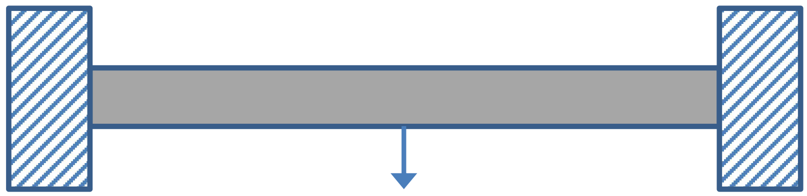
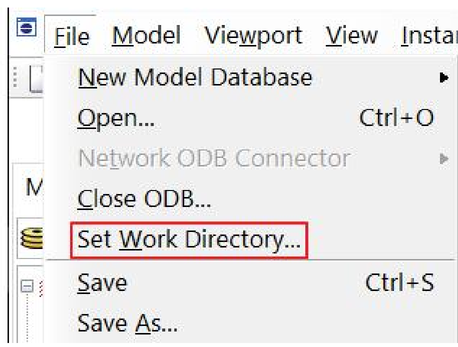
Exercise 1

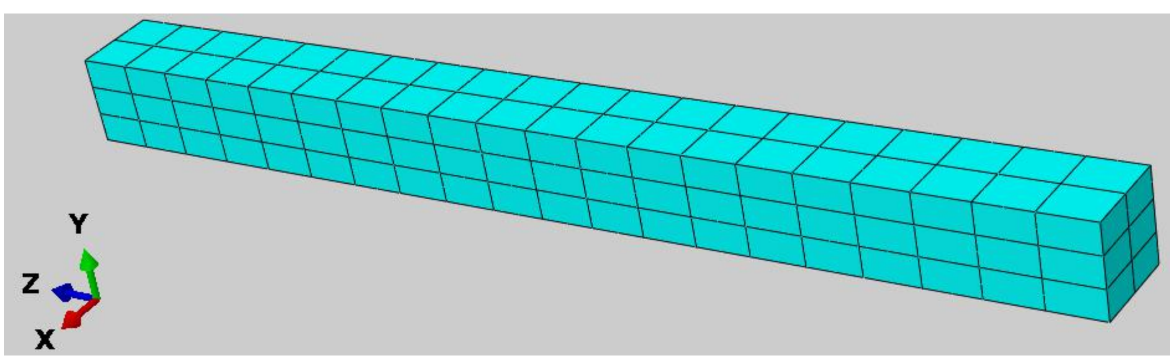
In this exercise we will determine the deflection of a beam which loaded at centre and fixed at both ends.



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



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



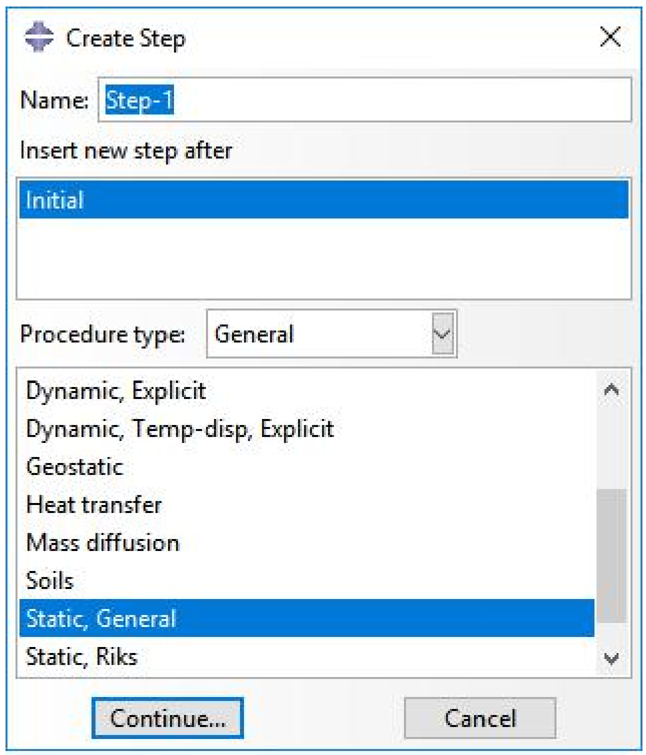
The model consists of a single part, Beam. The beam shown in above figure has a length of 300mm and cross section of 50mm x 50mm. It is assumed to be made from Aluminum with a Young's modulus of 70 GPa and a Poisson's ratio of 0.3. It is meshed with 20-node quadratic brick elements with reduced integration (C3D20R).

Defining Step

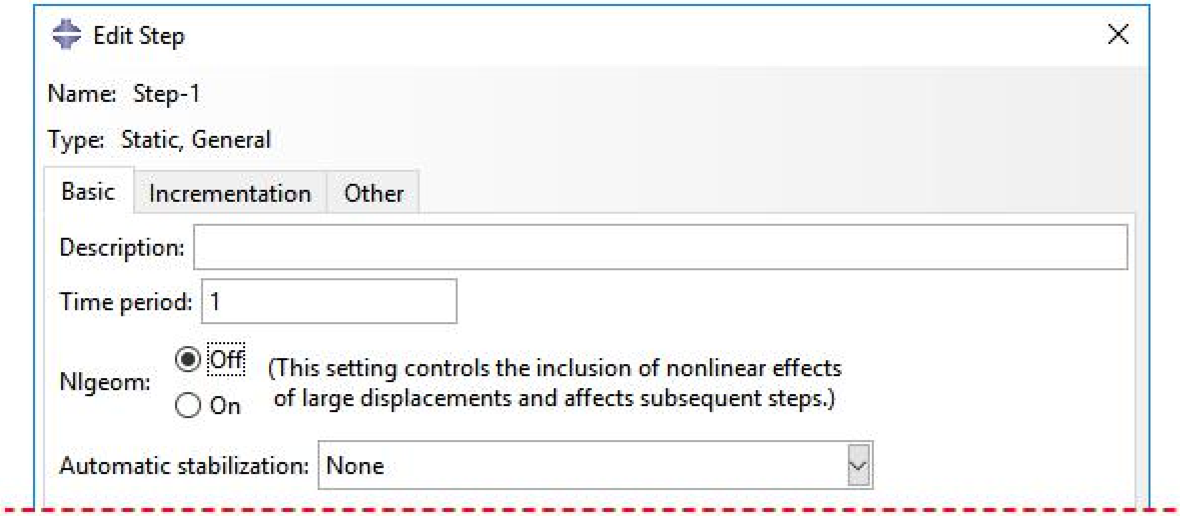
We assume that load is applied slowly such that inertia effects can be neglected. So analysis will be performed using the Static, General procedure.

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. Notice that by default total time period is set to **1.0** and NLgeom option is set to **Off**.



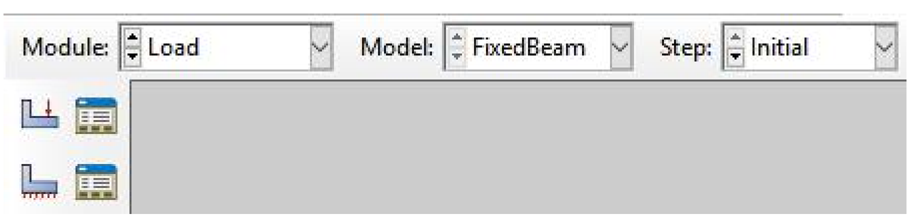
We assume that changes in geometry are small during the loading of beam and a linear analysis will provide accurate results. So we leave NLgeom option off.

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

Defining Load

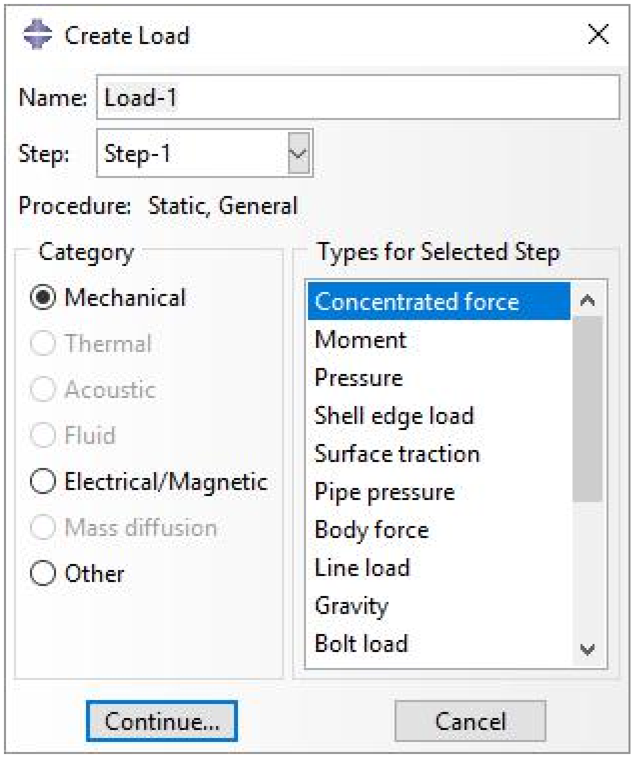
A concentrated force will be applied on a reference point which is coupled to the edge lying on the midspan of the beam. The coupling has already been created by defining a “Coupling” constraint.

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

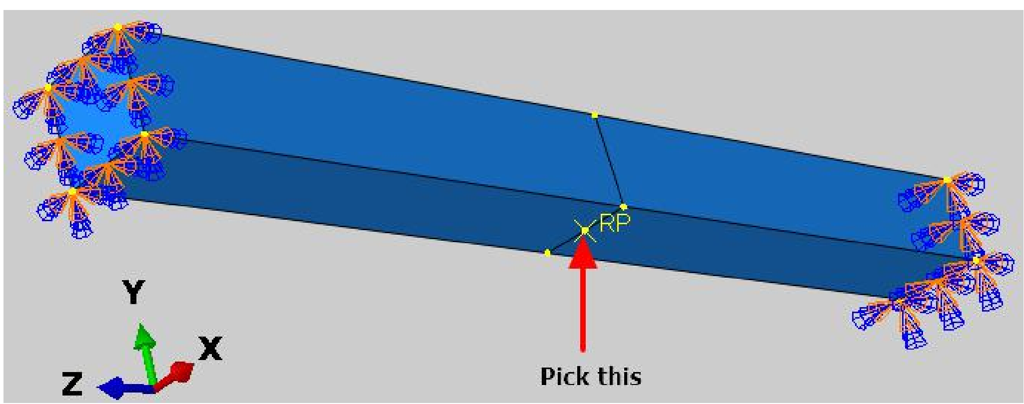


Pick  to create a load and select Step-1 in the Step field.

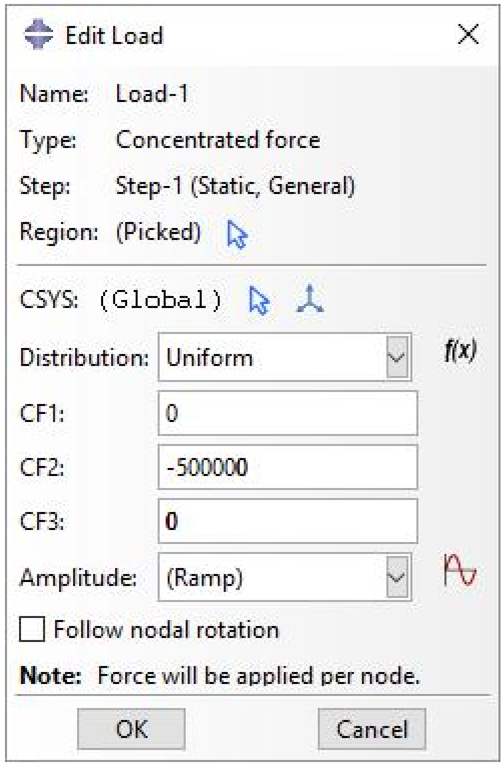
Select **Concentrated** in the Types field and pick **Continue.**



Select the "RP" reference point as shown in figure below.



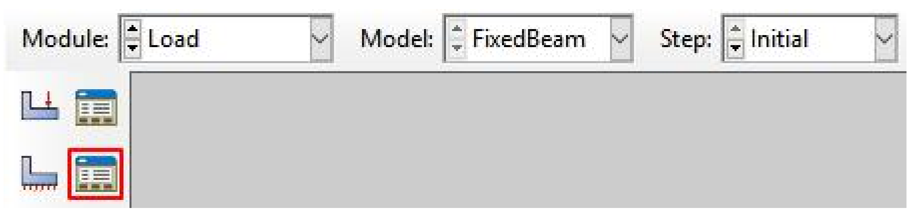
Pick **Done** and enter **-500000** as the force magnitude in y-direction.



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.



In the Boundary Conditions Manager it can be seen that two boundary conditions have already been created. The boundary condition “BC-Right” constrains all the degree of freedom of right face of the beam. The boundary condition “BC-Left” constrains all the degree of freedom of left face of the beam.

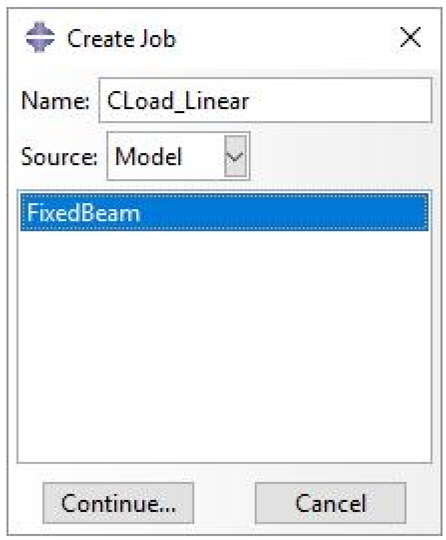
Pick **Dismiss** to close the manager.

⇨ Job Submission

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

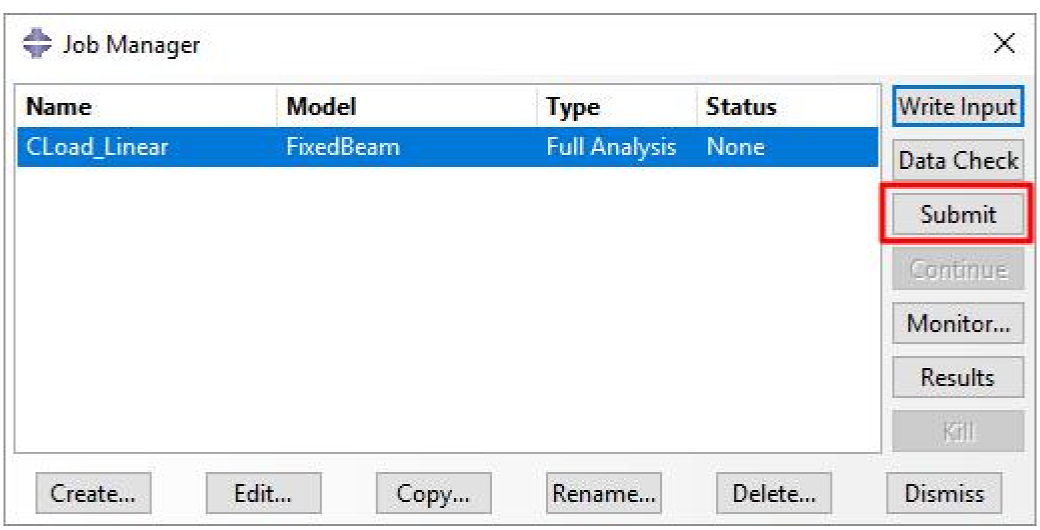
So change to **Job** module and pick  to open the job manager.

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



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

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

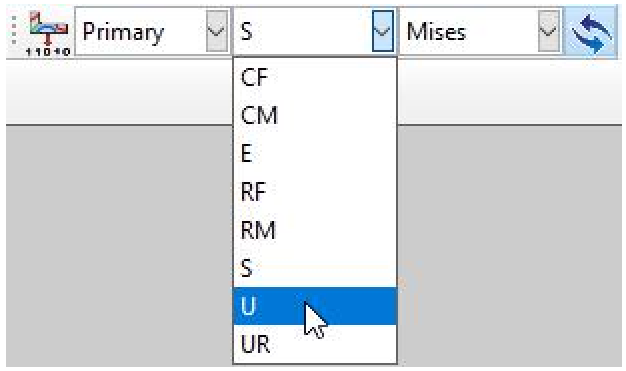


Pick **Results** to view the results in Visualization module.

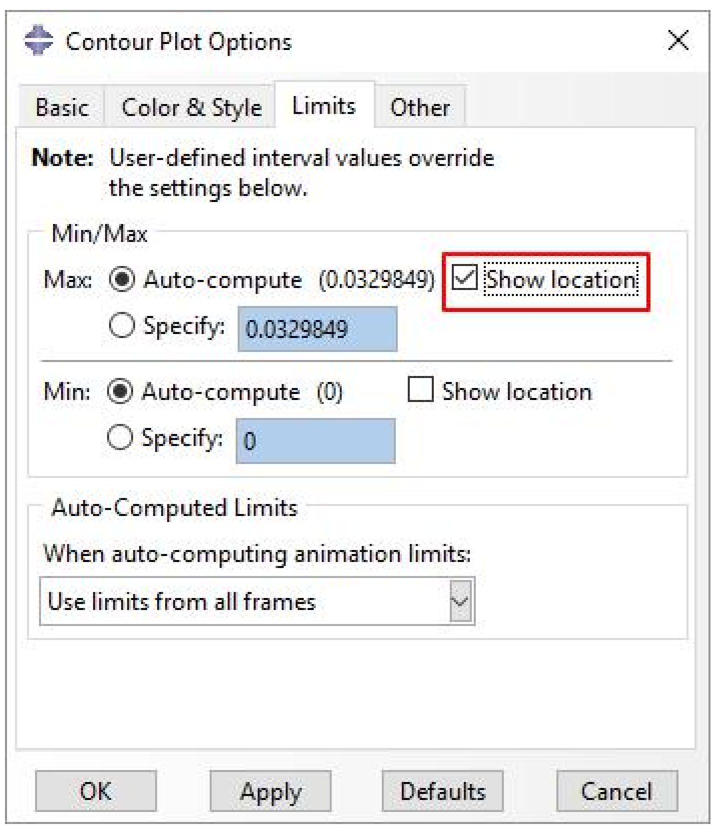
Postprocessing

Pick  to plot contours on deformed shape.

Select **U** as output variable in the Field Output toolbar.

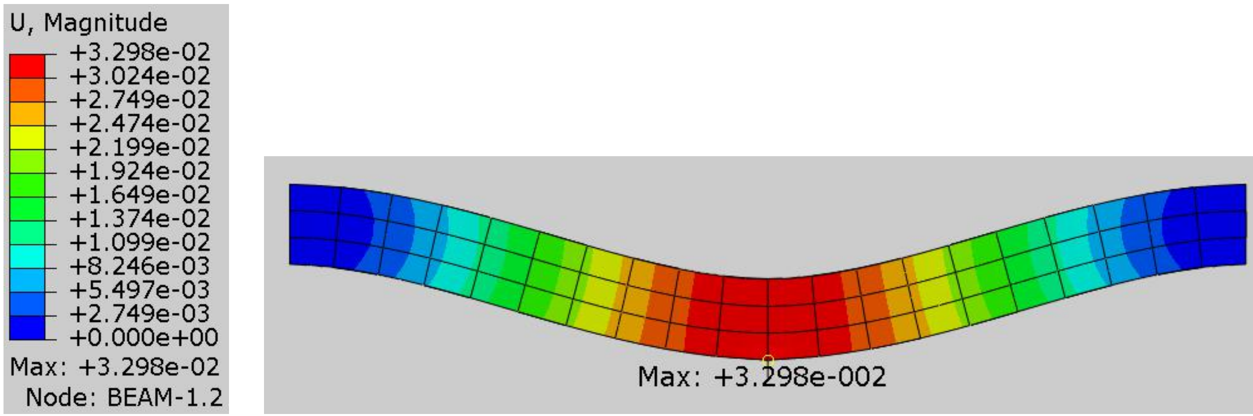


Pick  to open Contour Plot Options dialog box and check the **Show location** field for Max limit as shown below.



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

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



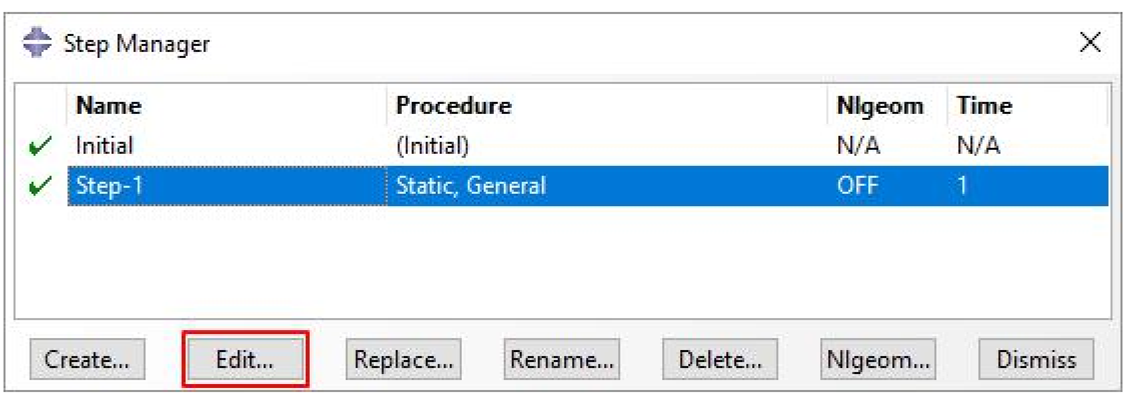
It can be seen that midspan deflection is approximately 33mm which is approximately 11% of the beam´s length. Remember that we did not turn on the NLgeom option which means that analysis did not take into account the geometric nonlinearity in the structure. As we can see that deflection of the beam is considerable, the assumption that large deformations will not take place does not seem correct. To verify if this is the case, we will perform a geometrically nonlinear analysis.

