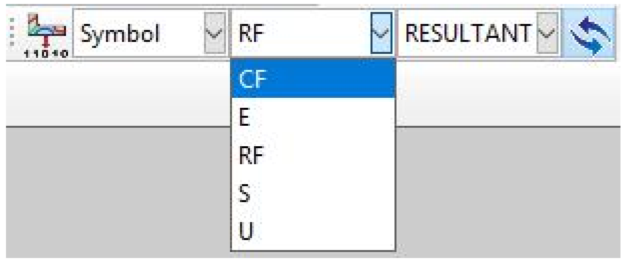
It can be seen that tip deflection is approximately 160.1mm which is larger than the previous case.

Now we will generate a symbol plot of the applied force vector to visulaize the changes during the step.

Select **Symbol** as category in the Field Output toolbar.



Select **CF** as output variable and make sure that **RESULTANT** is selected in component field.



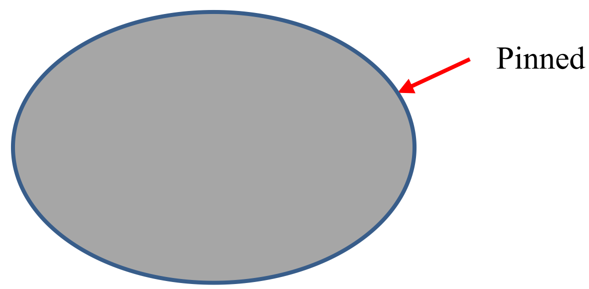
The simulation results will appear as shown below (Scale factor is set to 1).

It can be seen that direction of the applied force does not stay parallel to the y-axis instead it is constantly rotating with the node of the beam. In the undeformed configuration, the applied force vector is normal to the beam at the applied node. In the deformed configuration, force vector maintains this relationship and stays normal to the geometry of the beam. This changing-direction load is called a follower load.

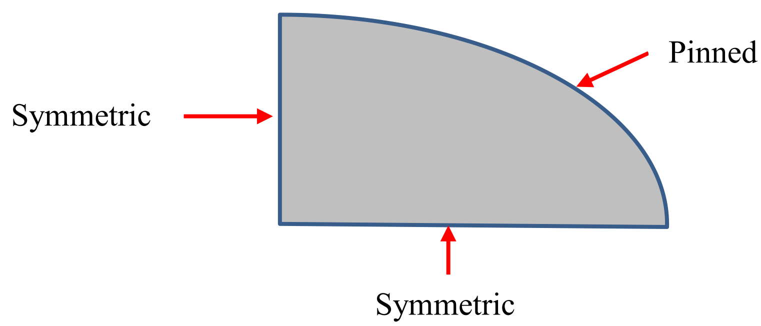
Select **File > Save** to save the changes we made so for.

Exercise 16

In this exercise we will analyze a rubber disc subjected to pressure loading and pinned on boundary edges. The disc is shown schematically in the figure below.



As the geometry is symmetric, so we will analyze only a quarter of the model using the symmetry boundary conditions as shown below.



Pick **File > Set Work Directory** and set the work directory to the **DiscBulging** folder

Open the model database **RubberDisc.cae**.

The model consists of a single part, Disc. The disc shown in above figure has a length of 0.4 m and a width of 0.27 m. It is assumed to be made from rubber. Rubber is best modeled as a hyperelastic material but we will use linear elastic model with a Young's modulus of 6 MPa and a Poisson's ratio of 0.49 as an approximation. The disc is 0.01 m thick and modeled using planar shell feature. We assume that strains will be large, so it is meshed with 4-node shell elements with reduced integration (S4R). The disc is loaded with a uniform pressure of 250 kPa.

⇨ Defining Step

We assume that load is applied slowly such that inertia effects can be neglected. So analysis will be performed using the Static, General procedure.

The step has already been defined with a total time period set to **1.0.** Nlgeom option has been set to Off. The incrementation has been set to **Automatic** type with an initial increment size of **0.01** and a maximum increment size of **0.04**

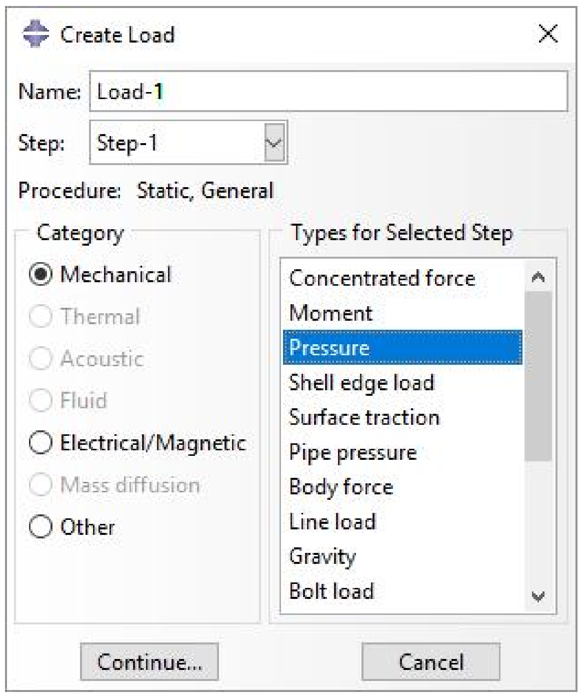
Defining Load

A uniform pressure load will be applied across the bottom of the disc.

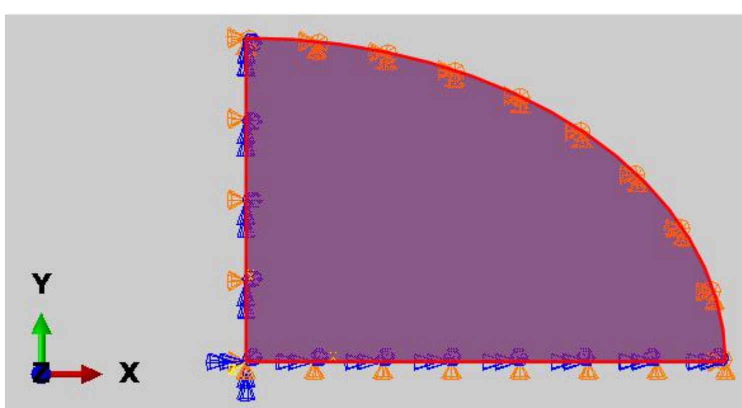
To define load, change to **Load** module.

