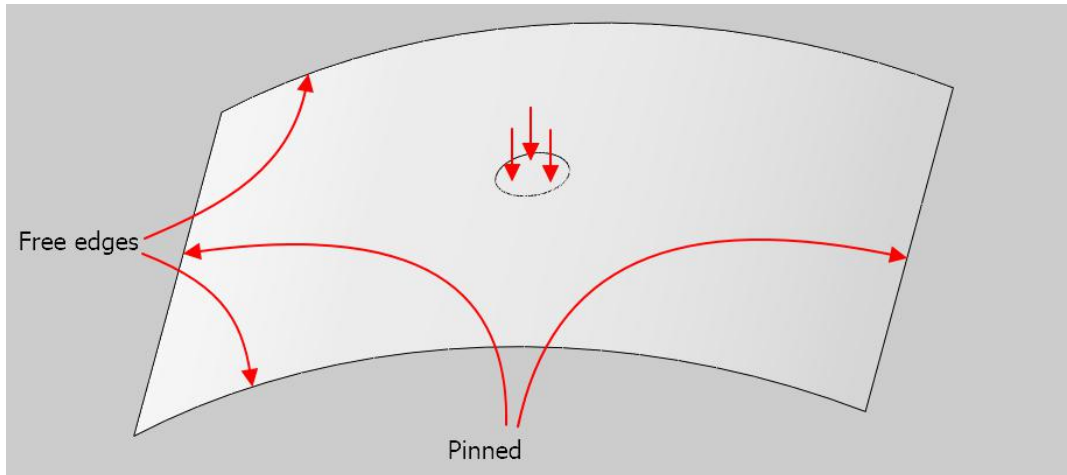


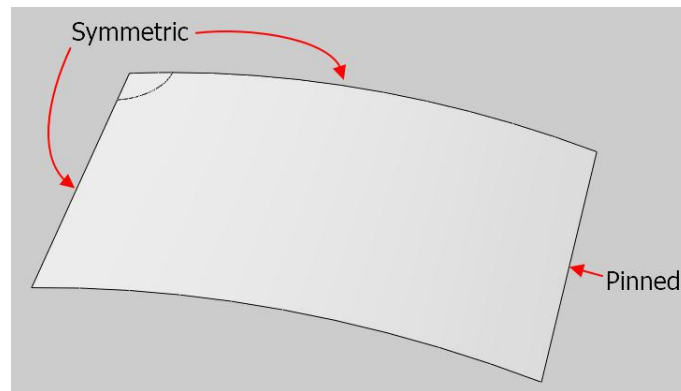
## Exercise 9

In this exercise we will analyze a cylindrical roof which exhibits snap-through behavior. The roof is shown schematically in the figure below.



The structure will be analyzed by both the Static Riks and Static General analysis techniques.

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

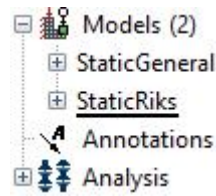


Pick **File > Set Work Directory** and set the work directory to the **Roof** folder

Open the model database **Roof.cae**.

The model consists of a single part, Roof. The part has a width of 1m and a thickness of 0.03m. It has a thickness-to-minimum span ratio of 0.03 (0.01/1) and hence considered thin. It is assumed to be made from an alloy with a Young's modulus of 100 GPa and a Poisson's ratio of 0.3. The roof is modeled using a shell extrude feature. We assume that strains will be small. So it is meshed with quadratic shell elements with reduced integration (S8R5). The roof is loaded with a uniform pressure of 10 MPa in the center.


In the model tree, it can be seen that there are two models. In the "StaticRiks" model, a Static, General step will be used to solve the problem. Other model, "StaticGeneral", will be used to solve the problem by using Static, Riks step.

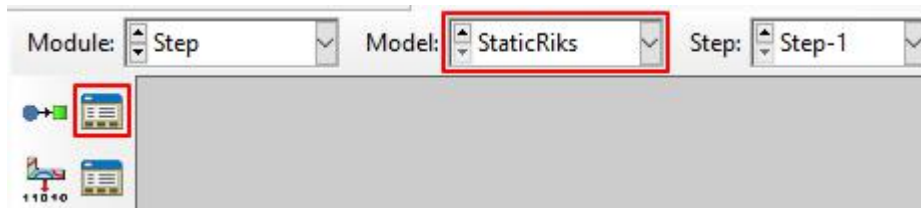


First we will analyze the structure using Riks method.

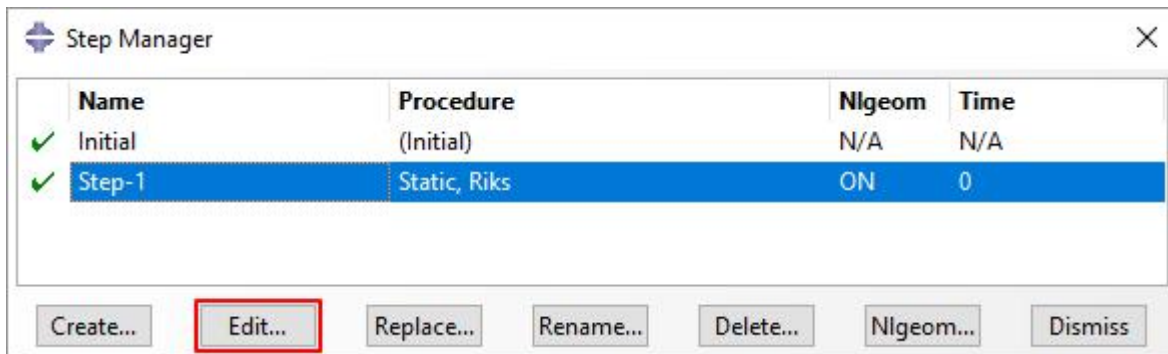
## ⇒ Defining Step

As the roof snaps, its load-displacement response becomes unstable. We will perform the snap-through analysis of the roof by using the Static, Riks procedure. A step has already been created. We will modify it to specify the stopping criteria.

Change to **Step** module and open the Step Manager by picking . First make sure that StaticRiks model is active.



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

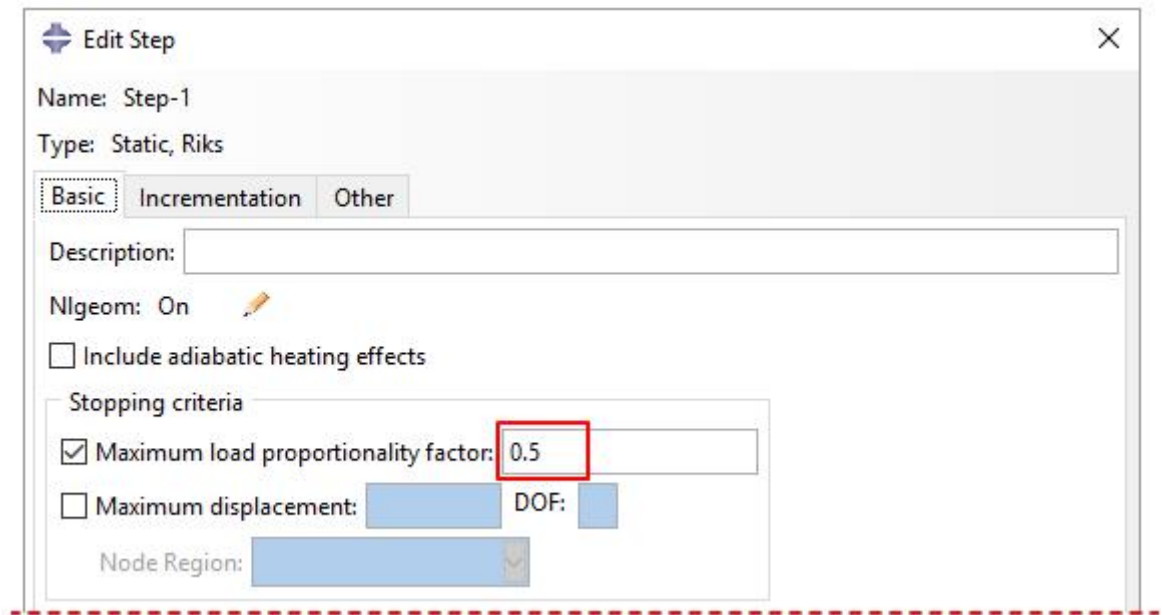


It is already known that critical pressure for the roof is about 4 MPa i.e. the initial snap occurs at this pressure. We can compute the LPF ( $\lambda$ ) by using  $P = \lambda P_{ref}$ . As we know  $P_{ref} = 10$  MPa, so  $\lambda = 0.4$  (4 MPa/10 MPa).

We are interested in the postsnap behavior of roof somewhat beyond the point when magnitude of pressure becomes larger than the critical pressure. So we can specify 0.5 as stopping value of LPF to have an insight into the postsnap behavior.

Check the **Maximum load proportionality** option.

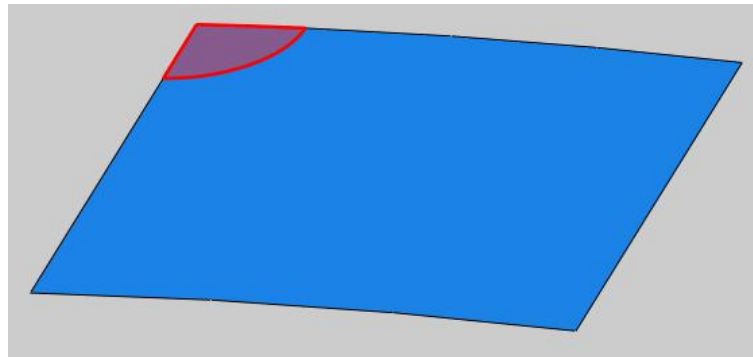
Enter **0.5** as the LPF factor as shown below.




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

## ⇒ Defining Load

A uniform pressure of 10 MPa is applied on the highlighted surface patch in the figure below. The load has already been defined.



To review the applied load, change to **Load** module and open the Load Manager by picking .

## ⇒ 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 already been created.

The boundary condition “BC-Right” constrains all the degrees of freedom of the right edge of the roof except the rotation about the z-axis (UR3). The boundary condition “BC-Left” applies a symmetric constraint to the left edge of the roof. The boundary condition “BC-Side” applies a symmetric constraint

to the side edge of the roof.

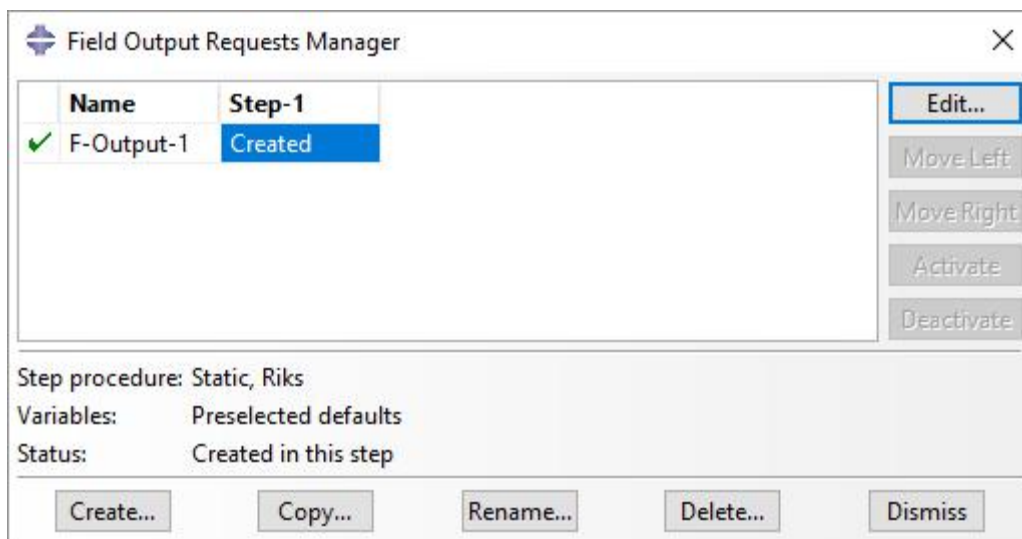
## ⇒ Field Output Request

We will plot the history of applied pressure during postprocessing. This information can be plotted by requesting P, Pressure loads output variable.

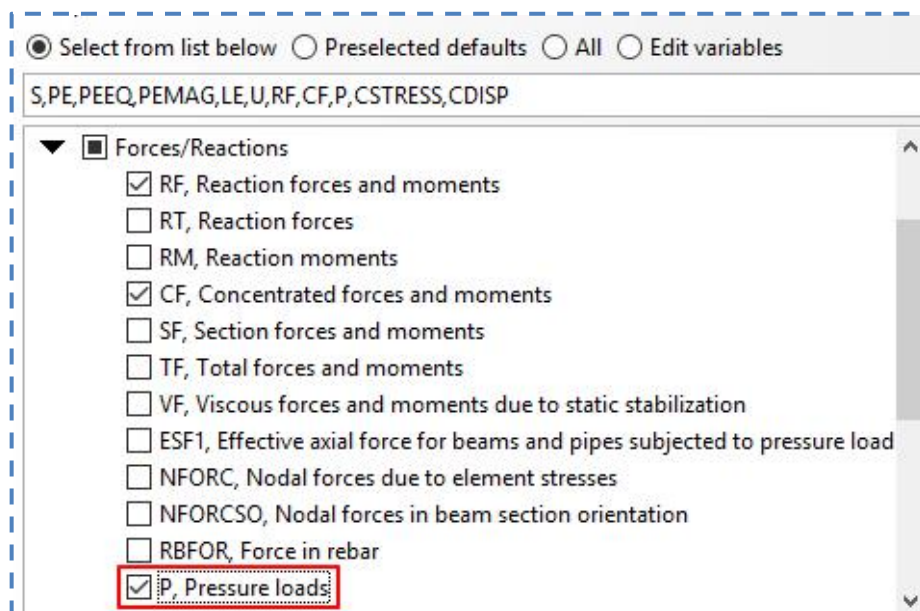
Change to **Step** module.

Open the Field Output Manager by picking  .

