# Exercise 2

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.
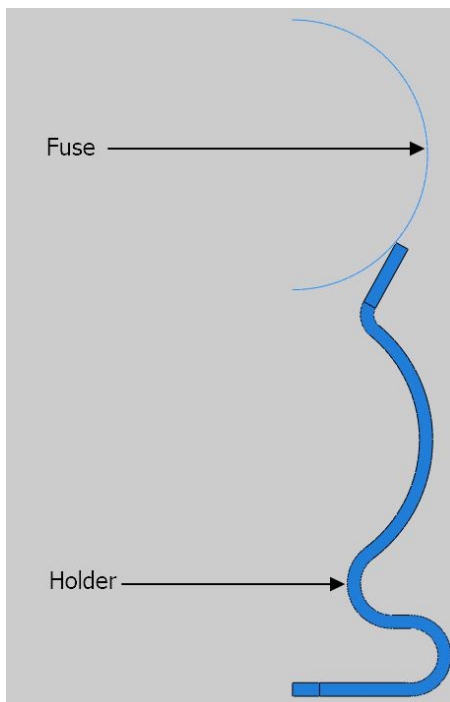
The model consists of two two-dimensional parts: fuse and holder. The fuse is modeled as an analytical rigid part. The holder is assumed to be made from brass with a Young's modulus of 100 GPa and a Poisson's ratio of 0.3. It is assumed that contact is frictionless. The holder is meshed with bilinear 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.
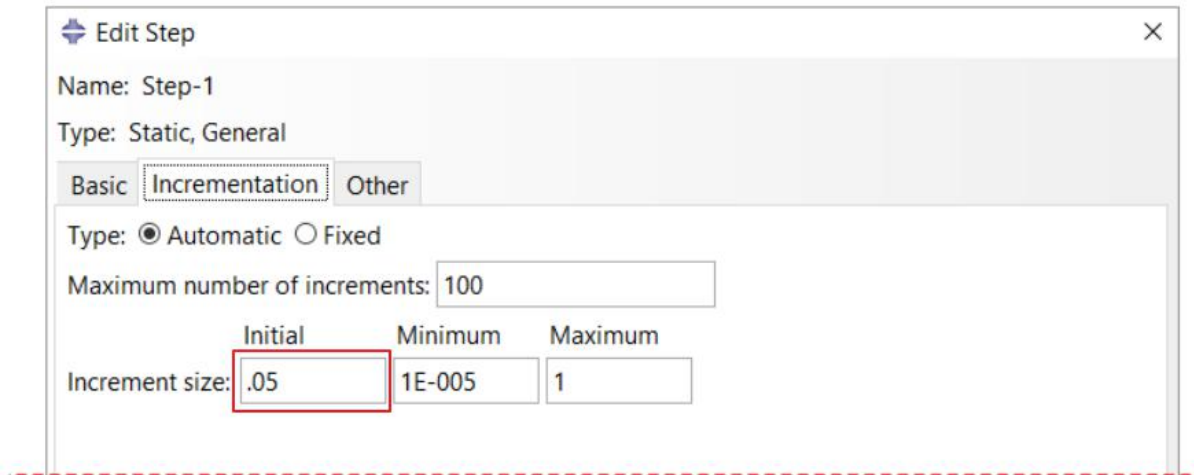
Change to **Step** module.

Open the Step Manager by picking [icon] . It will appear as shown below.
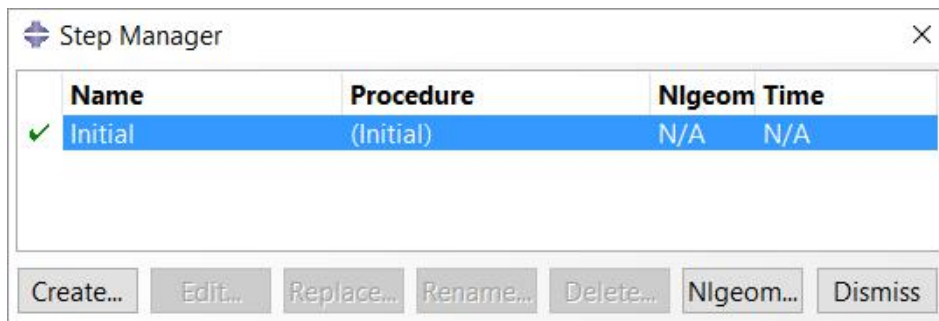
Pick **Create** to create a new step and select the **Static, General** step.