Pick  to create a load and select Step-1 in the Step field.

Select **Pressure** in the Types field and pick **Continue.**

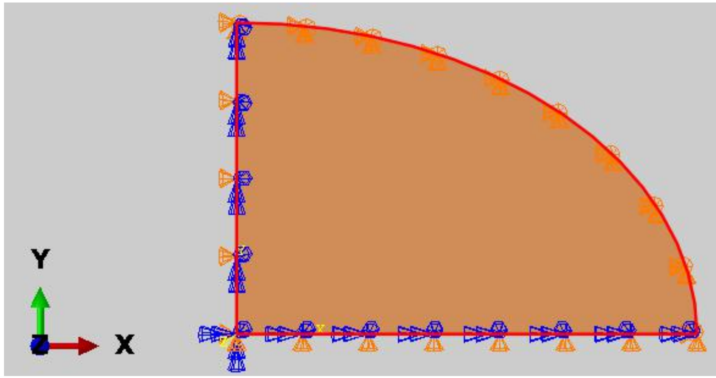


Select the surface of the disc as shown in figure below.

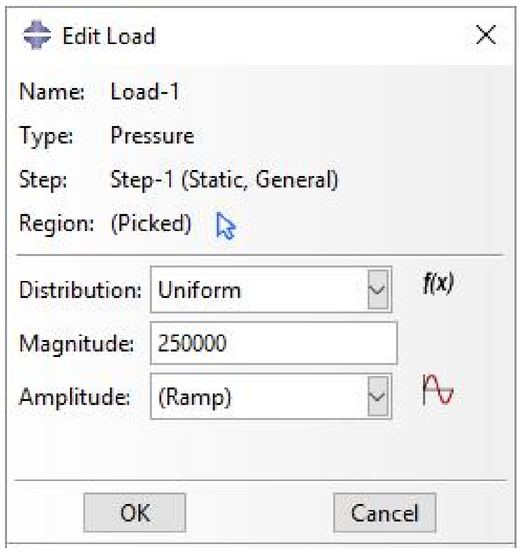


Pick **Done** to accept the selection.

Pick **Purple** to specify the bottom side of the disc for applying pressure load.



Enter **250000** as the pressure magnitude.



Pick **OK** and it completes the definition of load.

Boundary Conditions

Boundary conditions required for the analysis have already been defined. In the Boundary Conditions Manager it can be seen that three boundary conditions have been created.

The boundary condition “BC-Pinned” constrains all the translational degrees of freedom of the outer edge of the disc. The boundary condition “BC-SymmX” applies a symmetric constraint to the vertical edge of the disc. The boundary condition “BC-SymmY” applies a symmetric constraint to the horizontal edge of the disc.

Field Output Request

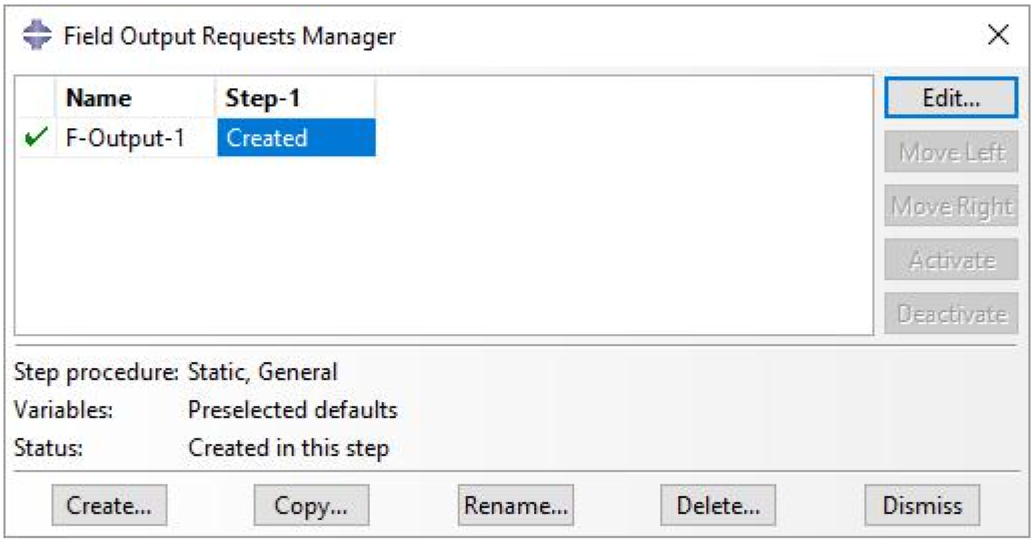
We will plot the section forces, i.e membrane forces and shear forces. This information can be plotted by requesting SF output variable.

SF variable contains all section forces and moment components.

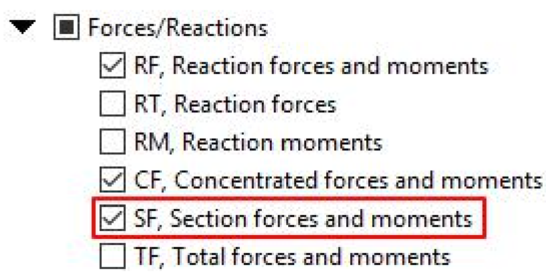
Change to **Step** module.

Open the Field Output Manager by picking  .

Select the F-Output-1 field and pick **Edit** as shown below.

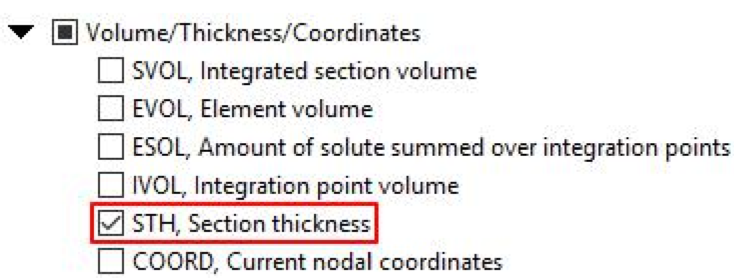


Check the **SF** variable (located under the **Forces/Reactions** container).