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



Check the **P, Pressure loads** 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 SnapRiks or whatever name you would like.

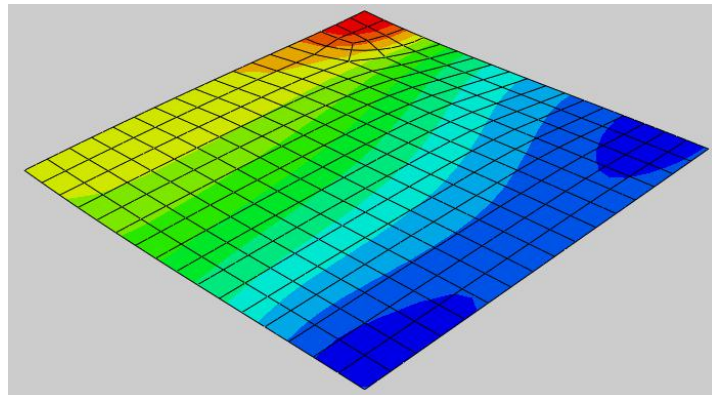
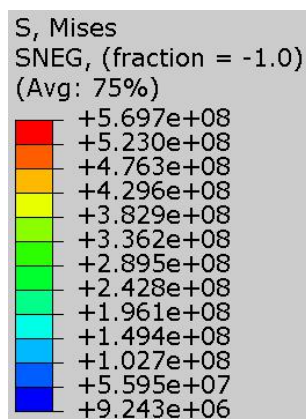
Pick **Continue** and then **OK**.

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. The contour plot of von Mises stress will appear as shown below.

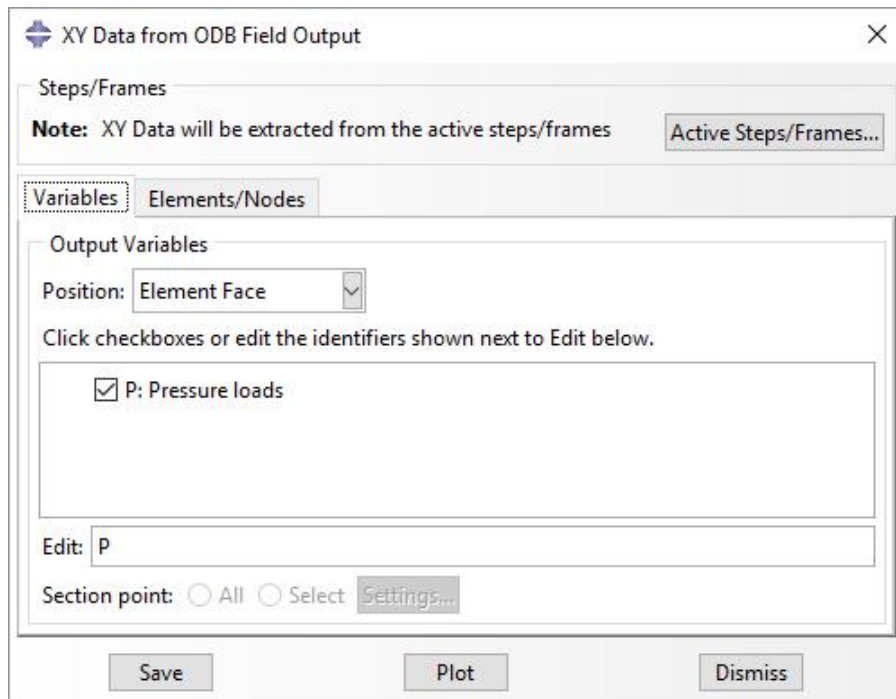


Now we will plot the applied pressure versus the displacement of center node (node at the center of roof).

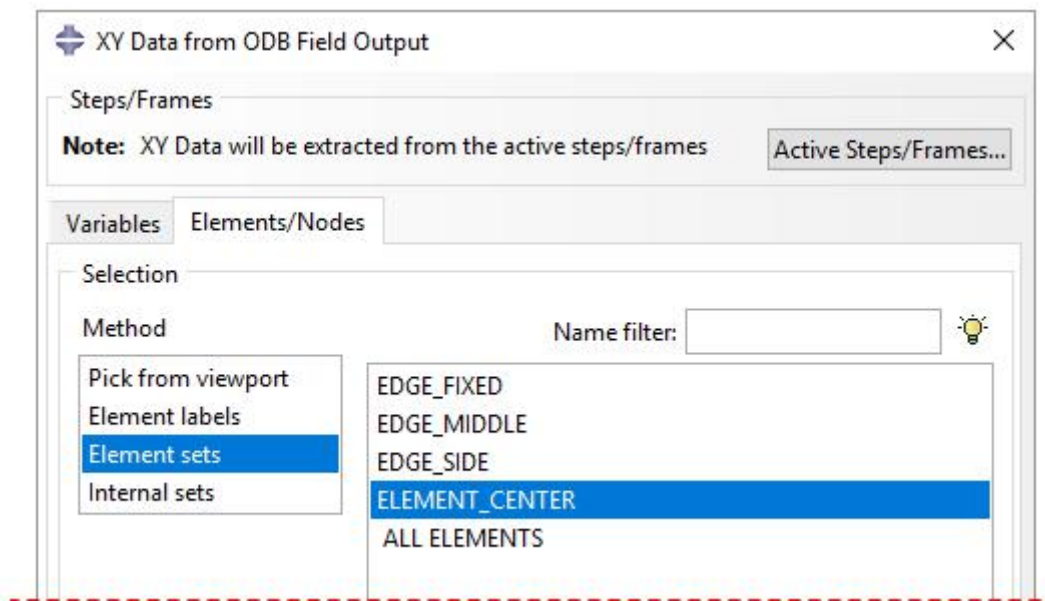
Pick  and Create XY Data dialog box will appear.

Pick **ODB field output > Continue**

In the Position field, select **Element Face** and check the **P:Pressure loads** checkbox.



In the Elements/Nodes tab, pick **Element sets** as method and select the **ELEMENT\_CENTER**. This element has already been defined for the ease of use. Note: Check **Highlight items in viewport** in the dialog box if you want to see the element on screen.



Pick **Save** to save the plot and then pick **OK** to confirm.

It will create a XY data plot of applied pressure versus Arc Length.

In the Variable tab, uncheck **P:Pressure loads** variable.

Select **Unique Nodal** as position and check the **U2** checkbox.



In the Elements/Nodes tab, pick **Node sets** as method and select the **NODE-CENTER**.

Note: Check **Highlight items in viewport** in the dialog box if you want to see the node on screen.

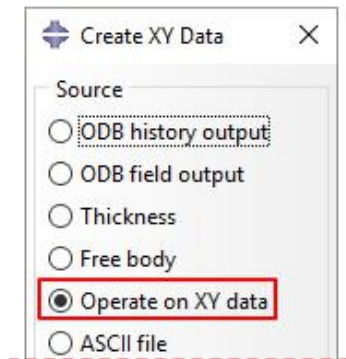
Pick **Save** to save the plot and then pick **OK** to confirm.

It will create a XY data plot of U2 versus Arc Length.

Pick **Dismiss** to close the Field Output dialog box.

Each of the XY data curves is a history plot (variable versus arc length). We will combine the plots to produce the desired pressure-displacement plot.

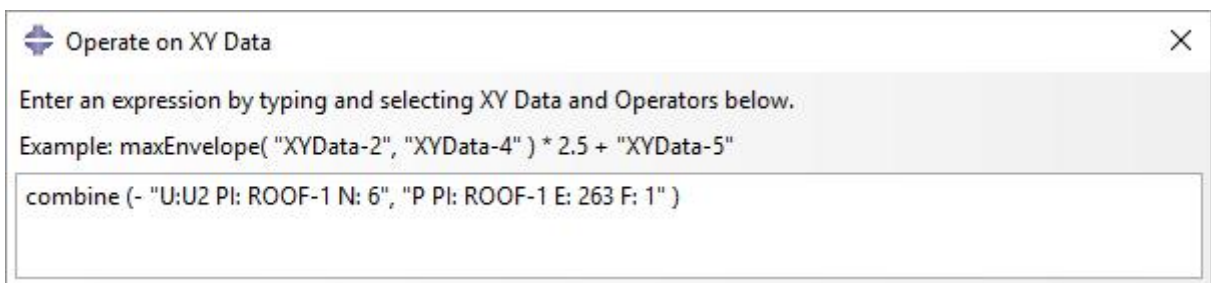
Again pick  and select **Operate on XY data** option.



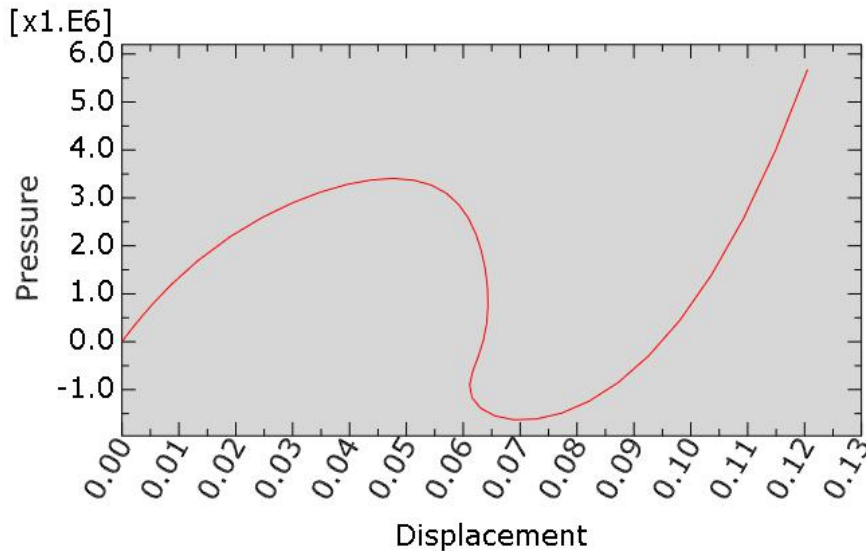
Pick **Continue** to proceed.

In the Operate on XY Data dialog box, pick **combine(X,X)** operator.

Then double-click the “U:U2.....” data and then “P P1:.....” data in the list. It is customary to plot the magnitude of displacement and ignore the sign in such graphs. As the U2 displacement occurs in negative direction, so to obtain displacement with positive magnitude add a “-” before the “U:U2.....” as shown below.



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



As the plot looks satisfactory so pick **Save As** to save the plot and then pick **OK** to accept the default name.

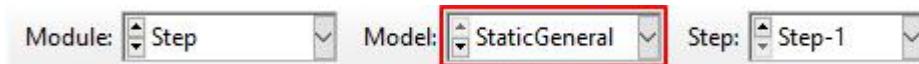
It can be seen that snap-through happens when the load is about 3.5 MPa. After that load falls rapidly to a minimum value of about -1.5 MPa. Before reaching the minimum load, the center point of the roof moves upward by about 0.005 m. This upward movement is called snap-back. When the center of roof has moved upto 0.065 m, it starts moving back (upward) and its displacement becomes about 0.06. After that the center point again starts moving downward. In the undeformed configuration, the center point of the roof is 0.05 m above the pinned edges. The snap-through occurs when the center point has displaced downward about 0.048 m. at the end of snap-through the center point has displaced about 0.07 m. So during the snap-through, the center point has moved about 0.022 m. After the snap-through structure stiffens rapidly.

Notice that the maximum load in the graph is about 5.5 MPa. We will apply the same magnitude of load for the analysis when Static, General step is used.

## ⇒ Static, General Step

Now we will perform a analysis of the structure by using the Static, General step.

Change the current model to StaticGeneral as shown below.



A step has already been defined with a total time period set to **1.0**. Nlgeom option has been set to On. The incrementation has been set to **Automatic** type with an initial increment size of **0.01** and a maximum increment size of **0.05**

We will introduce the automatic stabilization to get a solution of this problem.

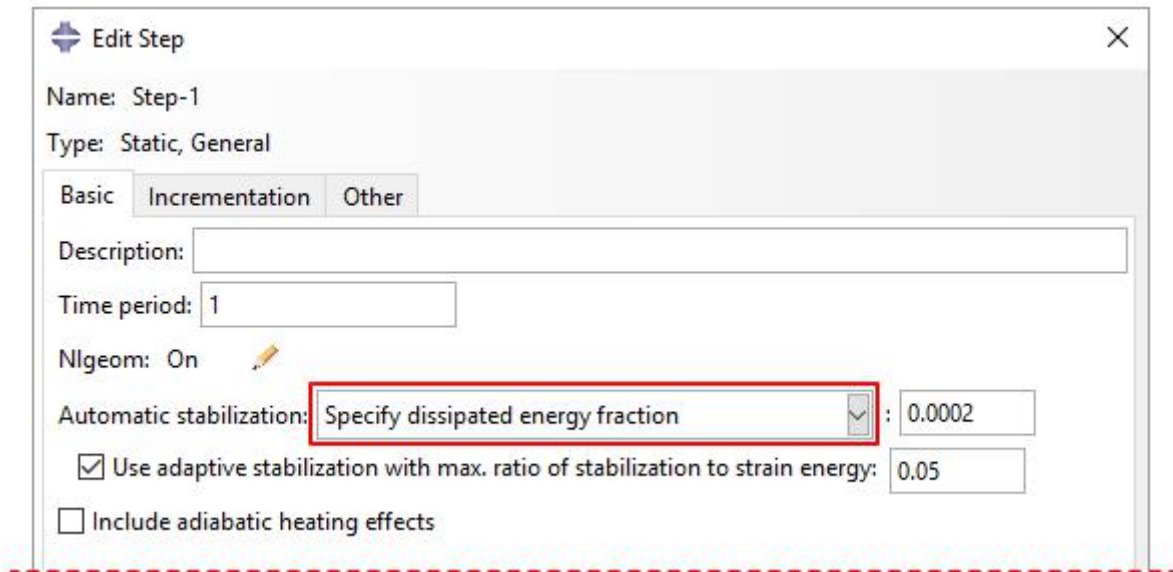
Change to **Step** module.

Open the Step Manager by picking  .