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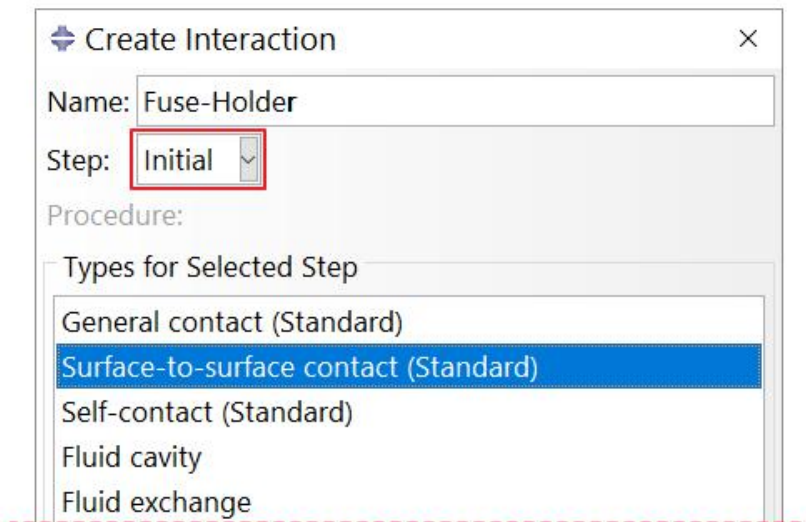
Change to **Interaction** module and open the Interaction Manager by picking .

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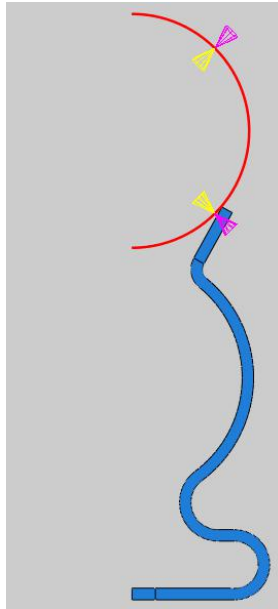
As a surface can contact on either side, so it is important to specify the desired side of the surface to be in contact. It is done by choosing the color associated with the desired side.

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.
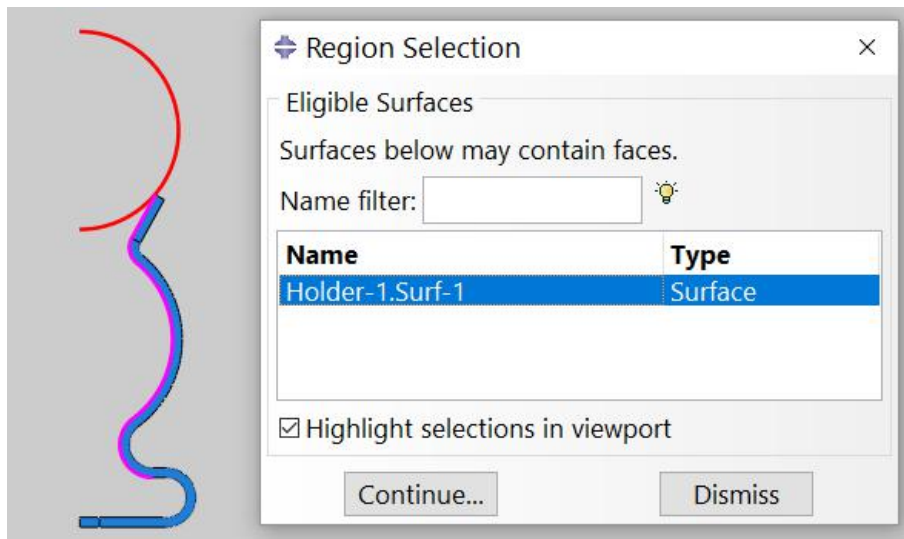

Pick **Continue** and Edit Interaction dialog box will appear.