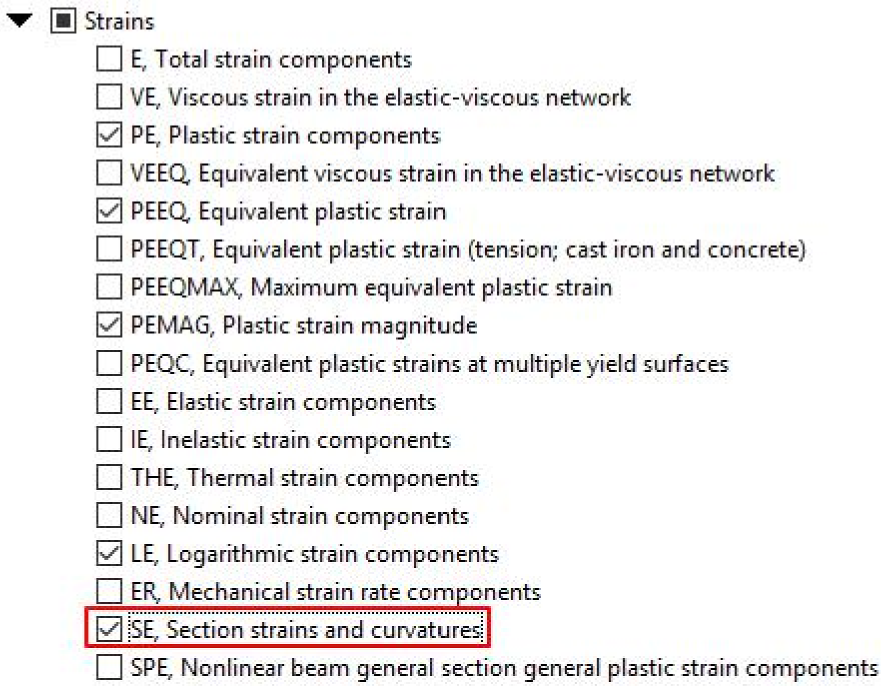


During the bulging of the disc, changes in the thickness are also of interest. The thickness for the shell elements can be monitored with output variable STH. This information is not included by default in the ODB file. We will also request STH variable.

Check the **STH** variable (located under the Volume/Thickness/Coordinates container).



It is also of interest to plot strain in the thickness direction. So check the **SE** variable (located under the Strains container).



Pick **OK** to apply and exit.

Pick **Dismiss** to close the manager.

Job Submission

All the information required for analysis has been set up in the model. Now we can submit the job for analysis.

So change to **Job** module and pick  to open the job manager.

Pick **Create** and create a job named Pressure_Linear or whatever name you would like.

Pick **Continue** and then **OK**.

Pick **Submit** to submit the job for analysis.

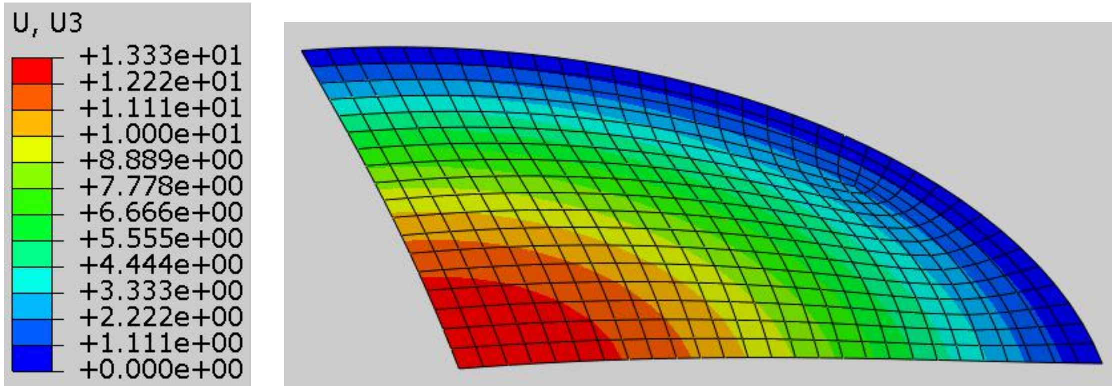
Pick **Results** to view the results in Visualization module.

Postprocessing

Pick  and select **U** as output variable in the Field Output toolbar.

Pick **U3** in the component field.

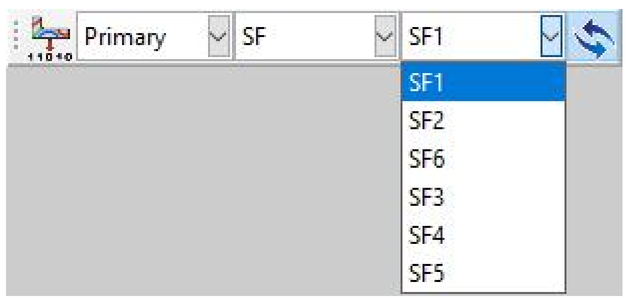
The simulation results will appear as shown below (Deformation scale factor is set to 0.0015).