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

Pick **Specify dissipated energy fraction** in the Edit Step dialog box as shown below.




Notice that Use adaptive stabilization option is checked by default. Adaptive stabilization applies damping factors which can increase/decrease with time to account for changes occurring in the model during the step.

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

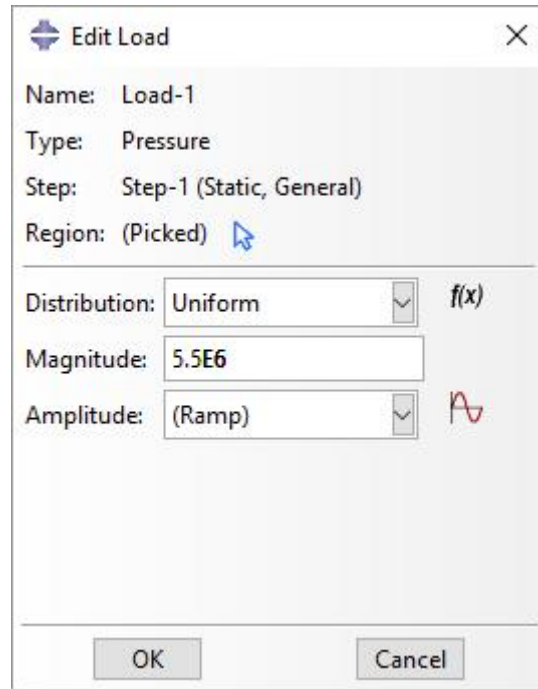
## ⇒ Defining Load

The load has already been defined. A uniform pressure of 10 MPa is specified in this load specification. We noticed that the maximum load in the pressure-displacement plot for the Riks procedure is about 5.5 MPa. We will apply the same magnitude of load in the current analysis so that a comparison could be made among both solutions.

To review the applied load, change to **Load** module and open the Load Manager by picking .

Select the Load-1 and pick **Edit**.

Enter 5.5E6 in the magnitude field as shown below.



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

## ⇒ Field Output Request

We will plot the history of applied pressure during post-processing. This information can be plotted by requesting P, Pressure loads output variable.

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 **P, Pressure loads** 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 **SnapGeneral** 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.

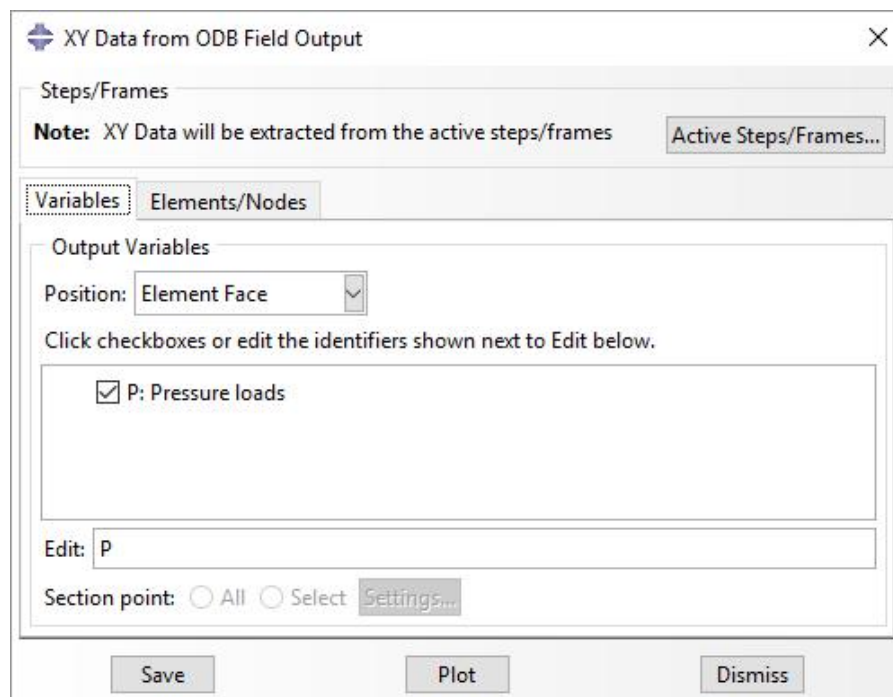
## ⇒ Postprocessing

Now we will plot the applied pressure versus the displacement of center node.

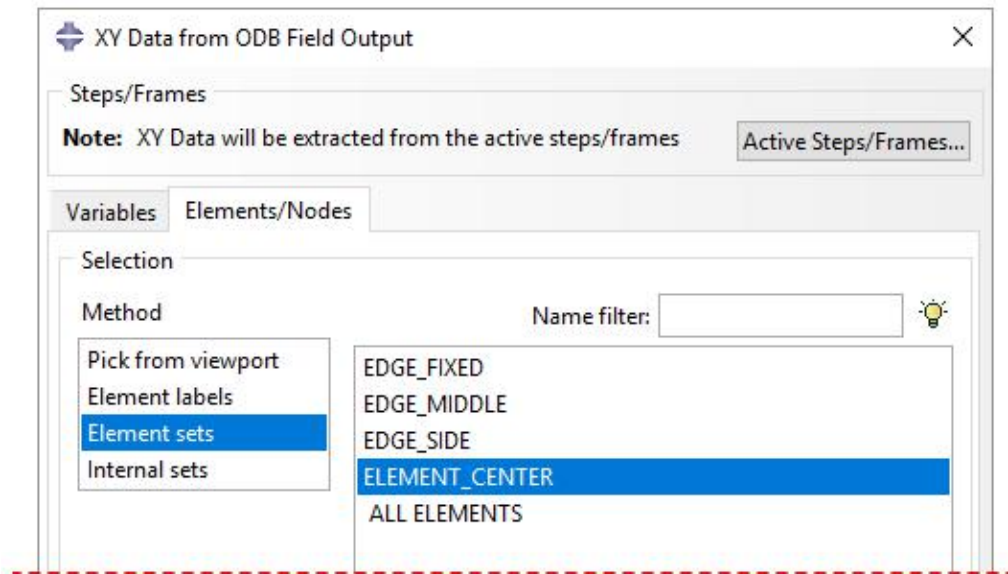
Pick  and Create XY Data dialog box will appear.

Pick **ODB field output > Continue**

In the Position field, select **Element Face** and check the **P:Pressure loads** checkbox.



In the Elements/Nodes tab, pick **Element sets** as method and select the **ELEMENT\_CENTER**. This element has already been defined for the ease of use. Note: Check **Highlight items in viewport** in the dialog box if you want to see the element on screen.



Pick **Save** to save the plot and then pick **OK** to confirm.

It will create a XY data plot of applied pressure versus time.

In the Variable tab, uncheck **P:Pressure loads** variable.

Select **Unique Nodal** as position and check the **U2** checkbox.

In the Elements/Nodes tab, pick **Node sets** as method and select the **NODE-CENTER**.

Pick **Save** to save the plot and then pick **OK** to confirm.

It will create a XY data plot of U2 versus time.

Pick **Dismiss** to close the Field Output dialog box.

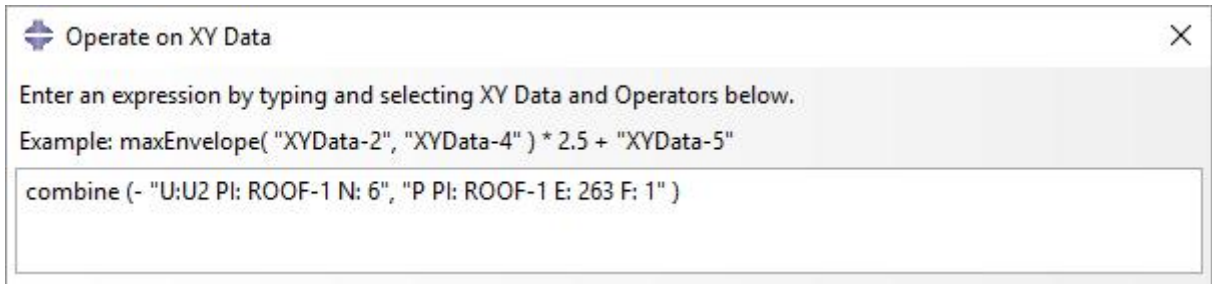
Each of the XY data curves is a history plot (variable versus time). We will combine the plots to produce the desired pressure-displacement plot.

Again pick  and select **Operate on XY data** option.

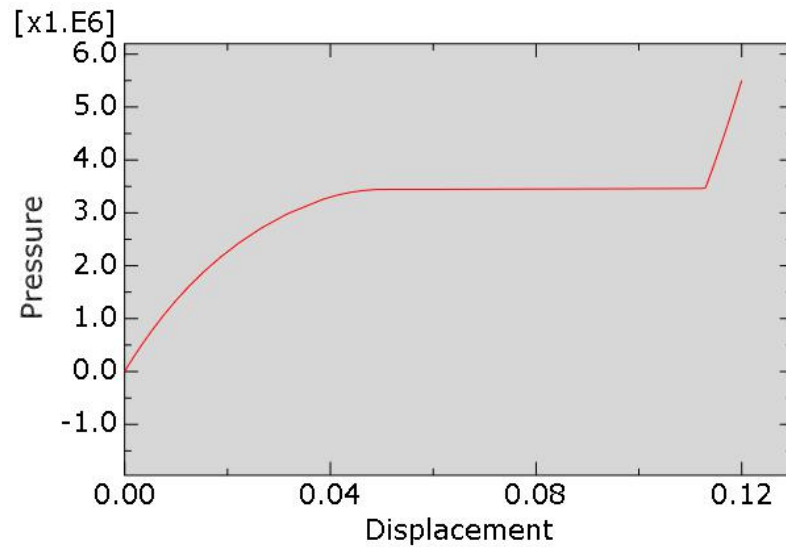
Pick **Continue** to proceed.

In the Operate on XY Data dialog box, pick **combine(X,X)** operator.

Then double-click the “U:U2……” data and then “P P1:……” data in the list. It is customary to plot the magnitude of displacement and ignore the sign in such graphs. As the U2 displacement occurs in negative direction, so to obtain displacement with positive magnitude add a “-” before the “U:U2……” as shown below.



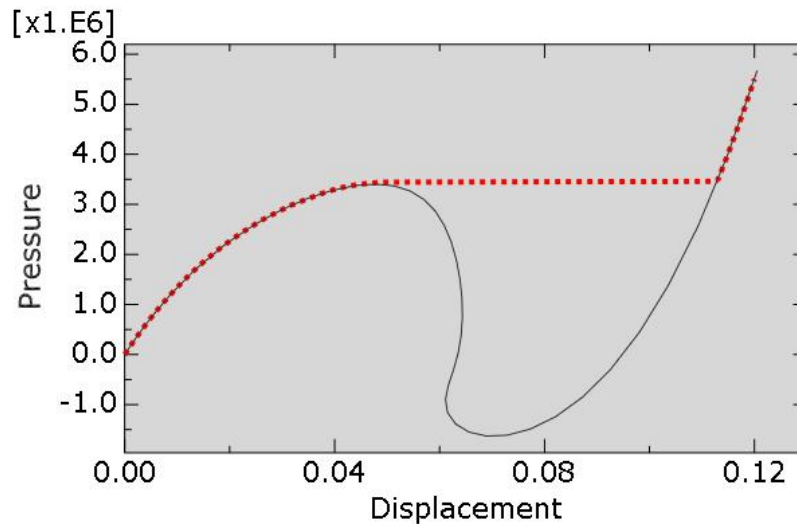
Pick **Plot Expression** tab and graph will appear as shown below.



As the plot looks satisfactory so pick **Save As** to save the plot and then pick **OK** to accept the default name.

## ⇒ Comparison

The following figure shows the load-displacement curve for both the analysis. The dotted line represents the static, general analysis and solid line represents the Riks analysis.



It can be seen that during the initial loading (before the critical point) and after the snap-through takes place, both curves are very similar. It can also be seen that static, general solution using automatic stabilization does not capture snap-through. For an unstable problem, when the stiffness matrix becomes negative and system releases energy it accelerates the system. As the velocity of system increases, adaptive stabilization introduces damping to absorb the energy. As a result the external load stays almost constant during the unstable part of the loading. In other words, during the snap-through the strain energy that the system relieves through decreasing the load now instead is being dissipated through damping when automatic stabilization is introduced.

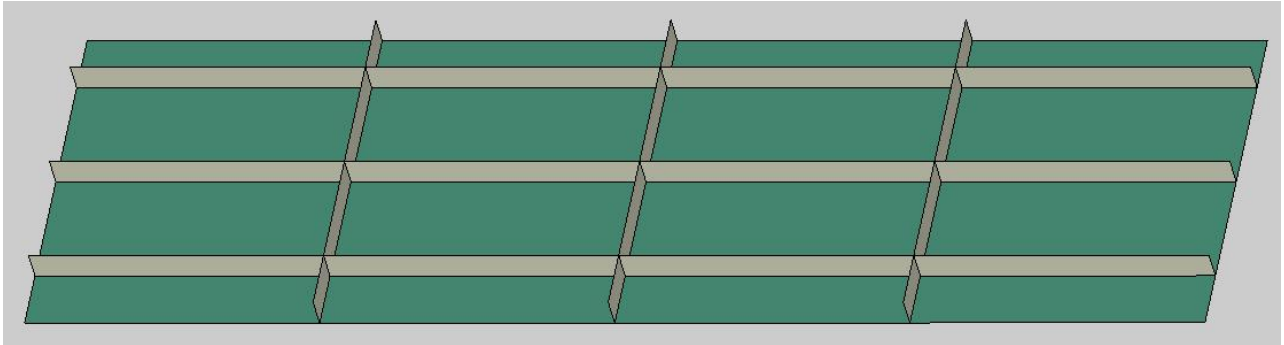
The disadvantage of introducing damping is that it produces an almost constant load during snap-through without giving any information about the severity of the snap-through.

