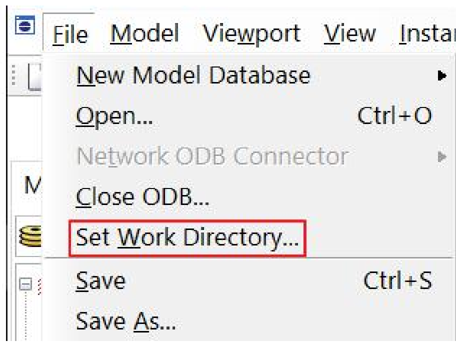
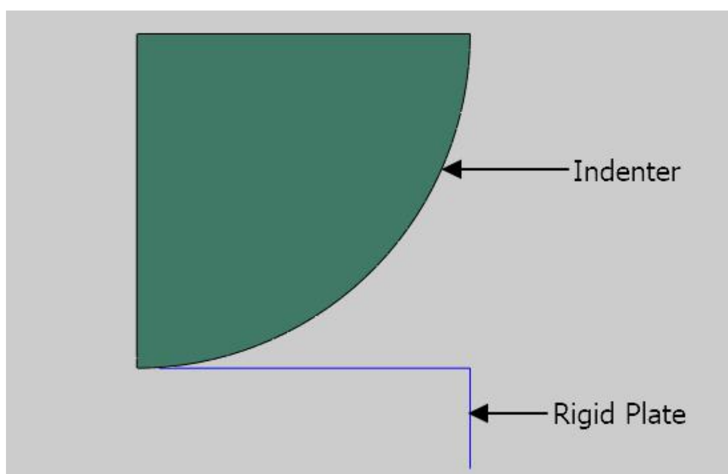
Exercise 25

In this exercise we solve the Hertzian problem of an elastic cylindrical indenter contacting a rigid surface by using the contact pair algorithm. We assume frictionless contact. The numerical results will also be compared to an analytical solution. By taking advantage of the symmetry, only a half of the indenter will be considered for analysis.

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



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

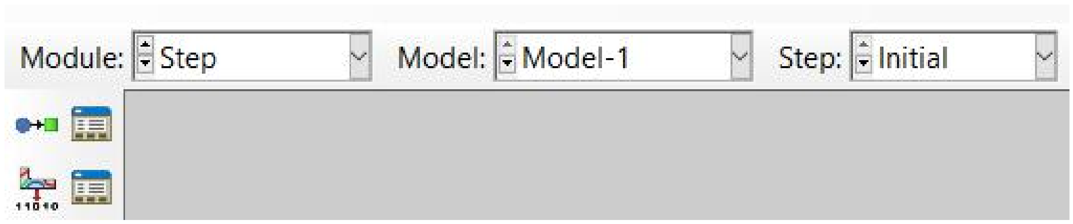


The model consists of two two-dimensional parts: indenter and rigid plate. The indenter has a radius of 50mm. The rigid plate is modeled as an analytical rigid part. The indenter is assumed to be made from Aluminum with a Young's modulus of 70 GPa and a Poisson's ratio of 0.3. It is assumed that contact is frictionless. The indenter is meshed with bilinear plane strain quadrilateral elements with reduced integration (CPE4R). It is finely meshed in the area which could come in contact with rigid surface.

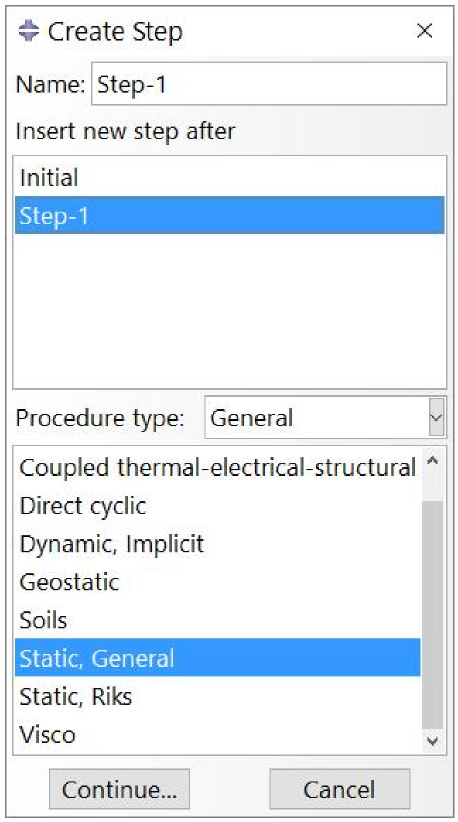
⇨ Defining Step

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.

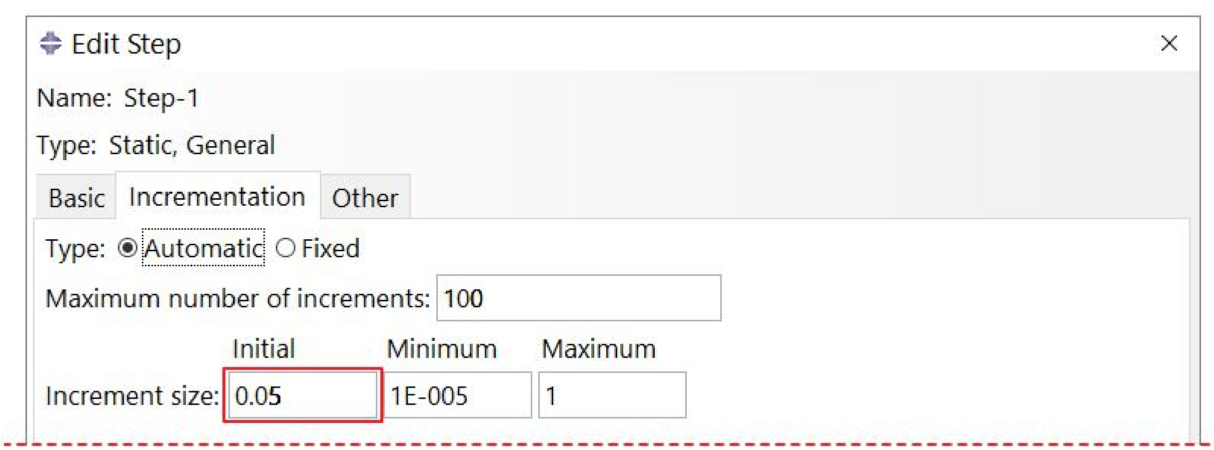


Pick  to create a new step and select the **Static, General** step.



Pick **Continue** and Edit Step dialog box will appear. Notice that total time period is set to **1.0.**

Under the Incrementation tab, set the initial increment size to **0.05**



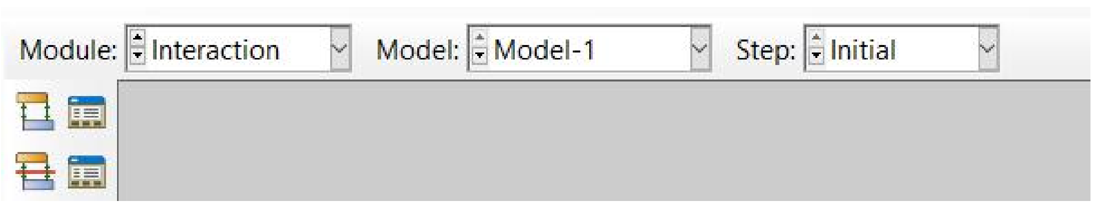
The ratio of the initial time increment to the total time period specifies the proportion of load applied in the first increment. As the contact problems are quite nonlinear in nature it is suggested, in general, to set the initial time increment to be 5% or less of the total time period.

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

Defining Contact Interaction

A contact interaction should be defined between the elastic indenter and rigid plate. The rigid surface must be the master surface and the surface defined on indenter edges will be the slave surface.

Now we will create contact interaction. So change to **Interaction** module.

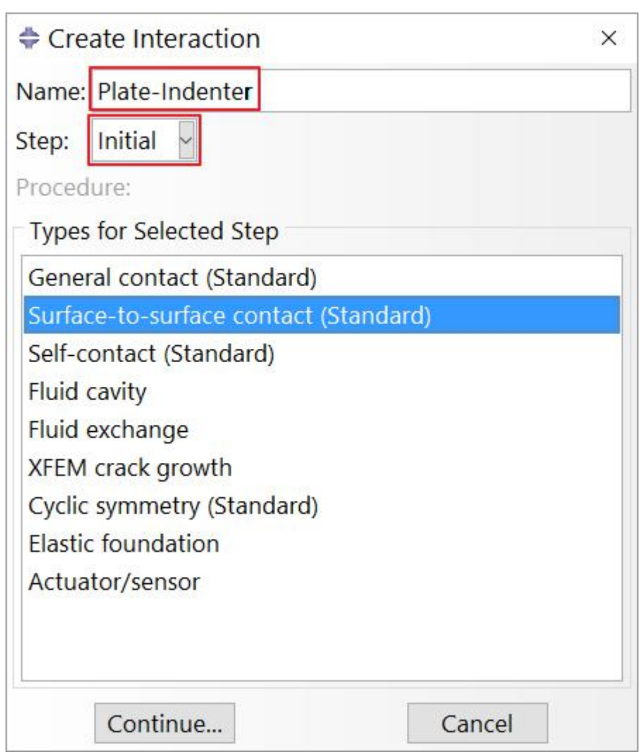


Pick  to create a new interaction.

Enter **Plate-Indenter** as the name of the interaction.

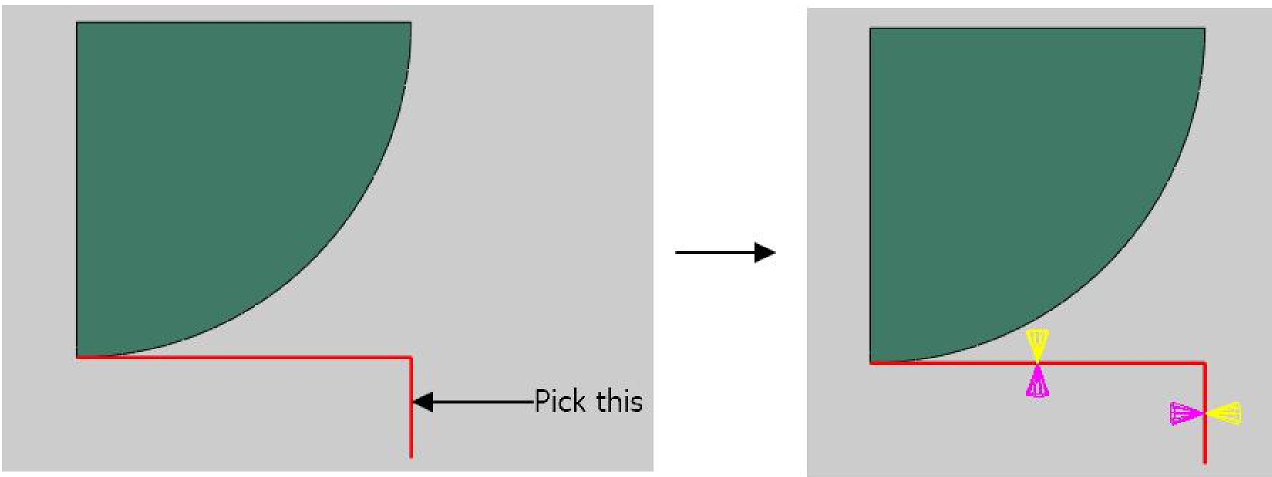
Pick **Initial** in the Step field.