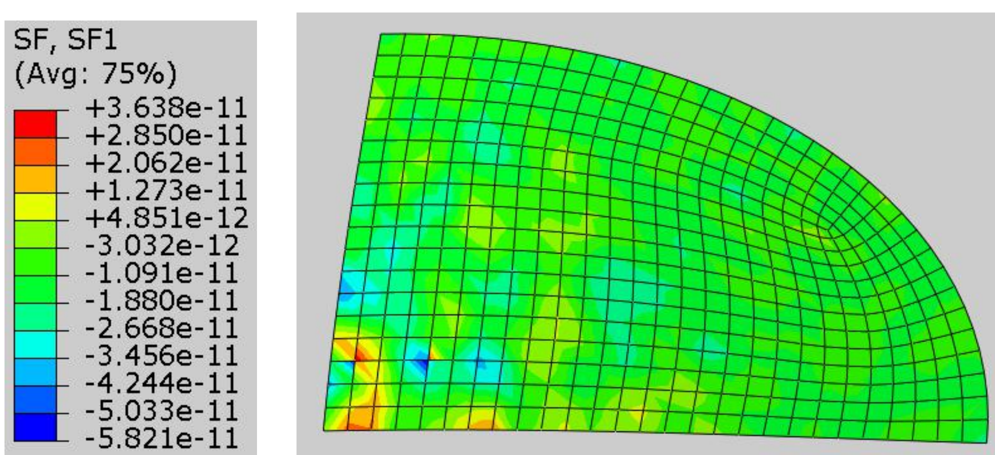
It can be seen that maximum deflection is about 13 m which is unrealistic as compared to the total length of 0.4 m of disc. Remember that we did not turn on the NLgeom option which means that analysis did not take into account the geometric nonlinearity in the structure.

Now we will plot the membrane force in the disc.

Select **SF** as output variable in the Field Output toolbar and make sure that **SF1** is selected as component.



The simulation results will appear as shown below (Scale factor is set to 0.0015).



It can be seen that magnitude of the membrane force is almost zero.

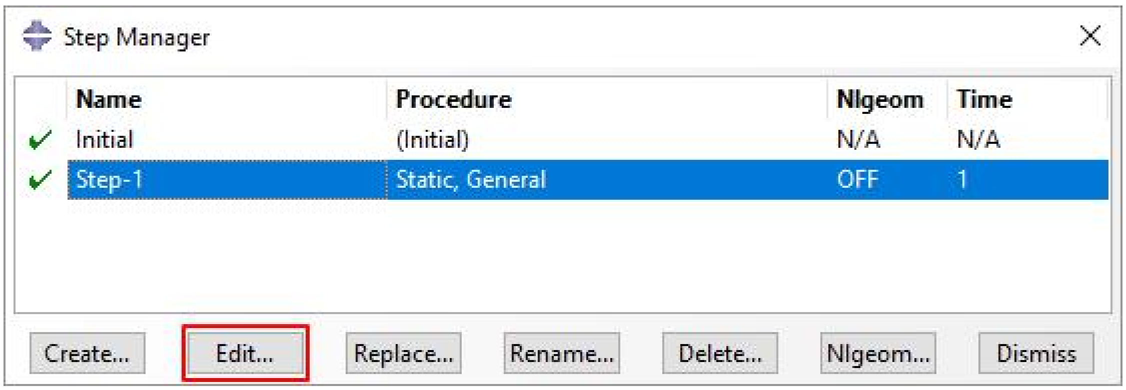
Initially the disc resists the pressure load with bending stiffness but after the pressure causes it to be bulged, the deformed geometry exhibits additional stiffness and membrane forces appear in the disc. For a large deformation problem, these forces become significant. A linear analysis fails to take into consideration the membrane forces therefore the simulation results seems unrealistic.

Performing Nonlinear Analysis

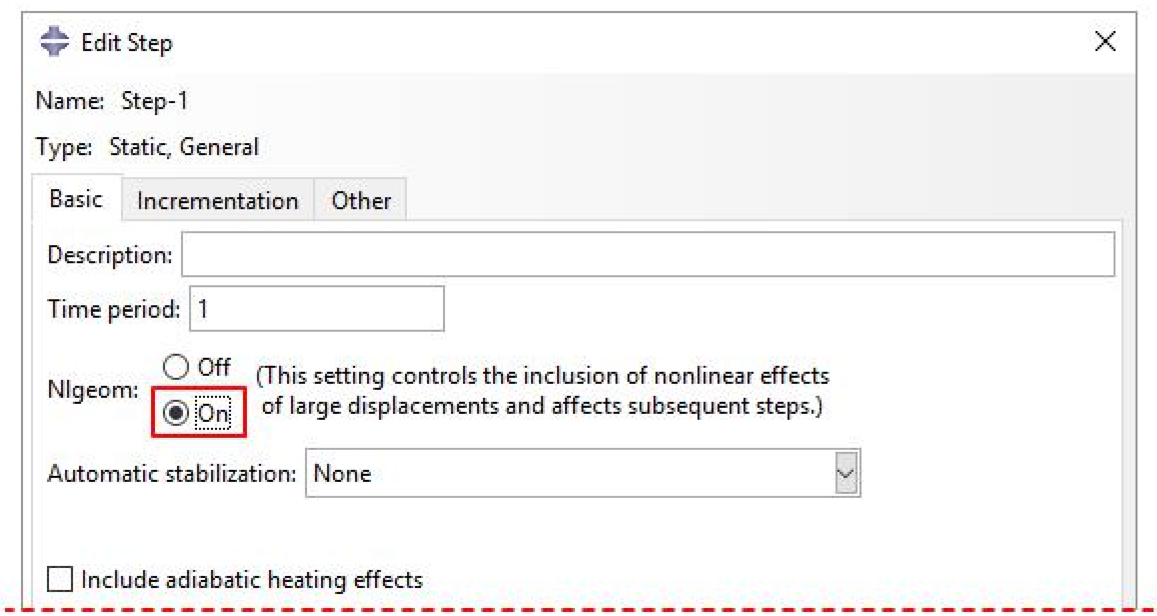
Change to **Step** module.

Open the Step Manager by picking  .

Pick the Step-1 and select **Edit**.



Turn on the Nlgeom option as shown below.



Pick **OK** to apply the changes and exit the dialog box.

Job Submission

Now we can submit the job for analysis.

So change to **Job** module and pick  to open the job manager.

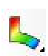
Pick **Create** and create a new job named Pressure_NonLinear or whatever name you would like.

Pick **Continue** and then **OK**.

Pick **Submit** to submit the job for analysis.