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

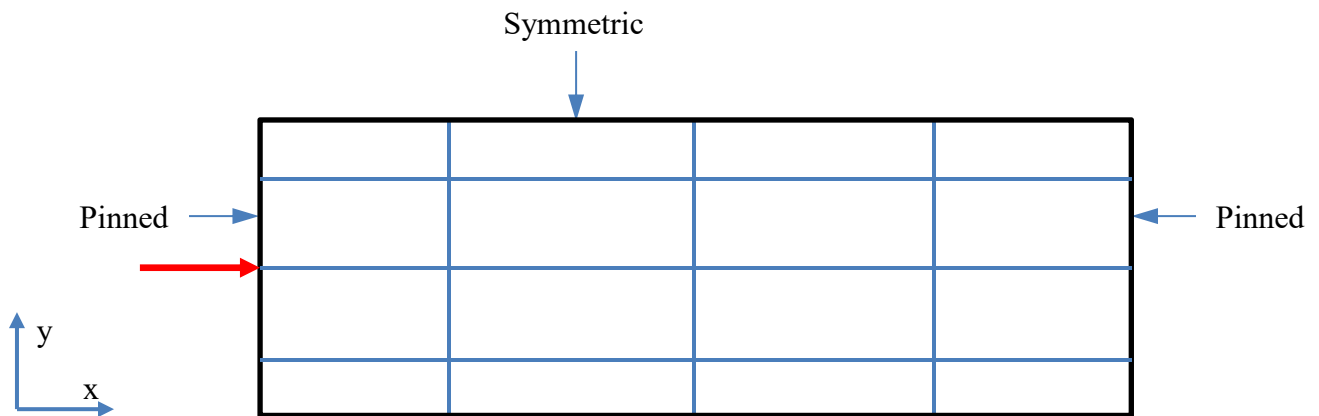


## Exercise 10

In this exercise we will analyze a stiffened plate structure subjected to in-plane loading. The structure is shown in the figure below.



The plate is 10 m long and 5 m wide and constructed of 5 mm thick steel. One large and four small stiffeners are welded in the longitudinal direction and three equally spaced stiffeners are welded to the plate in the second principal direction. The stiffeners are made from 10 mm thick plate. The structure is pinned along the shorter sides and has symmetry boundary condition along one of the longer sides. The compressive load is applied to one of the pinned sides by specifying a concentrated force to a reference point. This reference point is constrained to the nodes on the side of the plate structure using MPC constraint. The following figure shows the plate geometry, boundary conditions and loading schematically.



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

Open the model database **Plate.cae**.


The model consists of a single part, Plate. The plate is modeled using shell extrude features. It is assumed to be made from steel with a Young's modulus of 210 GPa and a Poisson's ratio of 0.3. It is meshed with linear quad shell elements (S4). The plate is loaded with a concentrated force of  $5 \times 10^6$  N to the left side of the plate. Two springs are also modeled to represent flexible connections to the rest of the structure.

## ⇒ Defining step

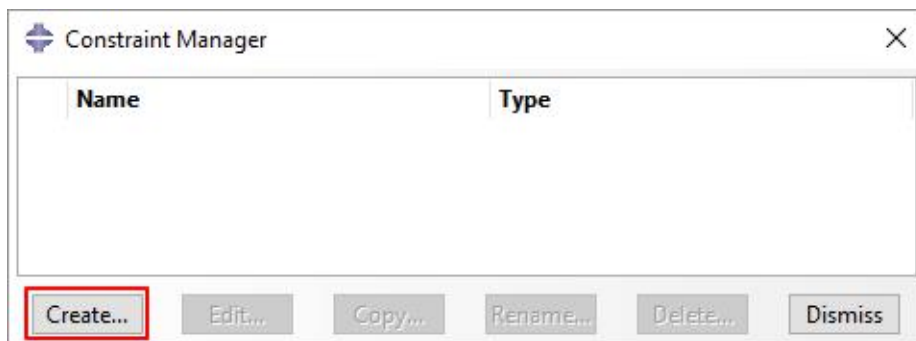
The step has already been defined with a total time period set to **1.0**. Nlgeom option has been set to On. The incrementation has been set to **Automatic** type with an initial increment size of **0.01**

## ⇒ Defining Constraint

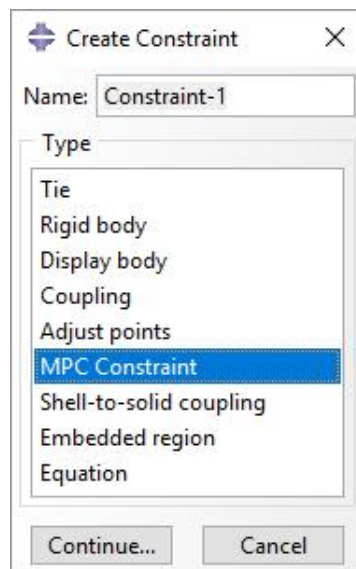
Now we will define an MPC constraint to relate degrees of freedom of plate nodes to the reference point.

Change to **Interaction** module and open the Constraint Manager by picking .

Pick **Create** to define a new constraint.

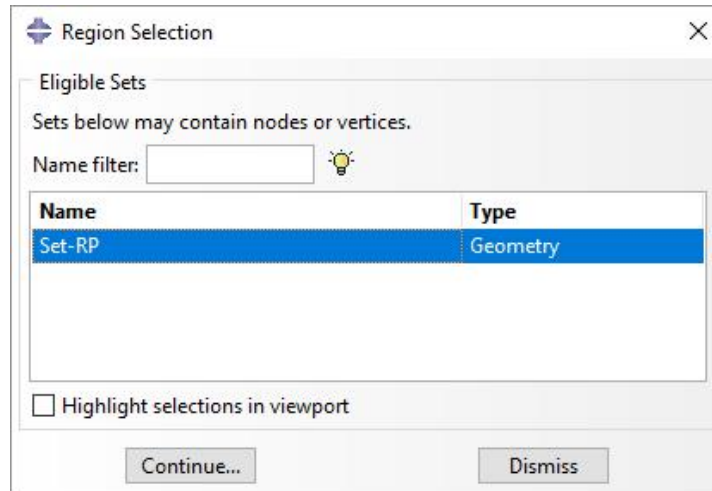


Pick **MPC Constraint** as the type of the constraint.



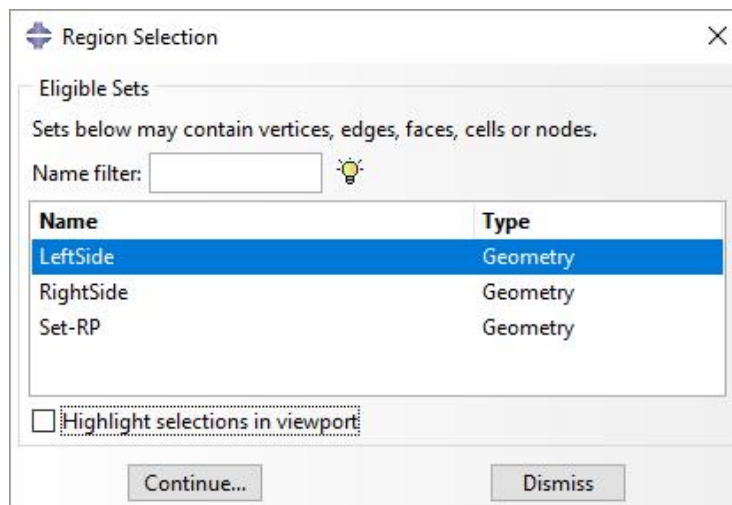
Pick **Continue** to proceed.

System will ask you to pick the control point. As we have already defined a set, so pick **Sets** and select the Set-RP.



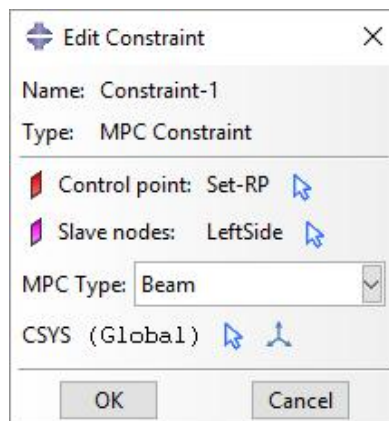
Pick **Continue** to proceed.

System will ask you to pick the region for slave nodes. As we have already defined a set, so pick **Sets** and select the LeftSide.



Pick **Continue** to proceed.

We will use the Beam as MPC Type which is selected by default as shown below.



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

## ⇒ **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-Right” constrains all the degrees of freedom of the right side of the plate except the translation and rotation about the z-axis (U3 and UR3). The boundary condition “BC-Left” constrains all the degrees of freedom of the reference point (Set-RP) except the translation in x and z-direction (U1 and U3) and rotation about the z-axis (UR3). This reference point is constrained to the degrees of freedom of plate nodes with MPC constraint. The boundary condition “BC-Symmetric” applies a symmetric constraint to the longitudinal side of the plate.


## ⇒ **Defining Load**

A concentrated force of magnitude  $5 \times 10^6$  N is applied on the right side of the plate. The load has already been defined.

The load is applied to the reference point (Set-RP) which is constrained to the degrees of freedom of plate nodes with MPC constraint.

## ⇒ **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 InPlaneLoading or whatever name you would like.

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

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


If you look in the message area, the following message can be seen.

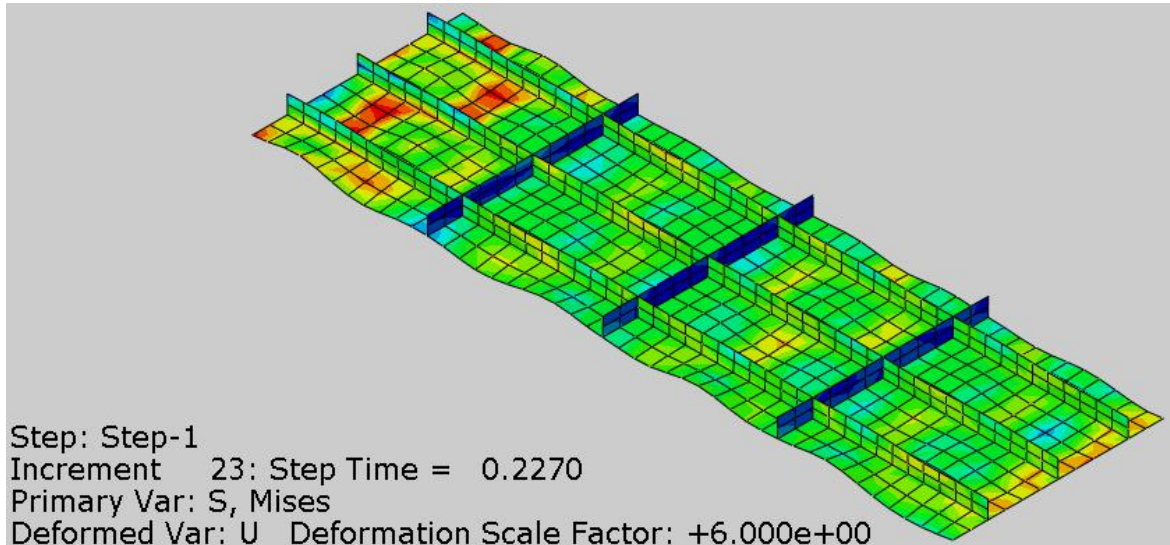
“Job InPlaneLoading aborted due to errors”

We will try to find the cause of this error in the Visualization module.

So pick **Results** to view the results in the Visualization module.

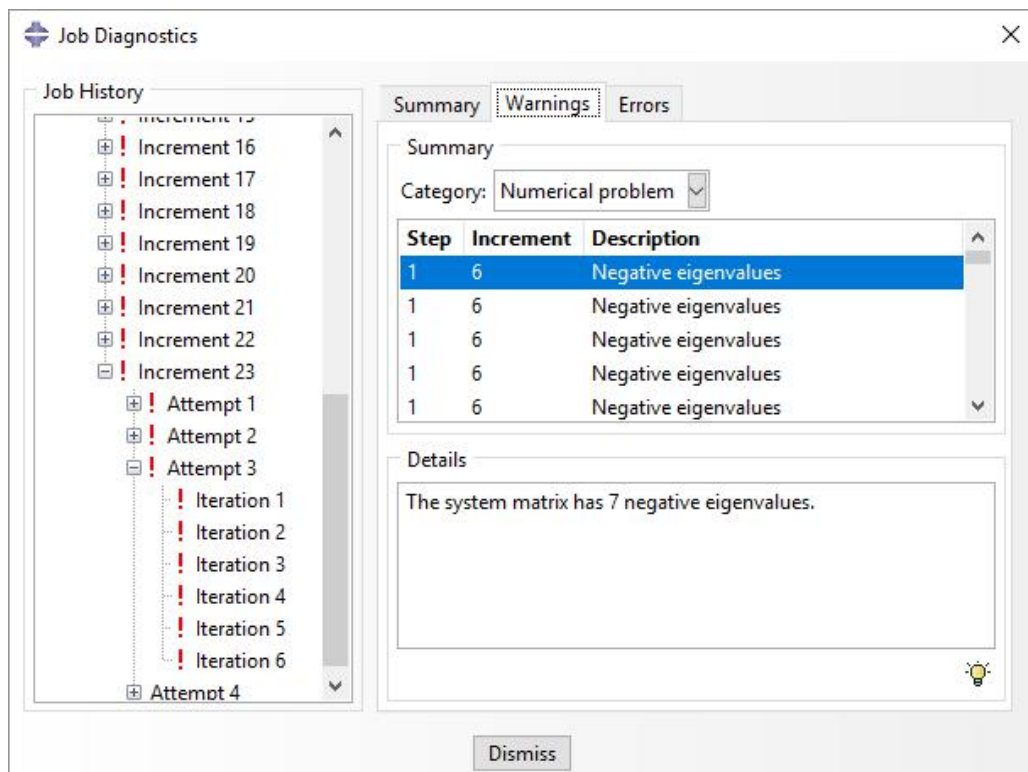
## ➔ Diagnosing the error

Pick  to plot contours on deformed shape. The contour plot of von Mises stress will appear as shown below. (Deformation scale factor is set to 6).



The deformed shape of the plate shows local buckling. It can be seen in the figure below that job runs till 23<sup>th</sup> increment.