Performing Nonlinear Analysis

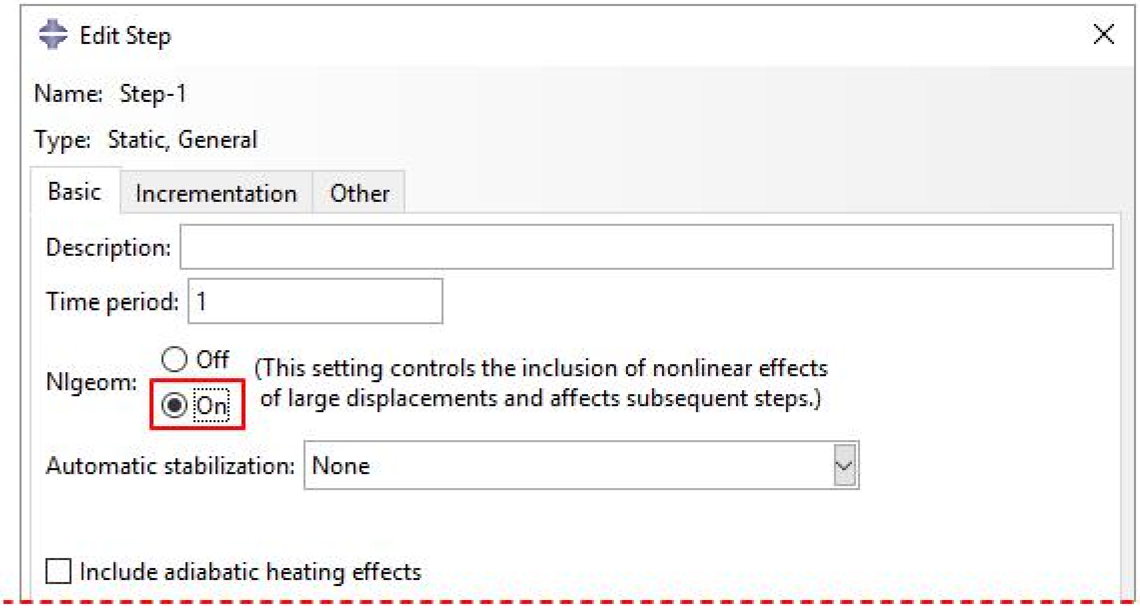
Change to **Step** module.

Open the Step Manager by picking  .

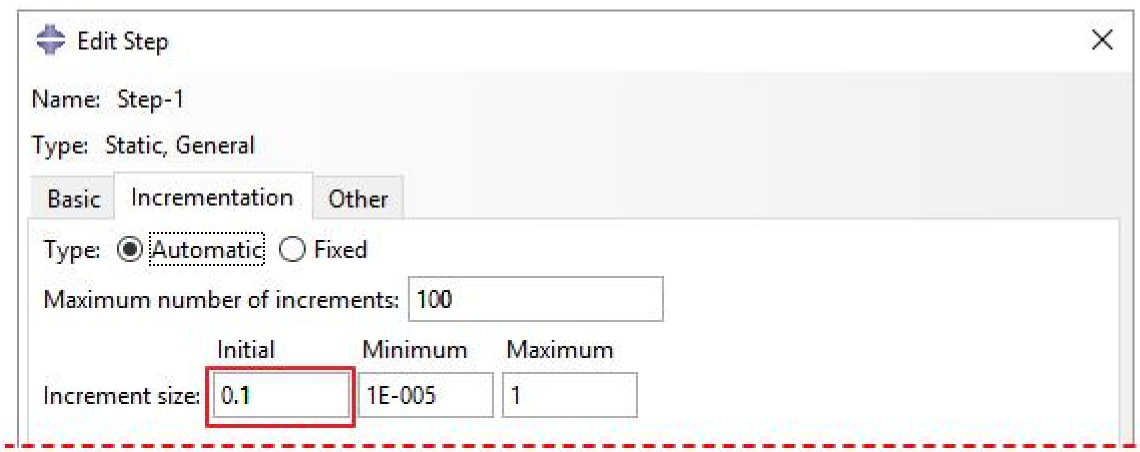
Pick the Step-1 and select **Edit**.



Turn on the Nlgeom option as shown below.



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



In a nonlinear problem the solution is found by applying the load gradually and incrementally. Therefore Abaqus breaks the simulation into a number of load incompetents. The ratio of the initial time increment to the total time period specifies the proportion of load applied in the first increment.

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 CLoad_NonLinear or whatever name you would like.

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

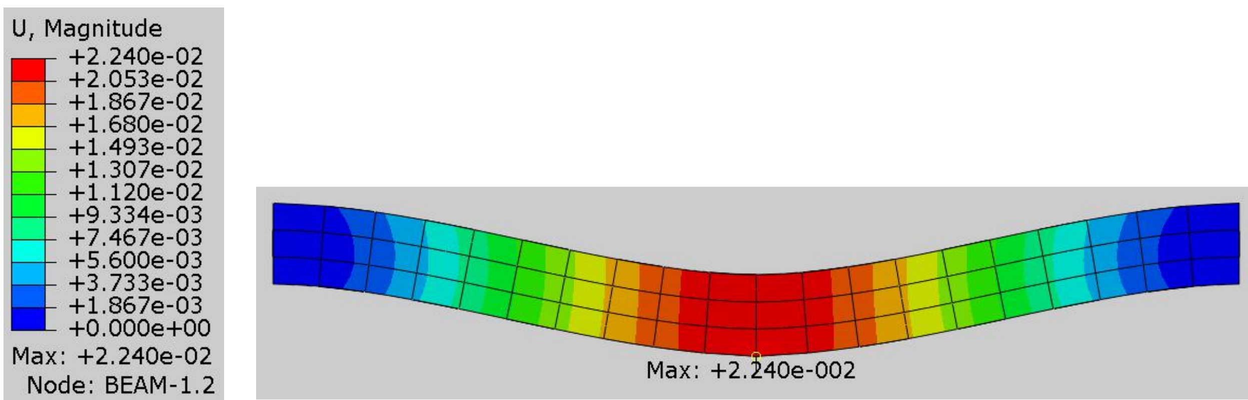
Pick **Submit** to submit the job for analysis.

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

Postprocessing

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

The simulation results will appear as shown below.



It can be seen that midspan deflection is approximately 22.4mm which is considerably less than the deflection with linear analysis. The difference between the linear and nonlinear simulations is sufficiently large to conclude that a linear analysis is not appropriate for the beam and therefore a nonlinear analysis must be performed to get realistic results.

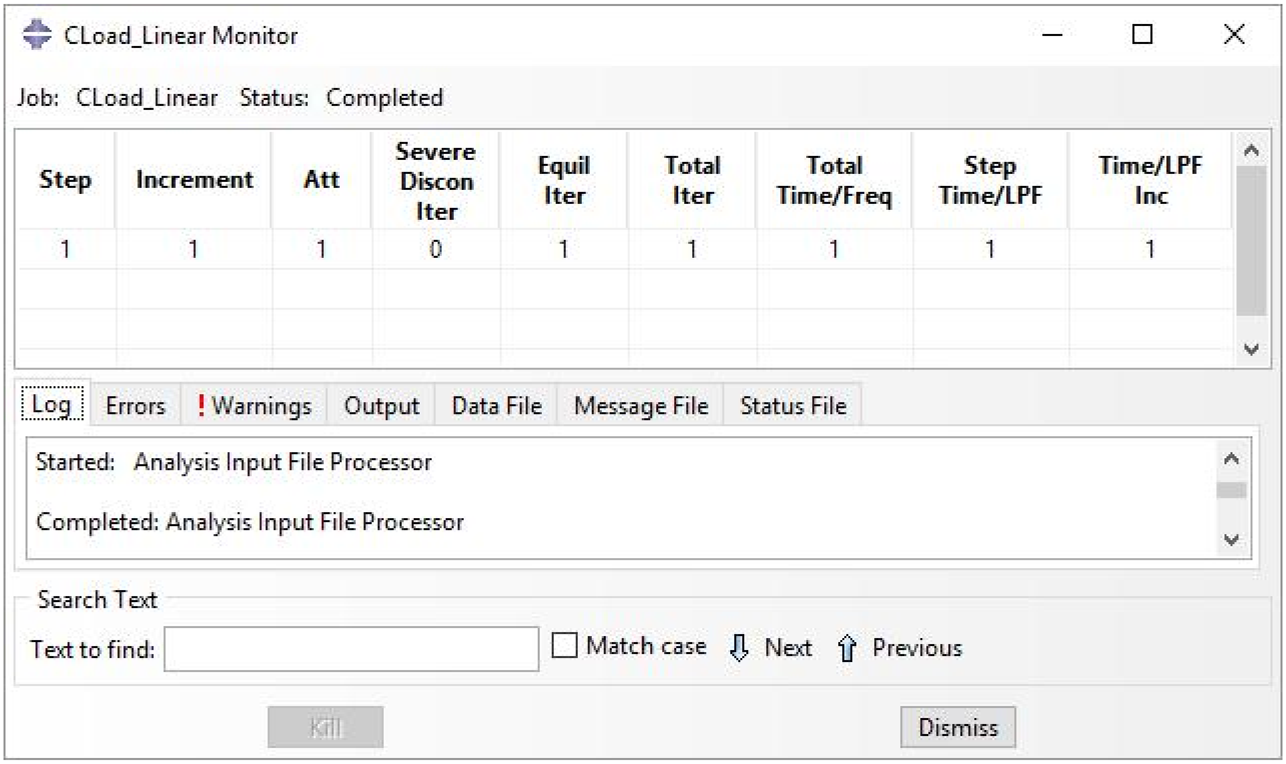
As the load is applied, the shape of the beam changes to a curved shape and hence its stiffness also changes. This change in shape and stiffness is accounted for in the nonlinear simulation. As a consequence of this, simulation results are different for both analysis.

Job Monitoring

We will look at the information of analysis job by using Job Monitor.

Change to **Job** module and pick  to open the job manager.

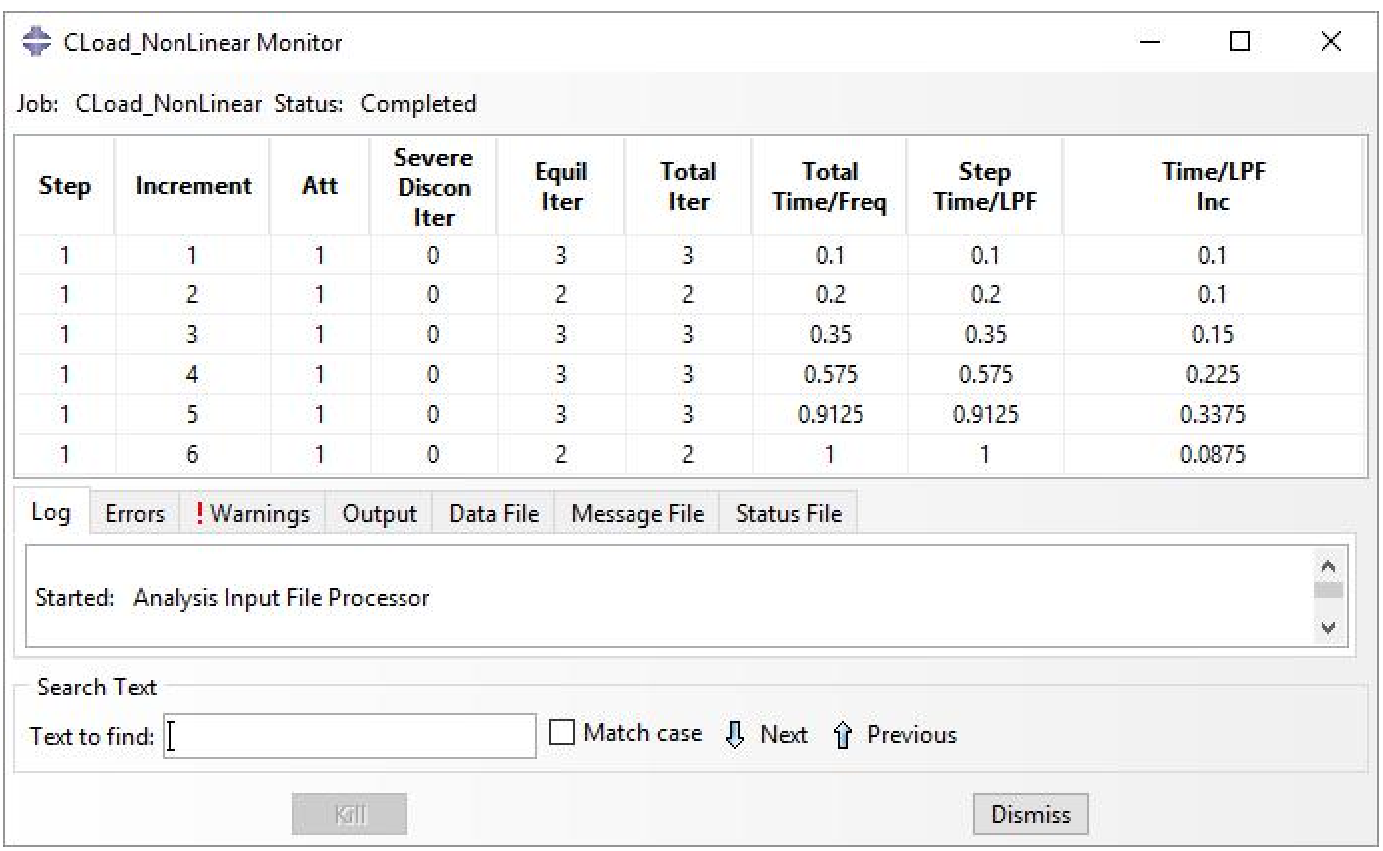
Select CLoad_Linear job in the manager and pick **Monitor** and Job Monitor will appear as shown below.



Step	Increment	Att	Severe Discon Iter	Equil Iter	Total Iter	Total Time/Freq	Step Time/LPF	Time/LPF Inc
1	1	1	0	1	1	1	1	1

The first column shows the step number and second column shows the number of increments. It can be seen that job completes only in one increment as this is a linear analysis.

Select CLoad_NonLinear job in the manager and pick **Monitor** and Job Monitor will appear as shown below.



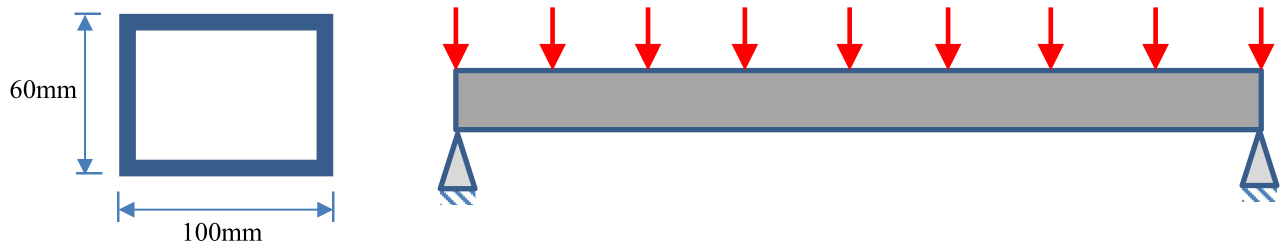
Step	Increment	Att	Severe Discon Iter	Equil Iter	Total Iter	Total Time/Freq	Step Time/LPF	Time/LPF Inc
1	1	1	0	3	3	0.1	0.1	0.1
1	2	1	0	2	2	0.2	0.2	0.1
1	3	1	0	3	3	0.35	0.35	0.15
1	4	1	0	3	3	0.575	0.575	0.225
1	5	1	0	3	3	0.9125	0.9125	0.3375
1	6	1	0	2	2	1	1	0.0875

It can be seen that total six increments are required to apply the full load. In the first increment, 10% of the total load is applied, again 10% in the second, 15% in the third, 22.5% in the fourth, 33.75% in the fifth and 8.75 % of the total load is applied in the sixth increment. Abaqus/Standard automatically controls the increment size. It can also be seen that during the first increment, three iteration are required to converge to a solution. During the second increment two iterations, during third,fourth and fifth increments three iterations for each and during sixth increment two iteration are required to converge to a solution.

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

Exercise 2

In this exercise we will analyze the beam shown schematically in the figure below.

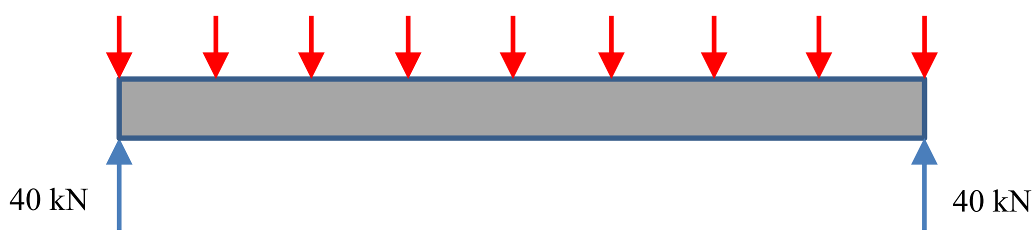


Pick **File** > **Set Work Directory** and set the work directory to the **BeamPinned** folder

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 2m and hollow cross section of 100mm x 60mm. The walls have a uniform thickness of 5mm. It is modeled as one-dimensional line (beam elements) instead of three-dimensional continuum elements. It is assumed to be made from steel with a Young's modulus of 200 GPa and a Poisson's ratio of 0.3. It is meshed with 3-node quadratic beam elements (B22). A uniform pressure of 40000 N/m is applied.

The following diagram shows the reactions at the supports computed using the analytical technique.



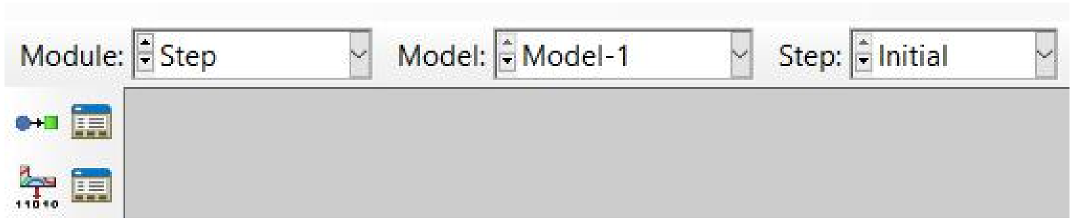
$$Deflection = \frac{5}{384}\frac{pL^4}{EI} = 0.0483$$

We will perform a finite element analysis and compare the results with analytical solution.

⇨ Defining Step

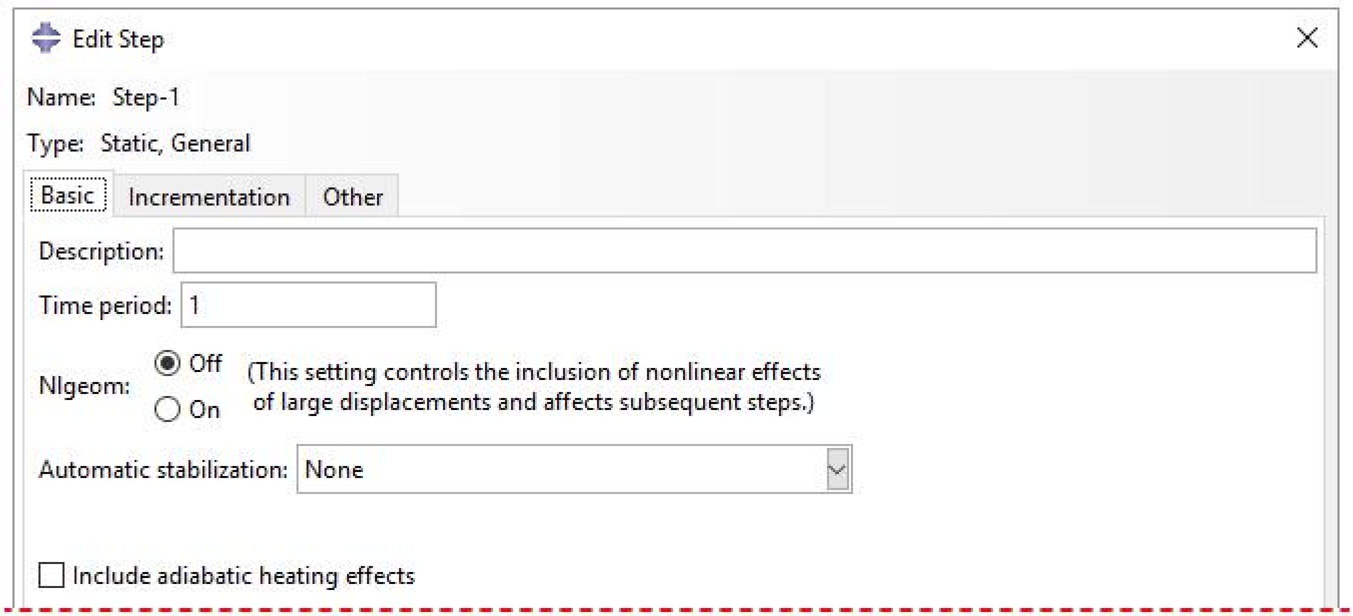
We assume that load is applied slowly such that inertia effects can be neglected. So analysis will be performed using the Static, General procedure.