Select **Surface-to-surface contact** as the type of the interaction.



Pick **Continue** and select the rigid surface as master.

Pick **Done** and select **Yellow** for the side of surface in contact.



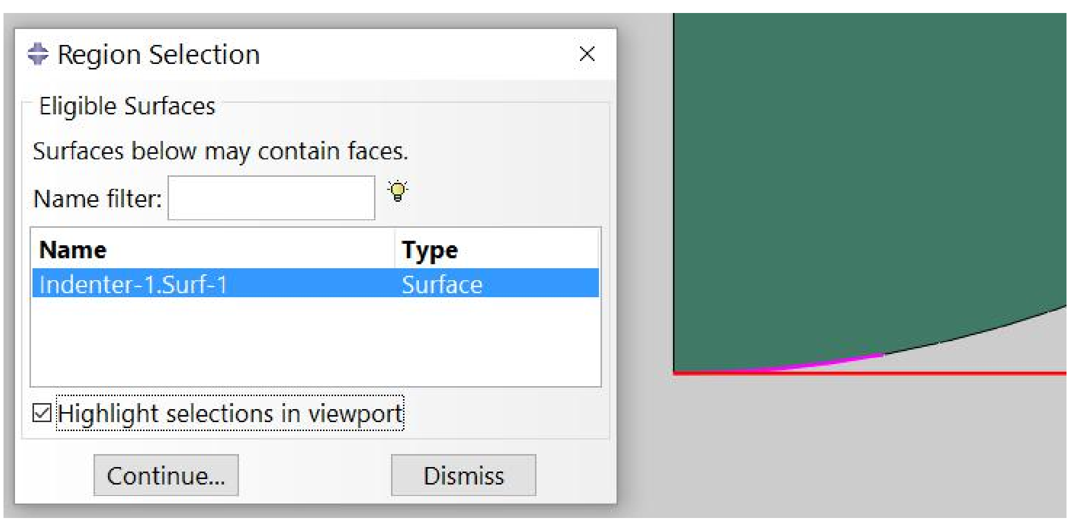
As a rigid surface can contact on either side, so it is important to specify the desired side of the surface to be in contact. It is done by choosing the color associated with the desired side.

For the slave type, pick **Surface**.

We can either pick a surface or a node region as slave. A surface is always preferred over a node region for better accuracy.

We have already defined a surface so pick **Surfaces** and select the Indenter-1.Surf-1.

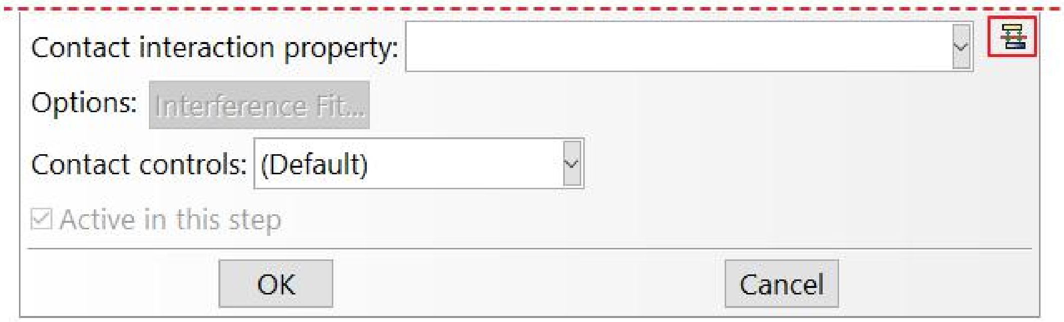
Select **Highlight selections in viewport** to view the selected surface on screen as shown below.



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

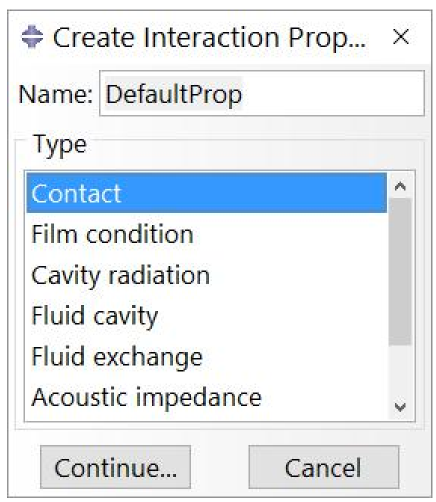
Each contact interaction must refer to a contact interaction property that governs the interaction behavior. This property can be created either before defining the interaction or during the definition of contact interaction.

To define the contact interaction property, pick  in the Edit Interaction dialog box.

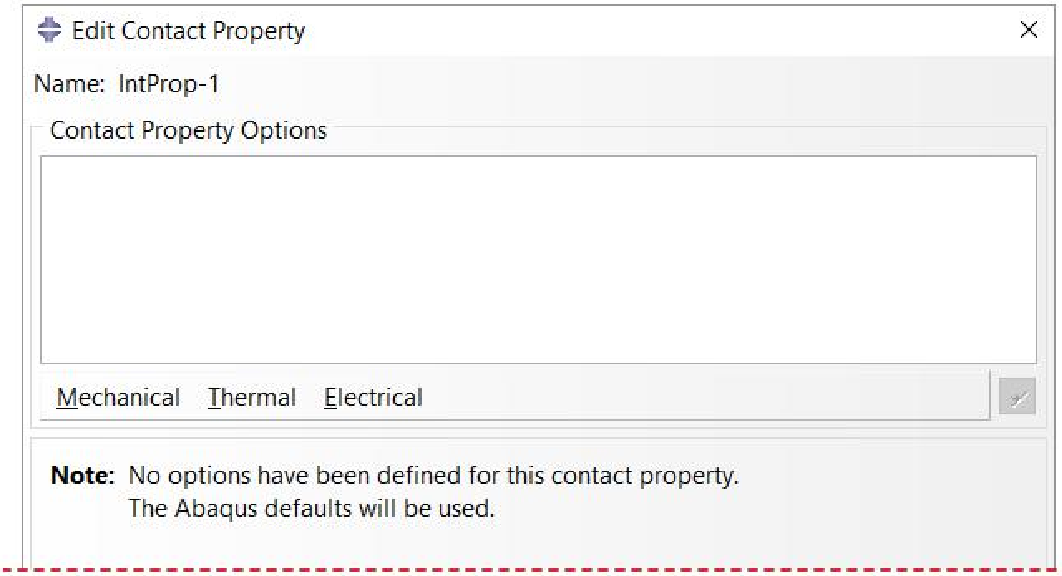


Select **Contact** as type for the interaction property.

Enter **DefaultProp** as the name for the property.



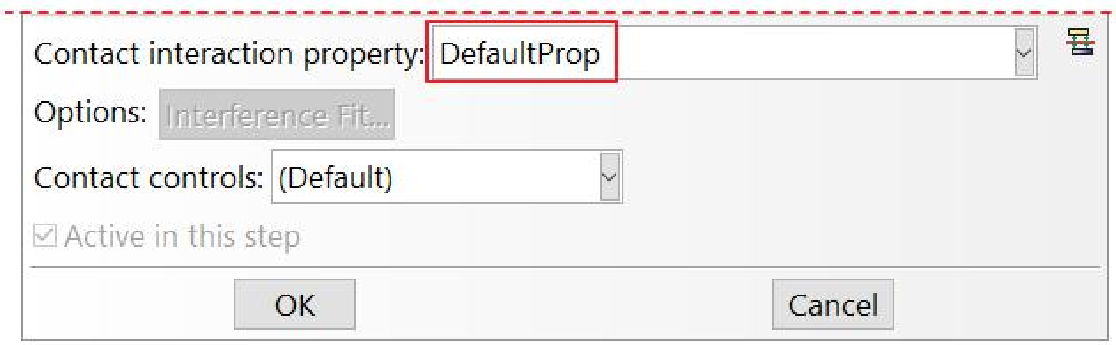
Pick **Continue** and Edit Contact Property dialog box will appear.



We will use the default options so pick **OK** in the Edit Contact Property dialog box.

By default, "hard" contact is enforced in normal direction and frictionless contact in tangential direction when no other option is specified.

In the Edit Interaction dialog box, select the **DefaultProp** as interaction property.

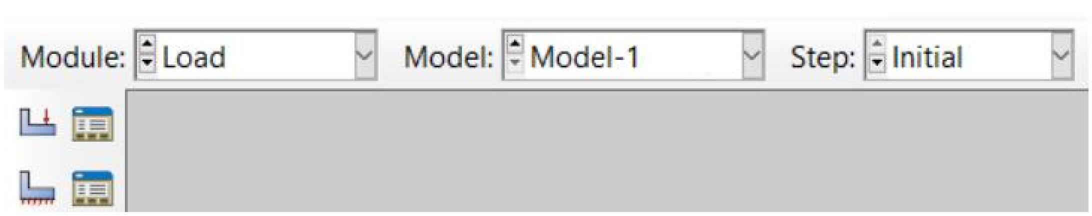


Pick **OK** and it completes the definition of interaction.

Defining Load

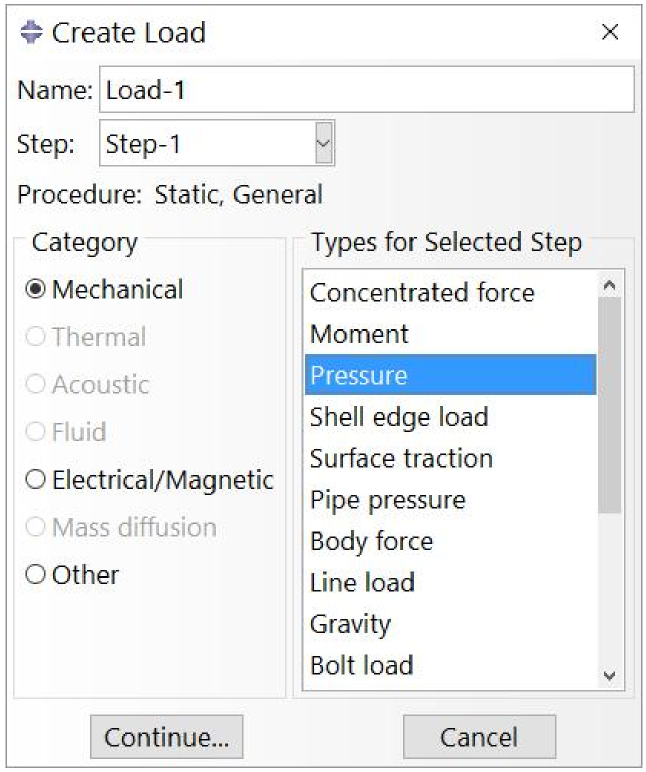
A distributed load will be applied on the top boundary edge of the indenter.

So change to **Load** module.