To investigate the cause of analysis termination, pick **Tools > Job Diagnostics** from the main menu bar and Job Diagnostics dialog box will appear.



It can be seen that for the 23<sup>th</sup> increment, there have been total 4 attempts but solution has not converged. In the last increment and even in the previous increments “Negative eigenvalues” warning message appear as shown in the figure above.


“Negative eigenvalue” messages appear due to some instability in the system.

The instability seems to occur due to localized buckling in the plate. If the instabilities are localized in a static problem, there will be a transfer of strain energy from unstable area of the model to neighboring areas. If such a problem is analyzed dynamically, the strain energy released is transformed into kinetic energy. So the inertia has a stabilizing effect on the model. This mostly works well but solving a quasi-static problem dynamically could be computationally expensive. Another alternative is to introduce damping to dissipate the strain energy. For that purpose, Abaqus/Standard provides an automatic stabilization capability to stabilize unstable quasi-static problems by applying volume-proportional damping to the model.

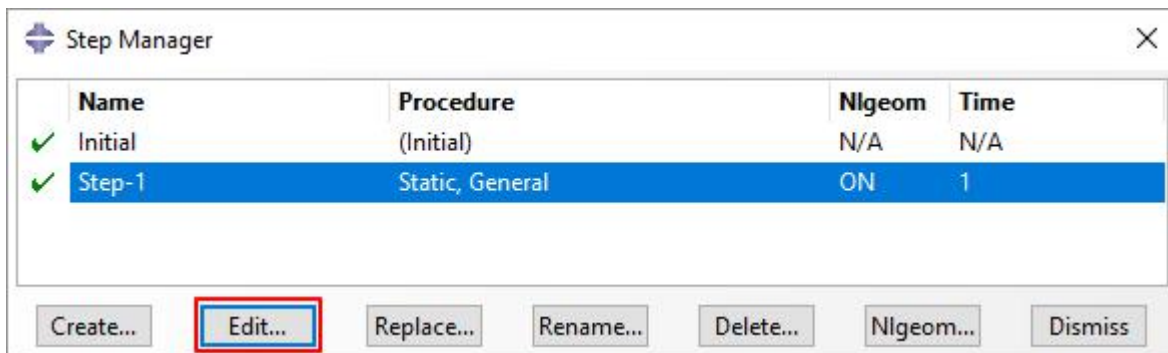
## ⇒ Introducing Stabilization

We will introduce the automatic stabilization to get a solution of this problem.

Change to **Step** module.

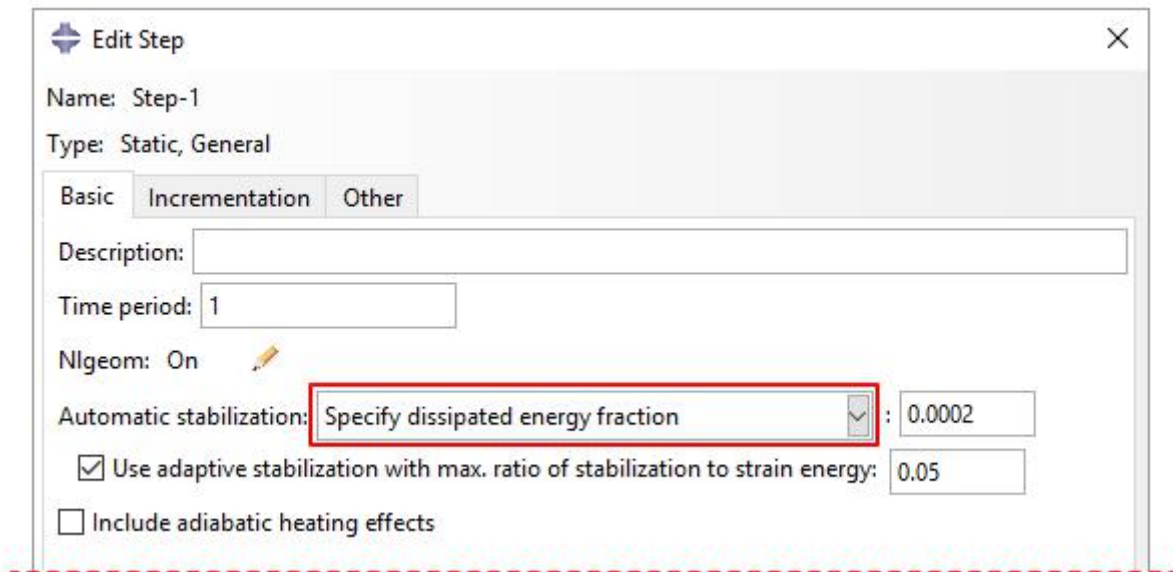
Open the Step Manager by picking  .

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



Pick **Specify dissipated energy fraction** in the Edit Step dialog box as shown below.






Notice that Use adaptive stabilization option is checked by default. Adaptive stabilization applies damping factors which can increase/decrease with time to account for changes occurring in the model during the step. So the damping factor increases if the convergence behavior is problematic or decreases if it distorts the solution.

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.


Select the job named InPlaneLoading and pick **Submit**.

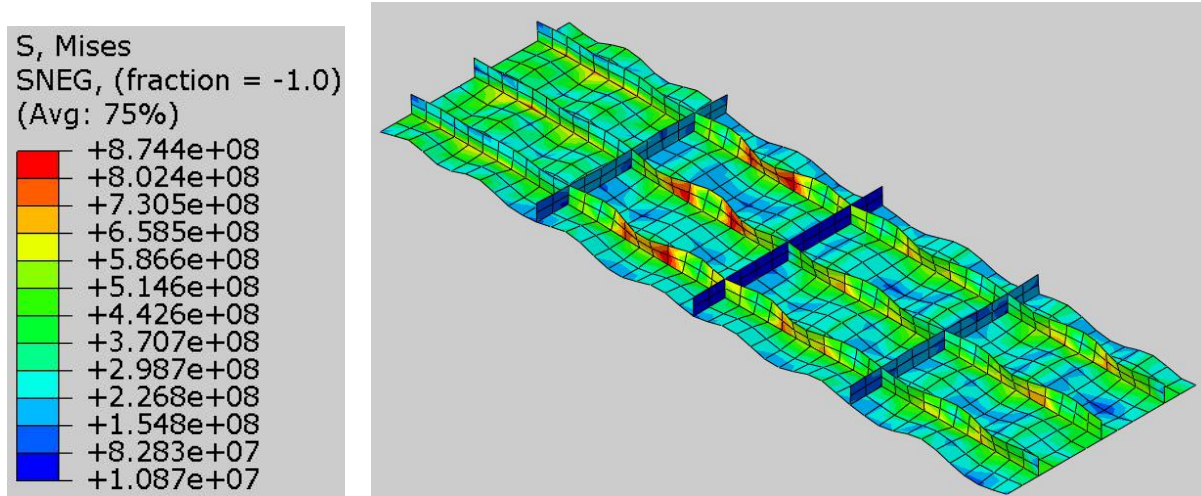
Pick **OK** to accept overwriting.

Notice that job completes successfully.

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

## ⇒ Postprocessing

Pick  to plot contours on deformed shape. The contour plot of von Mises stress will appear as shown below. (Deformation scale factor is set to 3).



It can be seen that stresses are very high and plastic yielding is expected but we will ignore the plastic yielding for this exercise.

Now we will plot the applied force versus the displacement of reference point.

Pick  and Create XY Data dialog box will appear.

Pick **ODB field output > Continue**

In the Position field, select **Unique Nodal** and check the **CF1** and **U1** checkbox.

In the Elements/Nodes tab, pick **Node sets** as method and select the **SET-RP**.

Pick **Save** to save the plots and then pick **OK** to confirm.

Pick **Dismiss** to close the Field Output dialog box.

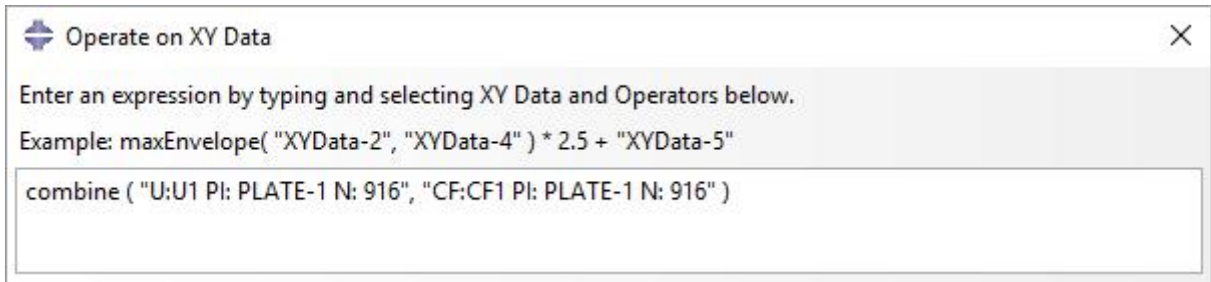
Each of the XY data curves is a history (variable versus time) plot. We will combine the plots to produce the desired force-displacement plot.

Again pick  and select **Operate on XY data** option.

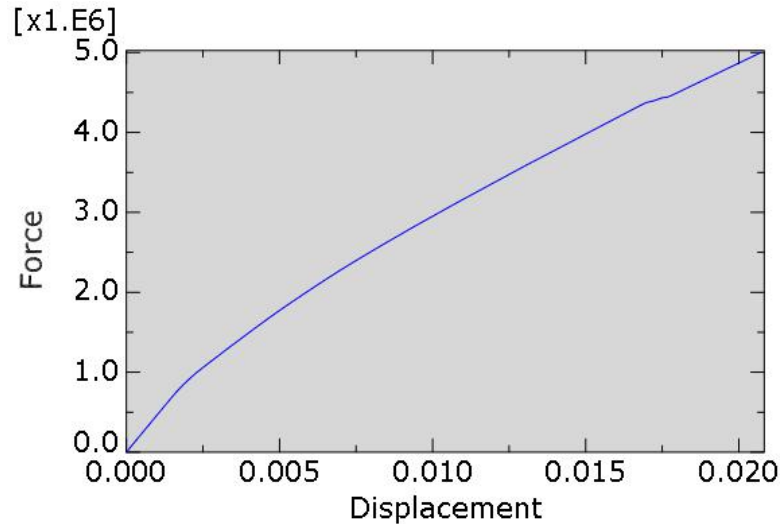
Pick **Continue** to proceed.

In the Operate on XY Data dialog box, pick **combine(X,X)** operator.

Then double-click the “U:U1.....” data and then “CF:CF1.....” data in the list.



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



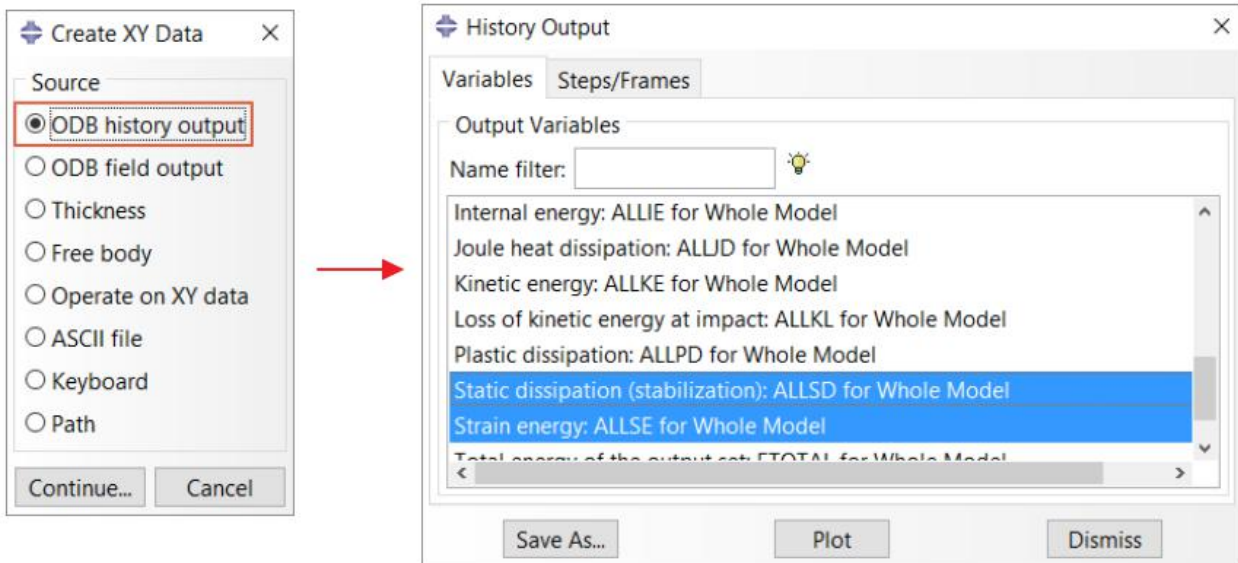
It can be seen that displacement of reference point produced by the applied force is smooth and does not reflect the local buckling in the plate. It can be said that the gross or overall static behavior of a structure characterized by a load-deflection response curve can not always show the localized instabilities present.

The introduction of stabilization in a problem can change the solution significantly. Although the adaptive stabilization allows to increase the damping factor if the convergence behavior is problematic or to decrease the damping factor if it distorts the solution, it is not certain that the value is unnecessarily high and distorting the solution. Therefore it is necessary to verify that the inclusion of stabilization does not significantly alters the solution. The simplest method is to compare the energy dissipation due to stabilization (ALLSD) to the elastic strain energy of the model (ALLSE) or the internal energy of the model (ALLIE). Smaller the value of dissipation energy as compared to the internal energy of the model, the better.