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. Notice that by default total time period is set to **1.0** and NLgeom option is set to **Off**.



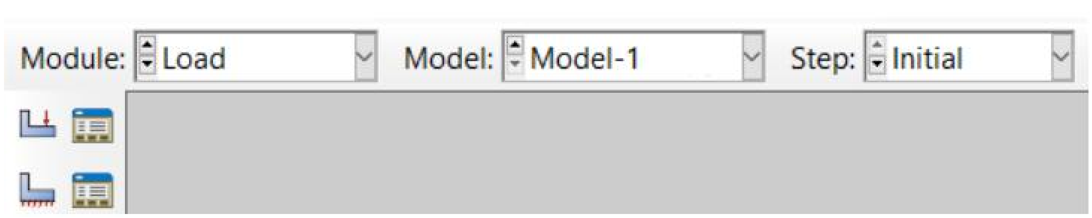
We assume that changes in geometry are small during the loading of beam and a linear analysis will provide accurate results. So we leave NLgeom option off.

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

Defining Load

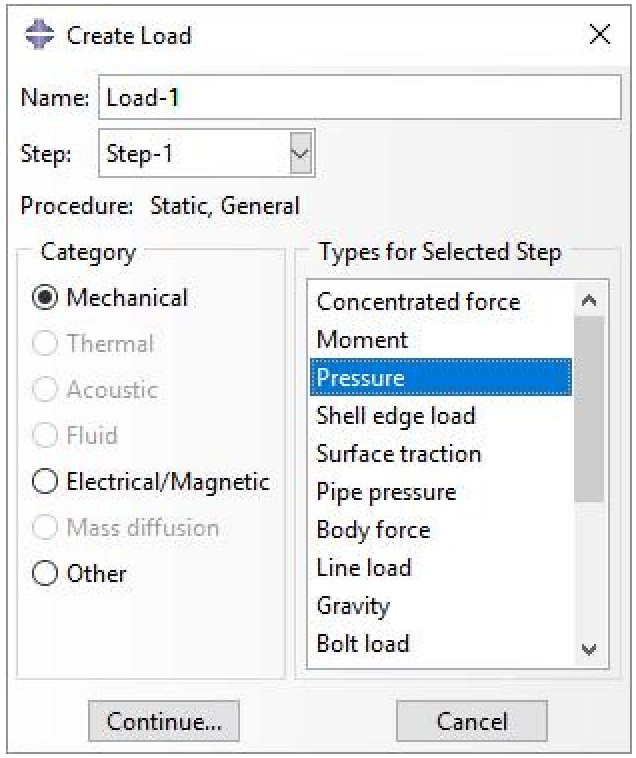
A uniform pressure load will be applied across the top of the beam.

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



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

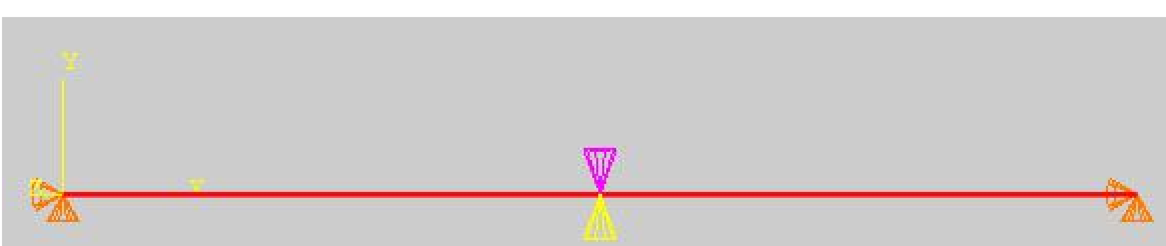
Select **Pressure** in the Types field and pick **Continue.**



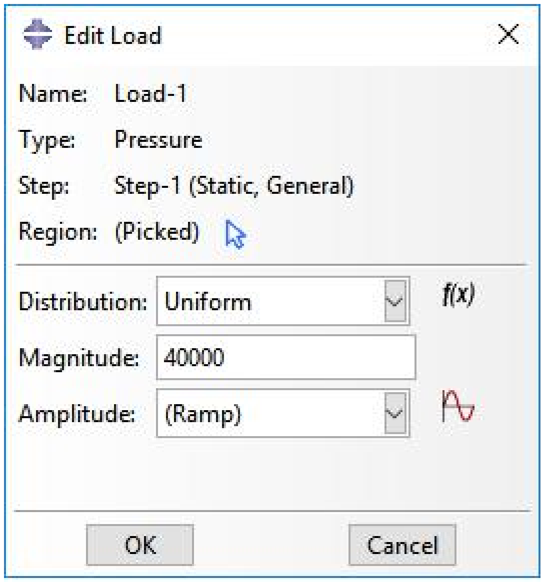
Select the geometry of the beam as shown in figure below.

Pick **Done** to accept the selection.

Pick **Magenta** to specify the top side of the beam for applying pressure load.



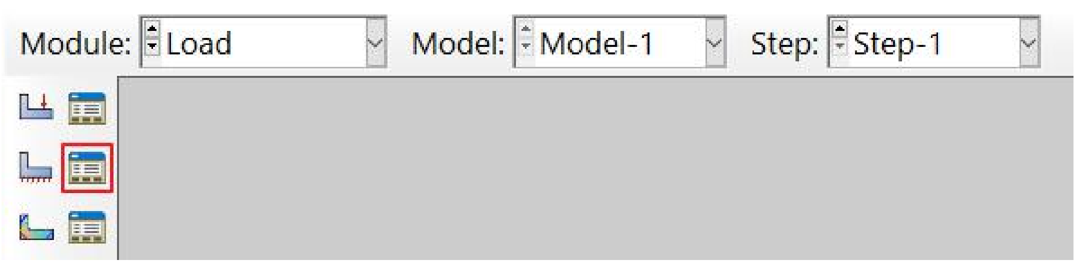
Enter **40000** as the pressure magnitude.



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

Boundary Conditions

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



In the Boundary Conditions Manager it can be seen that two boundary conditions have already been created.

The boundary condition "BC-Right" constrains translational degrees of freedom of right node of the beam.

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

Pick **Dismiss** to close the manager.

Field Output Request

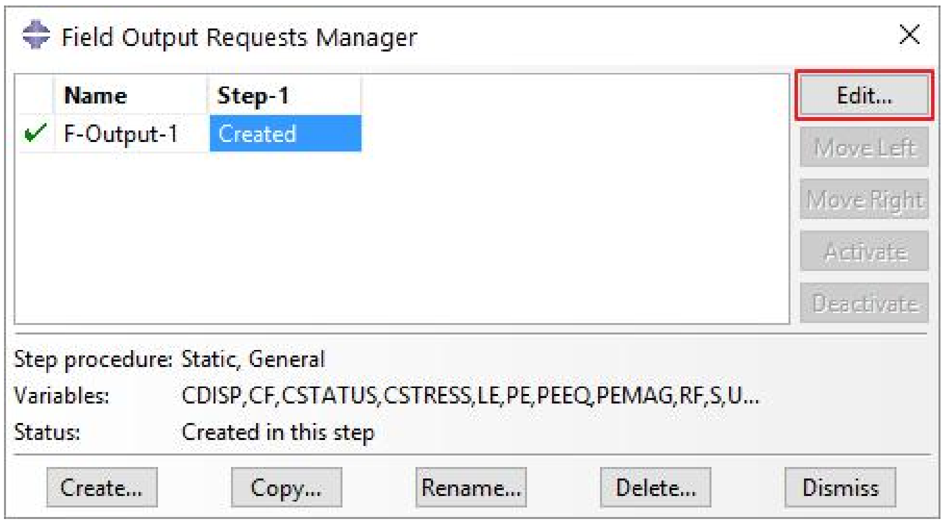
We will plot the section forces, i.e axial forces and shear forces. This information can be plotted by requesting SF output variable.

SF variable contains all section forces and moment components.

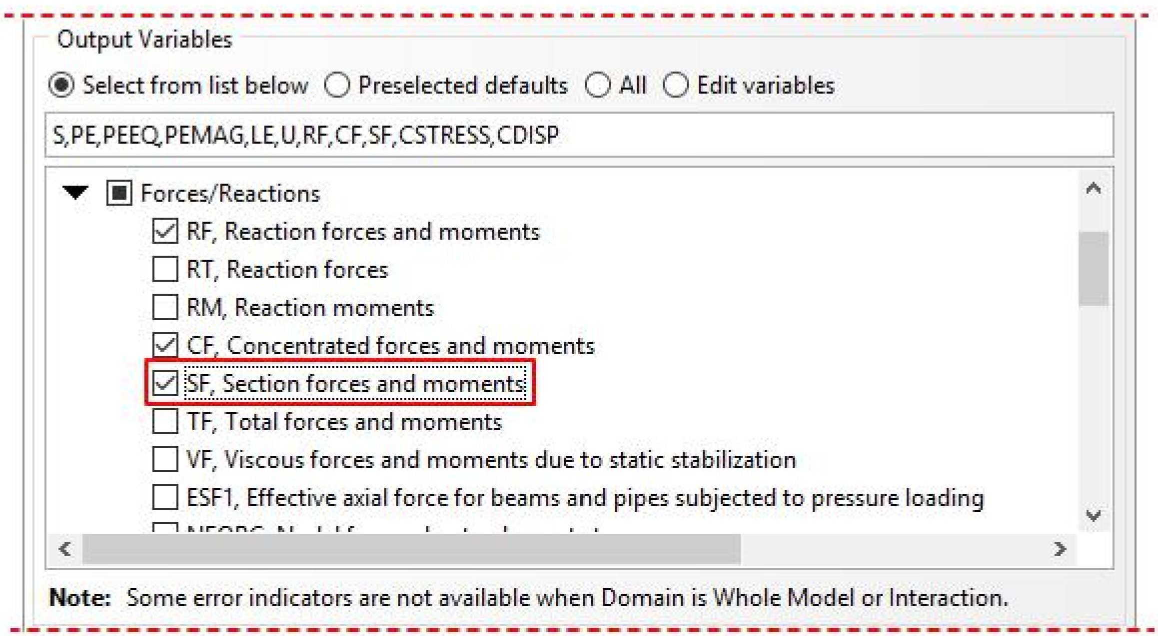
Change to **Step** module.

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).



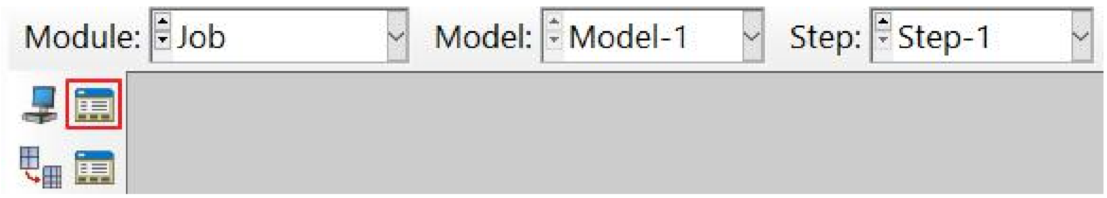
Pick **OK** to apply and exit.

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.

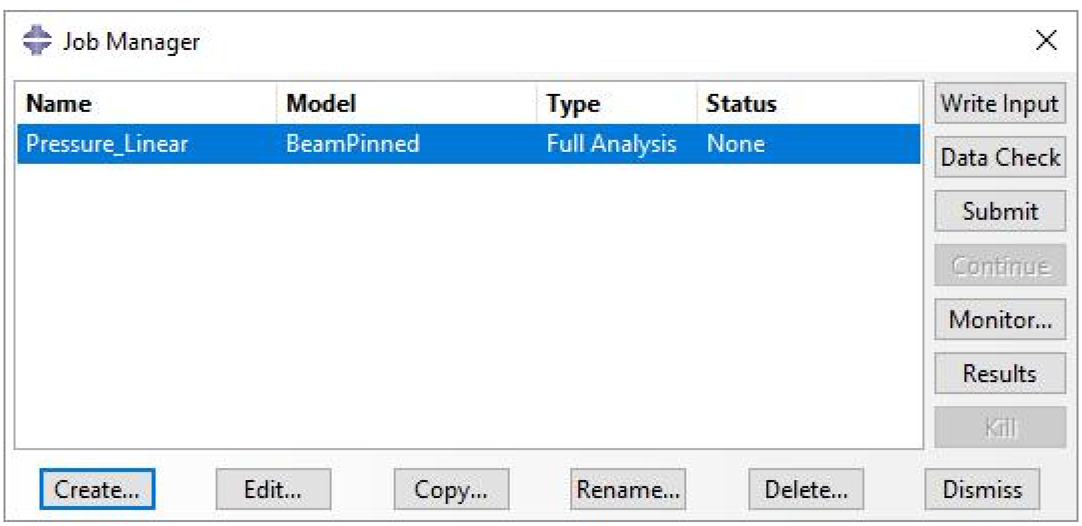
So change to **Job** module and pick  to open the job manager.



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.



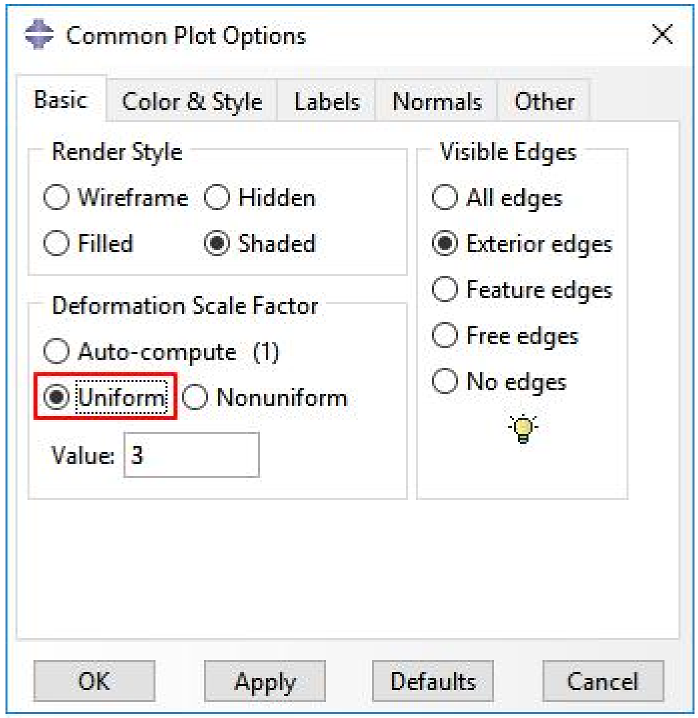
Pick **Results** to view the results in Visualization module.

Postprocessing

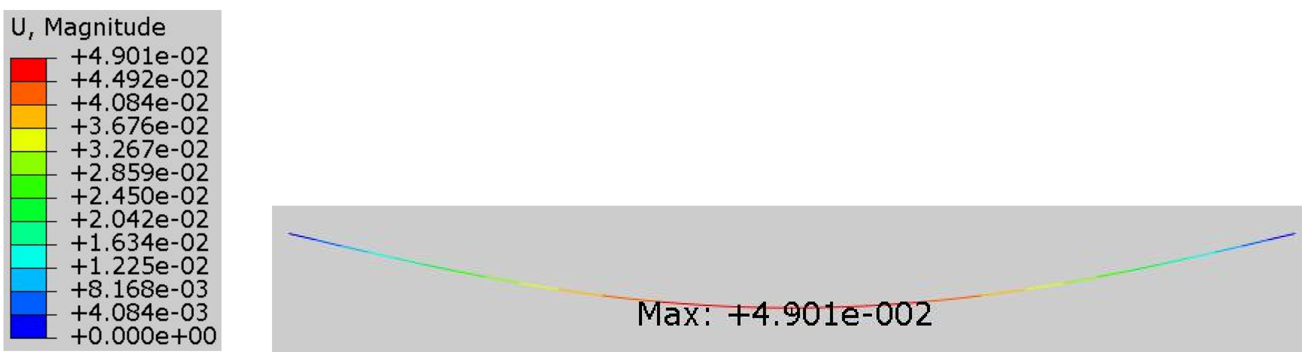
Pick  and select **U** as output variable in the Field Output toolbar.

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

Pick  and set the Deformation Scale Factor to **Uniform** and enter 3 as value.



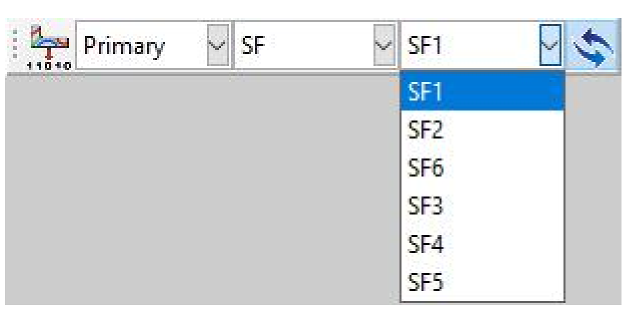
The simulation results will appear as shown below.



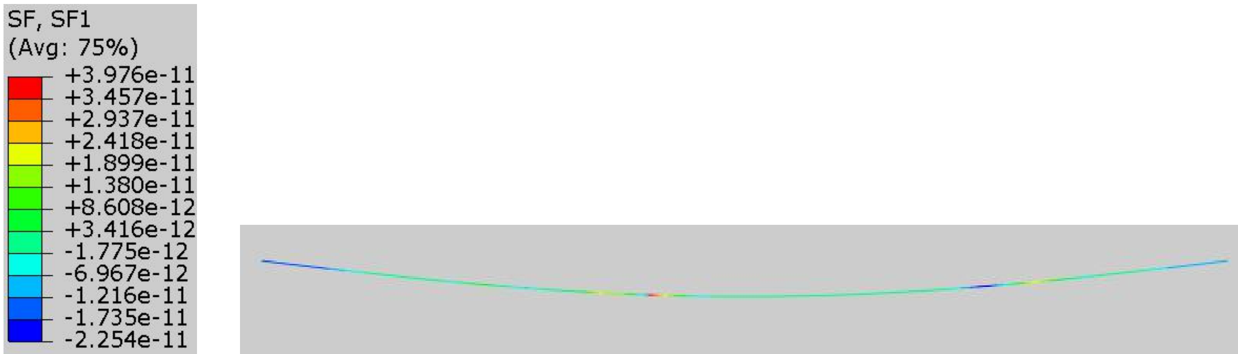
It can be seen that midspan deflection is approximately 49.01mm which is approximately 2.45% of the beam´s length. Remember that we did not turn on the NLgeom option which means that analysis did not take into account the geometric nonlinearity in the structure. We can say that deflection of the beam is small as compared to the length of the beam.

Now we will plot the axial force in the beam.

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 3).

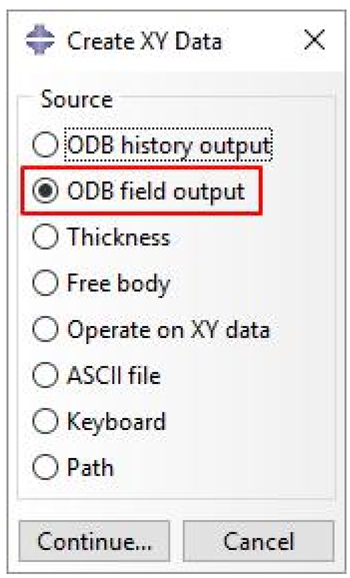


It can be seen that magnitude of the axial force is almost zero.