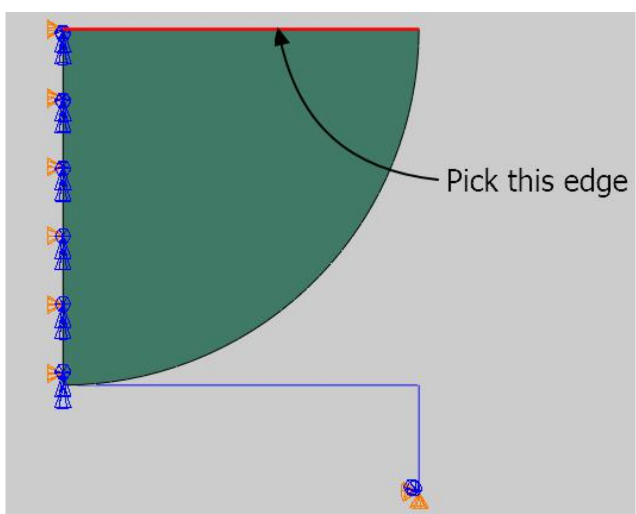


Pick  to create a load and select Step-1 in the Step field.

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



Select the top horizontal edge of the indenter as a reference for load.

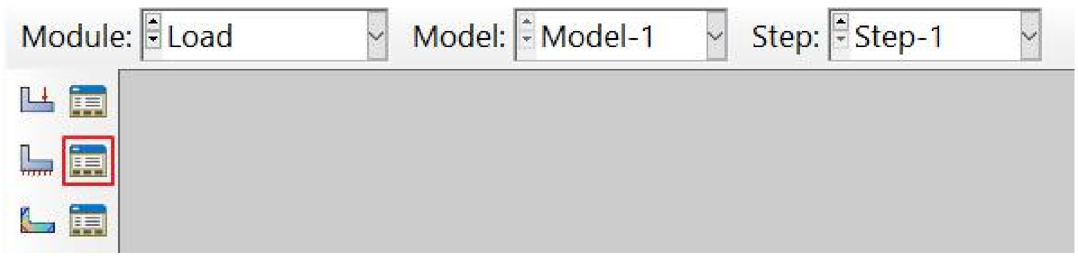


Pick **Done** and enter **60E6** as the pressure magnitude.

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 “Indenter_Symm” applies symmetry boundary condition to the edge of the indenter lying on symmetry plane.

The boundary condition “RigidPlate” constrains the motion of the Reference point attached to the Rigid Plate both in x and y-directions.

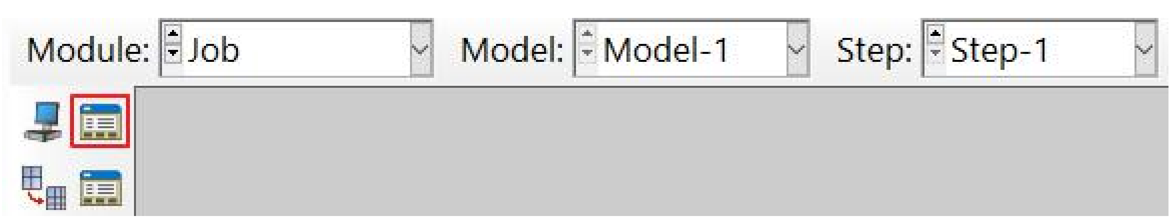
The motion of the Reference Point governs the motion of the rigid surface.

Pick **Dismiss** to close the manager.

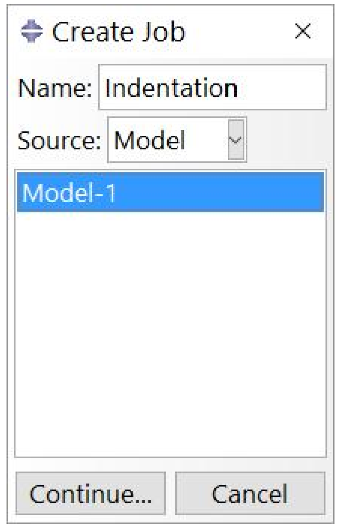
Job Submission

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.

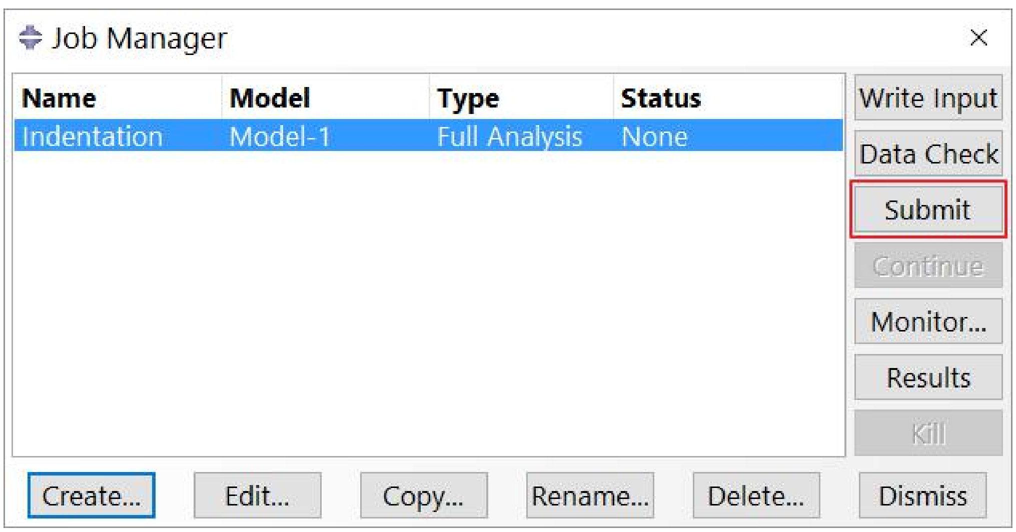


Pick **Create** and create a job named Indentation or whatever name you would like.



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

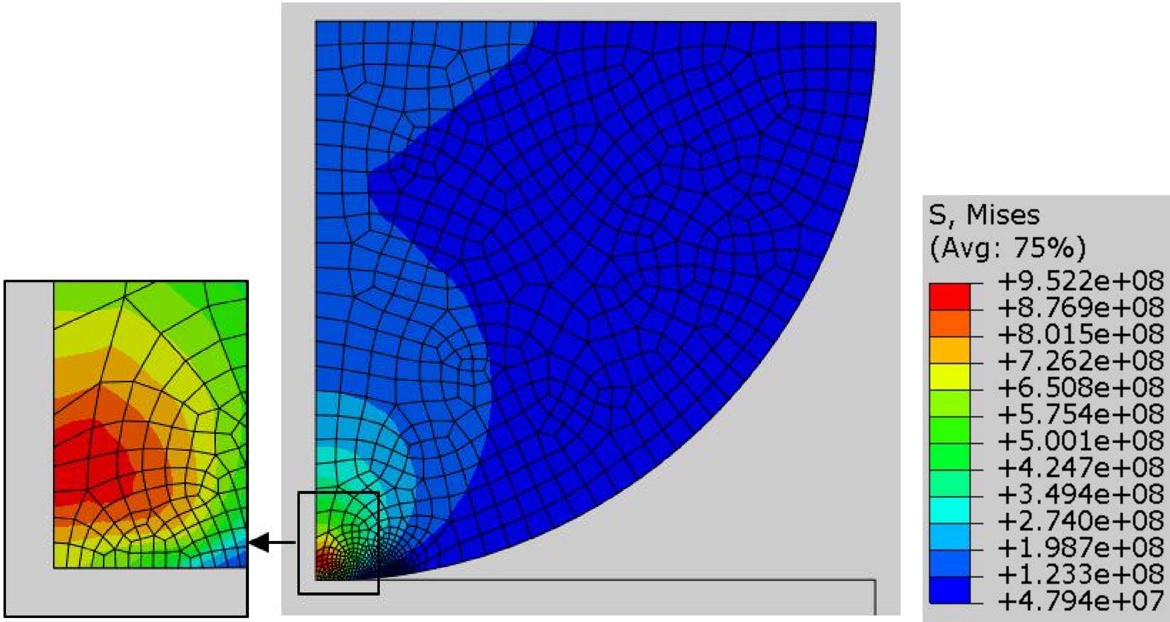
Pick **Submit** to submit the job for analysis.



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

Postprocessing

Pick  and contour plot of von Mises stress at the end of simulation will appear as shown in the figure below. (Deformation Scale factor is set to 1)



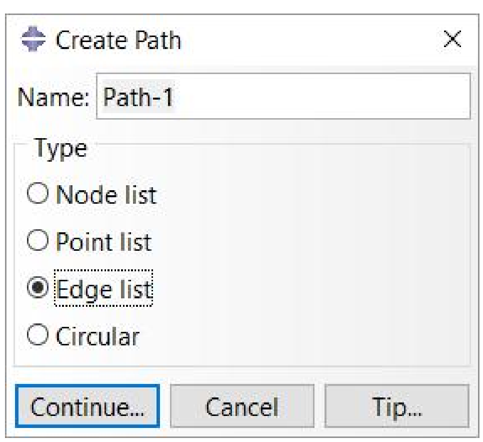
An interesting result from this simulation is that the highest stress occurs below the contacting surface as shown in the above figure.

Contour plot of contact pressure on deformed shape is of not much help for the current problem. So we will plot the contact pressure along the boundary of the indenter in the form of an X-Y plot. For that purpose we will define a "Path", which consists of a series of line segments.

From the main menu bar, pick **Tools** > **Path** > **Create** to define a path along which contact pressure will be plotted.

A path is a line defined by specifying a series of points or line segments.

Pick **Edge list** in the dialog box.

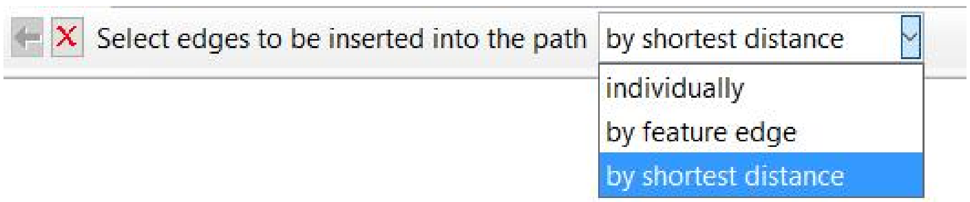


In an edge list path the segments are defined by element edges.

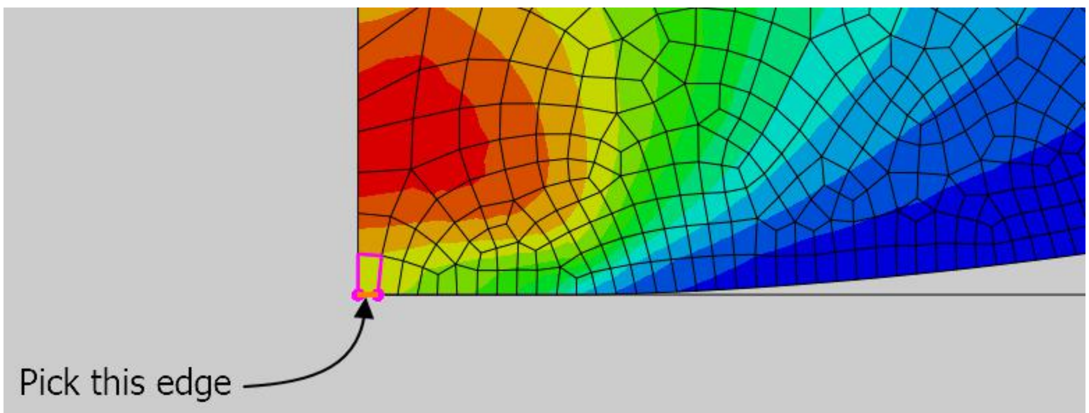
Pick **Continue** and Edit Edge List Path dialog box will appear.