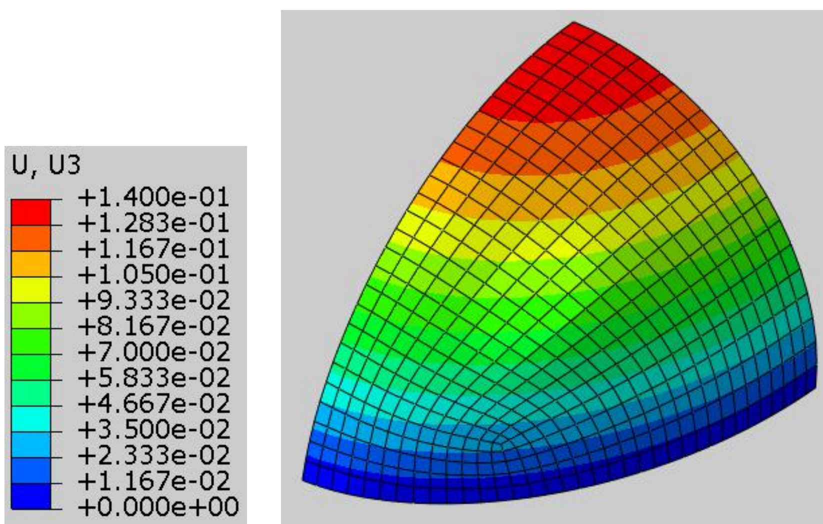
Pick **Results** to view the results in Visualization module.

Postprocessing

Pick  and select **U** as output variable in the Field Output toolbar.

Pick **U3** in the component field.

The simulation results will appear as shown below (Deformation scale factor is set to 1).



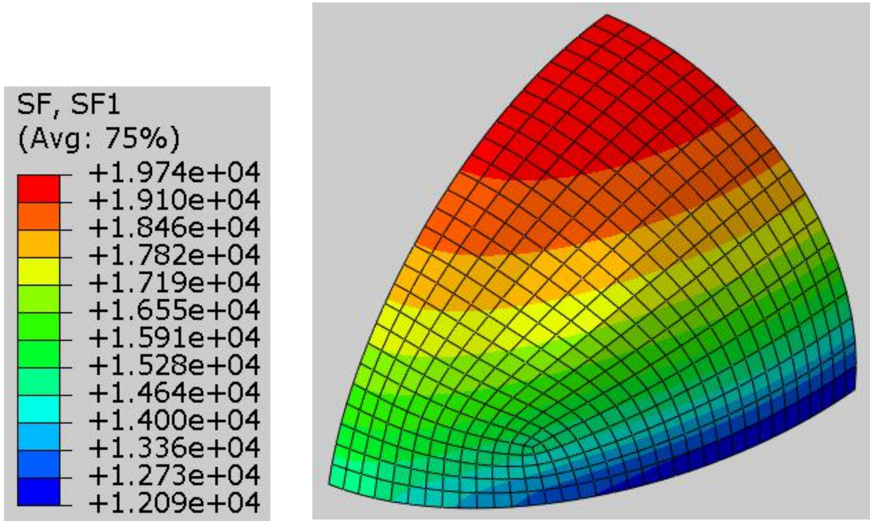
It can be seen that maximum deflection is about 0.14 m which seems realistic as compared to the total length of 0.4 m of disc.

As the load is applied, the shape of the disc changes to a bulged shape and hence its stiffness also changes. This change in shape and stiffness is accounted for in the nonlinear simulation. Furthermore, the pressure loading which is always normal to the disc, starts to have a component in the x and y-direction as the disc deforms. In a nonlinear analysis, normals to boundaries are updated and pressure load take the deformation into account and stays normal to surface. As a consequence of this, simulation results are different for the linear and nonlinear analysis.

Now we will plot the membrane force in the disc.

Select **SF** as output variable in the Field Output toolbar and make sure that **SF1** is selected as component.

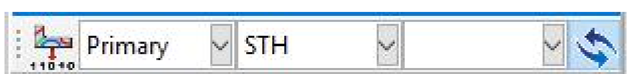
The simulation results will appear as shown below (Scale factor is set to 1).



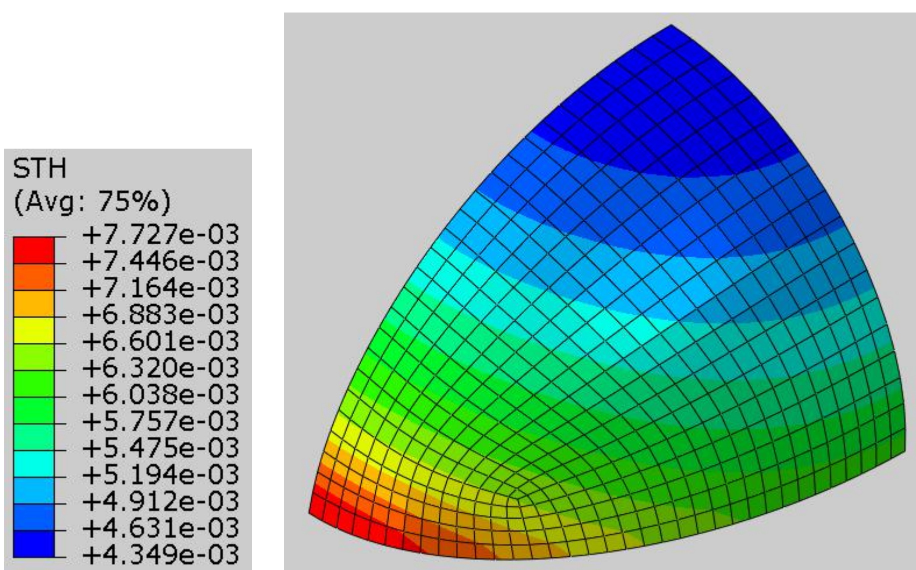
It can be seen that a significant membrane force has developed in the disc. As the disc is deformed, additional stiffness and membrane forces appear.

Now we will plot thickness for the shell elements.

Select **STH** in the Field Output toolbar to show contours of shell thickness in the disc.