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.
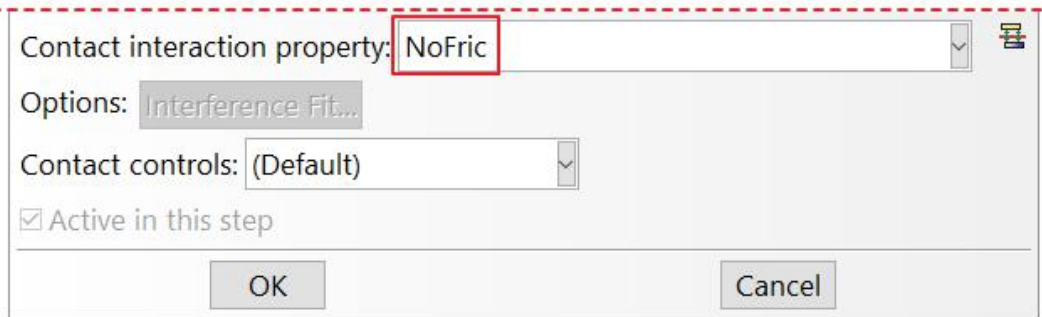
In this problem as there is considerable sliding between the fuse and its holder so we will use Finite sliding option which is active by default.
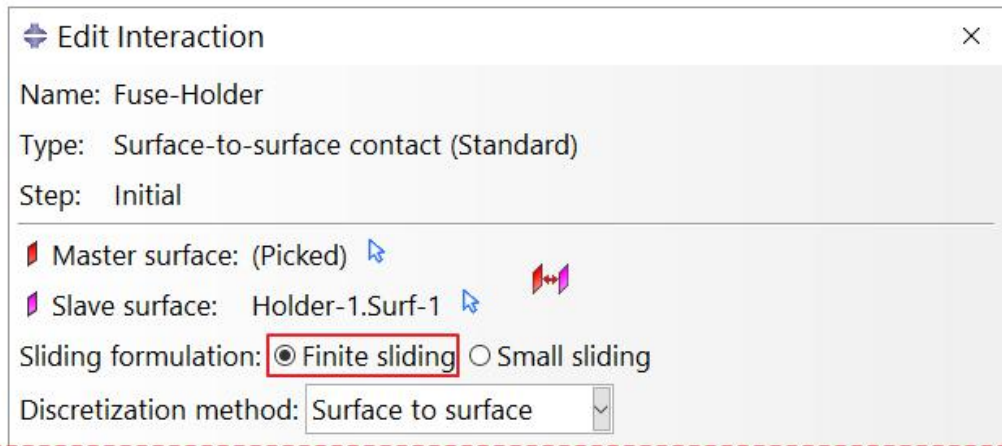
Pick **OK** and it completes the definition of interaction. It can be seen in the Interaction Manager that newly created interaction has been propagated through all the steps.
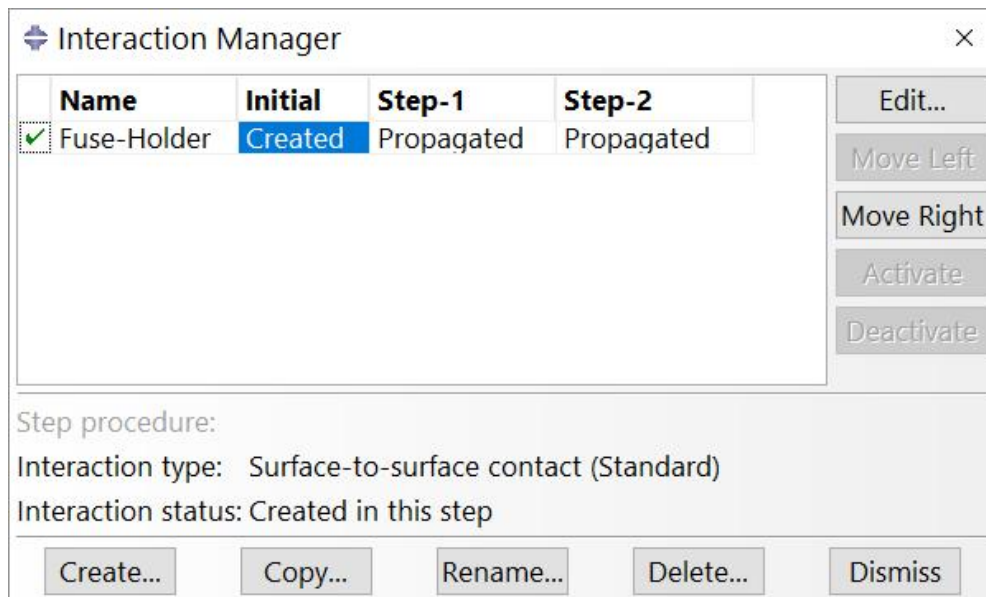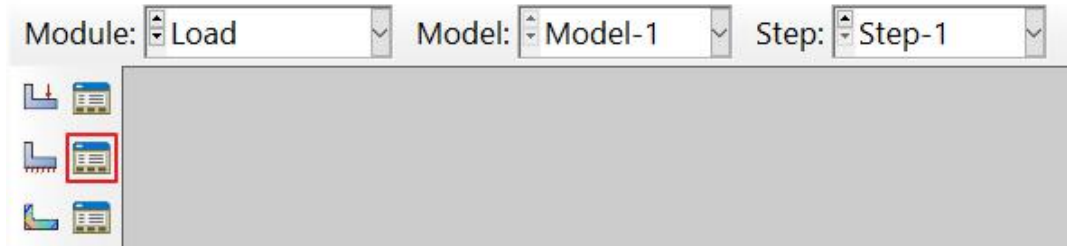