Pick **Add Before** to start picking the edges in the viewport.

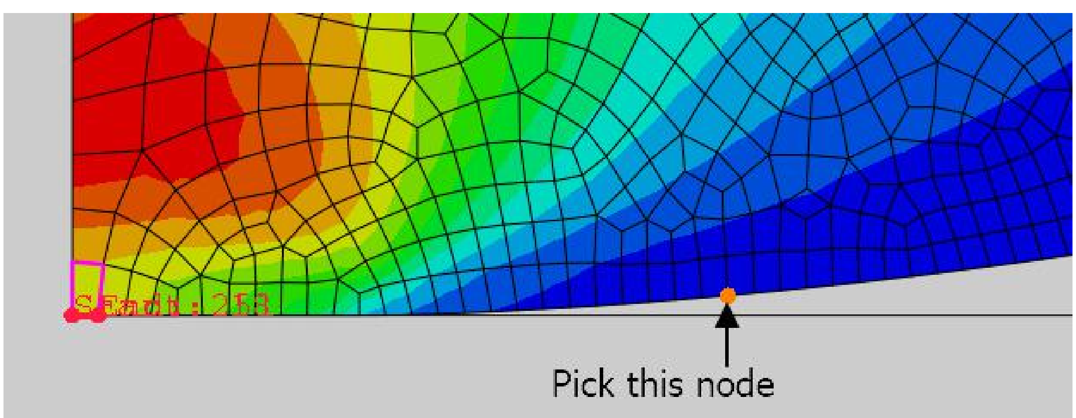
Pick **by shortest distance** to pick the desired edges.



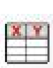
Pick the edge shown in the figure below.



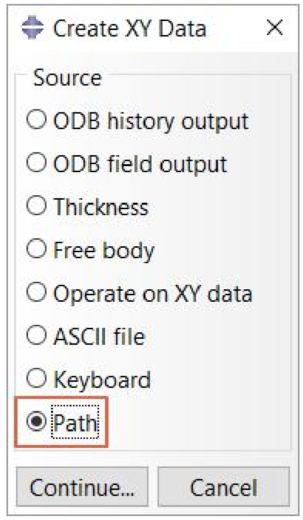
Pick the node shown in the following figure below to complete the specification of the path. (you do not need to pick exactly the same node, any other nearby node on this edge should suffice).



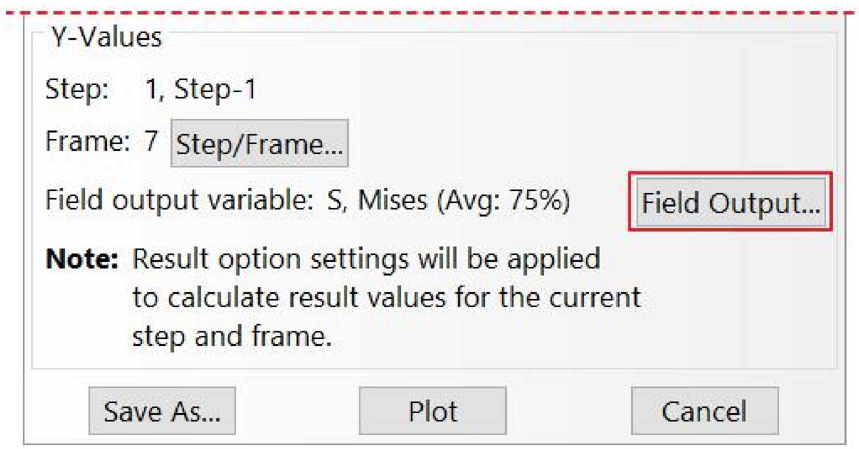
Pick **Done** and then **OK** to complete the definition of the path.

Pick  to access the Create XY Data dialog box.

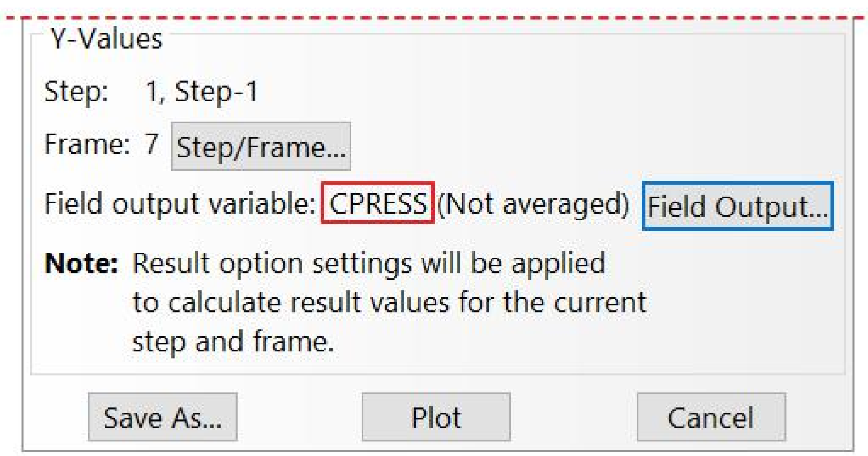
Pick **Path > Continue**



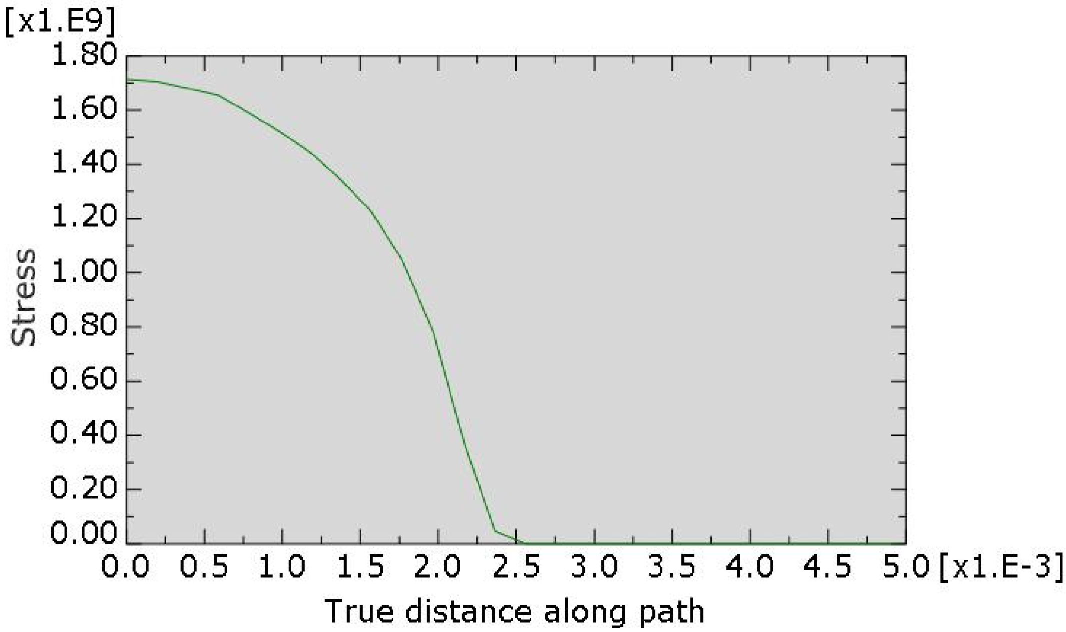
In the “XY Data from Path” dialog box, pick **Field Output.**



Select **CPRESS** (contact pressure) as output variable and pick **OK**.



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



Comparing to Analytical Results

In many industrial applications the contact problem is highly complex due to intricate geometries of contacting bodies. For such problems analytical solutions are not available to be compared against numerical results. However, analytical solution is available for a Hertzian problem and provide an opportunity to verify the contact results from Abaqus.

The distribution of the normal traction σ_n at the contact interface due to applied load $W = \int w ds$, where w is load per unit length, can be determined analytically by the following relation which is based on the Hertzian contact theory.

$$\sigma_n(x) = \sigma_n^{\max}\sqrt{1-\left(\frac{x}{b}\right)^2}$$

where x is the distance from the symmetry line of the cylinder, b is the width of the contact zone given as

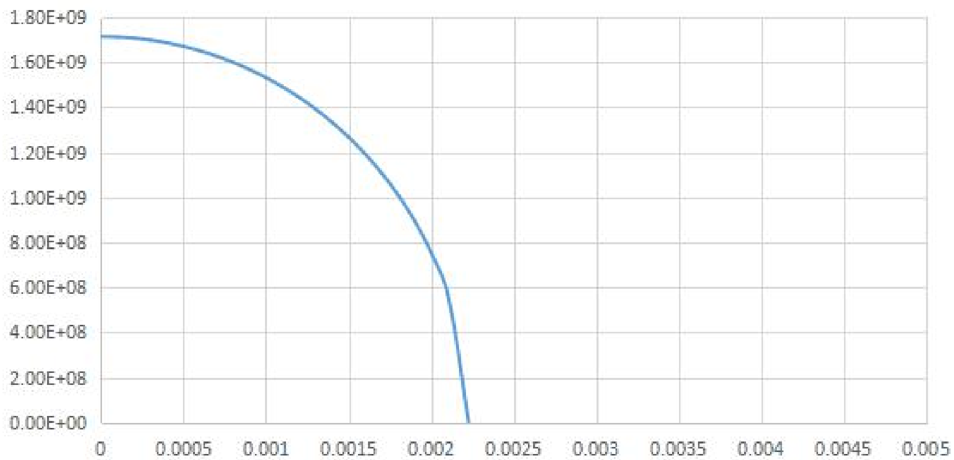
$$b = 2\sqrt{\frac{WR(1-v^2)}{\pi E}}$$

And $\sigma_n^{\max}$ is the maximum normal traction given as

$$\sigma_n^{\max} = \sqrt{\frac{WE}{\pi R(1-v^2)}}$$

Here R is the radius of the cylinder, v is the Poisson's ratio and E is the Young's modulus.

The plot of contact pressure for the given problem will appear as shown below when we use the above relation. The x-axis of the plot has been adjusted so that it is comparable to the graph obtained from Abaqus.



By looking at the graphs, it can be concluded that numerical simulation results from Abaqus and analytical results are quite similar.

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

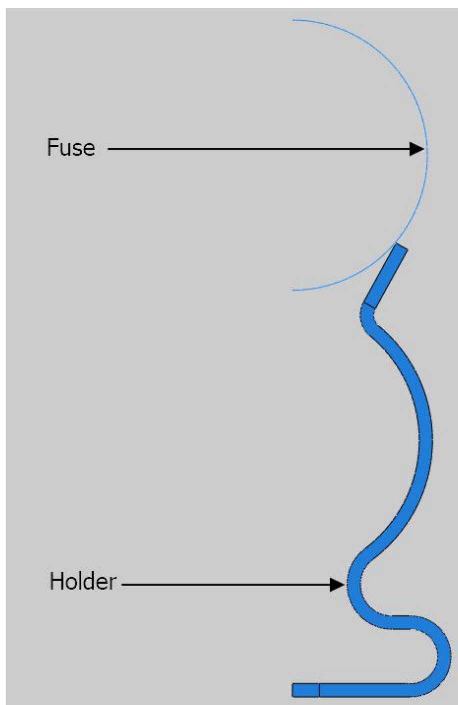
Exercise 26

In this exercise we will determine the contact pressure distribution between a fuse and its holder by using the contact pair algorithm.