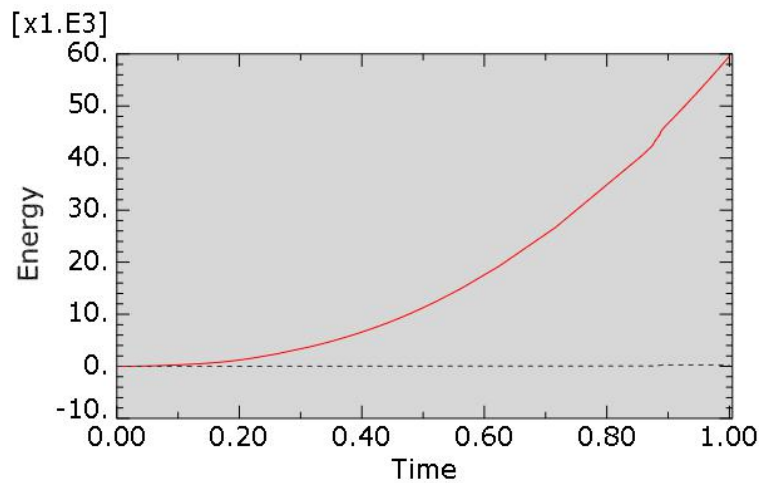
To plot these energies, pick  and Create XY Data dialog box will appear.

Pick **ODB history output > Continue**

In the History Output dialog box, select **ALLSD** and **ALLSE**.



Pick **Plot** and the graph will appear as shown below. Note that ALLSD is shown with dotted line.

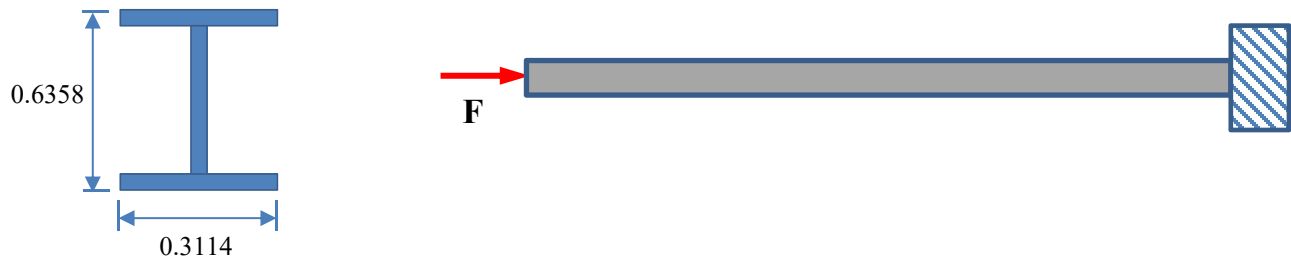


It can be seen that the dissipation energy is negligible compared to the strain energy of the model. So it can be concluded that stabilization has a negligible effect on the solution.

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

## Exercise 11

In this exercise we will perform buckling analysis of a column subjected to axial load  $F$ . The column is shown schematically in the figure below.



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

Open the model database **Column.cae**.

The model consists of a single part, Column. It has a length of 10 m and consists of an I-shaped cross-section. It is modeled as one-dimensional line (beam elements). It is assumed to be made from steel with a Young's modulus of 210 GPa and a Poisson's ratio of 0.3. It is meshed with 2-node cubic beam elements (B33). The column is loaded with a concentrated force of 1 MN.


A buckling study is usually performed in two stages. In the first stage an eigenvalue buckling analysis is performed to estimate the shape of buckling modes and corresponding eigenvalues of the structure. This analysis has limited usefulness if the structure exhibits material nonlinearity or geometric nonlinearity prior to buckling. Furthermore eigenvalue buckling analysis gives no information about postbuckling response. For these reasons, in the second stage a nonlinear load-displacement analysis is performed. A load-displacement analysis takes the nonlinearities into consideration and can also investigate the postbuckling response. The information obtained during eigenvalue buckling analysis is used to introduce imperfection in the perfect structure in second stage.

In this exercise, first the critical buckling loads and shapes of buckling modes will be estimated.

### ⇒ Eigenvalue buckling analysis

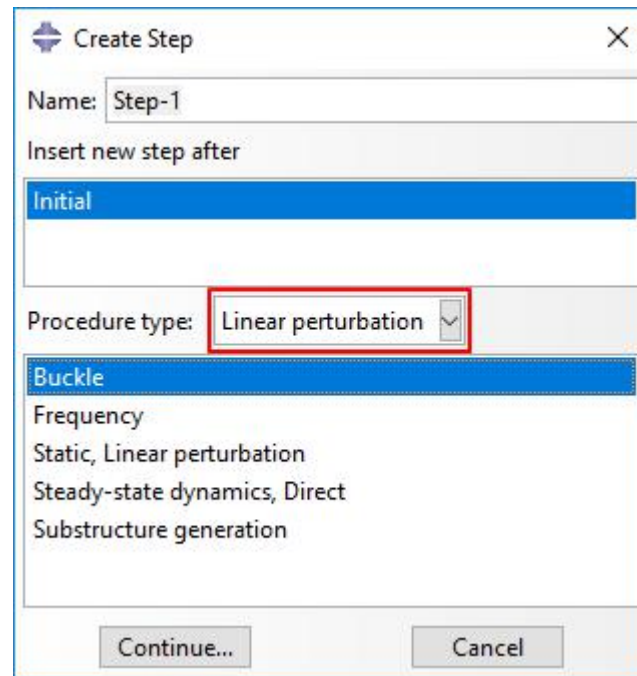
Eigenvalue buckling procedure is used to estimate buckling modes. This is a linear perturbation analysis. Now we will define a step to perform eigenvalue buckling analysis.

Change to **Step** module.

Pick  to create a new step.

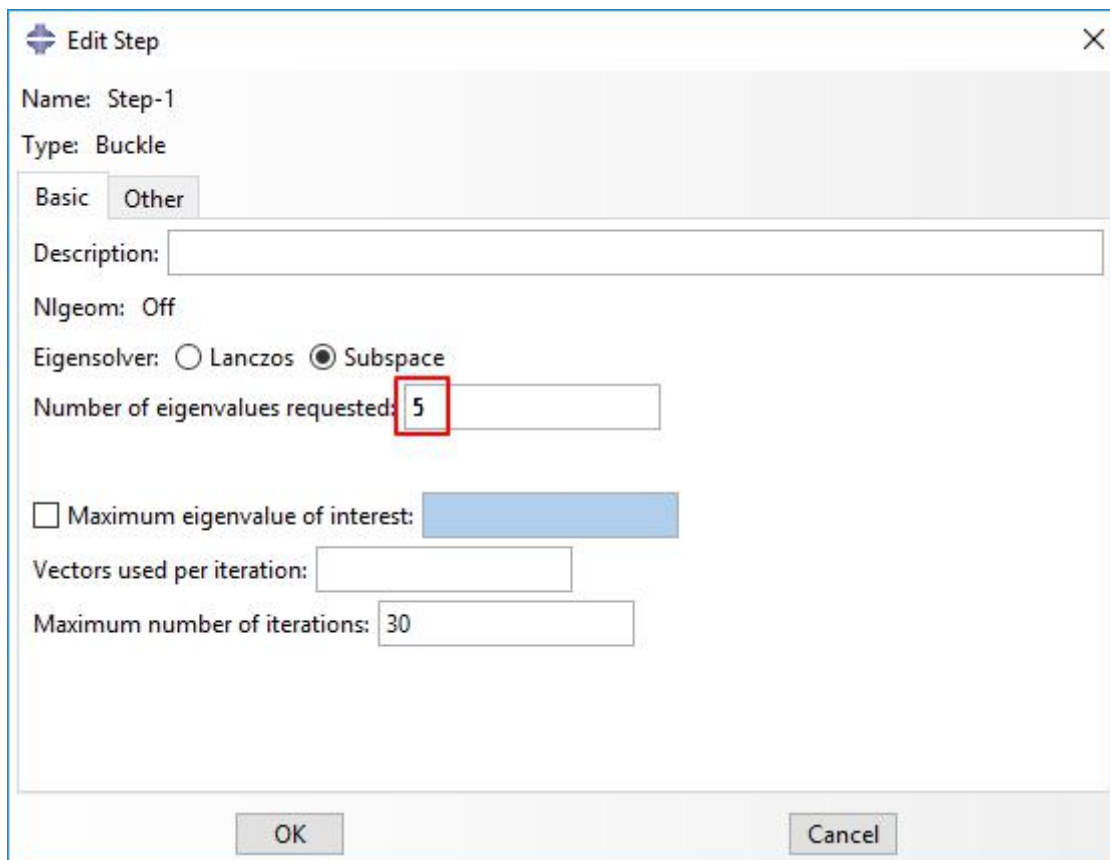
Select the **Linear perturbation** as procedure type.

Pick **Buckle** as procedure.



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

We are interested in first five eigenvalues so enter **5** as the number of eigenvalues requested.




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



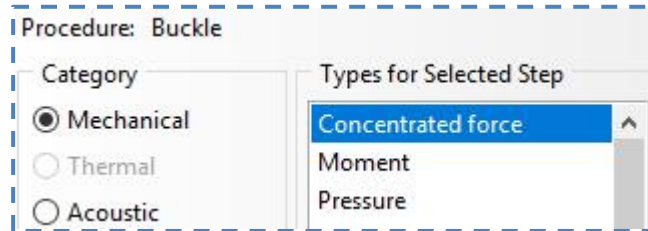
## ➔ Defining Load

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

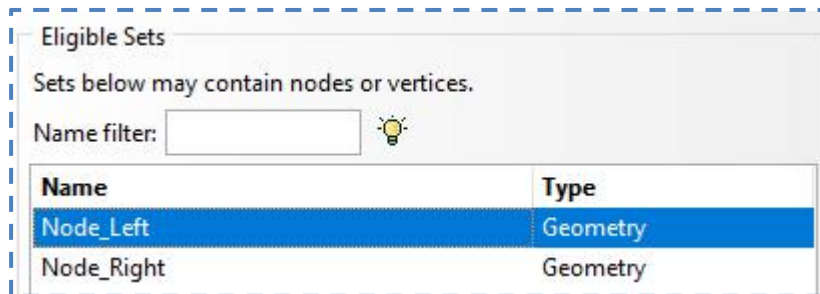
To define load, change to **Load** module.

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

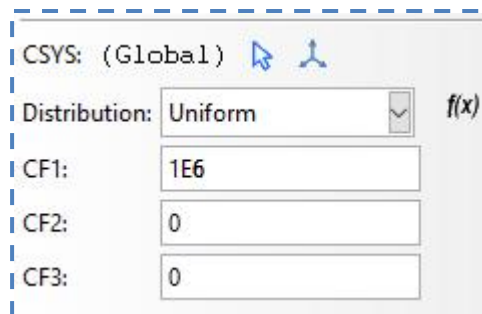
Select **Concentrated** in the Types field and pick **Continue**.



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\_Left**.



Pick **Done** and enter **1E6** as the force magnitude in y-direction.



For an eigenvalue buckling analysis, magnitude of the load is immaterial, because the response (eigenvalue) is proportional to the magnitude of the load. For example, if the magnitude of first eigenvalue is 25 when a load of  $10^5$  is specified, then for the applied load of  $10^3$  the eigenvalue magnitude will become 2500.

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

## ⇒ Boundary Conditions

The required boundary condition has already been defined. The boundary condition “BC-Fixed” constrains all the translational and rotational degrees of freedom of right node of the beam.

## ⇒ Job Submission

All the information required 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 Eigen or whatever name you would like.

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

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

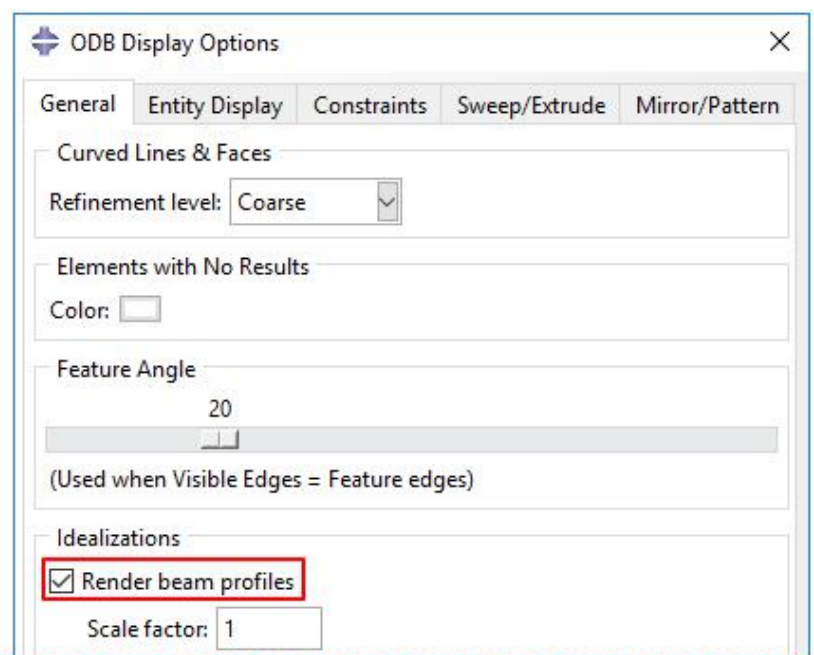
Pick **Results** to view the results in Visualization module.

## ⇒ Postprocessing

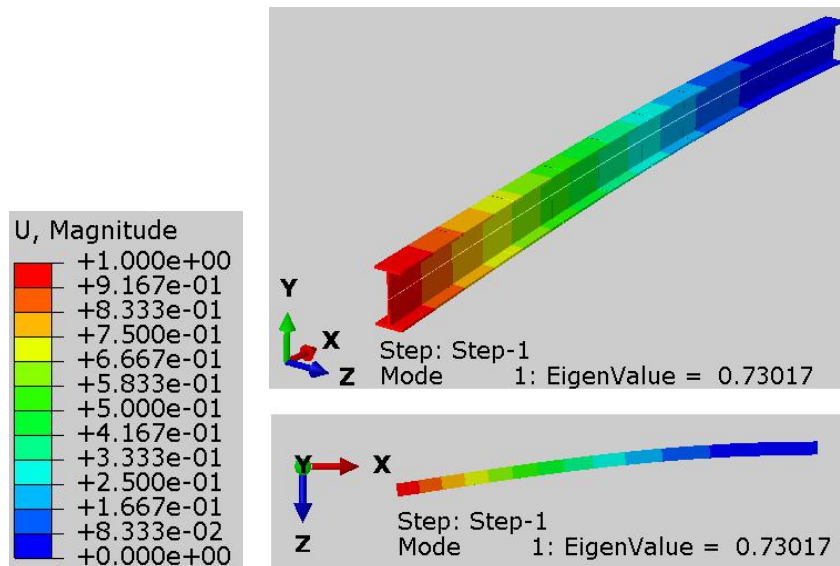
Pick  to plot contours on deformed shape.