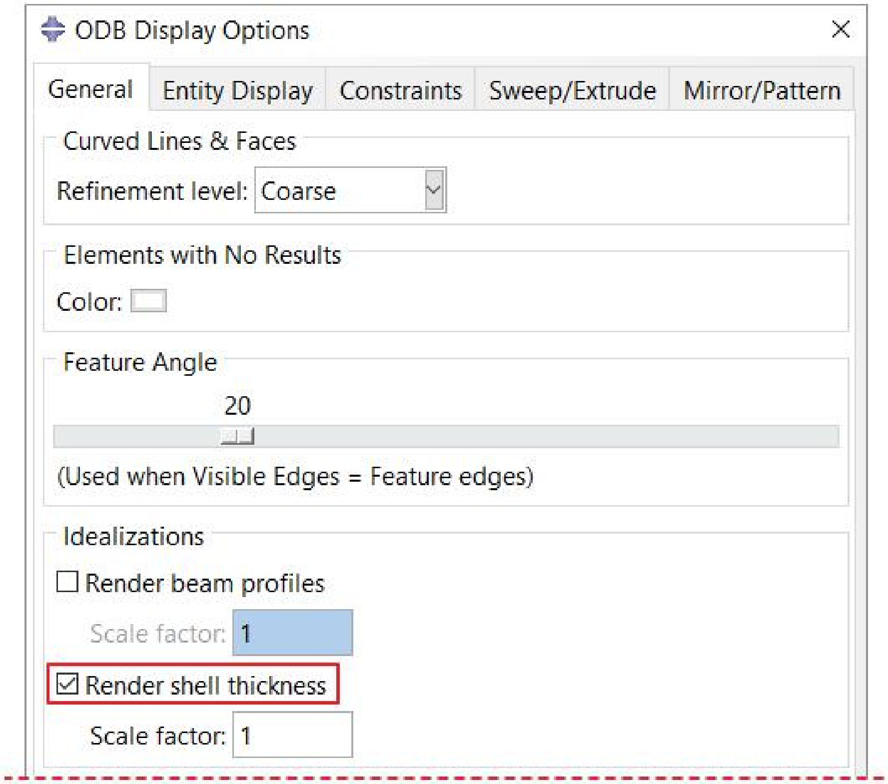
The thickness contour plot will appear as shown below (Scale factor is set to 1).



It can be seen that shell thickness is minimum near the center of the disc. Remember that the disc is 0.01 m thick in the undeformed state.

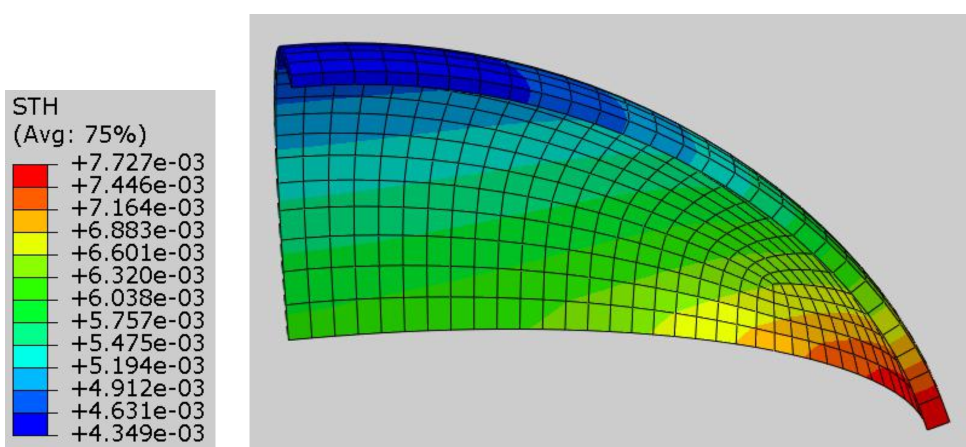
It is also possible to visulaize the thickness on screen.

While in the Visualization module, pick **View > ODB Display Options** and check the **Render shell thickness** option located under the General tab.



Pick **OK** to apply and exit.

The disc will appear as shown below.

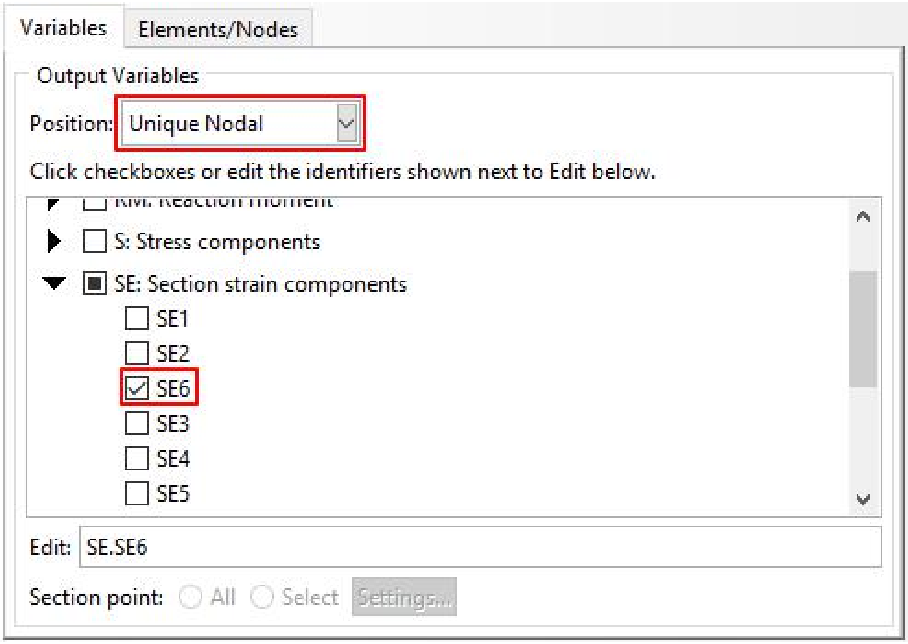


Now we will plot the thickness strain at two different locations on the disc.

Pick  and Create XY Data dialog box will appear.