Cylindrical fuses are widely used for protection against overloads. A cylindrical fuse is mounted in a clip holder by sliding form the top and is firmly held in place. In this exercise we will simulate the mounting of a fuse into its holder and determine the contact area and contact pressure distribution. By taking advantage of the symmetry, only a half of the model will be considered for analysis.

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

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



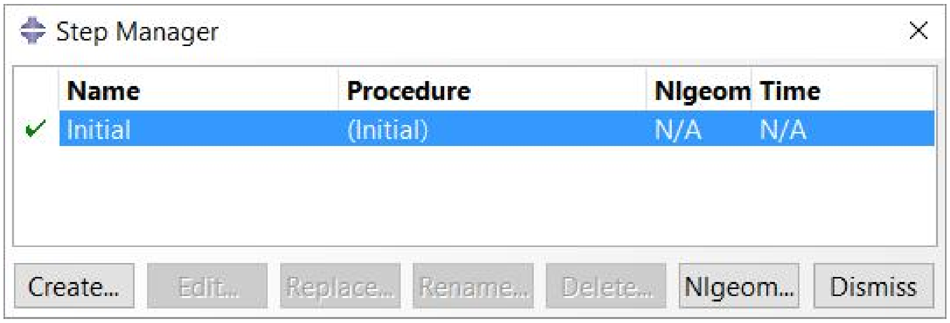
The model consists of two two-dimensional parts: fuse and holder. The fuse is modeled as an analytical rigid part. The holder is assumed to be made from brass with a Young's modulus of 100 GPa and a Poisson's ratio of 0.3. It is assumed that contact is frictionless. The holder is meshed with bilinear plane strain quadrilateral elements with reduced integration (CPE4R).

⇨ Defining Steps

We need to define two steps to complete this analysis. In first step, the fuse will move downwards by applying appropriate boundary conditions. In the second step, boundary conditions will be relaxed to allow the fuse to adjust its position freely.

Change to **Step** module.

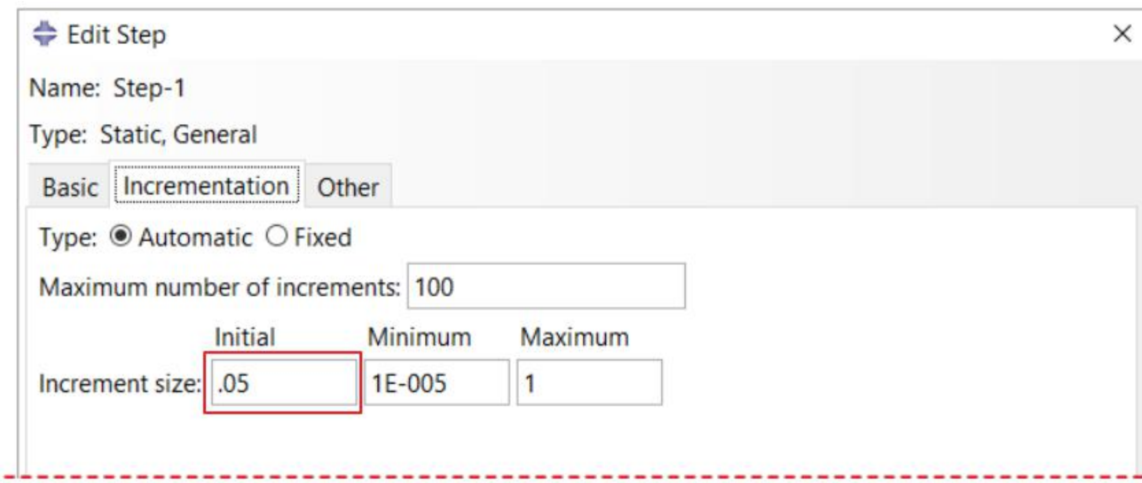
Open the Step Manager by picking  . It will appear as shown below.



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

Pick **Continue** and notice that total time period is set to **1.0.**

Under the Incrementation tab, set the initial time increment to **0.05**



As contact will establish between two parts during this step, we need to set the initial time increment to be a fraction of the total time period.

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

To create second step, again pick **Create** and select the **Static, General** step.

Pick **Continue** and Edit Step dialog box will appear. Notice that total time period is set to **1.0.**

Under the Incrementation tab, set the initial time increment to **0.05**

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

Defining Contact Interaction

Now we will define a contact interaction between fuse and holder. As the fuse consists of a rigid surface so it must be the master surface.

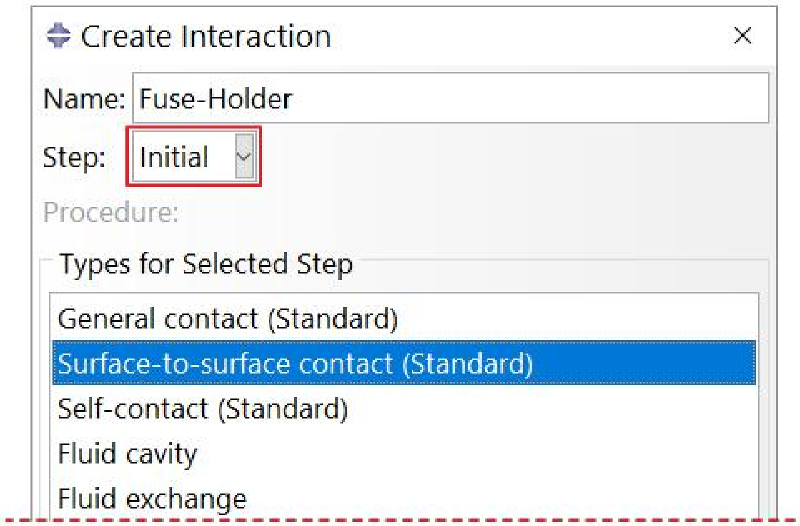
Change to **Interaction** module and open the Interaction Manager by picking .

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

Select **Surface-to-surface contact** as the type of the interaction.

Enter **Fuse-Holder** as the name of the interaction.

Make sure that this interaction is created in the Initial step as shown below.

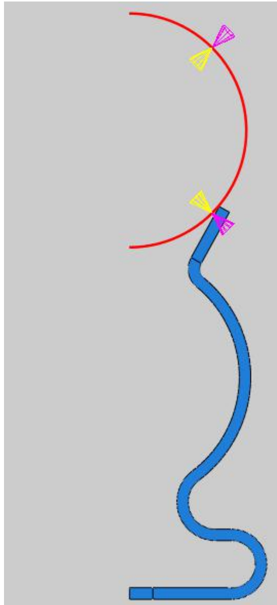


When an interaction is created in the Initial step, it automatically propagates to the later steps, where it can be modified if desired.

Although we can create the interaction in Step-1 with the same end results, it is recommended to create it in Initial step. If we later decide to delete a step, it leaves the interaction intact.

Pick **Continue** and select the rigid surface as master.

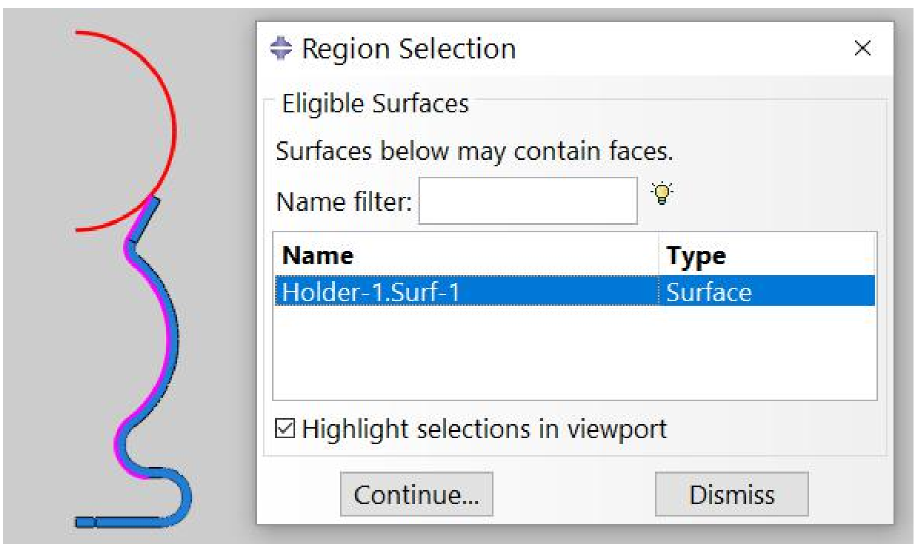
Pick **Done** and select **Magenta** for the side of surface in contact.



As a surface can contact on either side, so it is important to specify the desired side of the surface to be in contact. It is done by choosing the color associated with the desired side.

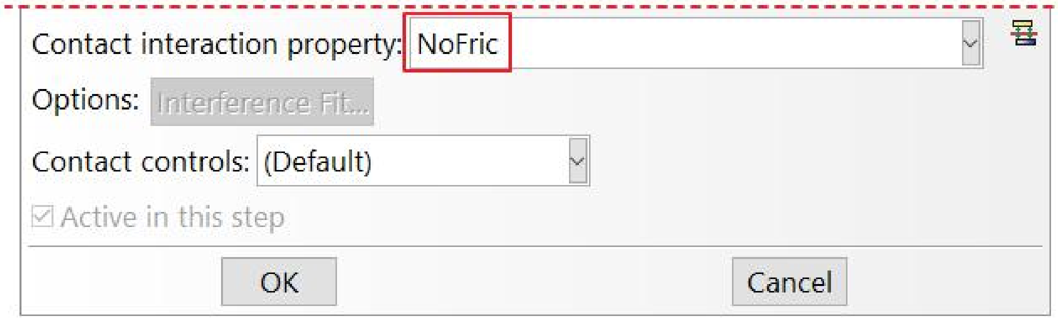
For the slave type, pick **Surface**. We have already defined a surface so pick **Surfaces** and select the holder-1.Surf-1.

Select **Highlight selections in viewport** to view the selected surface on screen as shown below.

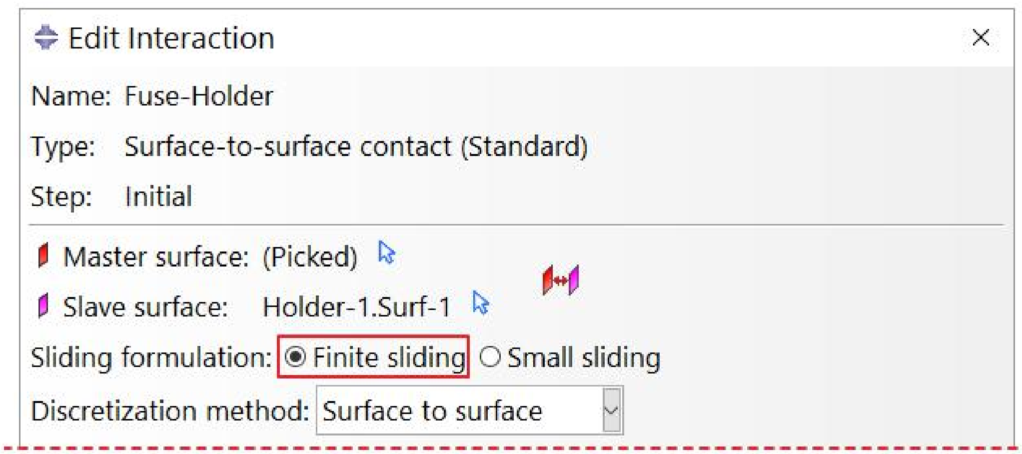


Pick **Continue** and Edit Interaction dialog box will appear.