Pick **Dismiss** to close the manager.

⇨ 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**.
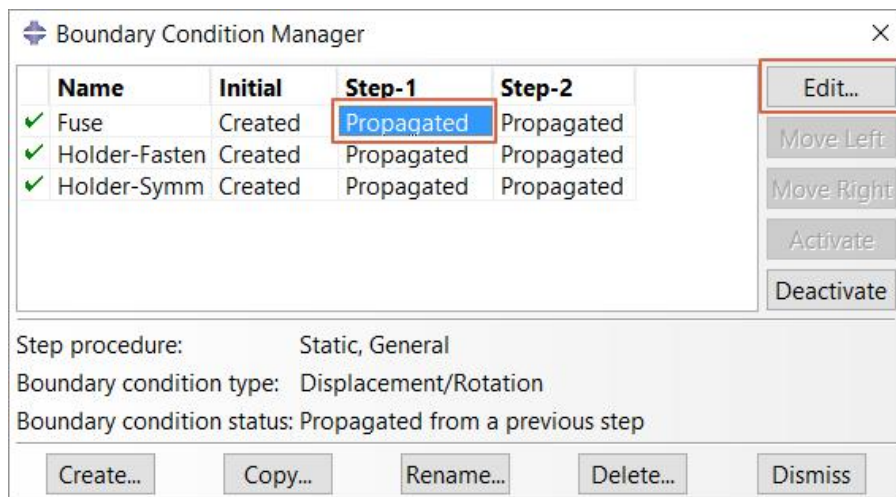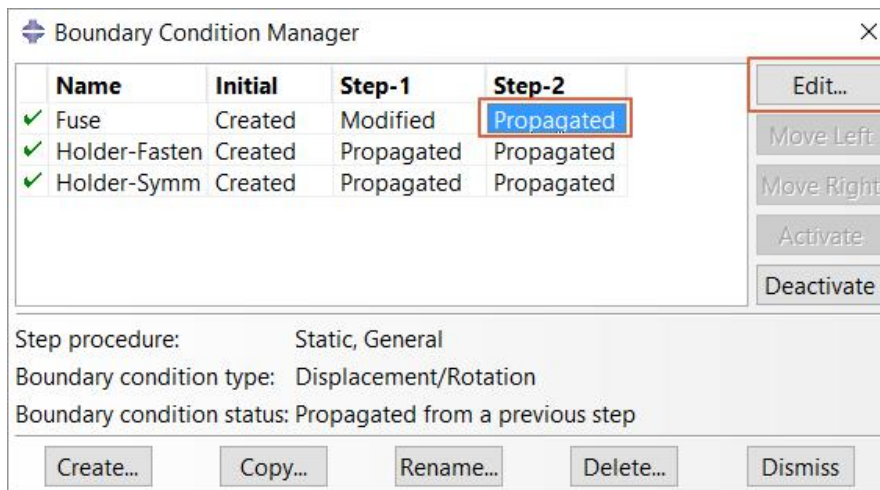