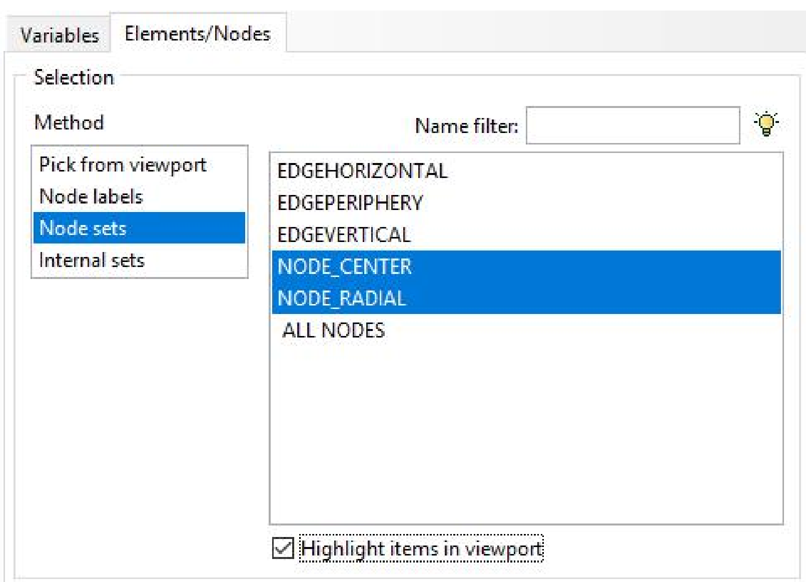
Pick **ODB field output > Continue**

In the Position field, select **Unique Nodal** and check the **SE6** checkbox.

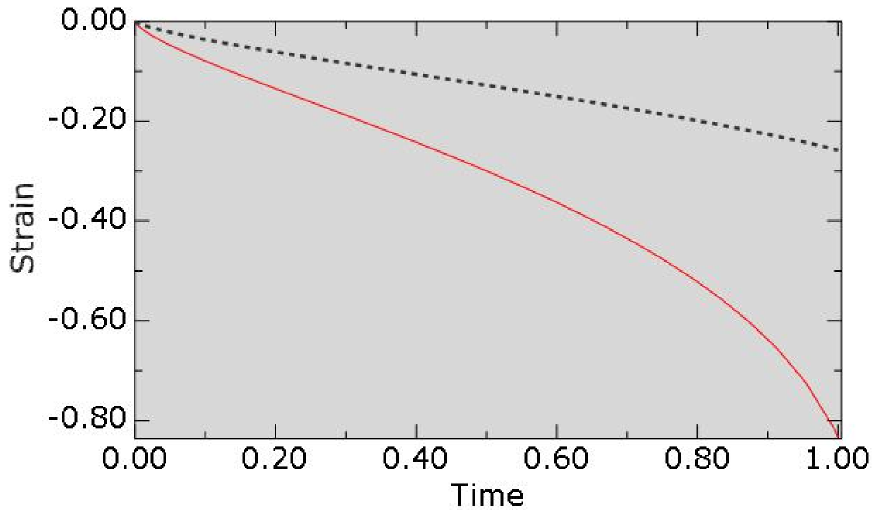


Under the Elements/Nodes tab, pick the **Node sets** as method and select the **NODE_CENTER** and the **NODE_RADIAL.**

Check **Highlight items in viewport** in the dialog box to see the location of nodes in viewport.



Pick **Plot** tab and graph will appear as shown below.

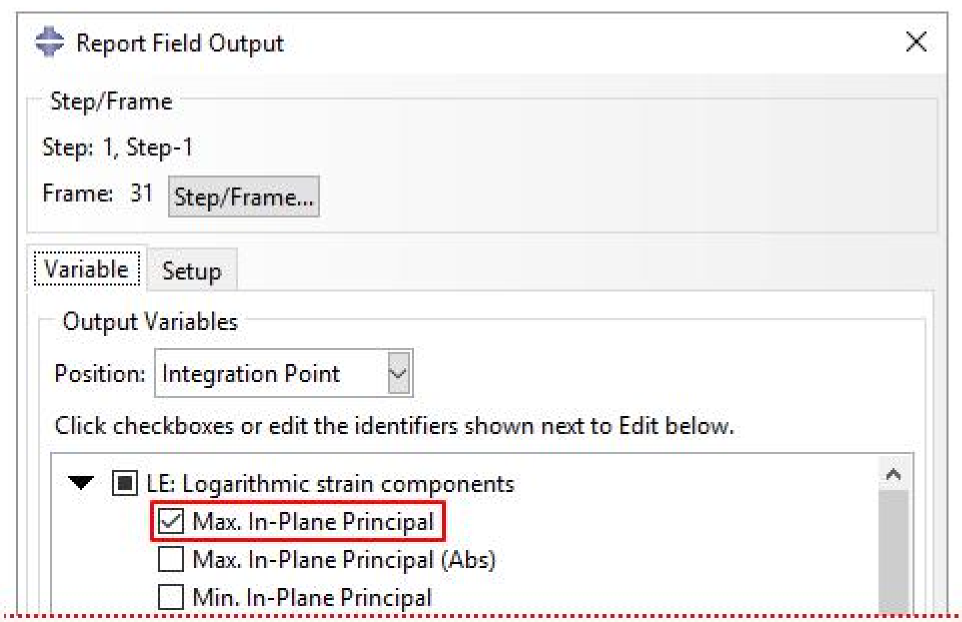


The dotted line plots the thickness strain for the node located on the periphery of the disc (Node_Radial) and red solid line plots the thickness strain for the node at the center of disc. It can be seen that thickness strain is higher in the center of the disc.