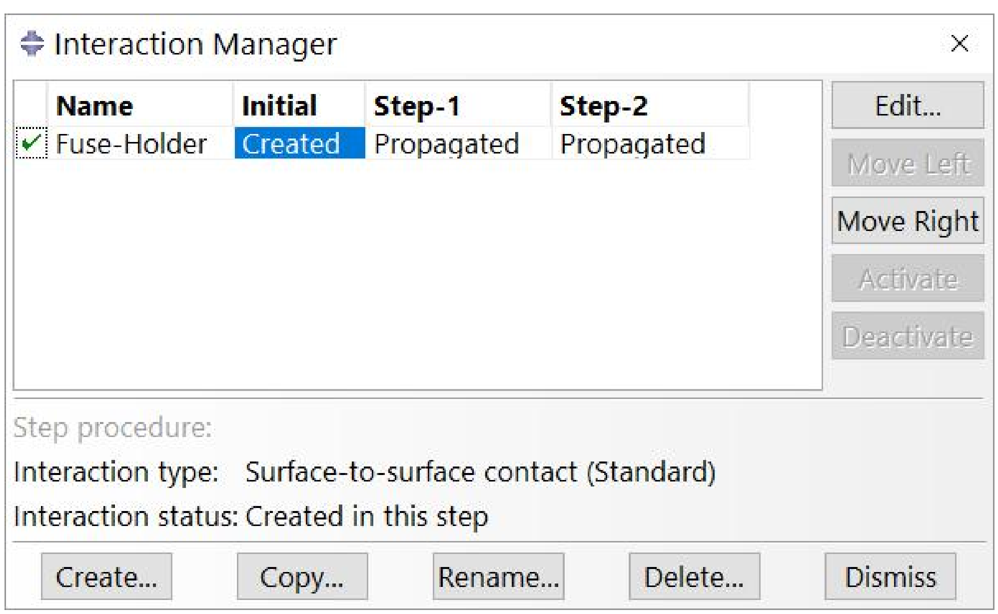
Each contact interaction must refer to a contact interaction property that governs the interaction behavior. The desired property has already been created and is selected by system automatically as shown below.



In this problem as there is considerable sliding between the fuse and its holder so we will use Finite sliding option which is active by default.



Pick **OK** and it completes the definition of interaction. It can be seen in the Interaction Manager that newly created interaction has been propagated through all the steps.

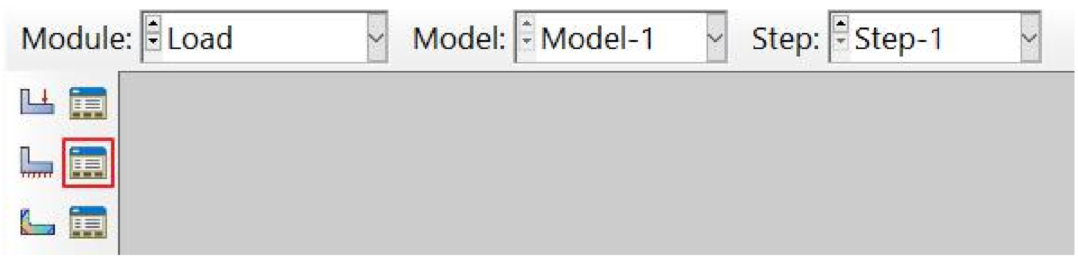


Pick **Dismiss** to close the manager.

Boundary Conditions

Boundary conditions required for the analysis have already been defined. We will modify the boundary condition applied to the fuse.

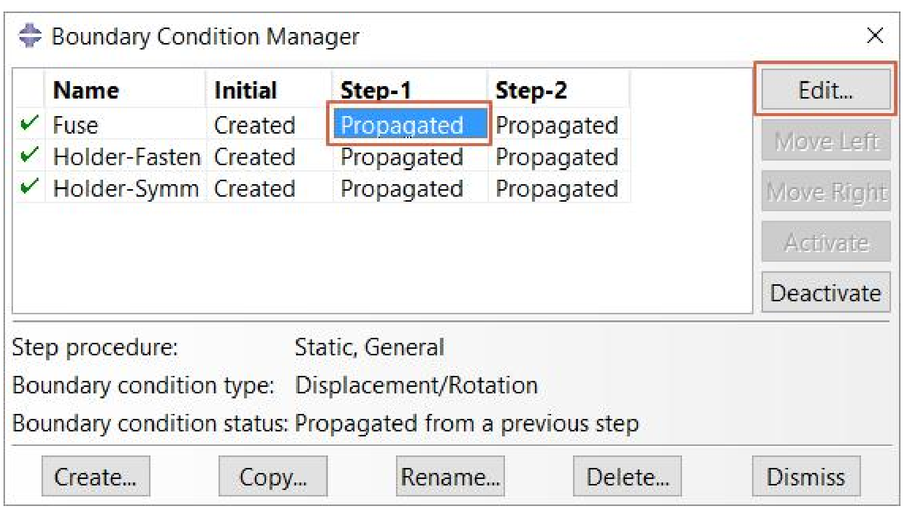
To review the boundary conditions change to **Load** module and open the Boundary Conditions Manager by picking .



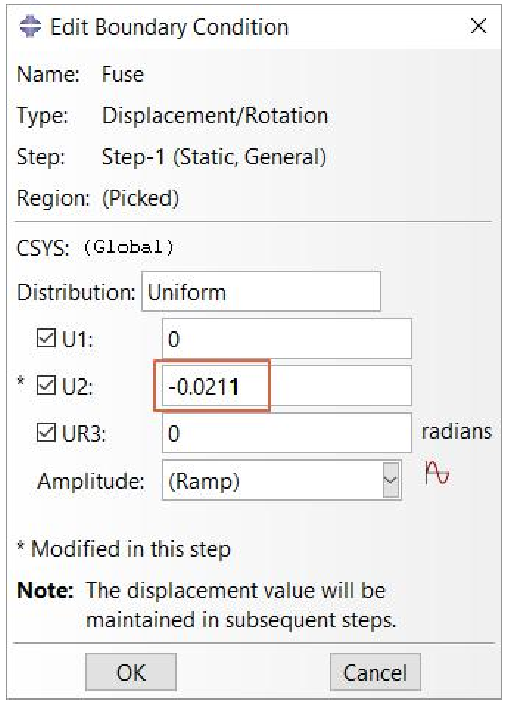
In the Boundary Conditions Manager it can be seen that three boundary conditions have already been created.

The boundary condition "Fuse" constrains the the motion of the Reference point attached to the rigid part. The rigid part will move in the negative y-direction in Step-1 and fit insider the holder. So we will modify the boundary condition accordingly.

Pick the boundary condition "Fuse" under column "Step-1" and pick **Edit**.

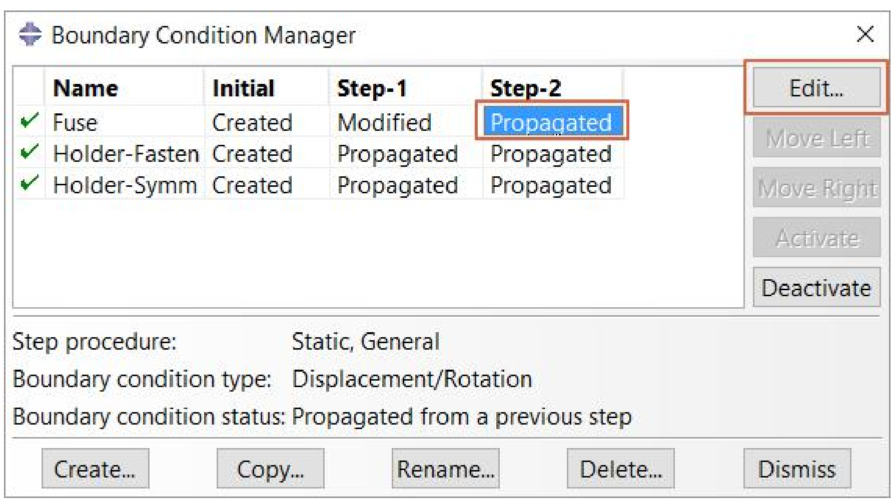


Enter **-0.0211** in the U2 field as shown below.

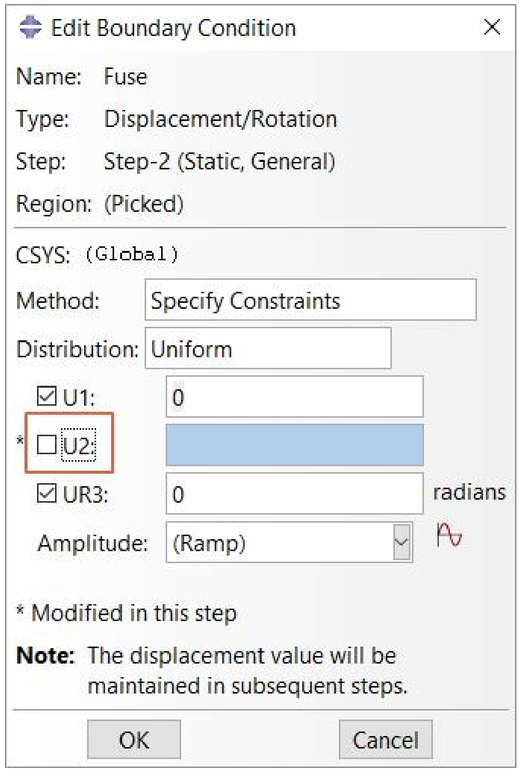


Pick **OK** to apply and exit.

Now pick the boundary condition "Fuse" under column "Step-2" and pick **Edit**.



Uncheck U2 field as shown below. It will allow the fuse to adjust its position freely in the y-direction.



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 open the Job Manager by picking .

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

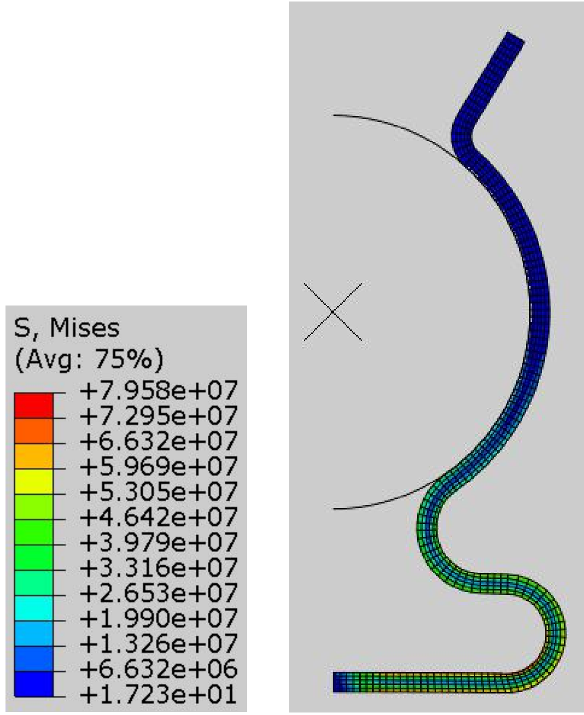
Pick **Continue** and then **OK**.