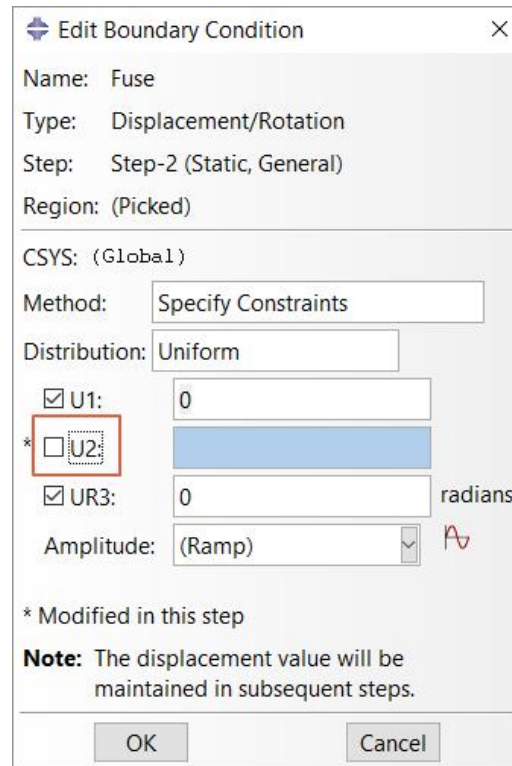Enter **-0.0211** in the U2 field as shown below.

www.caeuniversity.com
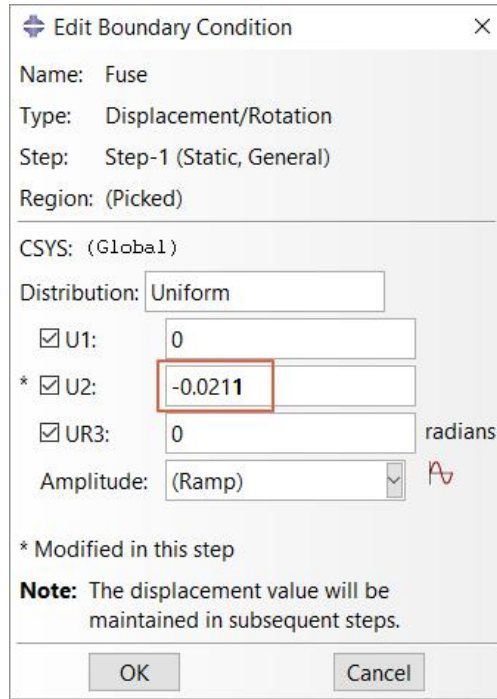
Pick **OK** to apply and exit.