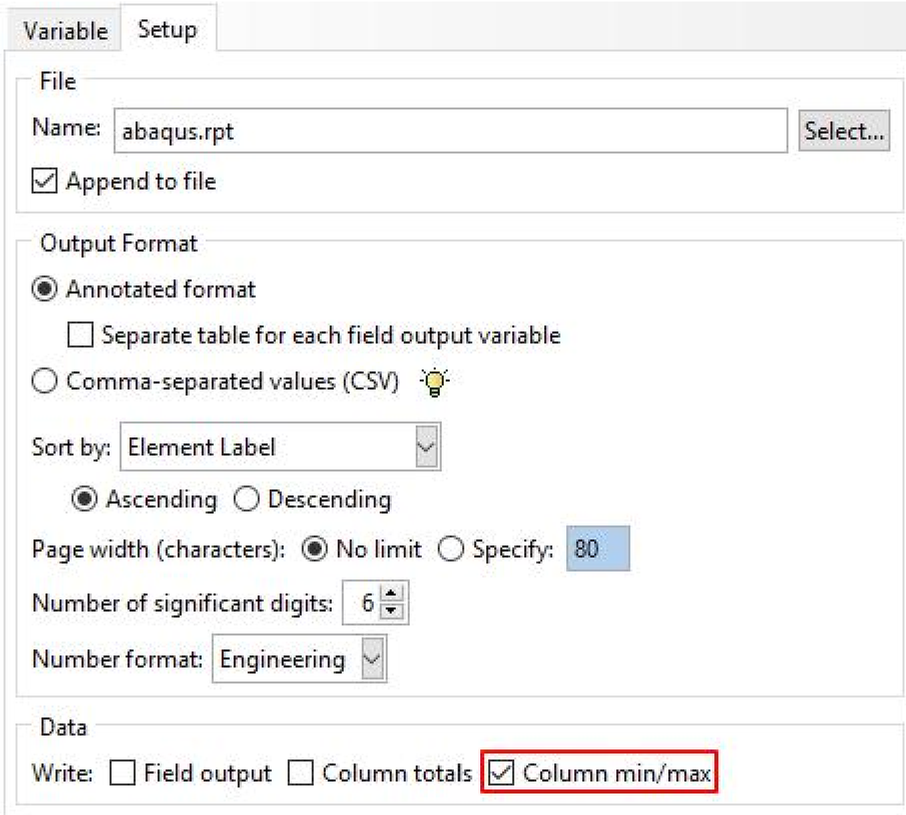
Now we will generate a report with maximum and minimum values for the strains in the disc.

While in the Visualization module, pick **Report > Field Output** and Report Field Output dialog box will appear.

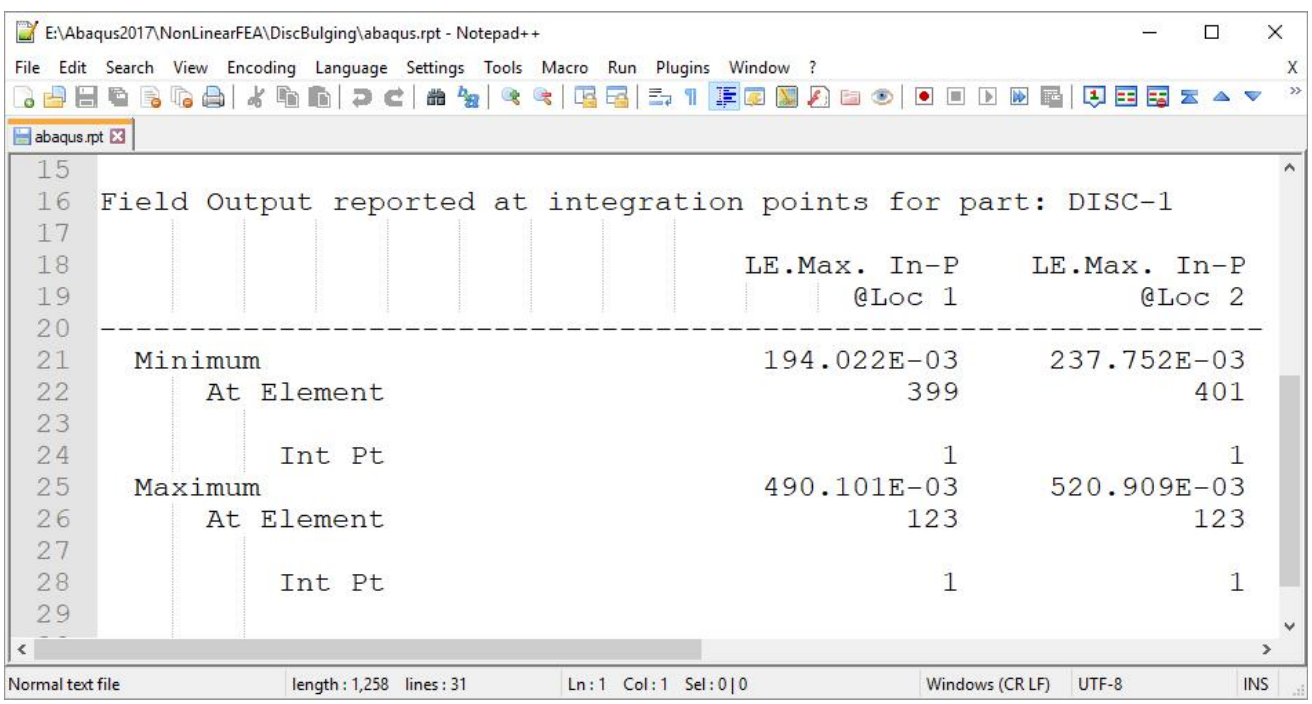
Select **Max. In-Plane Principal** as output variable under the **LE** identifier.



In the Setup tab, uncheck the Field output and Column totals options and check the **Column min/max** option.



Pick **Ok** and report will be written to the abaqus.rpt file. Open this file from the working directory and it will appear as shown below.



It can be seen that maximum strain of 520.9 E-03 occurs at the Loc 2 of element number 123. The locations Loc 1 and Loc 2 identify the section point in the element where the variable was calculated. As described in the report file, Loc 1 lies on the SNEG surface of the shell, and Loc 2 lies on the SPOS surface.

As the maximum strain is approximately 52%, this is clearly a large strain problem.

Select **File > Save** to save the changes we made so for.

FYI: The side of shell region in the direction of the positive element normal is called SPOS, while the side in the direction of the negative element normal is called SNEG. In the following figure section points through the thickness of the shell at the location of integration point are also shown.