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

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

Postprocessing

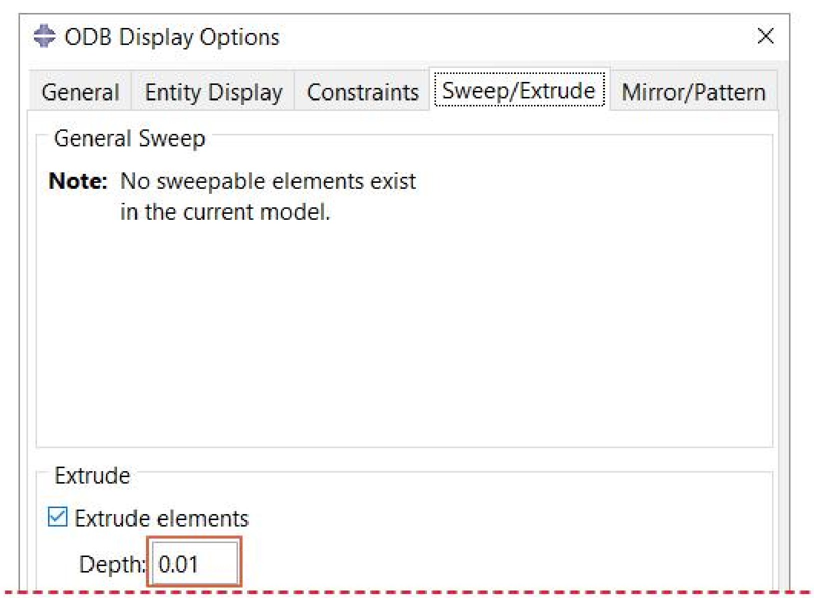
Pick  and contour plot of von Mises stress at the end of simulation will appear as shown in the figure below.



If we plot the contours of contact pressure on deformed shape, it would be very difficult to obtain any useful information. We can view the two-dimensional simulation results in three dimensions by extruding them to a specific depth. This gives a three-dimensional visual effect.

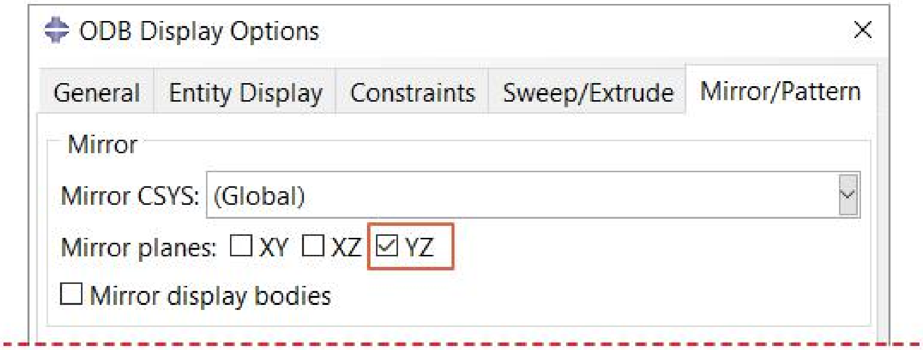
To extrude the current simulation results, pick **View > ODB Display Options** from the main menu bar while in the Visualization module.

Check the "Extrude elements" option located under the Sweep/Extrude tab and enter **0.01** in the Depth field.



We have simulated only half of the problem by taking advantage of the symmetry. Now we will also mirror the simulation results about symmetry plane so that complete assembly could be seen on screen.

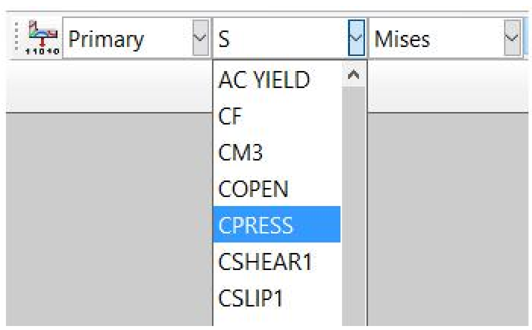
So check the **YZ** option located under the Mirror/Pattern tab.



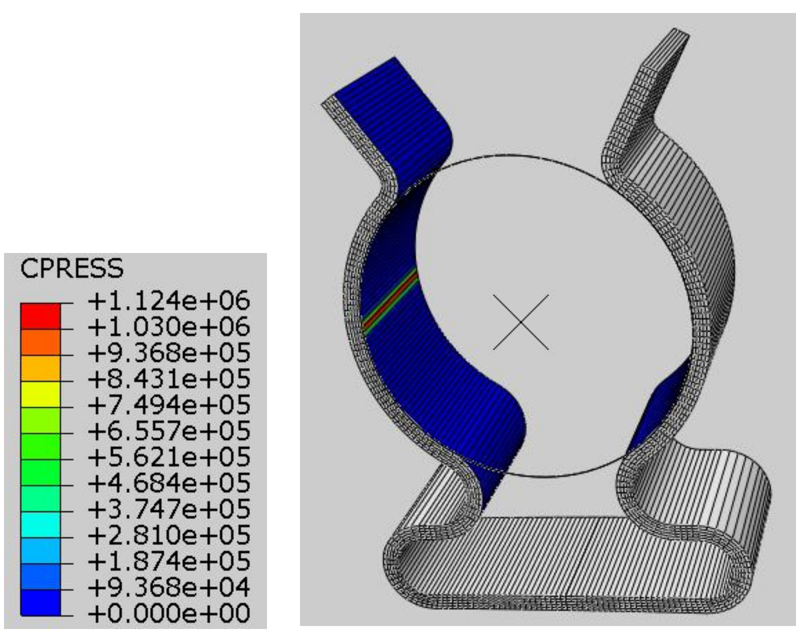
This will mirror the analysis results about the YZ datum plane of the model.

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

Now select **CPRESS** as output variable in the Field Output toolbar.

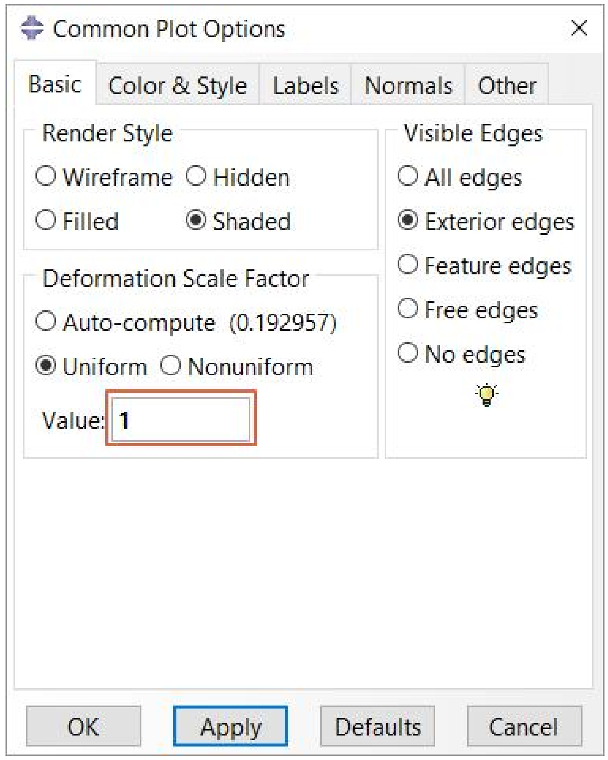


The simulation results will appear as shown below.



It can be seen that contact pressure develops only in a small region between the fuse and the holder.

Note: If you do not see the fuse moving according to the prescribed displacement, you might need to set the scale factor to 1 in in the Common Plot Options dialog box. This dialog box can be accessed by picking  while in the Visualization module.



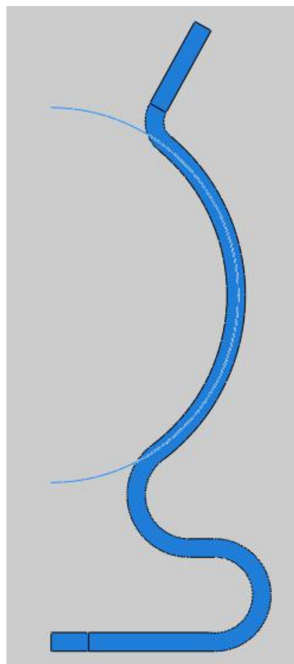
Select **File > Save** to save the changes we made so for.

Exercise 27

In this exercise we will determine the contact pressure distribution for the problem in previous exercise using a different technique.

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

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



In this assembly it can be seen that fuse is close to its final intended location. You can also notice the interference between the two parts. We will use the interference resolution capabilities of the contact interaction to solve this problem. All the steps, boundary conditions and contact interaction have already been defined. We will make the necessary changes in this exercise.

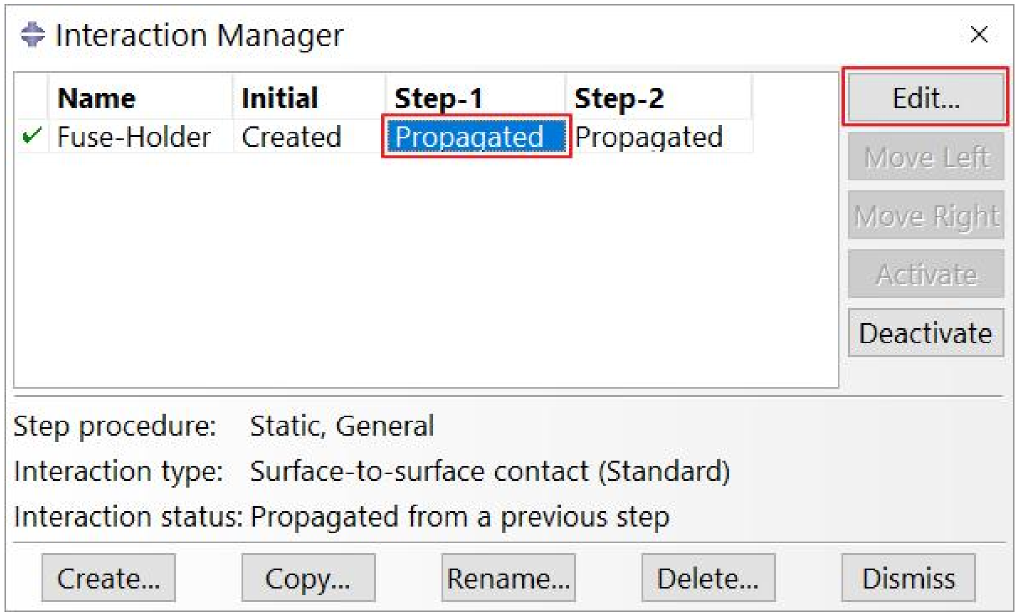
Modifying Contact Interaction

A contact pair interaction, named "Fuse-Holder", has already been created which defines contact between surfaces of the fuse and the holder. Now we will modify this contact interaction.