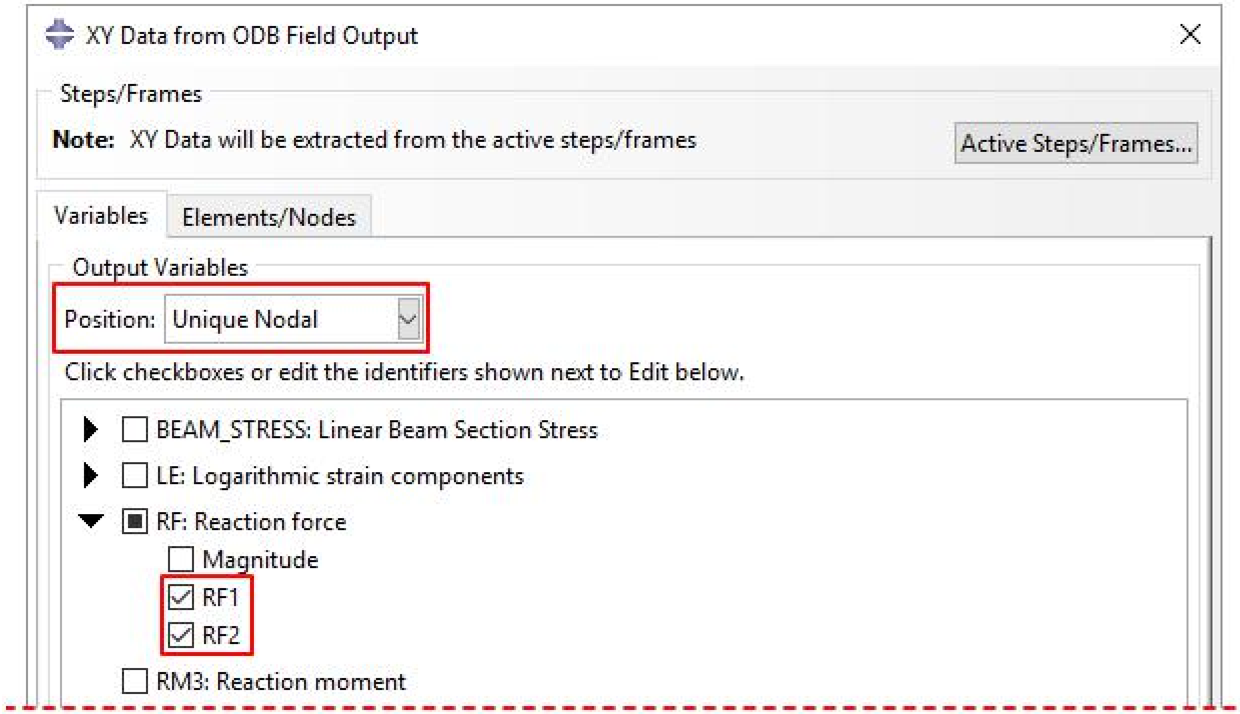
Now we will plot reaction forces at the end nodes where boundary conditions are defined.

Pick  and Create XY Data dialog box will appear.

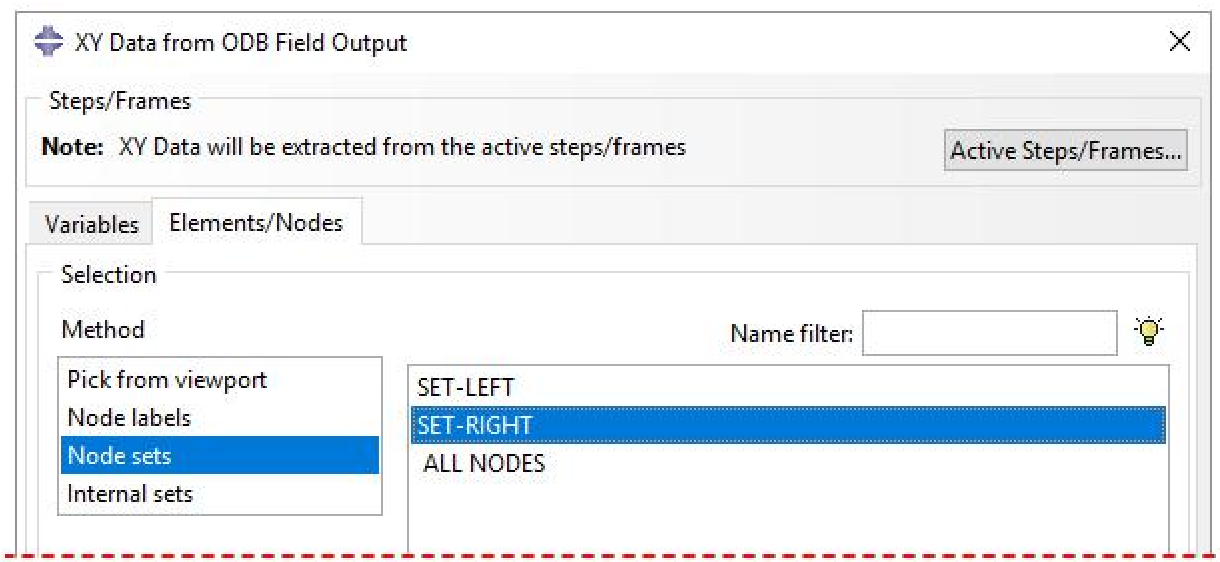
Pick **ODB field output > Continue**



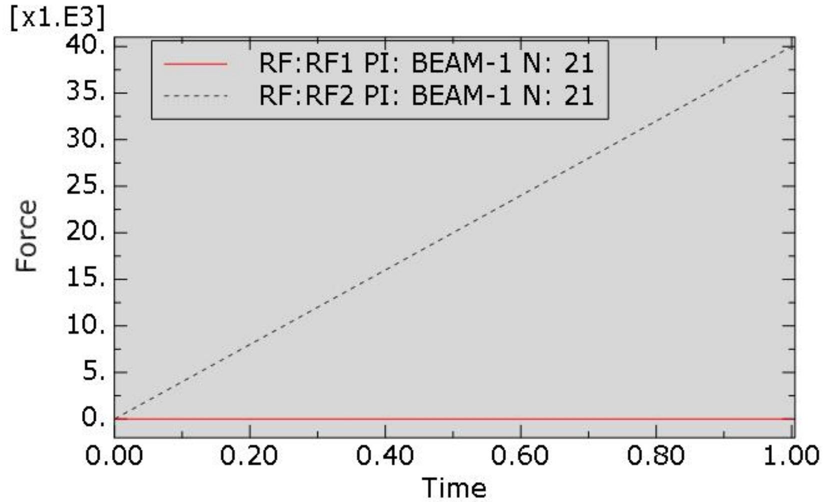
In the Position field, select **Unique Nodal** and check the **RF1** and **RF2** checkboxes.



In the Elements/Nodes tab, pick **Node sets** as method and select the **SET-RIGHT**. SET-RIGHT has been predefined and contains the right end node of the beam.



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



It can be seen that the reaction force appear in y-direction (RF2) only and there is zero reaction force in the x-direction (RF1). At the end of the step, when the load has been completely applied to the beam, reaction force in y-direction reaches the magnitude of 40kN. This is the same solution as we obtained using analytical technique.

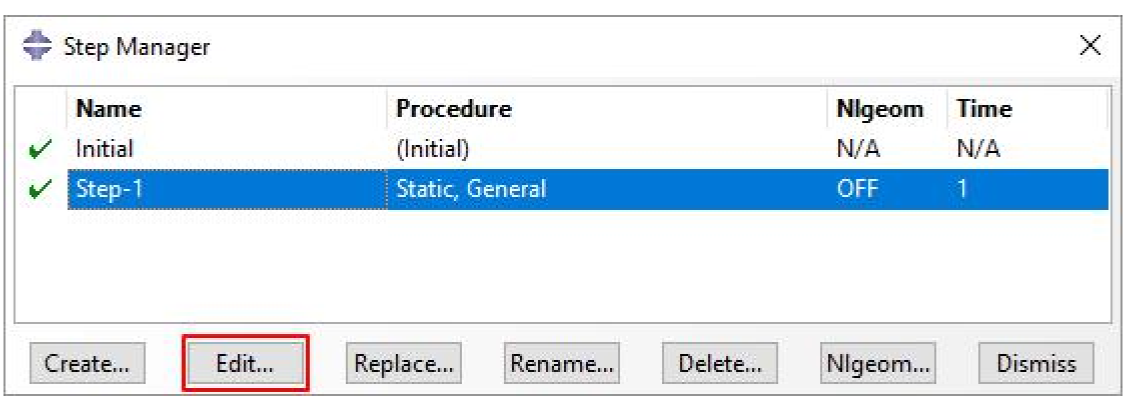
The pressure loading which is always normal to the beam, starts to have a component in the x-direction as the shape of the beam changes to a curved shape. This change in pressure direction is not accounted for in a linear analysis. For comparison purposes, we will perform a geometrically nonlinear analysis and see if geometric nonlinearities have a influence on the solution.

⇨ Performing Nonlinear Analysis

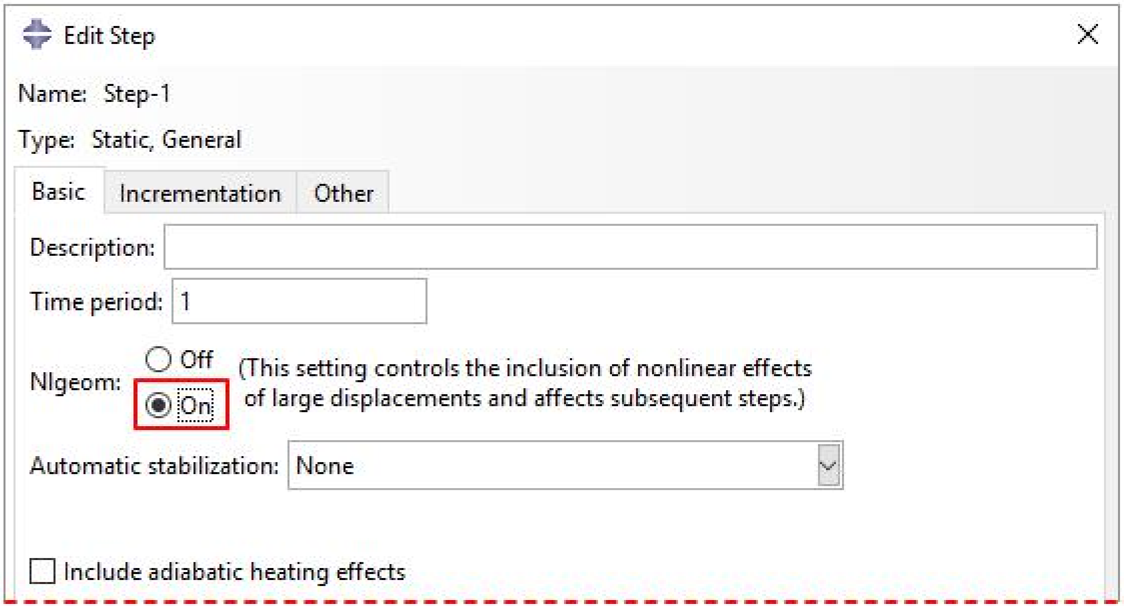
Change to **Step** module.

Open the Step Manager by picking  .

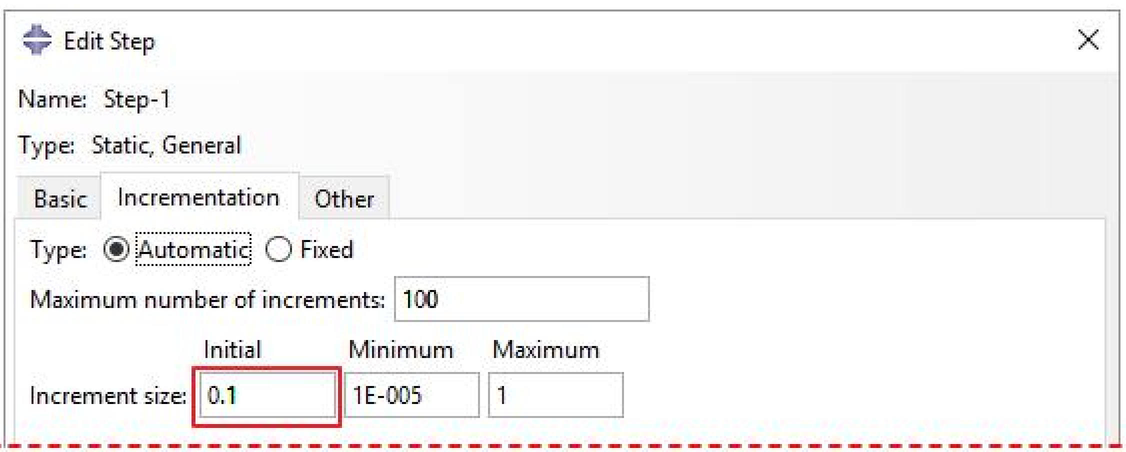
Pick the Step-1 and select **Edit**.



Turn on the Nlgeom option as shown below.



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



In a nonlinear problem the solution is found by applying the load gradually and incrementally. Therefore Abaqus breaks the simulation into a number of load incompetents. The ratio of the initial time increment to the total time period specifies the proportion of load applied in the first increment.

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

Job Submission

All the information required for analysis has been set up. 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**.

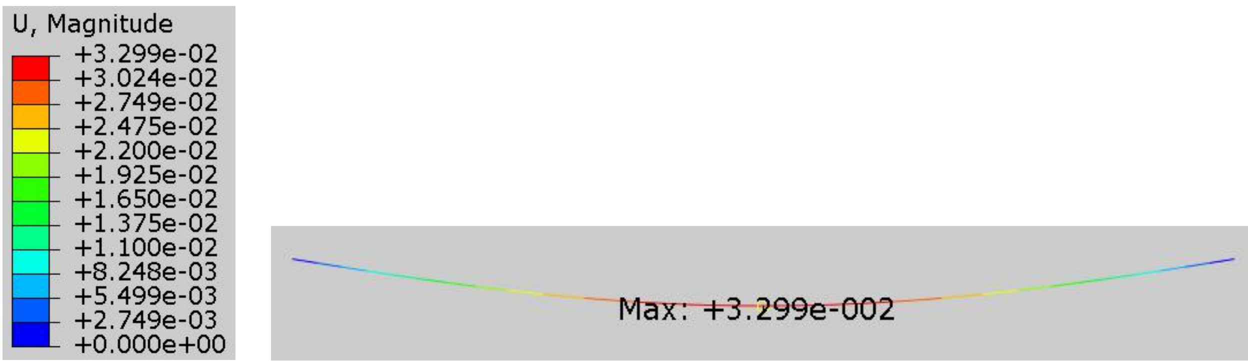
Pick **Submit** to submit the job for analysis.

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

Postprocessing

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

The simulation results will appear as shown below (Scale factor is set to 3).



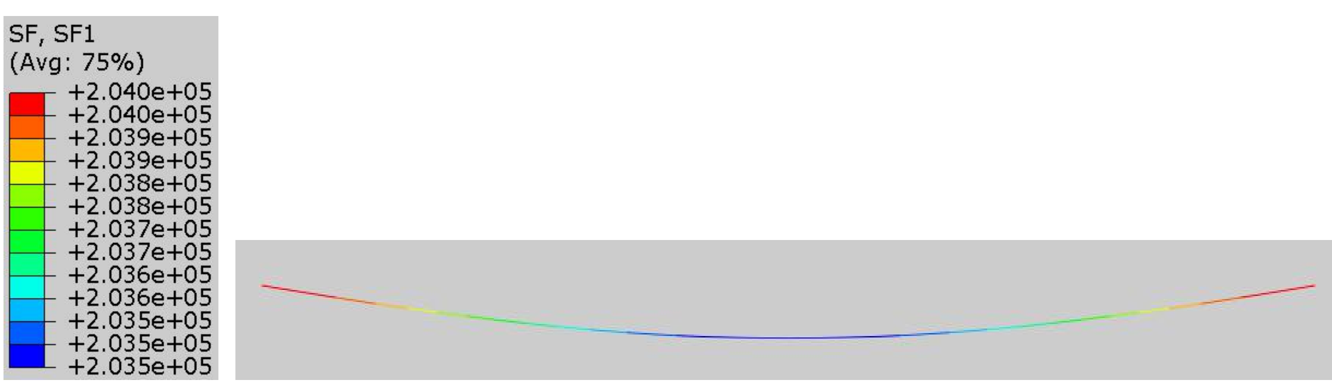
It can be seen that midspan deflection is approximately 32.9mm which is significantly less than the deflection with linear analysis.

As the load is applied, the shape of the beam changes to a curved 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 beam, starts to have a component in the x-direction as the beam bends. In a nonlinear analysis, normals to boundaries are updated and pressure load take the deformation into account. As a consequence of this, simulation results are different for the linear and nonlinear analysis.

Now we will plot the axial force in the beam.

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

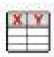
The simulation results will appear as shown below (Scale factor is set to 3).



It can be seen that a significant axial force has developed in the beam, similar to a wire in tension. Initially the beam resists the pressure load with bending stiffness but after the pressure causes the beam to be curved, the deformed geometry exhibits additional stiffness and axial forces appear in the beam. Membrane effects cause some of the load to be carried by axial forces rather than by bending alone.

As the load increases on beam there will be membrane forces in addition to bending moments.

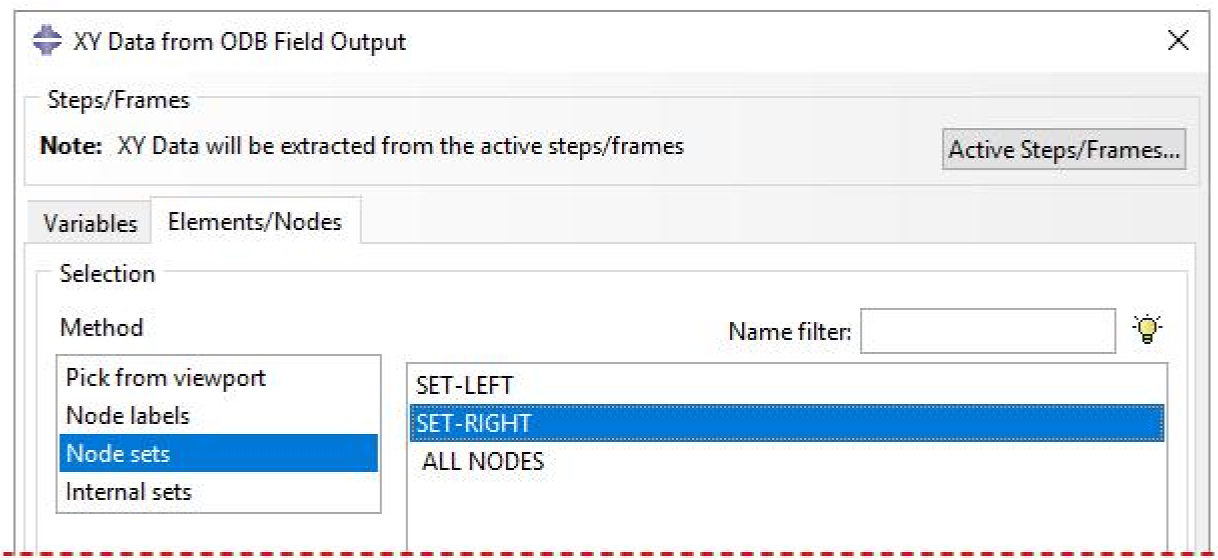
Now we will plot reaction forces at the end nodes where boundary conditions are defined.

Pick  and Create XY Data dialog box will appear.

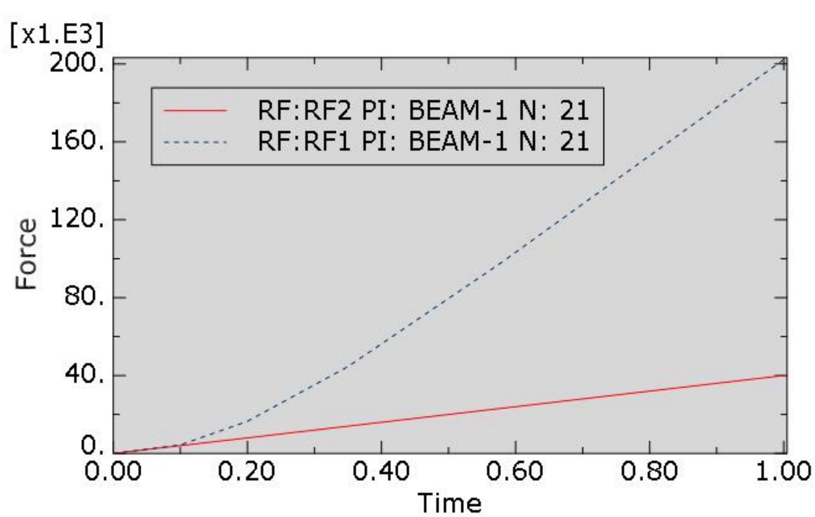
Pick **ODB field output > Continue**

In the Position field, select **Unique Nodal** and check the **RF1** and **RF2** checkboxes.

In the Elements/Nodes tab, pick **Node sets** as method and select the **SET-RIGHT**. SET-RIGHT has been predefined and contains the right node of the beam.



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



It can be seen that reaction forces also develop in the x-direction. This could be related to the presence of axial forces in the beam.