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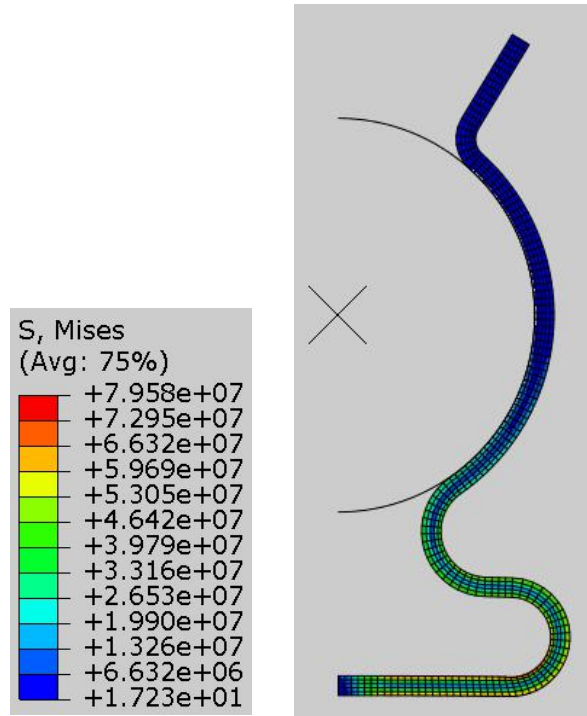
Pick **Continue** and then **OK**.

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.

8

S, Mises
(Avg: 75%)
+7.958e+07
+7.295e+07
+6.632e+07
+5.969e+07
+5.305e+07
+4.642e+07
+3.979e+07
+3.316e+07
+2.653e+07
+1.990e+07
+1.326e+07
+6.632e+06
+1.723e+01

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.
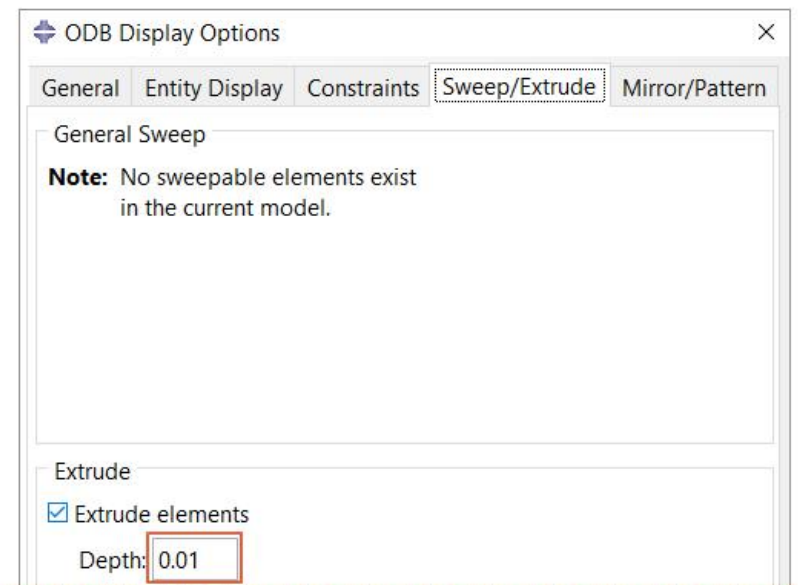
We have simulated only half of the problem by taking advantage of the symmetry. Now we will also mirror the simulation results about symmetry plane so that complete assembly could be seen on screen.
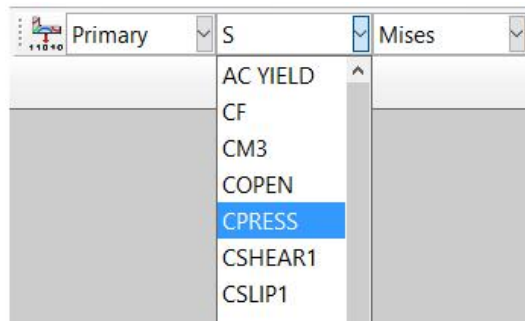
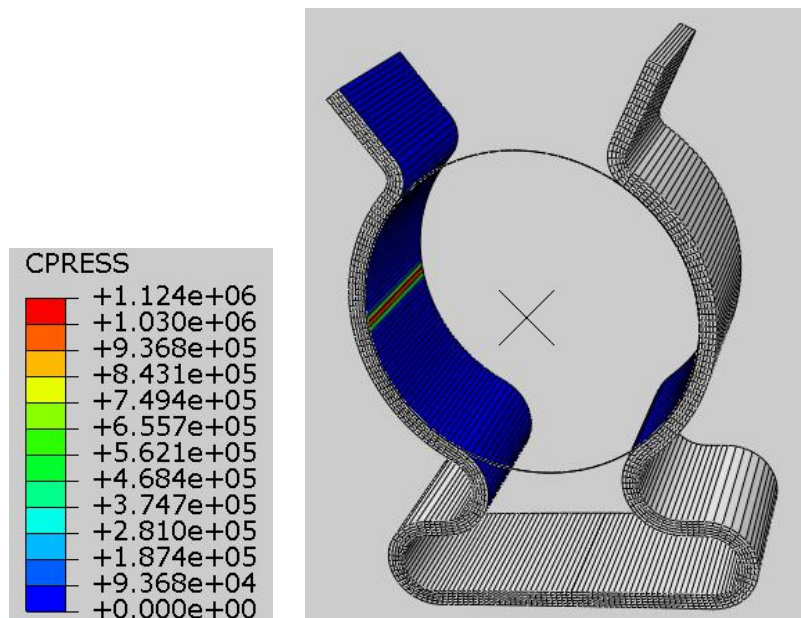So check the **YZ** option located under the Mirror/Pattern tab.
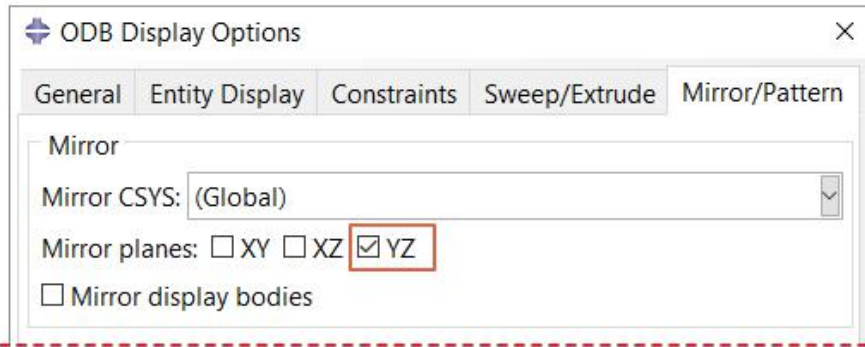
This will mirror the analysis results about the YZ datum plane of the model.