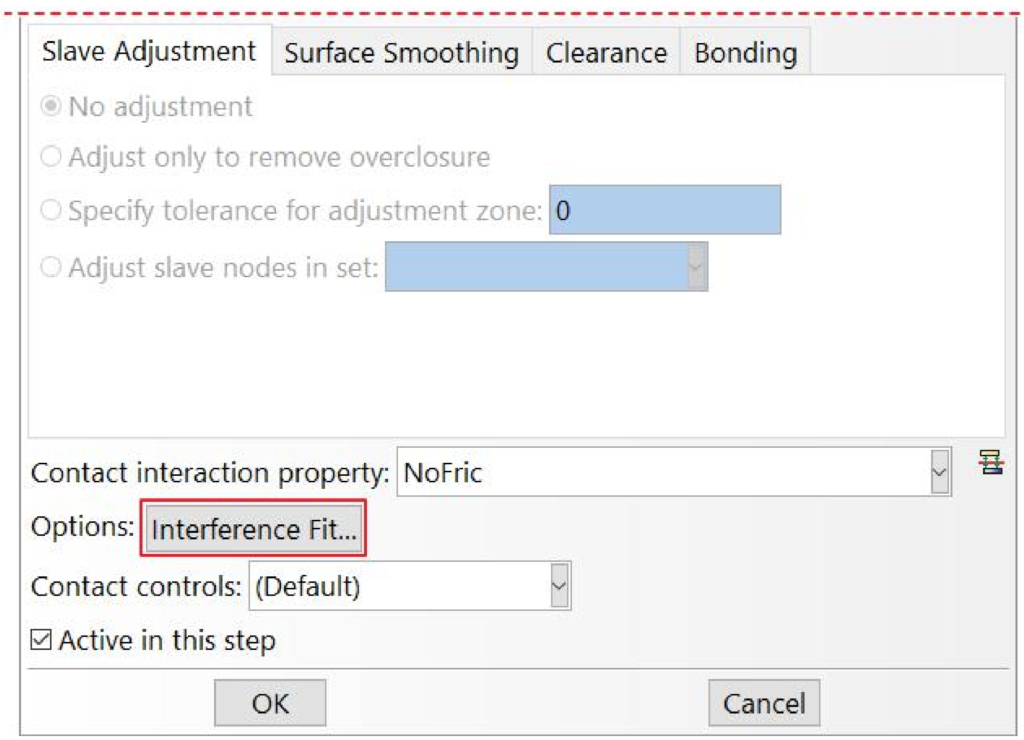
So change to **Interaction** module and open the Interaction Manager by picking .

We will modify the the already defined interaction in Step-1, such that it resolves the interference and establishes the contact.

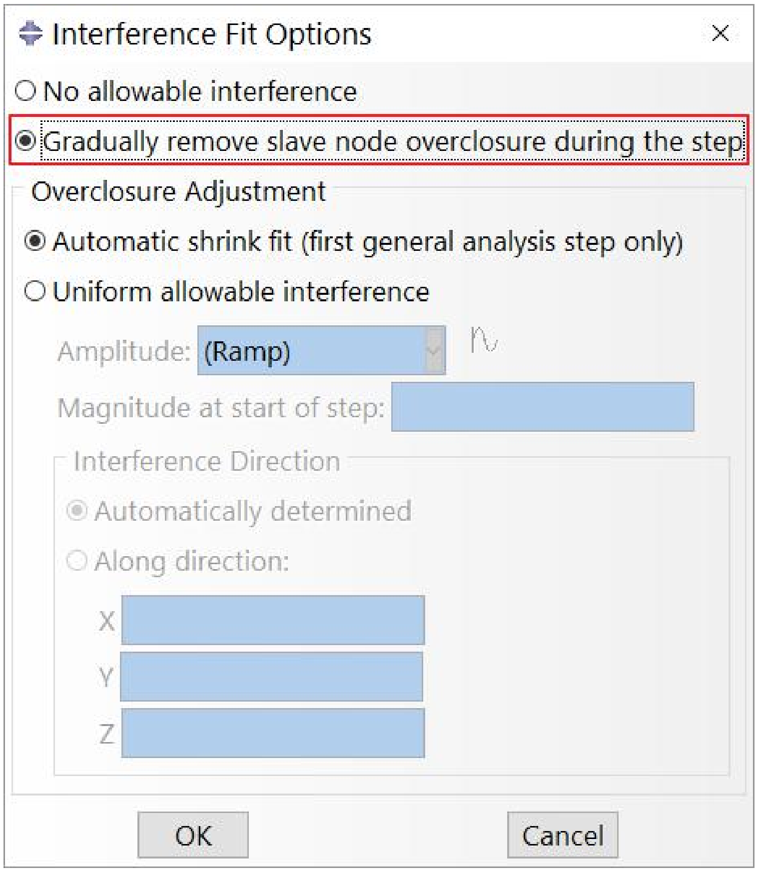
So pick the **Fuse-Holder** interaction under column "Step-1" and pick **Edit**.



Pick **Interference Fit** option in the Edit Interaction dialog box as shown below.



Pick **Gradually remove slave node overclosure during the step** in the Interference Fit Option dialog box.



The overclosure will be resolved over multiple increments. Contact algorithm pushes the surfaces apart until there is no penetration. As these overclosures are resolved, it results in stresses and strains in the model.

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

Pick **OK** to exit the Edit Interaction dialog box.

Pick **Dismiss** to close the Interaction Manager.

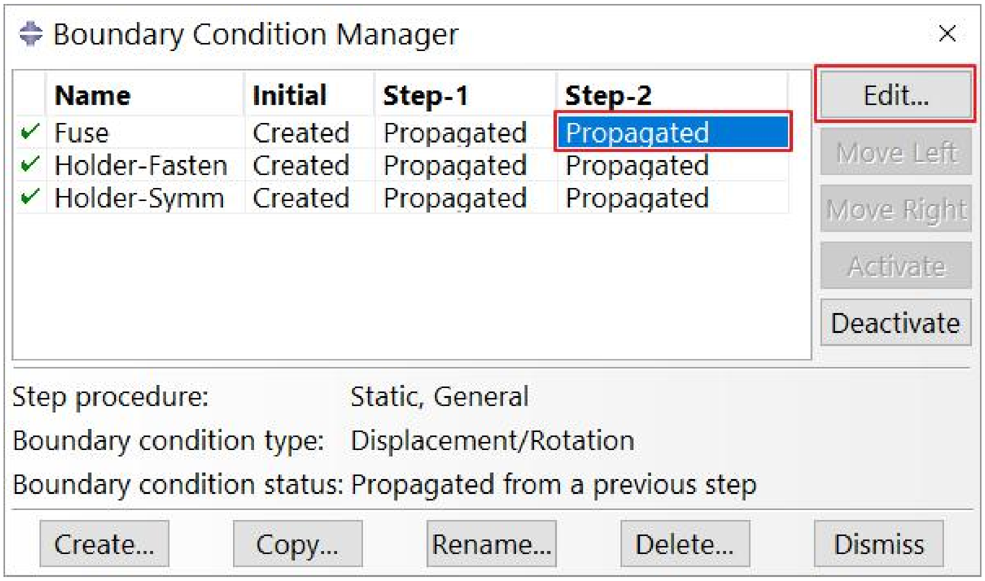
Boundary Conditions

Boundary conditions required for the analysis have already been defined. We need to modify only one of them.

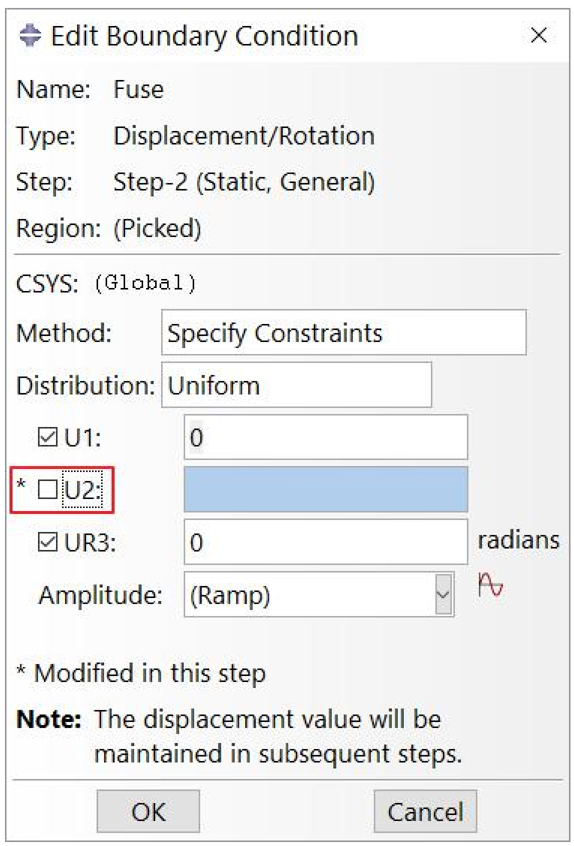
Change to **Load** module and open the Boundary Condition Manager by picking .

In the Boundary Conditions Manager it can be seen that three boundary conditions have already been created. The boundary condition “Fuse” constrains the the motion of the Reference point attached to the rigid part. We will modify it so that the rigid part is free to move in the y-direction during Step-2.

Pick the boundary condition “Fuse” under column “Step-2” and pick **Edit**.



Uncheck U2 field as shown below. It will allow the fuse to adjust its position freely in the y-direction.



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 open the Job Manager by picking .

Pick **Create** and create a job named Mounting_Int 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

For comparison, we will plot the contact pressure.

First we will mirror the simulation results about symmetry plane so that complete assembly could be seen on screen.

While in the Visualization module, pick **View > ODB Display Options** and check the YZ option located under the Mirror/Pattern tab.

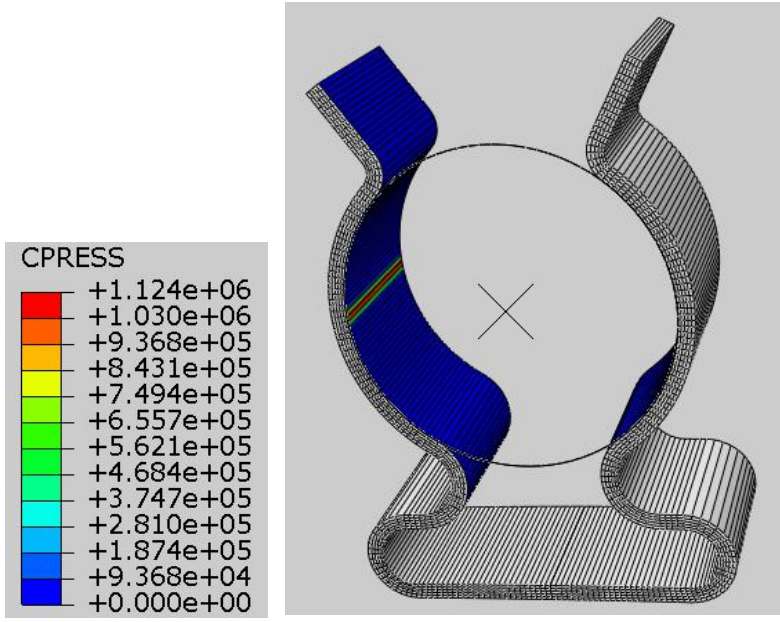
Now we will extrude the simulation results for better understanding.

Enter **0.01** in the Depth field located under Sweep/Extrude tab.

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

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

Now the simulation results will appear as shown below.



By comparing these results with those obtained in the previous exercise, it can be concluded that both techniques give the similar results.

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

Index