To visualize the beam profile on screen, we will turn off the rendering.

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



The contoured plot will appear as shown below. The beam is shown both in isometric and top view for clarity.



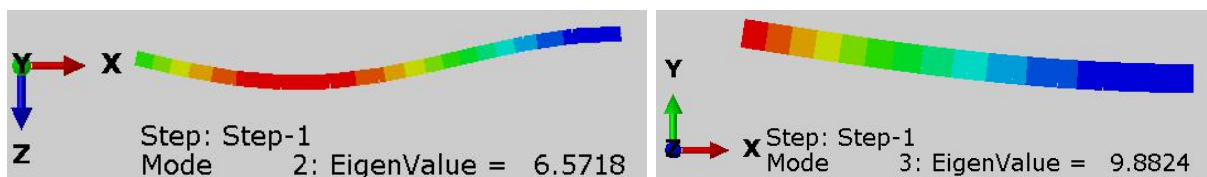
In the above figure first mode shape of the column can be seen. The corresponding eigenvalue is 0.73017. This eigenvalue is the factor by which the applied load must be multiplied to find the buckling load. So in this case the buckling load is 730170 ( $0.73017 \times 1 \times 10^6$ ). Notice that the flexural mode occurs in the XZ plane.

If you look in the legend in the above figure you can notice that maximum displacement is 1.0 m. It is important to know that they do not represent actual magnitudes of deformation.

The buckling mode shapes are normalized vectors and do not represent actual magnitudes of deformation at critical load. They are normalized so that the maximum displacement component is 1.0.

The displacement magnitude shown in the above figure are meaningless. The magnitudes shown cannot be used to compute e.g. stresses or strains in the structure. Only the mode shape and eigenvalue are the worthy output of this analysis.

The following figure shows the second and third modes.



It can be seen that eigenvalues for these modes are higher than the first eigenmode. So first eigenmode gives the lowest buckling load estimate.

Often the first mode shape is more feasible with the next eigenmodes at distinctly higher critical loads. These higher modes can be ignored for further analysis.

## ⇒ Load-Displacement Analysis

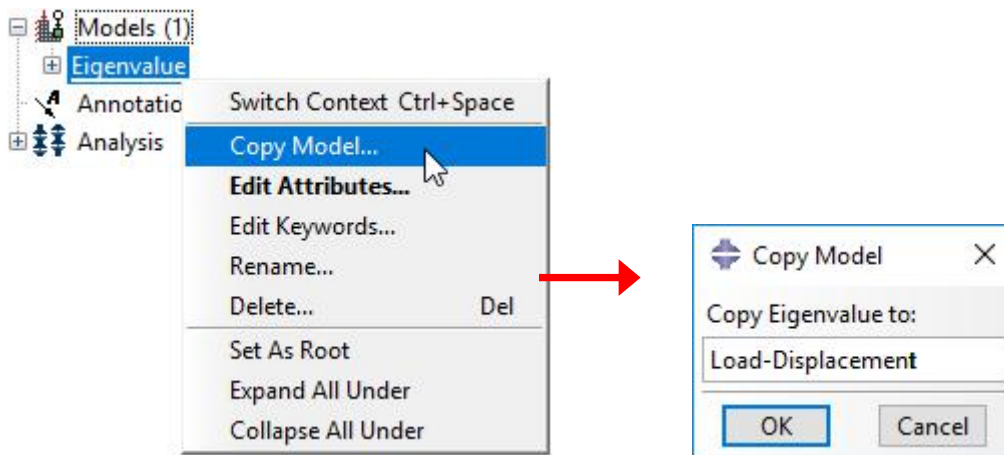
Now we will perform a load-displacement analysis to ensure that eigenvalue buckling prediction is accurate. Furthermore we will also investigate the post-buckling response.

An eigenvalue analysis provides estimate of the critical buckling load and possible mode shapes of the structure that will occur at buckling. It does not provide any information what happens after the structure buckles. If the structure is stable or unstable after the bifurcation point and what is the level of stresses and strains? To answer these questions, a load-displacement analysis is usually performed after the eigenvalue buckling analysis.

We will make a copy of the existing model and replace the existing step with a Static, General procedure.

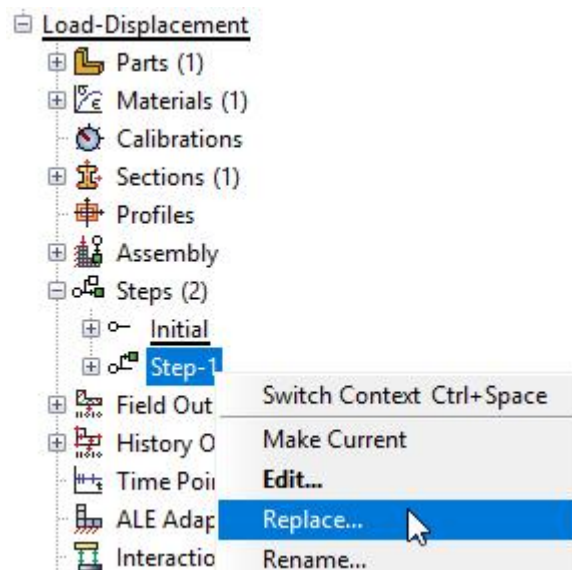
So right-click the Eigenvalue model in the Model Tree and pick **Copy Model**.

Enter **Load\_Displacement** as the name of the copied model.



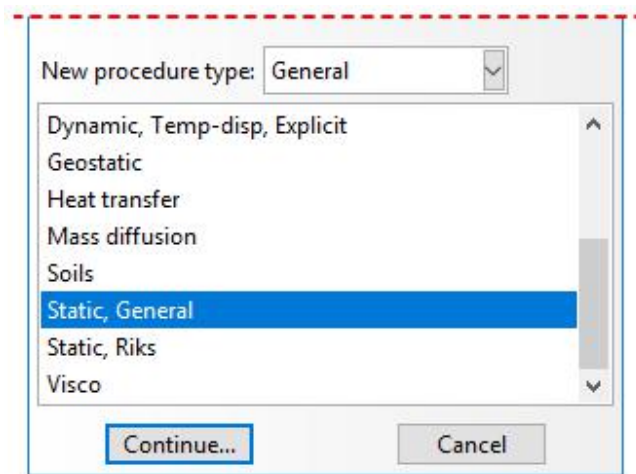
Pick **OK** to apply.

Right-click the Step-1 in the copied model and pick **Replace**.



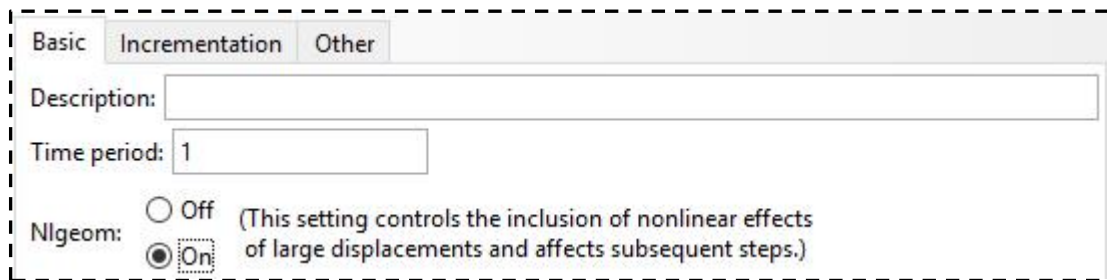
Select the **General** as procedure type.

Pick **Static, General** as procedure.



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

Turn on the **Nlgeom** option.




Under the Incrementation tab, set the initial increment size to **0.01** and a maximum increment size of **0.05**

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

## ⇒ Specifying Load

We expect the column to buckle at a load of about 730170 ( $0.73017 \times 1 \times 10^6$ ). As we also want to investigate the post-buckling response, so we will specify an arbitrary higher load of  $2 \times 10^6$  N.

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

Open the Load Manager by picking  .

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


Enter **2E6** as the pressure magnitude.

Distribution:	Uniform	$f(x)$
CF1:	2E6	
CF2:	0	
CF3:	0	

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

## ⇒ 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 **NoImperfection** or whatever name you would like.

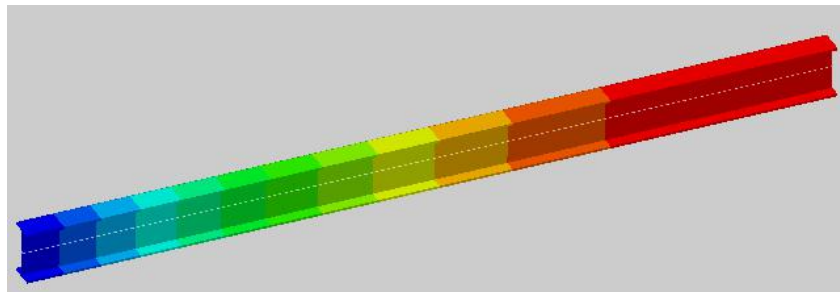
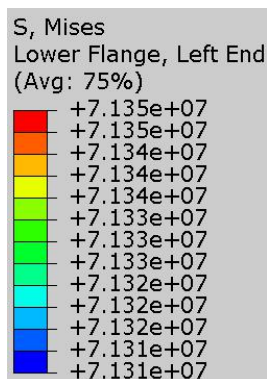
Pick **Continue** and then **OK**.

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. The contour plot of von Mises stress will appear as shown below.

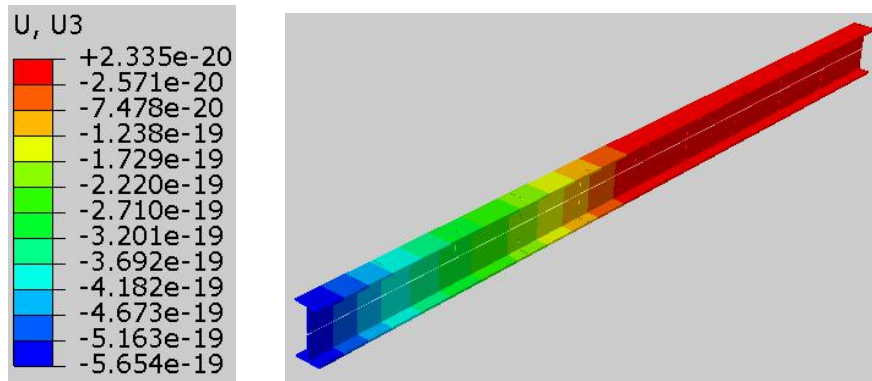


By looking at the legend, it can be seen that the von Mises stress is almost uniformly distributed.

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

Pick **U3** in the component field. The simulation results will appear as shown below.





It can be seen that displacement in the z-direction is almost zero. So we can conclude that column has not buckled.

The representation of a structure's geometry in a computer software does not include the imperfections due to manufacturing tolerances and transporting and handling processes. No material used to manufacture engineering structures is absolutely homogeneous. Furthermore in physical structures, the boundary conditions and load application can not be as perfect as represented during an finite element analysis. Due to these idealizations during finite element analysis, the buckling will usually not occur.

It is possible that an analysis performed using perfect geometry may never buckle at a load sufficiently higher than critical load level because the model has no initial displacement in the postbuckled mode. Therefore it has no ability to switch to that mode. To overcome this limitation, we will introduce a imperfection in the model.


## ⇒ Introducing geometric imperfection

We will introduce an imperfection in the shape of the first buckling mode predicted by the eigenvalue analysis as first mode corresponds to the lowest value of buckling load.

To trigger the buckling, it is needed to introduce an imperfection so that there is some response in the buckling mode before the critical load is reached.

There are different ways to introduce imperfections. We will apply fictitious trigger load which deforms the column in the shape of the first buckling mode. The trigger load will be applied in first step and buckling load will be applied in a second step. So first we will define a new step.

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 **On**.

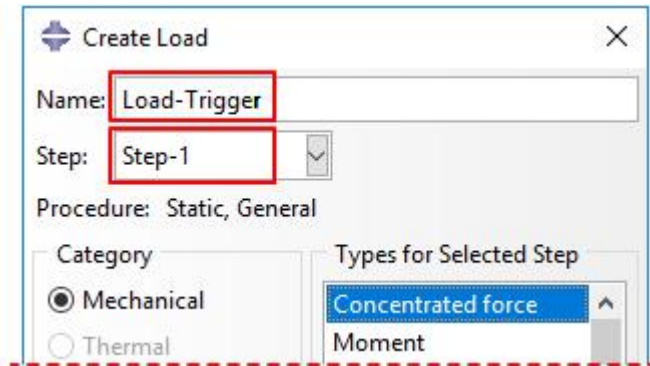
Under the Incrementation tab, set the initial increment size to **0.01** and and a maximum increment size of **0.05**

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

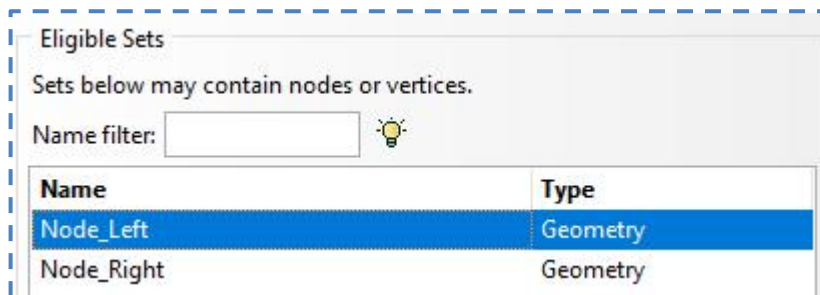
To define load, change to **Load** module.