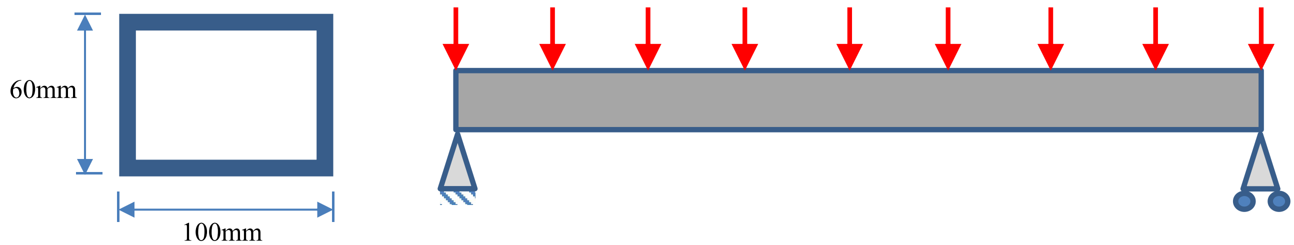
Although the deflection of the beam's middle point is small compared to the overall dimension of the beam but still geometric nonlinearity has significant influence on the final solution as compared to the linear solution.

It can be said that boundary conditions have significant influence on the membrane forces in the beam. These membrane forces make the structure stiffer which affects the deflection of the structure significantly.

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

Exercise 3

In this exercise we will analyze the beam shown schematically in the figure below.

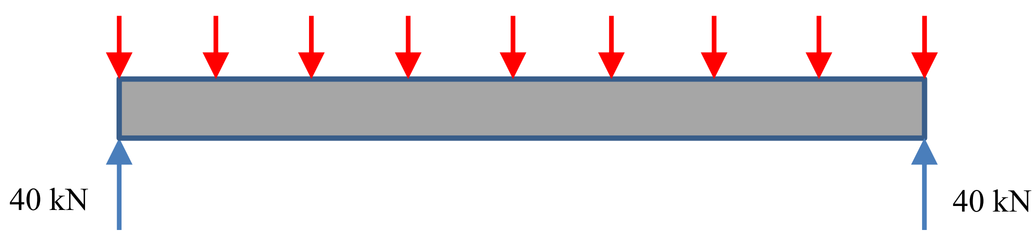


Pick **File > Set Work Directory** and set the work directory to the **BeamPinnedRoller** folder

Open the model database **BeamPR.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 2m and hollow cross section of 100mm x 60mm. It is modeled as one-dimensional line (beam elements) instead of three-dimensional continuum elements. It is assumed to be made from steel with a Young's modulus of 200 GPa and a Poisson's ratio of 0.3. It is meshed with 3-node quadratic beam elements (B22). A uniform pressure of 40000 N/m is applied.

The following diagram shows the reactions at the supports computed using the analytical technique.



$$Deflection = \frac{5}{384}\frac{pL^4}{EI} = 0.0483$$

We will perform a finite element analysis and compare the results with analytical solution.

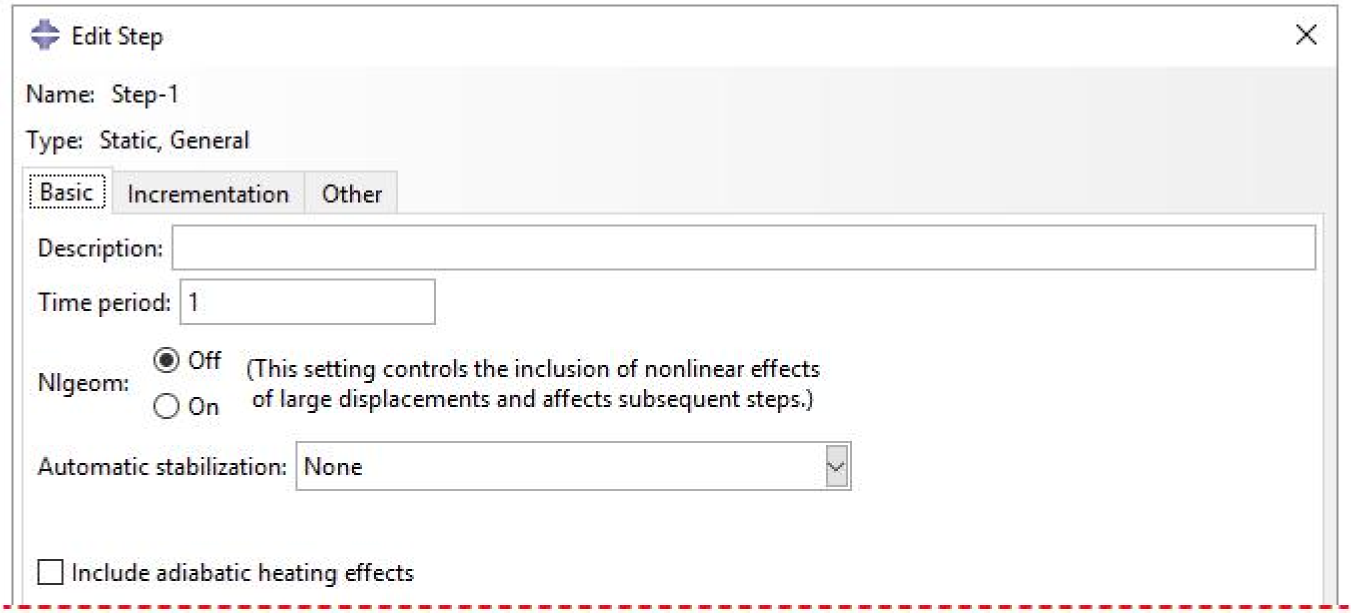
⇨ Defining Step

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. Notice that by default total time period is set to **1.0** and NLgeom option is set to **Off**.



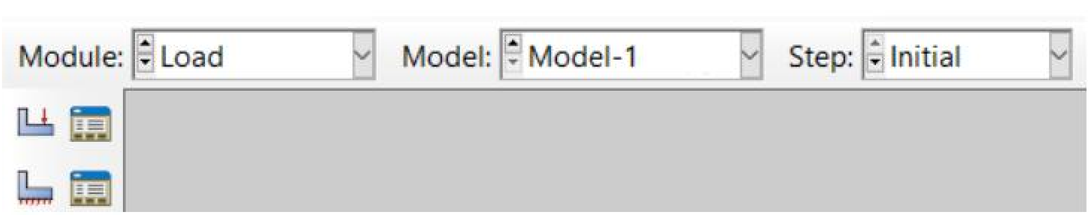
We assume that changes in geometry are small during the loading of beam and a linear analysis will provide accurate results. So we leave NLgeom option off.

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

Defining Load

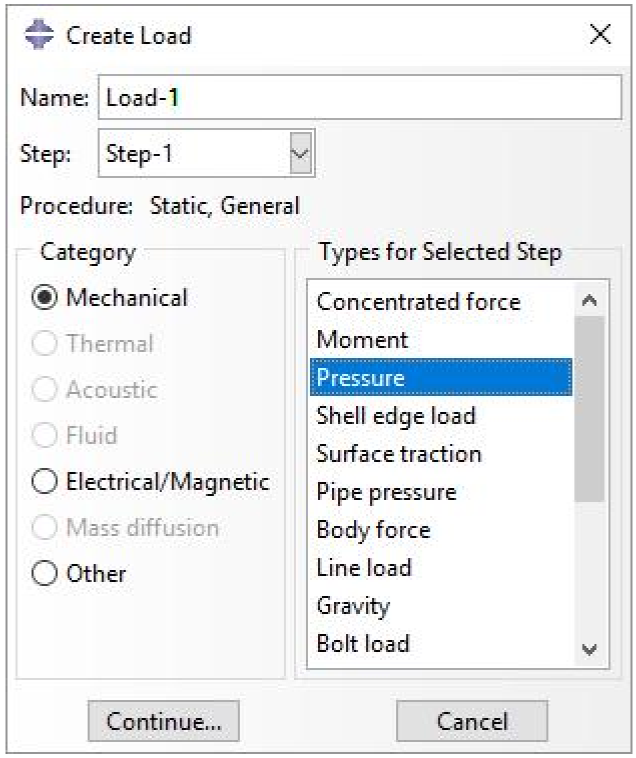
A uniform pressure load will be applied across the top of the beam.

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



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

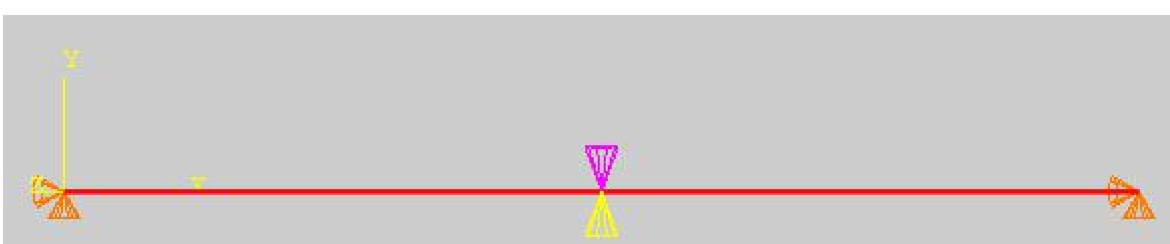
Select **Pressure** in the Types field and pick **Continue.**



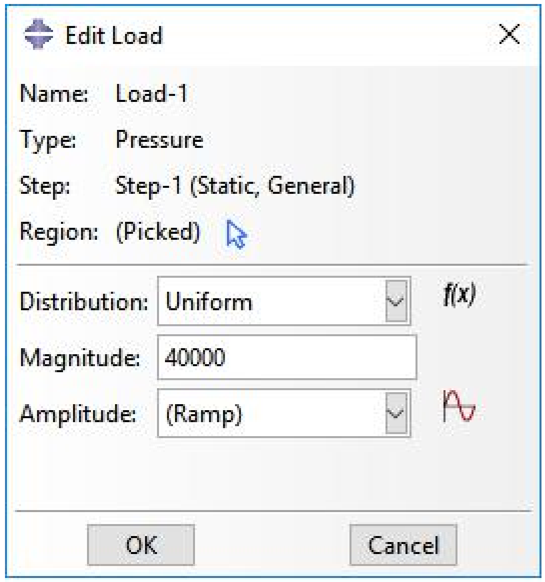
Select the geometry of the beam as shown in figure below.

Pick **Done** to accept the selection.

Pick **Magenta** to specify the top side of the beam for applying pressure load.



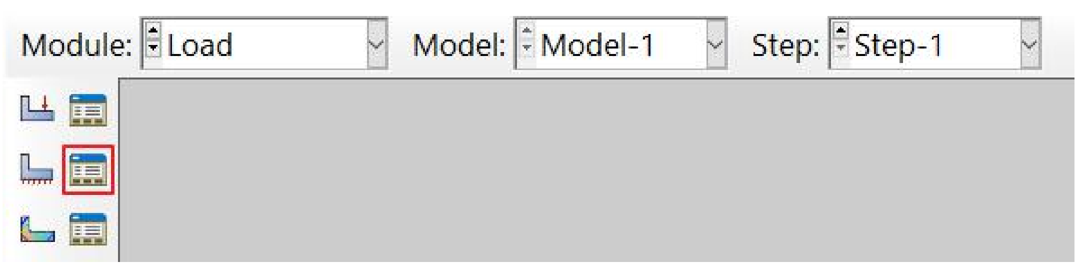
Enter **40000** as the pressure magnitude.



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

Boundary Conditions

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



In the Boundary Conditions Manager it can be seen that two boundary conditions have already been created. The boundary condition “BC-Right” constrains translational degree of freedom of right end of the beam along y-axis. The boundary condition “BC-Left” constrains all translational degrees of freedom of left end of the beam.

Pick **Dismiss** to close the manager.

Field Output Request

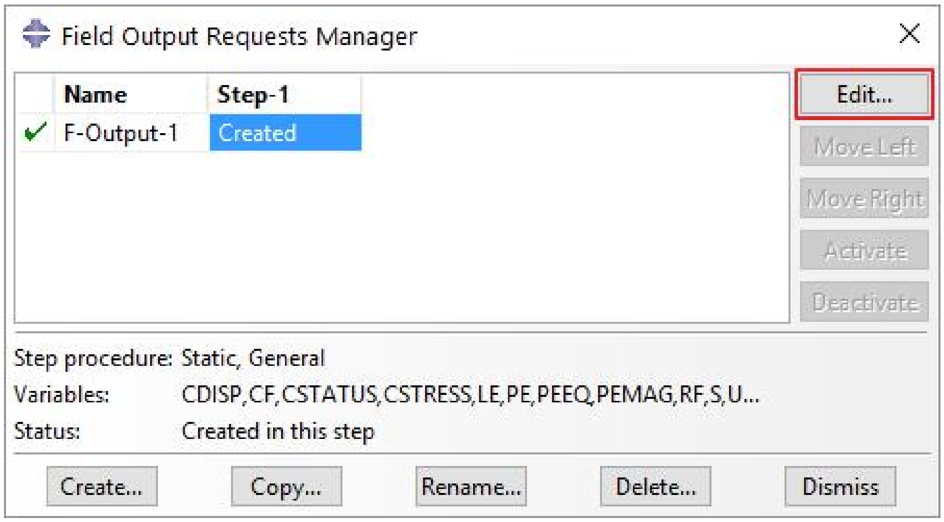
We will plot the section forces, i.e axial forces and shear forces. This information can be plotted by requesting SF output variable.

SF variable contains all section forces and moment components.

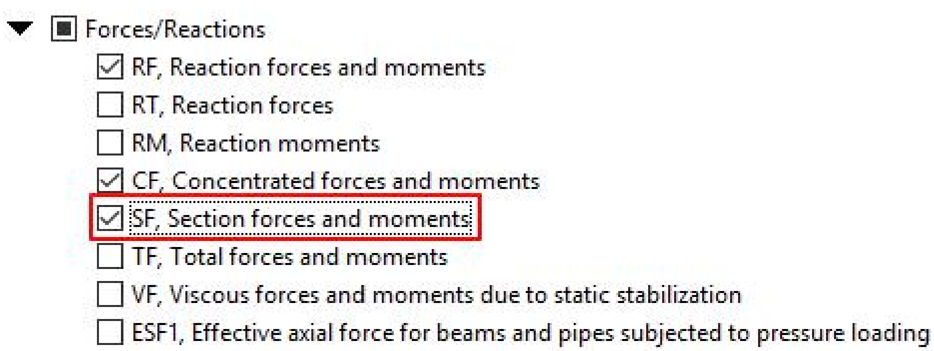
Change to **Step** module.

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).



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

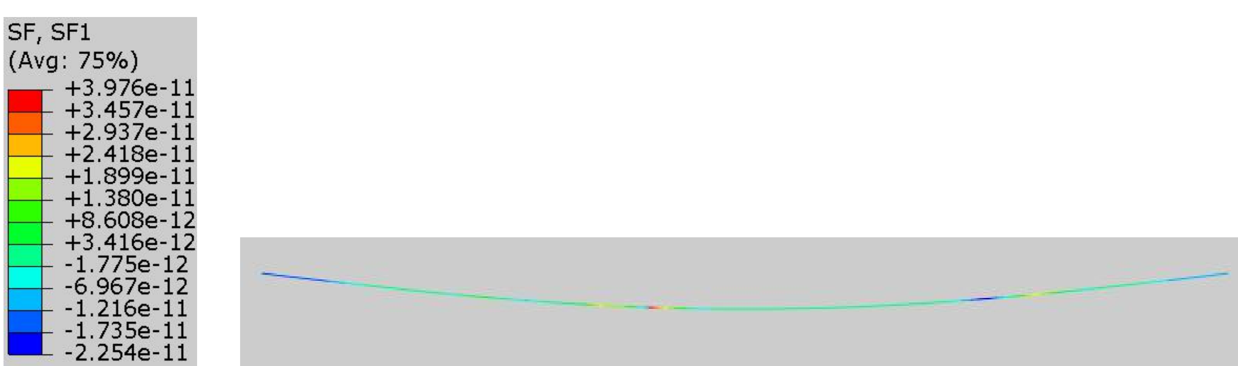
The simulation results will appear as shown below (Scale factor is set to 3).

It can be seen that midspan deflection is approximately 49.01mm which is approximately 2.45% of the beam´s length. 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. We can say that deflection of the beam is small as compared to the length of the beam.

Now we will plot the axial force in the beam.

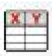
Select **SF** as output variable in the Field Output toolbar and make sure that **SF1** is selected as component.

The simulation results will appear as shown below (Scale factor is set to 3).

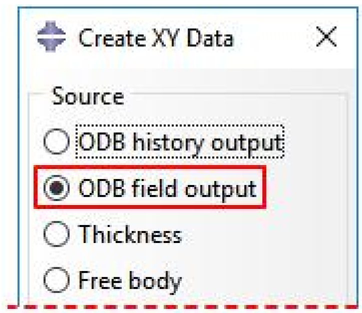


It can be seen that magnitude of the axial force is almost zero.

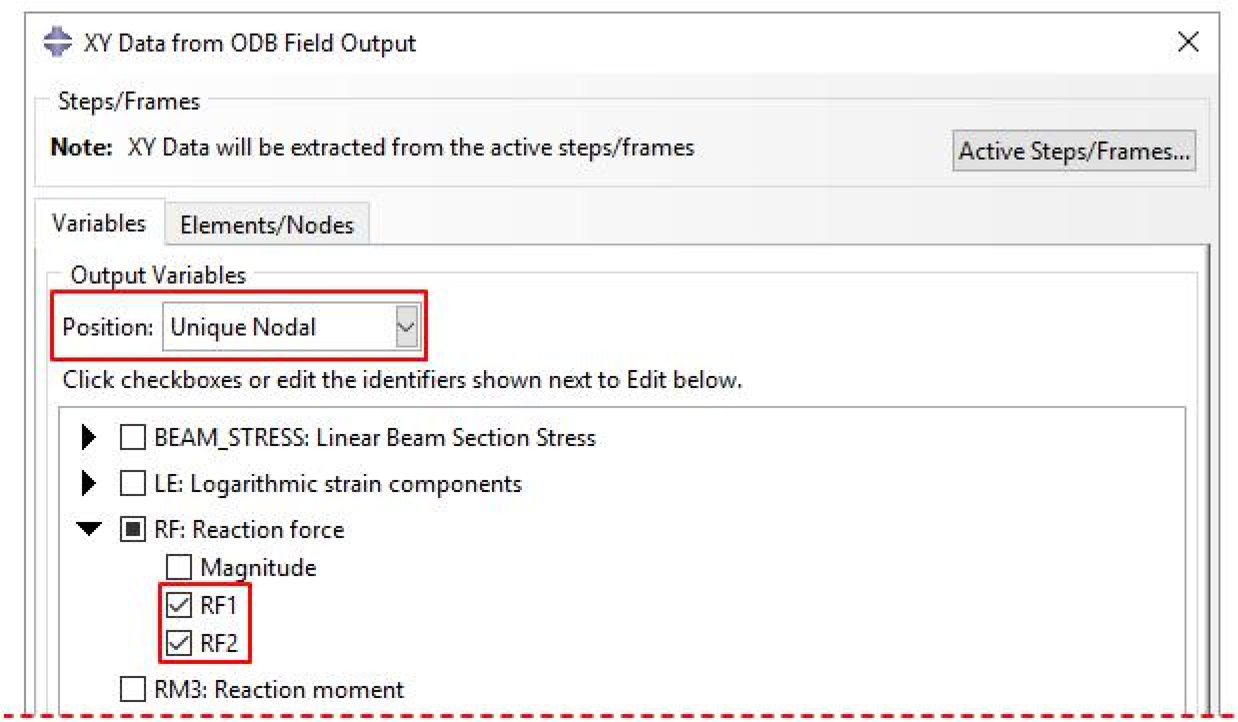
Now we will plot reaction forces at the end nodes where boundary conditions are defined.

Pick  and Create XY Data dialog box will appear.