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

10

the scale factor to 1 in in the Common Plot Options dialog box. This dialog box can be accessed by picking ![123] while in the Visualization module.
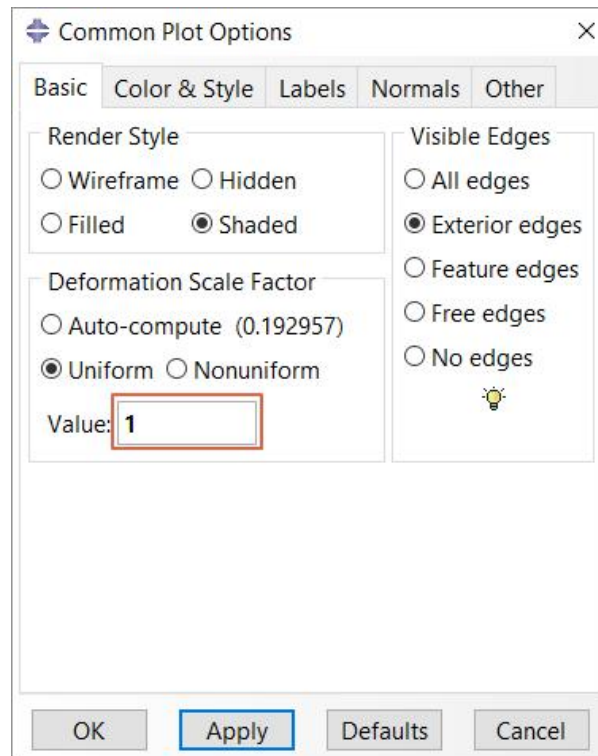
Select **File > Save** to save the changes we made so for.