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

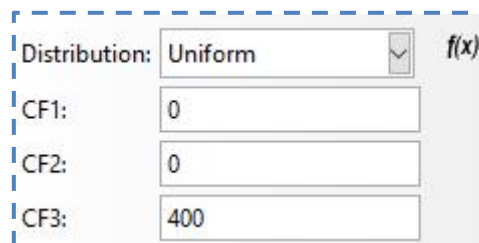
Select **Concentrated** in the Types field and pick **Continue**.




A set has already been defined, so pick **Sets** and select the Node\_Left.



Pick **Continue** and enter **400** as the force magnitude in z-direction.

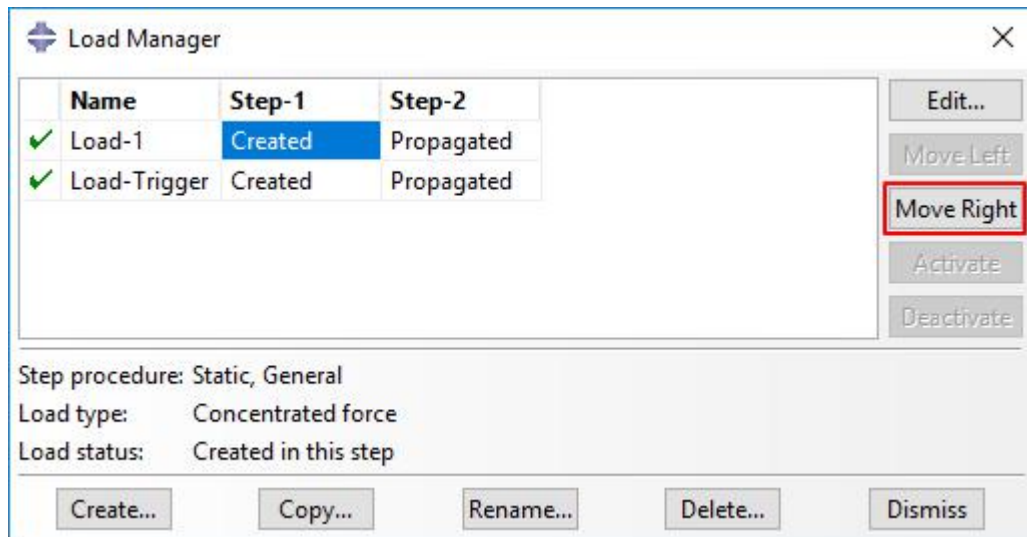


Pick **OK** and it completes the definition of load.

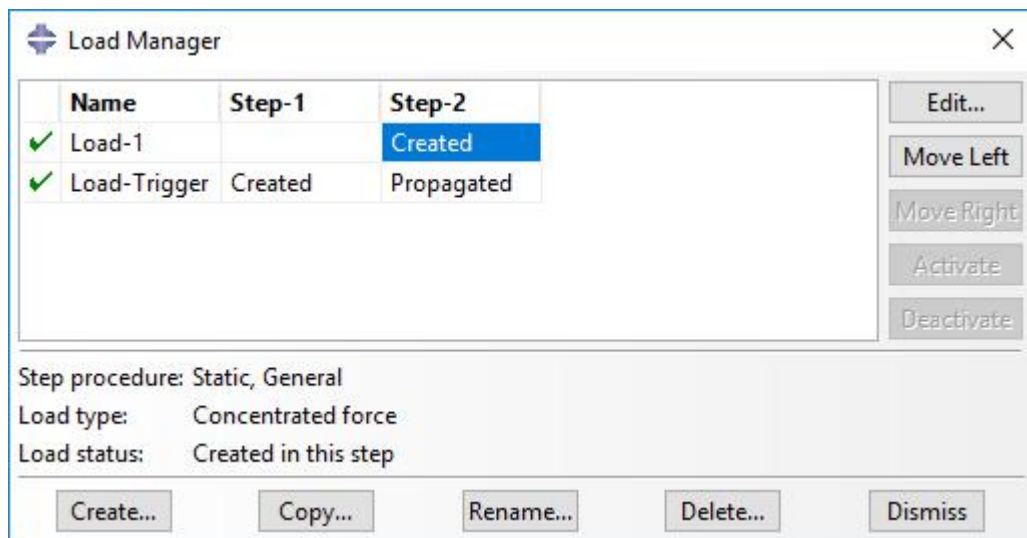
Open the Load Manager by picking .

In the Load Manager it can be seen that two load are created in the Step-1. Our goal is to apply the trigger load in the first step and buckling load in the Step-2. So we need to move the Load-1 under the Step-2 column.

Pick the “Load-1” under column “Step-1” and pick **Move Right**.




Now the Load Manager dialog box will appear as shown below.



Pick **Dismiss** to close the manager.

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 **Imperfection\_L400** or whatever name you would like.

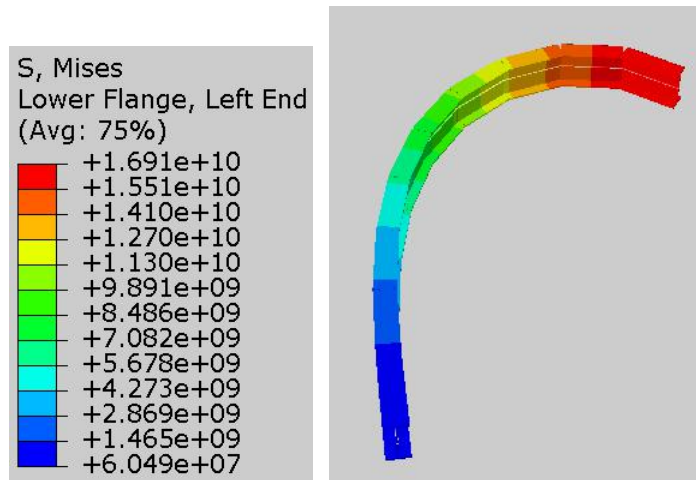
Pick **Continue** and then **OK**.

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

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

## ⇒ Postprocessing

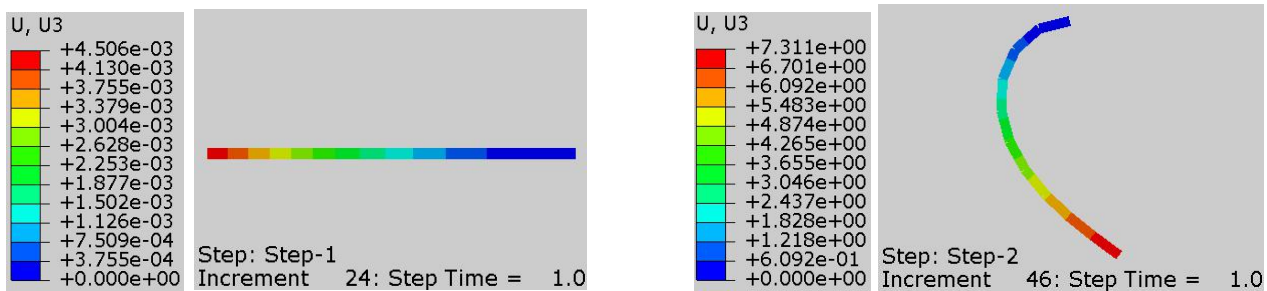
Pick  to plot contours on deformed shape. The contour plot of von Mises stress at the end of Step-2 will appear as shown below.



It can be seen that the von Mises stress is not uniformly distributed as in the previous case without imperfection.

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

Pick **U3** in the component field. The following figure shows the contour plot at the end of “Step-1” and “Step-2”.



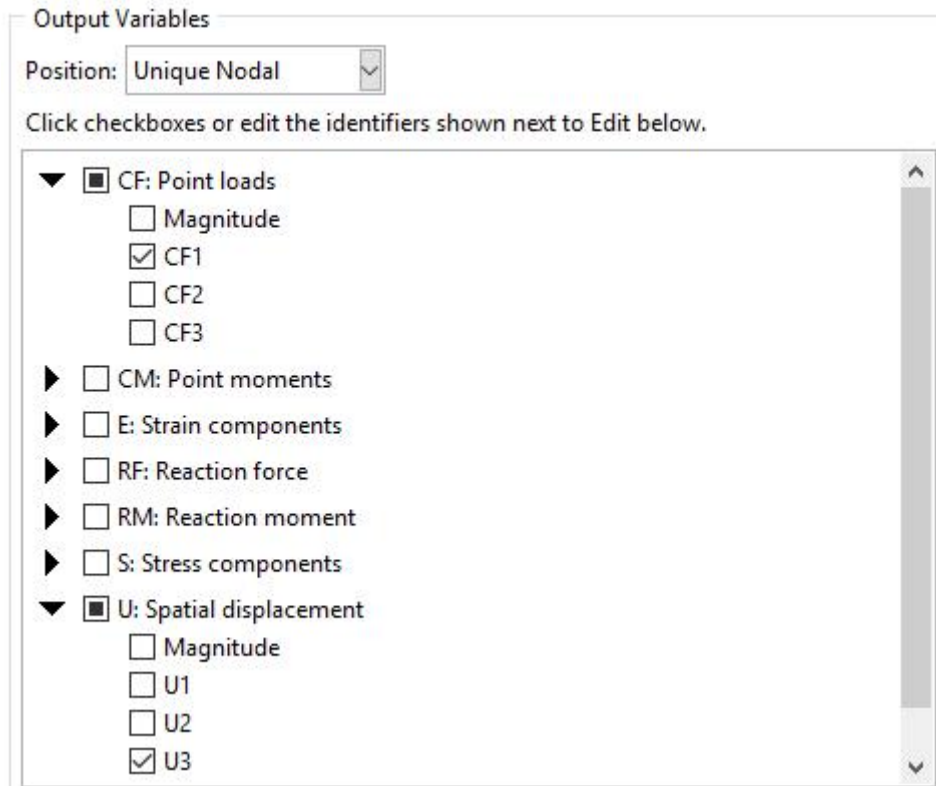
It can be seen that left end of beam has displaced about 4.50 E-3 m in the z-direction at the end of Step-1. This comes out to be about 1.4% of the width of column ( $4.50 \text{ E-}3/0.3114 \times 100 = 1.4\%$ ). In other words the trigger force has introduced an imperfection of about 1.4% of the width of column. Notice that this deformation corresponds to the shape of first buckling mode of column. When the load is applied in Step-2, the structure buckles due to the presence of imperfection.

Now we will plot the applied force versus the displacement of the left end of column.

Pick  and Create XY Data dialog box will appear.

Pick **ODB field output > Continue**

In the Position field, select **Unique Nodal** and check the **CF1** and **U3** checkboxes.



In the Elements/Nodes tab, pick **Node sets** as method and select the **NODE\_LEFT**.

Pick **Save** to save the plot and then pick **OK** to confirm.

It will create XY data plots of CF1 and U3 versus time.

Pick **Dismiss** to close the Field Output dialog box.

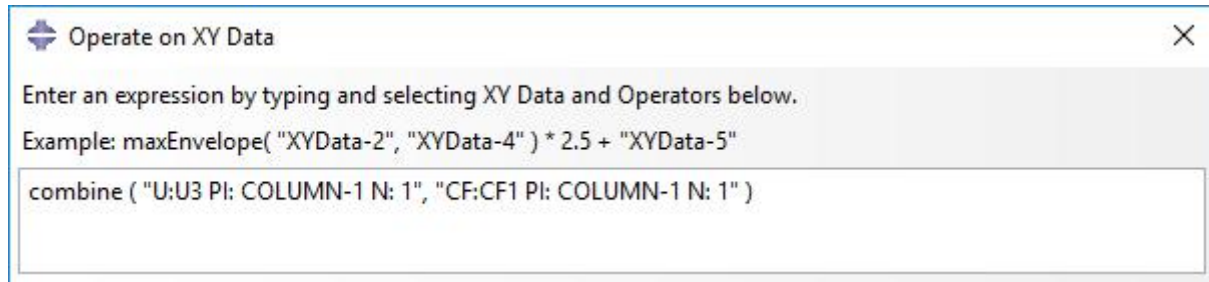
Each of the XY data curves is a history (variable versus time) plot. We will combine the plots to produce the desired force-displacement plot.

Again pick  and select **Operate on XY data** option.

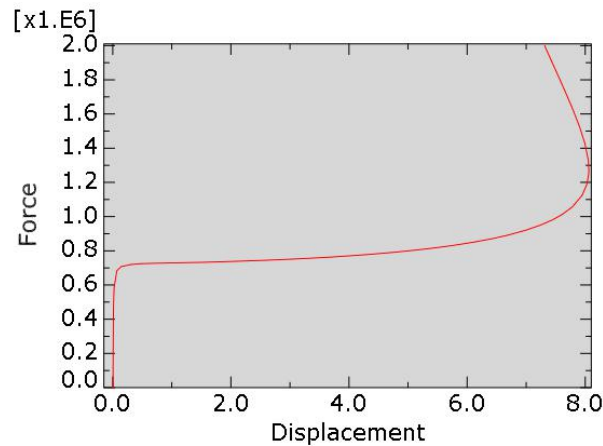
Pick **Continue** to proceed.

In the Operate on XY Data dialog box, pick **combine(X,X)** operator.

Then double-click the “U:U3.....” data and then “CF:CF1.....” data in the list and expression will appear as shown below.



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



As the plot looks satisfactory so pick **Save As** to save the plot.

It can be seen in the above figure that as the force reaches a magnitude of about  $0.7 \text{ E6 N}$ , the structure loses stiffness sharply. Remember that eigenvalue analysis predicted a buckling load of about 730170 ( $0.73017 \times 1 \times 10^6$ ). So it can be said that buckling load predicted by load-displacement analysis is almost the same as obtained by eigenvalue buckling procedure. Also notice that as the force magnitude is increased, the postbuckling behavior is stable i.e. system has positive stiffness matrix up to the maximum force applied.

If the response of a structure is linear elastic before it buckles, the buckling prediction using the eigenvalue buckling procedure is usually accurate. In other words, eigenvalue buckling analysis is useful for structures that exhibit only small, elastic deformations prior to buckling. If the structure exhibits material nonlinearity or geometric nonlinearity prior to buckling, a load-deflection analysis must be performed to investigate their influence on buckling load.

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