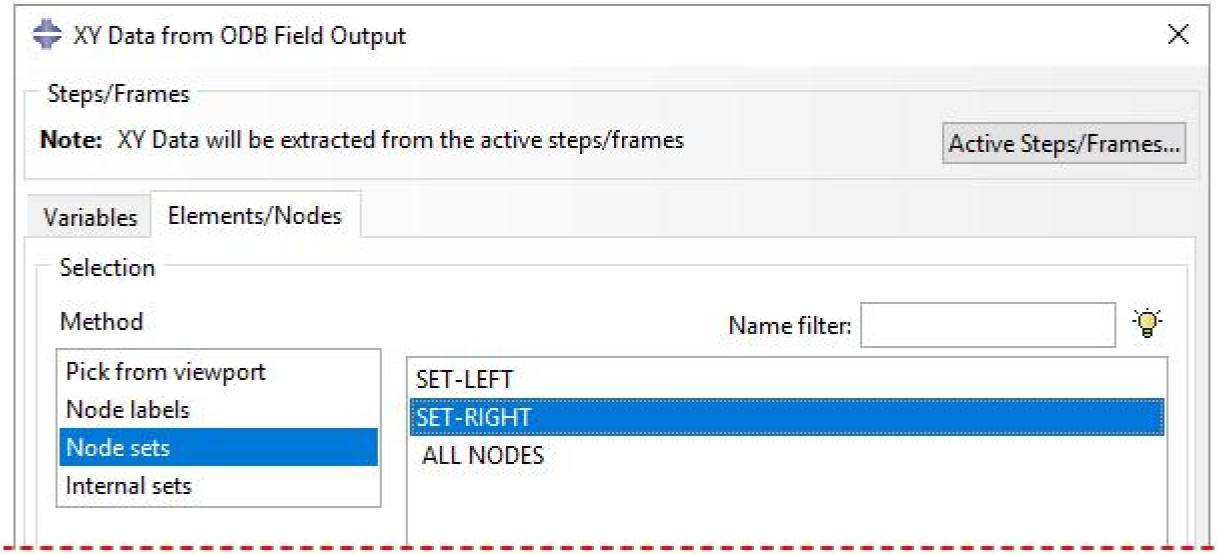
Pick **ODB field output > Continue**



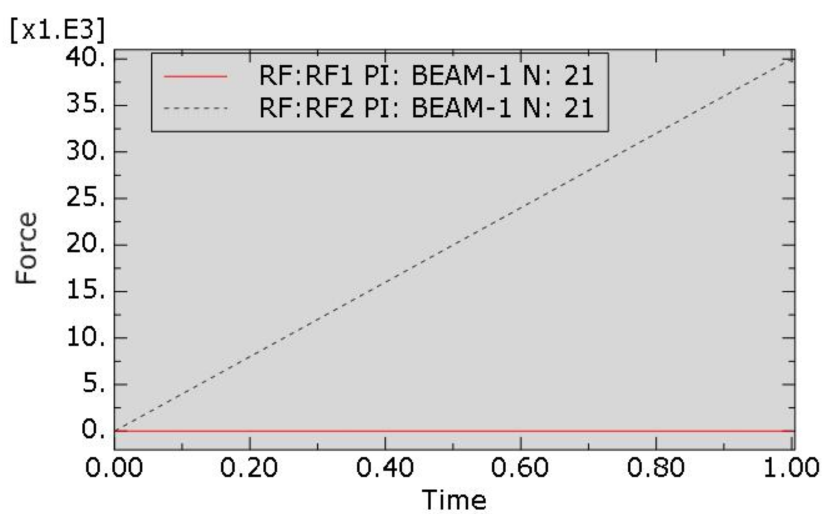
In the Position field, select **Unique Nodal** and check the **RF1** and **RF2** checkboxes.



In the Elements/Nodes tab, pick **Node sets** as method and select the **SET-RIGHT**. SET-RIGHT has been predefined and contains the right end node of the beam.



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



It can be seen that the reaction force appear in y-direction (RF2) only and there is zero reaction force in the x-direction (RF1). At the end of the step, when the load has been completely applied to the beam, reaction force in y-direction reaches the magnitude of 40kN. By comparing the reaction force and deflection of midspan to the values obtained from analytical solution it can be said that linear solution and analytical solution are similar.

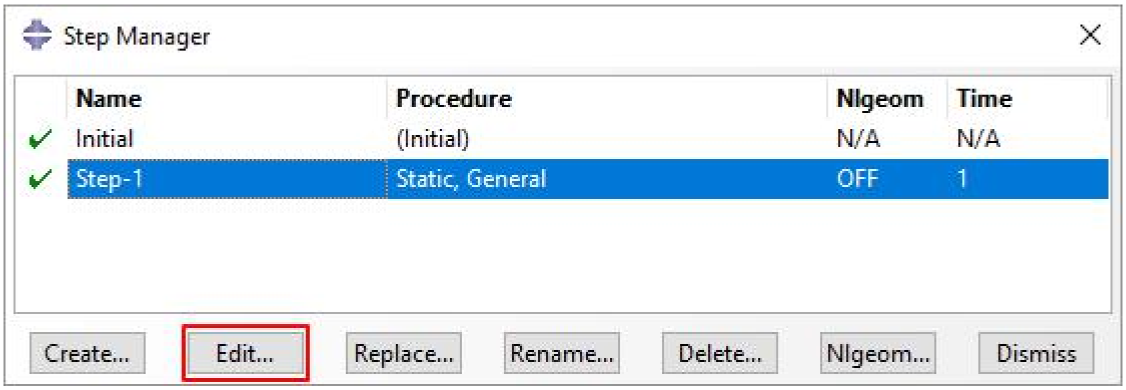
The pressure loading which is always normal to the beam, starts to have a component in the x-direction as the shape of the beam changes to a curved shape. This change in pressure direction is not accounted for in a linear analysis. For comparison purposes, we will perform a geometrically nonlinear analysis and see if geometric nonlinearities have a influence on the solution.

⇨ Performing Nonlinear Analysis

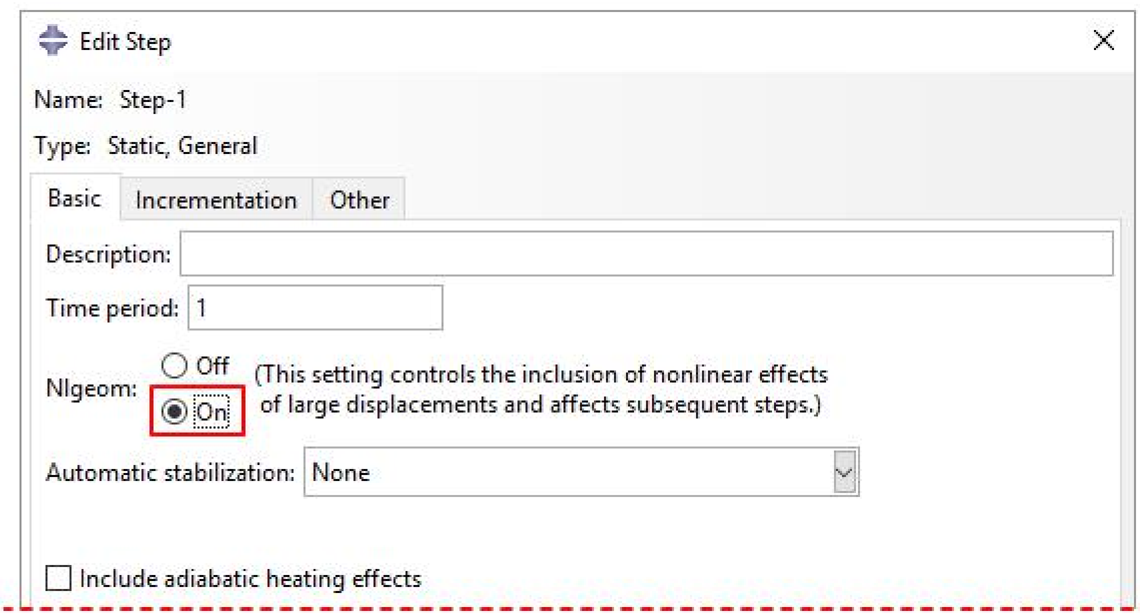
Change to **Step** module.

Open the Step Manager by picking  .

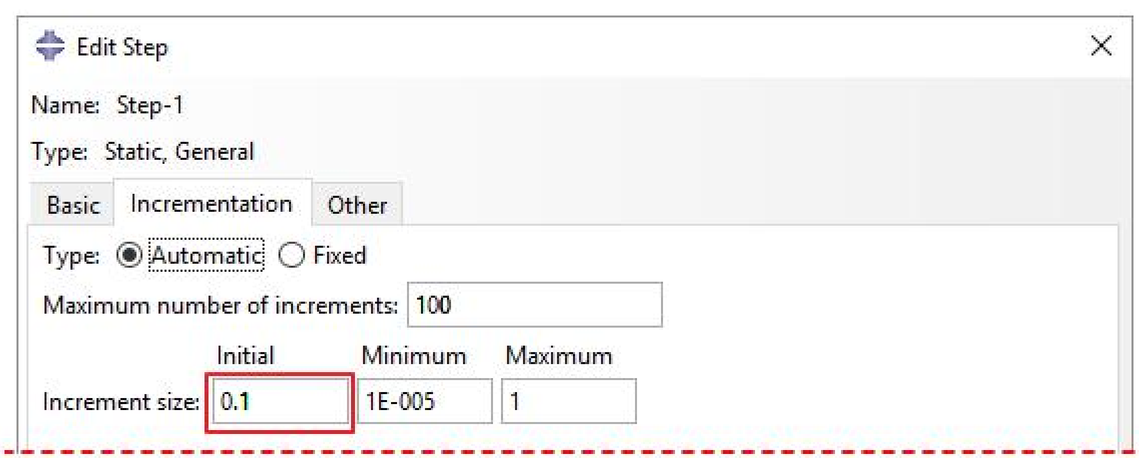
Pick the Step-1 and select **Edit**.



Turn on the Nlgeom option as shown below.



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



In a nonlinear problem the solution is found by applying the load gradually and incrementally. Therefore Abaqus breaks the simulation into a number of load incompetents. The ratio of the initial time increment to the total time period specifies the proportion of load applied in the first increment.

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

Job Submission

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

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

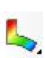
Pick **Create** and create a new job named Pressure_NonLinear or whatever name you would like.

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

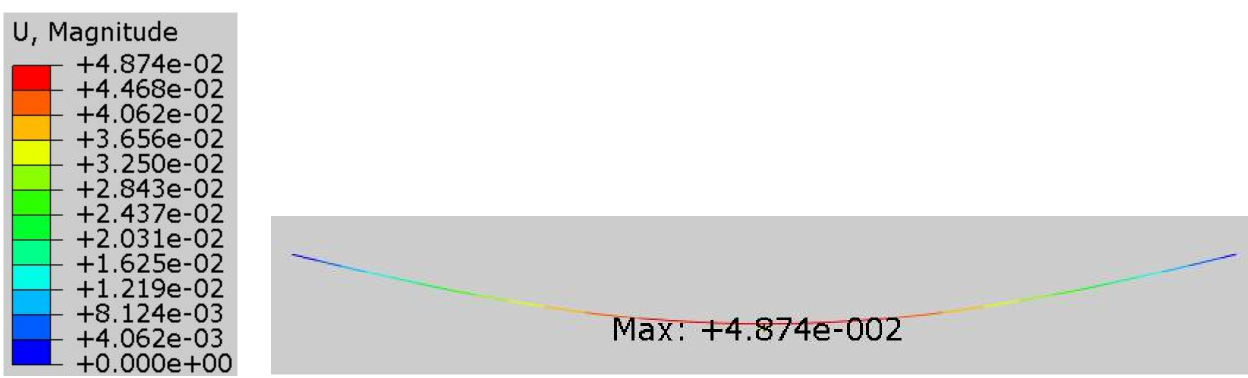
Pick **Submit** to submit the job for analysis.

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

Postprocessing

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

The simulation results will appear as shown below (Scale factor is set to 3).

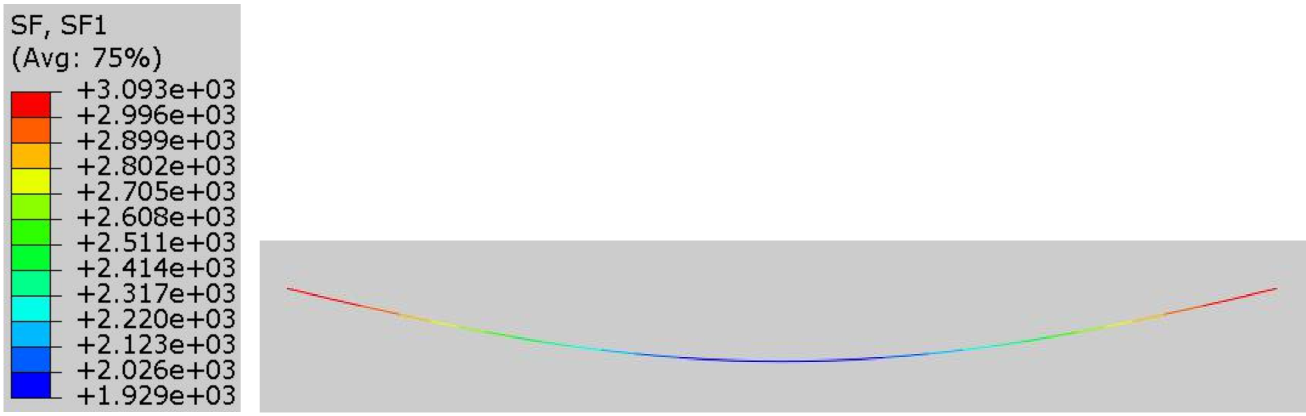


It can be seen that midspan deflection is approximately 48.7mm which is approximately equal to the deflection with linear analysis. As compared to the previous exercise where right end was pinned, this is a big change. It could be attributed to the free movement of right end along x-axis as compared to the previous exercise.

Now we will plot the axial force in the beam.

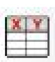
Select **SF** as output variable in the Field Output toolbar and make sure that **SF1** is selected as component.

The simulation results will appear as shown below (Scale factor is set to 3).



It can be seen that an axial force has developed in the beam, but this force is significantly lower than the case for pinned boundary condition as seen in the previous exercise. We can say that due to the freedom of the right end to move along x-axis the axial force (tension) is not as high as for the case when both ends are pinned.

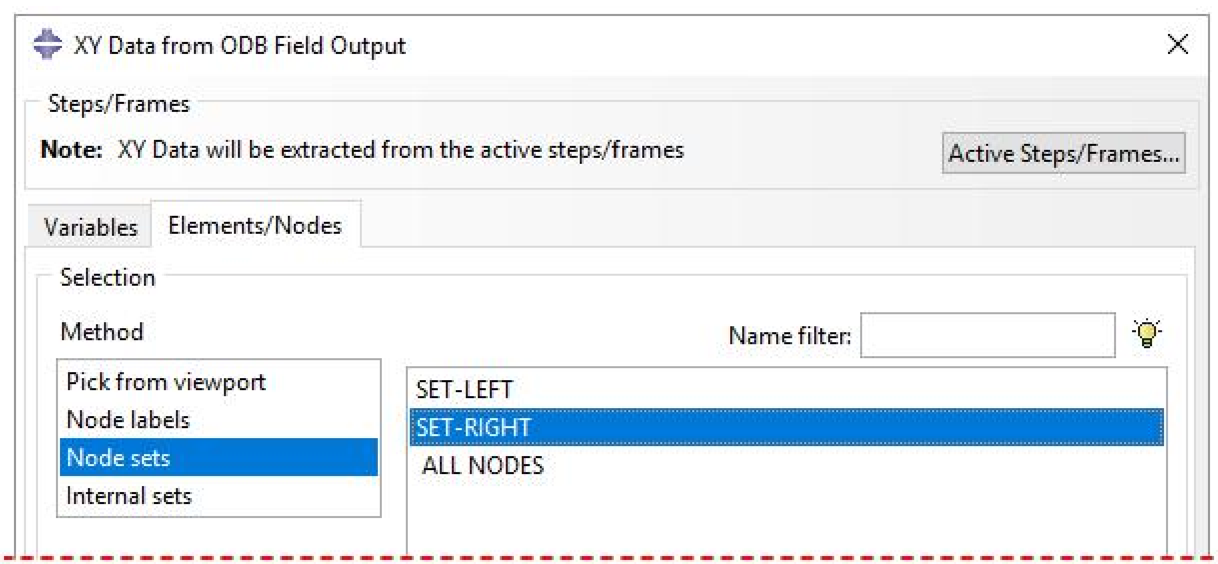
Now we will plot reaction forces at the end nodes where boundary conditions are defined.

Pick  and Create XY Data dialog box will appear.

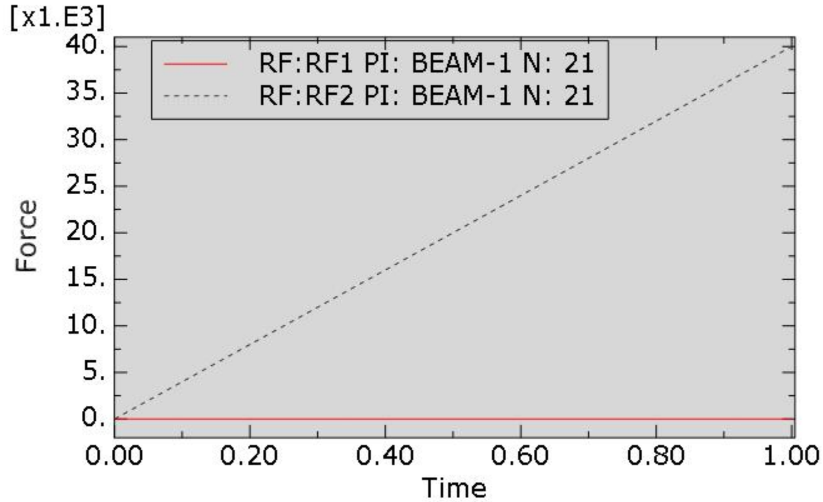
Pick **ODB field output > Continue**

In the Position field, select **Unique Nodal** and check the **RF1** and **RF2** checkboxes.

In the Elements/Nodes tab, pick **Node sets** as method and select the **SET-RIGHT**. SET-RIGHT has been predefined and contains the right node of the beam.



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



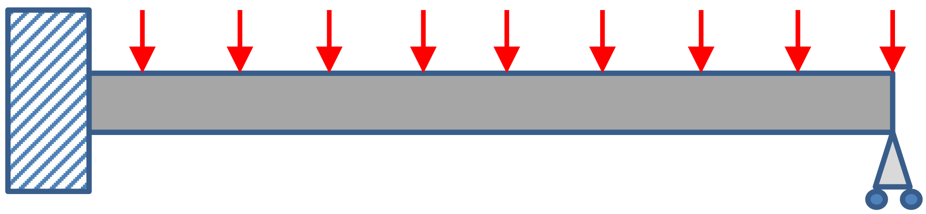
It can be seen that reaction forces do not develop in the x-direction due to roller boundary condition. A similar graph is obtained if the reaction forces are plotted on the left end of the beam.

It can be concluded that nonlinear analysis is essential to take into account the membrane forces in a structure. By comparing the results with previous exercise, it can also be said the membrane forces depend greatly on the boundary conditions

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

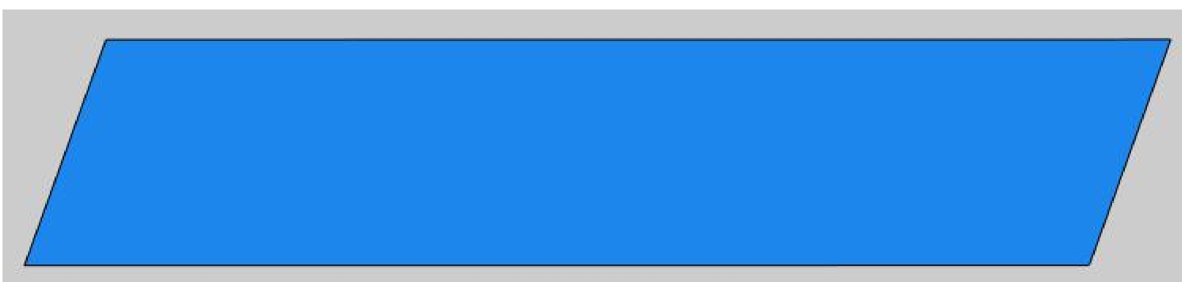
Exercise 4

In this exercise we will analyze a plate shown schematically in the figure below under the loading of a uniform pressure.



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

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

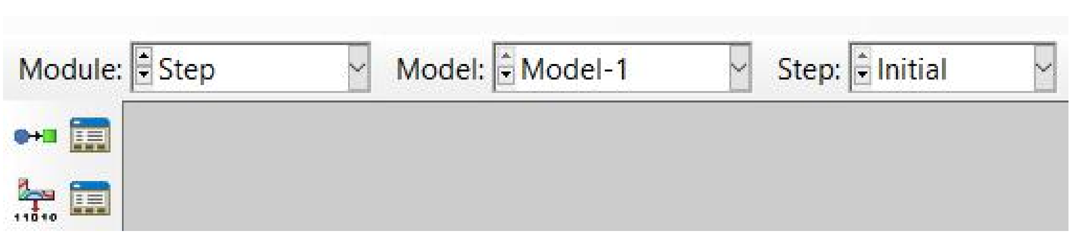


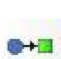
The model consists of a single part, Plate. The plate shown in above figure has a length of 1.0m, a width of 0.3m and thickness is 6mm. It has a thickness-to-minimum span ratio of 0.02 (0.006/0.3) and hence considered thin. It is assumed to be made from aluminum with a Young's modulus of 70GPa and a Poisson's ratio of 0.3. The plate is modeled using planar shell feature. We assume that strains will be small. So it is meshed with quadratic shell elements with reduced integration (S8R5). The plate is loaded with a uniform pressure of 20 kPa.

Defining Step

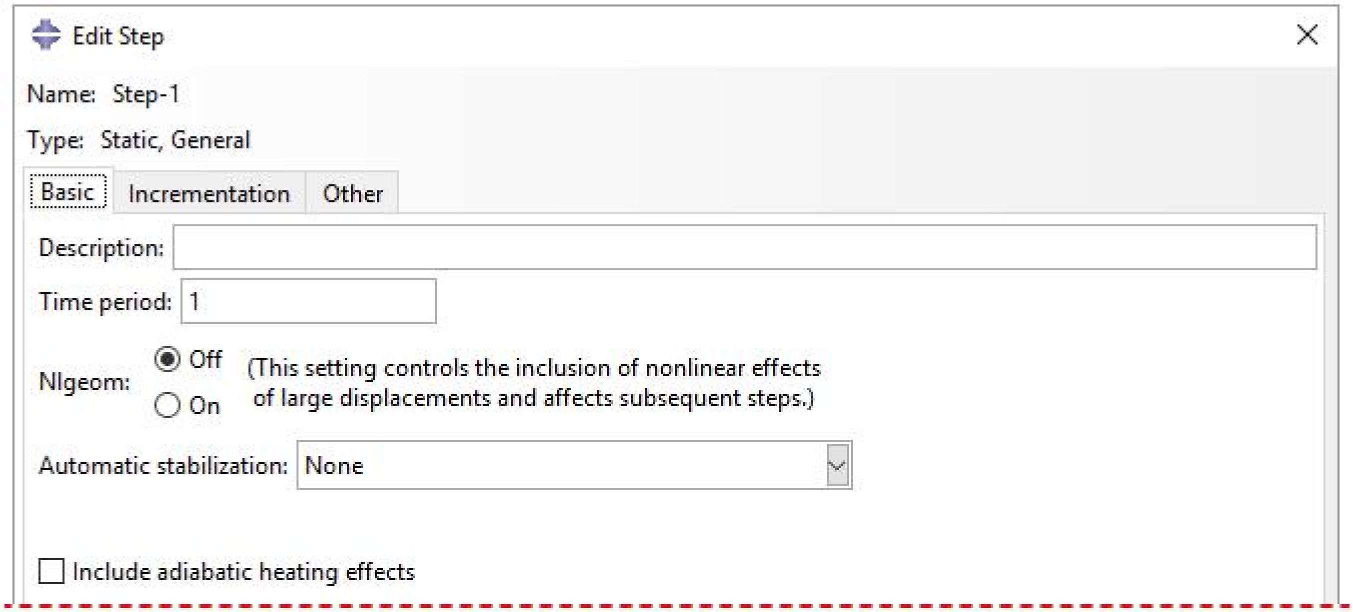
We assume that load is applied slowly such that inertia effects can be neglected. So analysis will be performed using the Static, General procedure.

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. Notice that by default total time period is set to **1.0** and NLgeom option is set to **Off**.



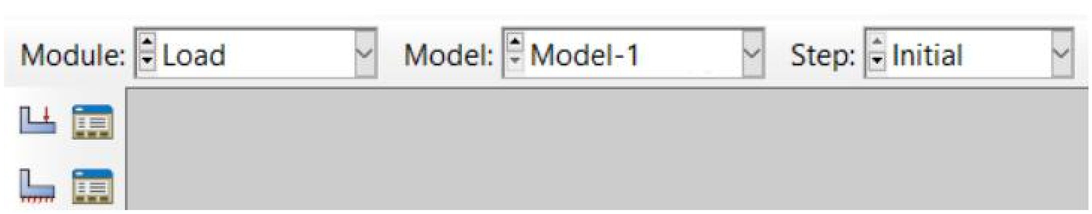
We assume that changes in geometry are small during the loading and a linear analysis will provide accurate results. So we leave NLgeom option off.

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

Defining Load

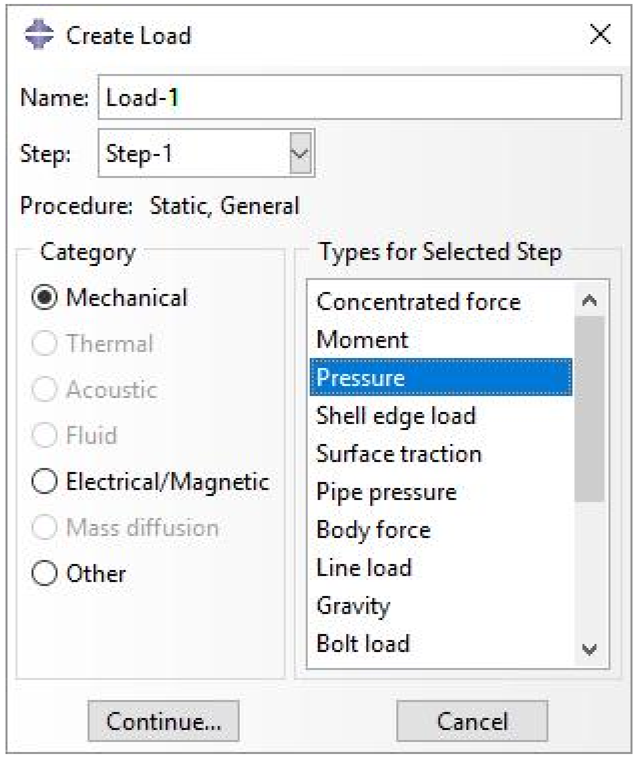
A uniform pressure load will be applied across the top of the plate.

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

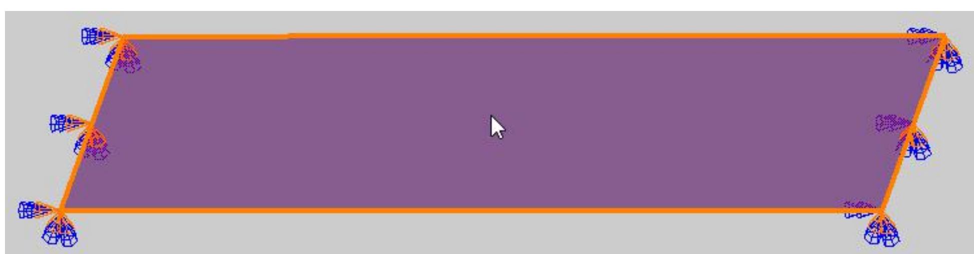


Pick  to create a load and select Step-1 in the Step field.

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



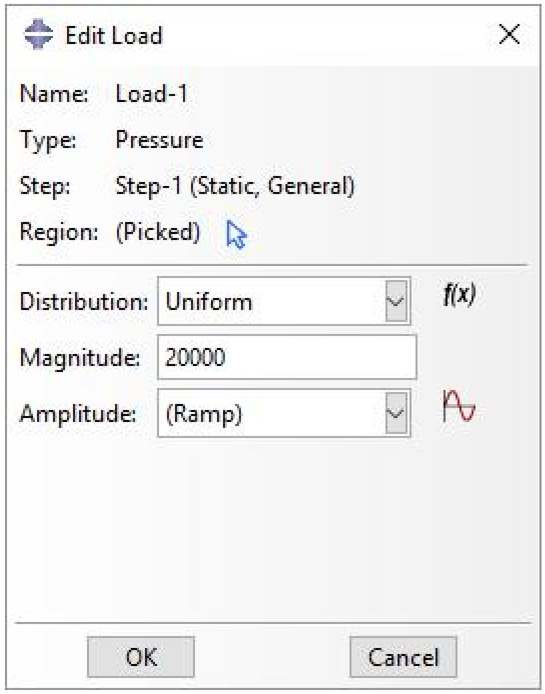
Select the surface of the plate as shown in figure below.



Pick **Done** to accept the selection.

Pick **Brown** to specify the top side of the plate for applying pressure load.

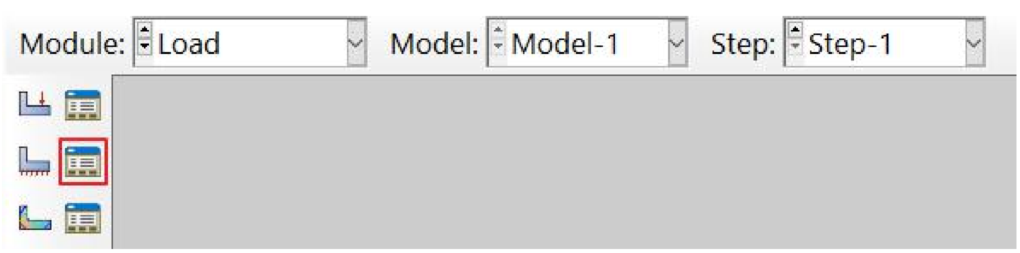
Enter **20000** as the pressure magnitude.



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

Boundary Conditions

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



In the Boundary Conditions Manager it can be seen that two boundary conditions have already been created. The boundary condition “BC-Right” constrains all translational and rotational degrees of freedom of right edge of the plate except the translation in x-direction. The boundary condition “BC-Left” constrains all degrees of freedom of left edge of the plate.

Pick **Dismiss** to close the manager.

Field Output Request

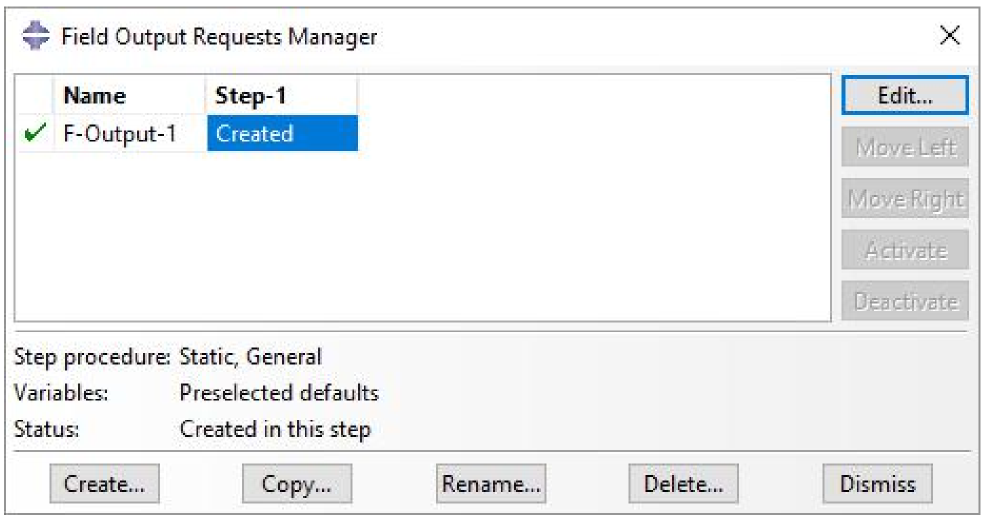
We will plot the section forces, i.e membrane forces and shear forces. This information can be plotted by requesting SF output variable.

SF variable contains all section forces and moment components.

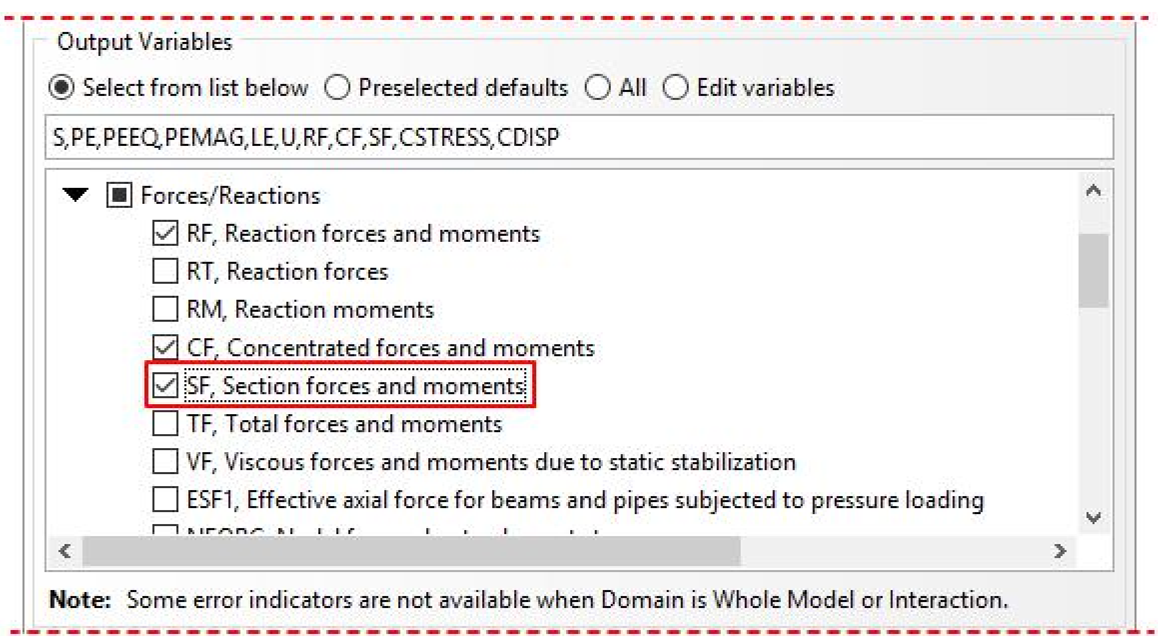
Change to **Step** module.

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).



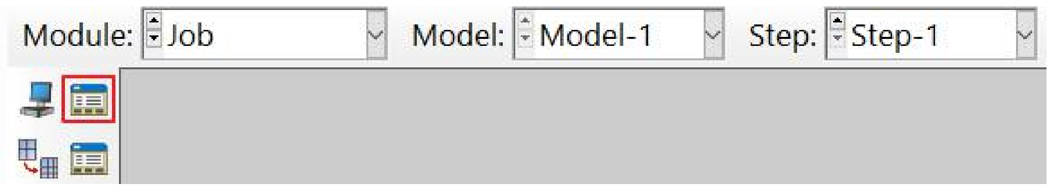
Pick **OK** to apply and exit.

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.

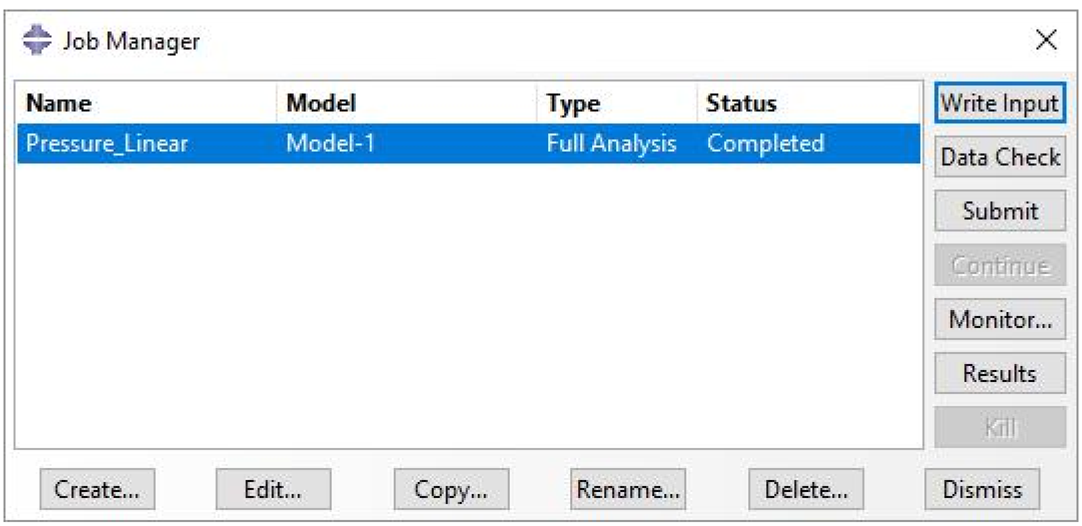
So change to **Job** module and pick  to open the job manager.



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.

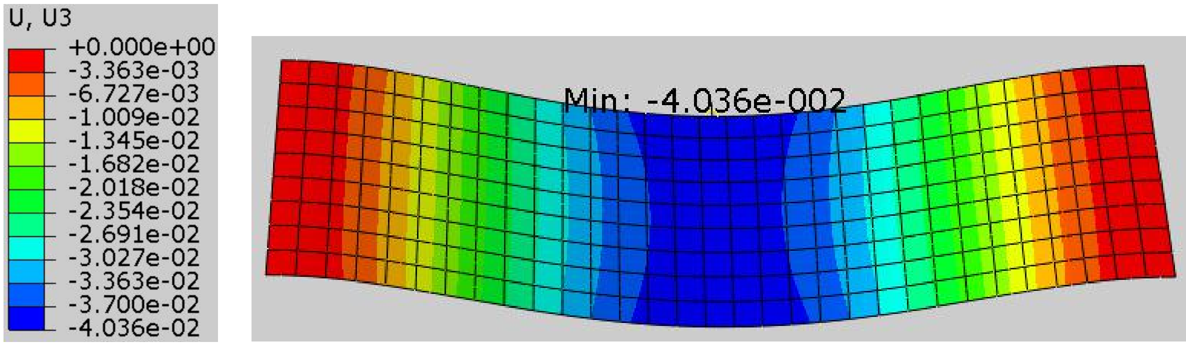
Postprocessing

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

Pick **U3** in the component field.

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

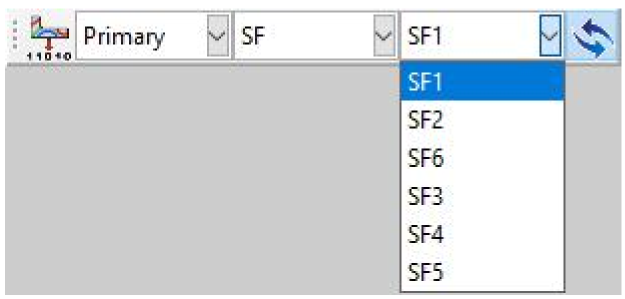
The simulation results will appear as shown below (Scale factor is set to 2).



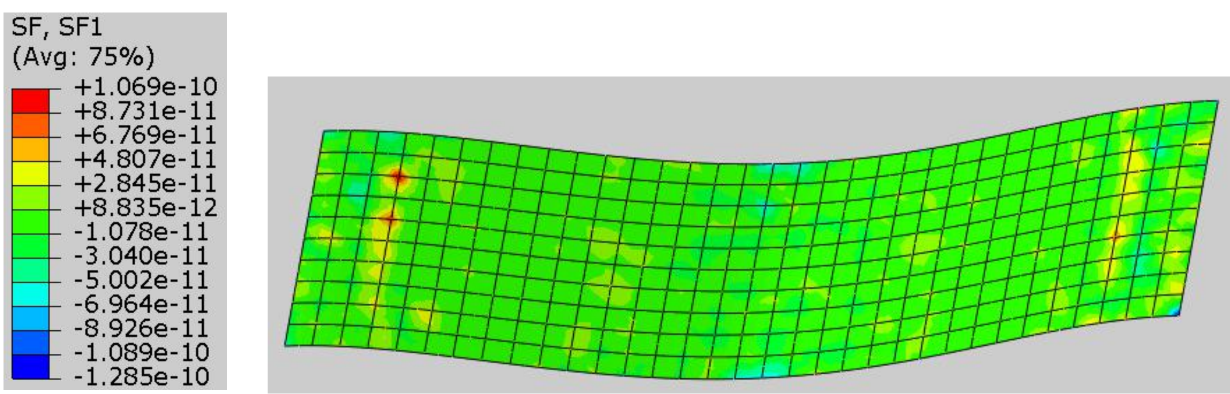
It can be seen that midspan deflection is about 40.4mm which is approximately 4% of the plates´s length. Remember that we did not turn on the NLgeom option which means that analysis did not take into account the geometric nonlinearity in the structure. We can say that deflection of the plate is small as compared to the length of the plate.

Now we will plot the membrane force in the plate.

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



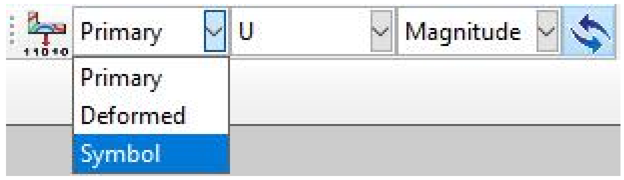
The simulation results will appear as shown below (Scale factor is set to 2).



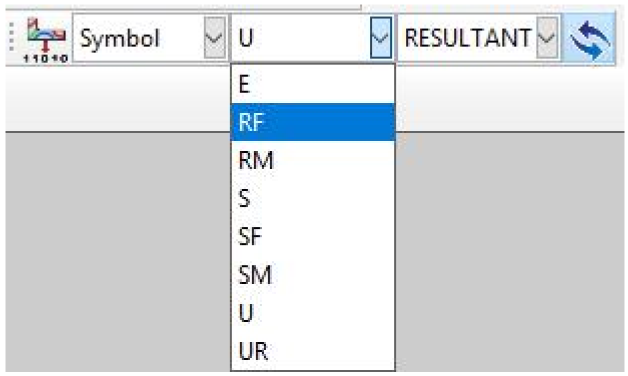
It can be seen that magnitude of the membrane force is almost zero. It is expected as the applied pressure acts in the z-direction.

Now we will plot reaction forces at the end nodes where boundary conditions are defined.

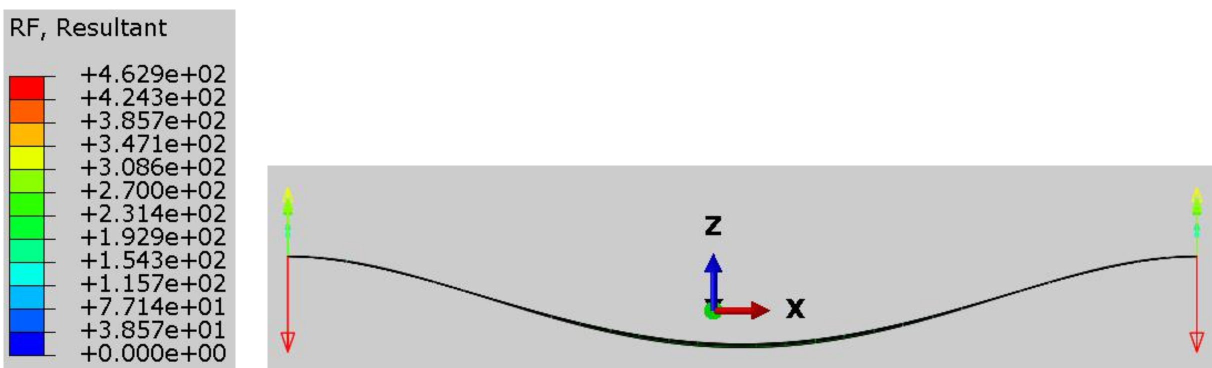
Select **Symbol** as category in the Field Output toolbar.



Select **RF** 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 2).



It can be seen that the reaction force appear in z-direction (RF3) only and there is zero reaction force in the x and y-directions (It can be confirmed by plotting RF1 and RF2 components).

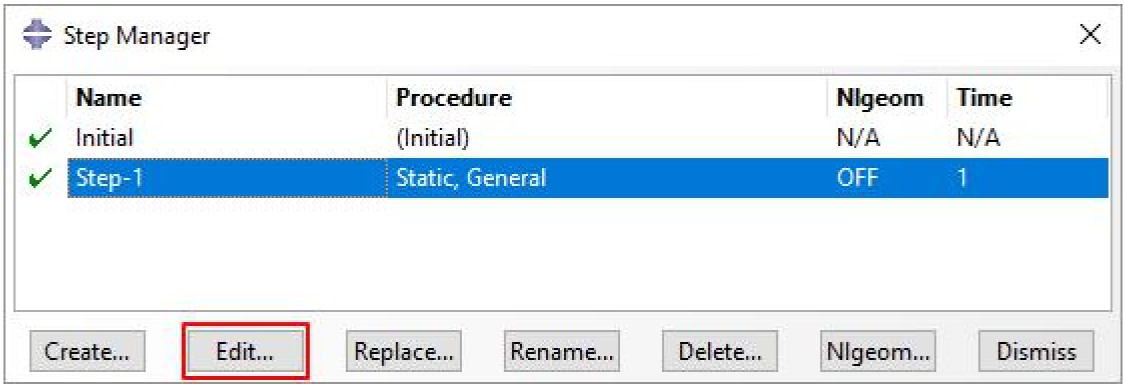
The pressure loading which is always normal to the plate, starts to have a component in the x-direction as the plate changes to a curved shape. This change in pressure direction is not accounted for in a linear analysis. For comparison purposes, we will perform a geometrically nonlinear analysis and see if geometric nonlinearities have a influence on the solution.

Performing Nonlinear Analysis

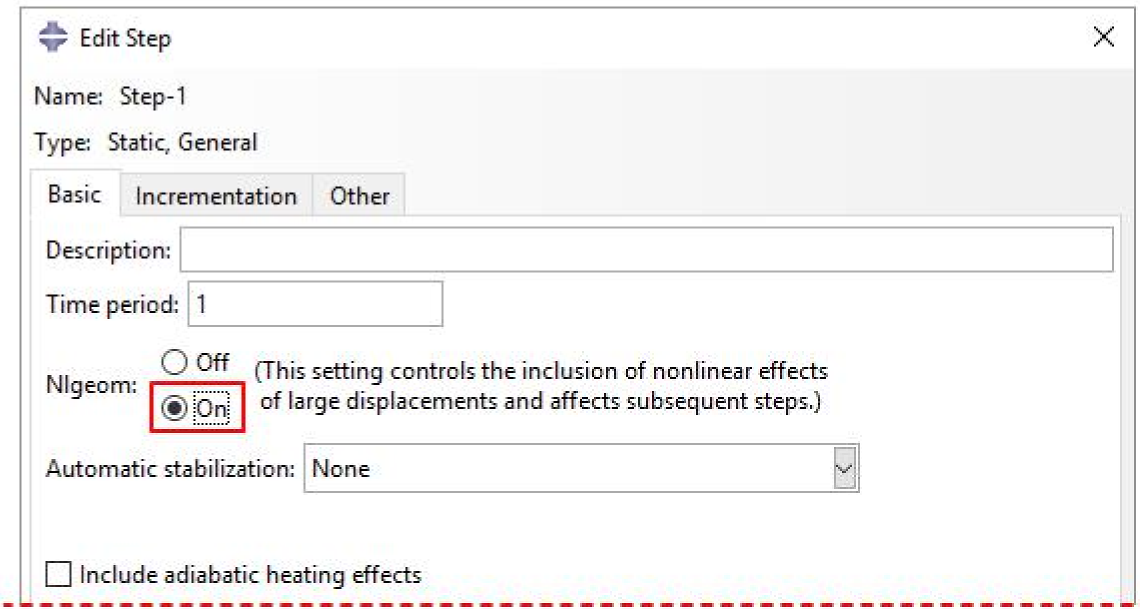
Change to **Step** module.

Open the Step Manager by picking  .

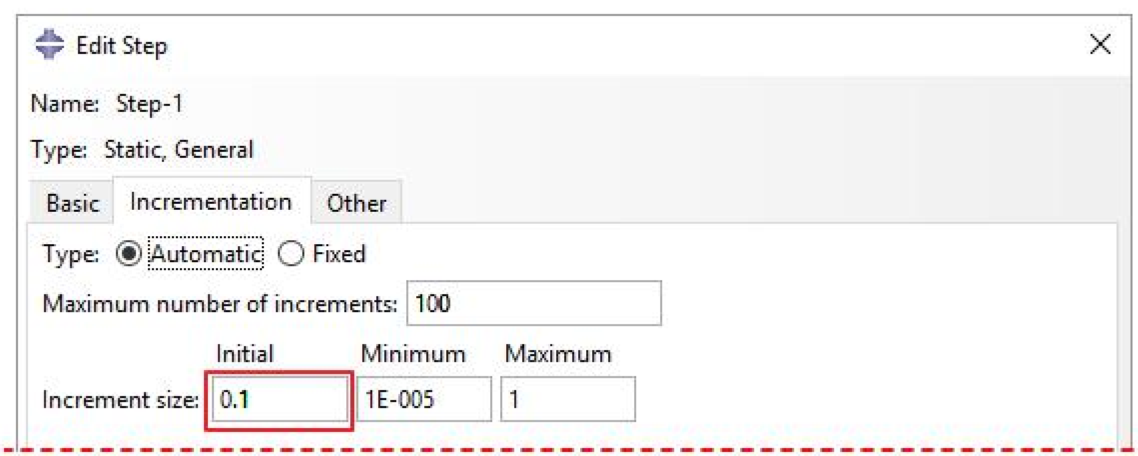
Pick the Step-1 and select **Edit**.



Turn on the Nlgeom option as shown below.



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



In a nonlinear problem the solution is found by applying the load gradually and incrementally. Therefore Abaqus breaks the simulation into a number of load incompetents. The ratio of the initial time increment to the total time period specifies the proportion of load applied in the first increment.

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.

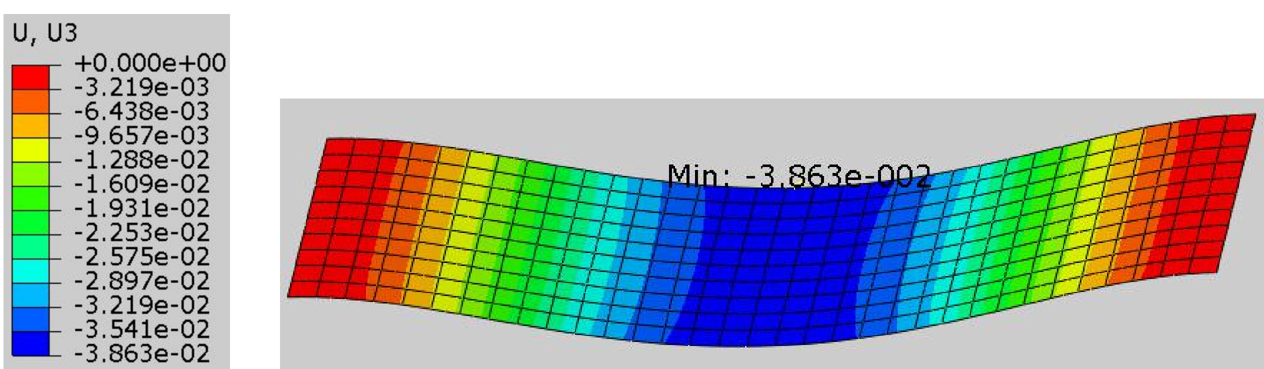
Postprocessing

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

Pick **U3** in the component field.

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

The simulation results will appear as shown below (Scale factor is set to 2).



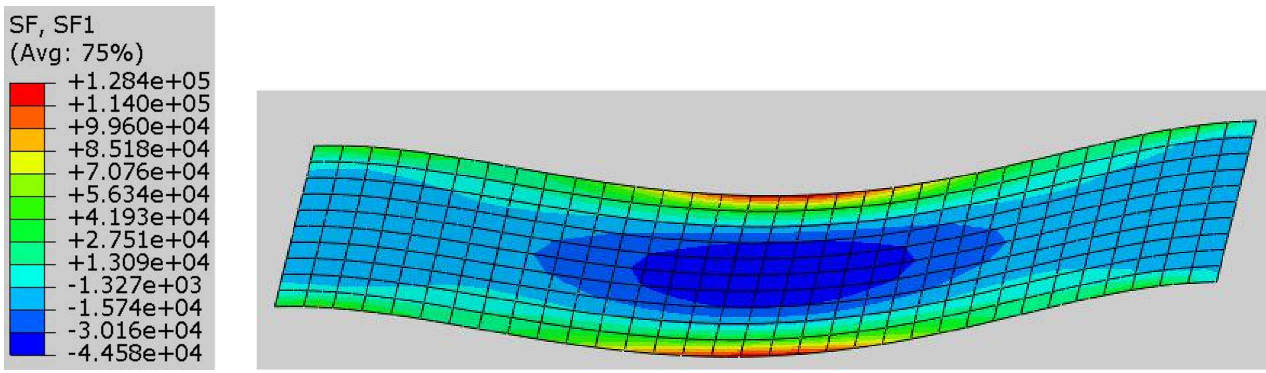
It can be seen that midspan deflection is approximately 38.6mm which is less than the deflection with linear analysis although difference is minute.

As the load is applied, the shape of the plate changes to a curved 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 plate, starts to have a component in the x-direction as the plate bends. In a nonlinear analysis, normals to boundaries are updated and pressure load takes the deformation into account. As a consequence of this, simulation results are different for the linear and nonlinear analysis.

Now we will plot the membrane force in the plate.

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 2).

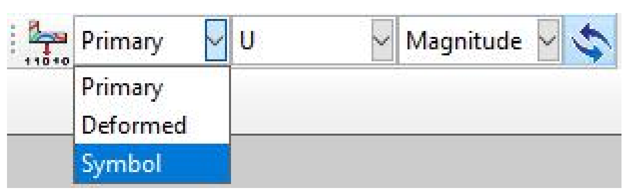


It can be seen that a significant membrane force has developed in the plate. Initially the plate resists the pressure load with bending stiffness but after the pressure causes the plate to be curved, the deformed geometry exhibits additional stiffness and membrane forces appear in the plate. Membrane effects cause some of the load to be carried by membrane forces rather than by bending alone.

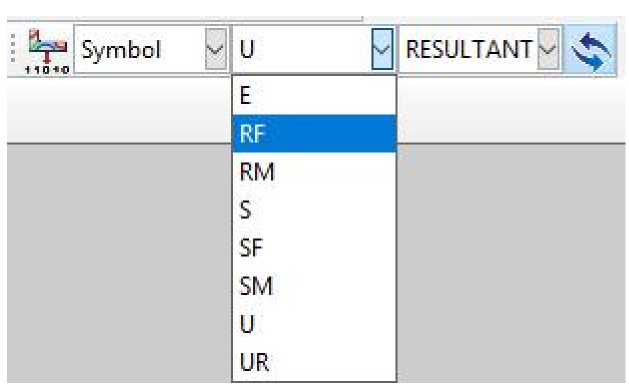
As the load increases on plate, there will be membrane forces in addition to bending moments.

Now we will plot reaction forces at the end nodes where boundary conditions are defined.

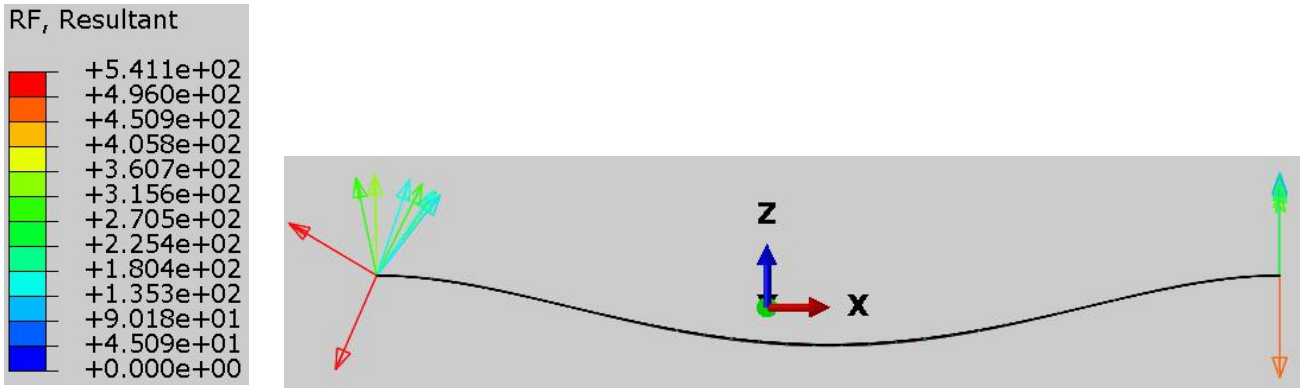
Select **Symbol** as category in the Field Output toolbar.



Select **RF** 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 2).



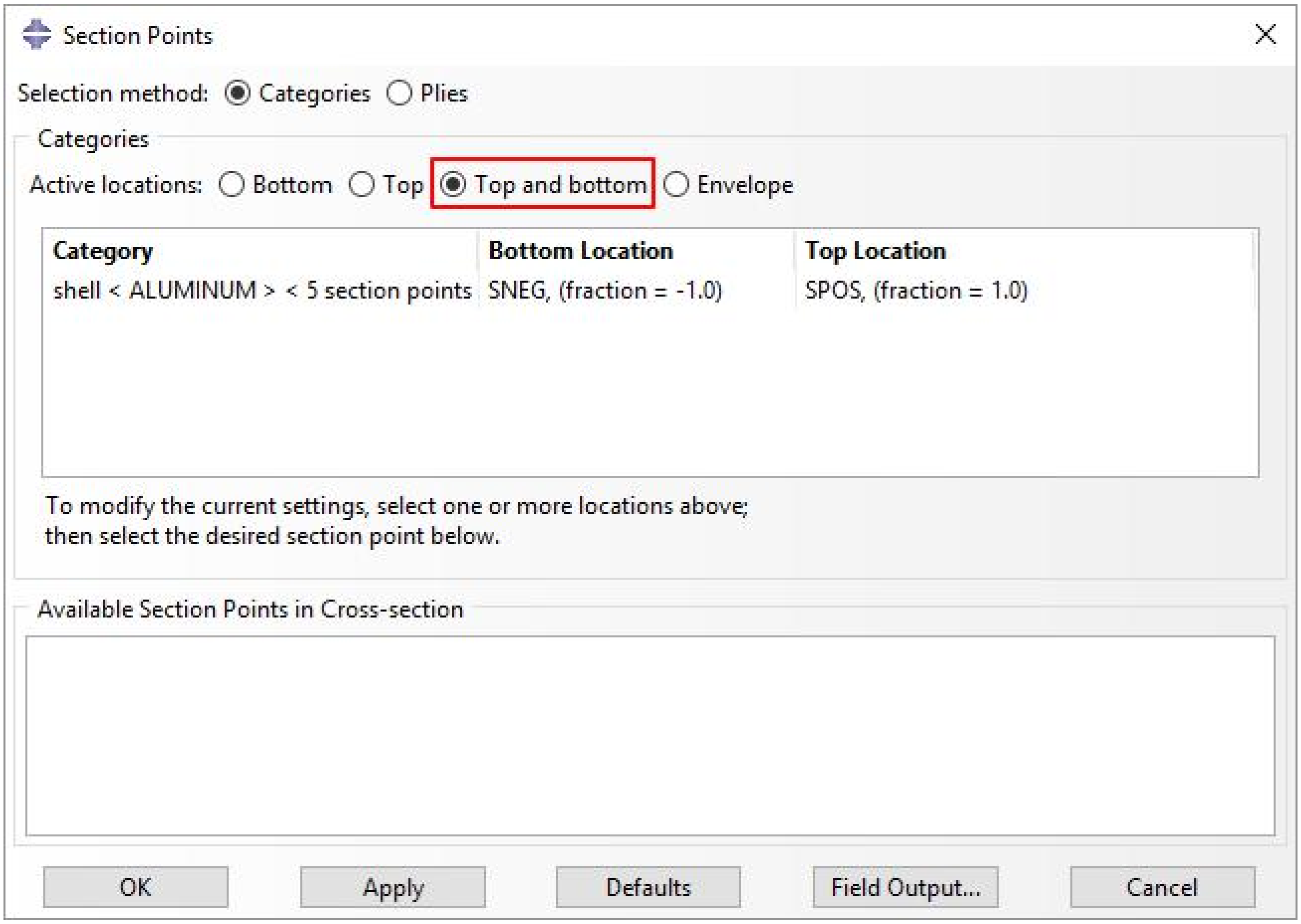
It can be seen that reaction forces develop in all directions on left edge while only in y-direction on the right edge. It could be related to the fact that the translation of right edge is not constrained in the x-direction.

The deformation of the plate may be composed of large strains, large translations or large rotations, or any combination of them. Now we will plot the axial strain in the plate to see if strains are large or small.

Select **E** as output variable in the Field Output toolbar and make sure that **E11** is selected as component.

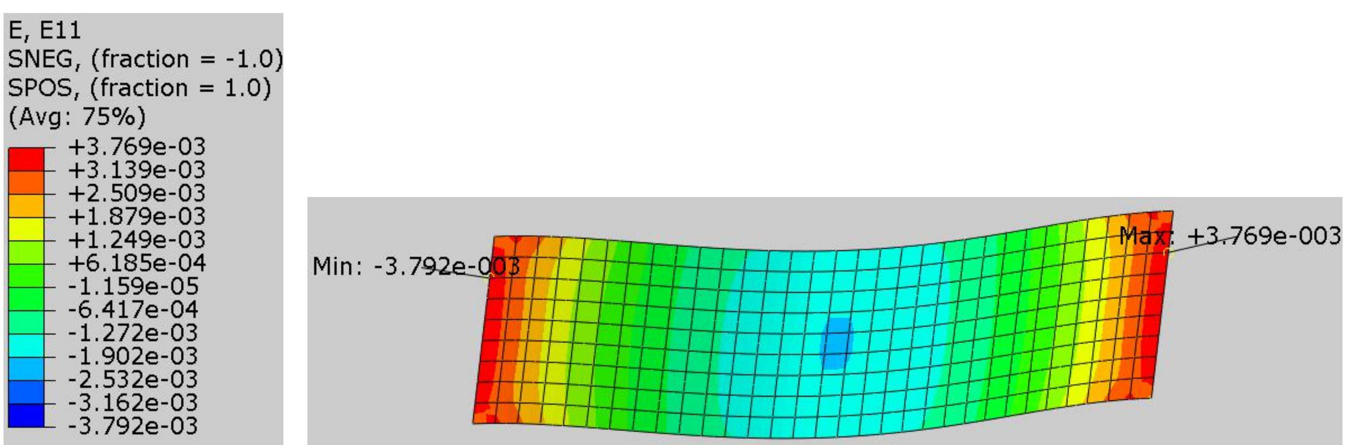
By default result is plotted for the bottom surface (SNEG) of the shell. We want to plot strain both for bottom and top surface (SPOS). So pick **Result > Section Points** from the main menu bar and Sections Points dialog box will appear.

Select **Top and bottom** as shown in the figure below.



Pick **OK** to apply and exit.

The simulation results will appear as shown below (Scale factor is set to 2).



It can be seen that maximum absolute axial strain is approximately 0.0038 which is 0.38%. Since the strain is typically considered small if it is less than 5%, the plate deformation with a strain of 0.38% can be classified as small strain problem.

Although the deflection of midspan is small compared to the overall dimension of the plate (less than 5%), still the changes in geometry as the structure deforms have significant influence on the final solution. So it can be said that geometric nonlinearity can have considerable influence on the final solution even when deformation is small.

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.