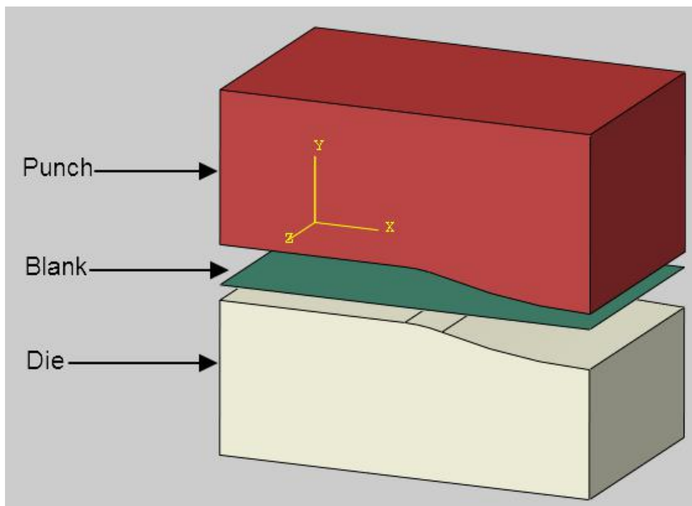
Exercise 21

In this exercise we will use general contact approach to define contact interaction and perform the stress analysis of assembly. We will also compare the results with contact pair approach.

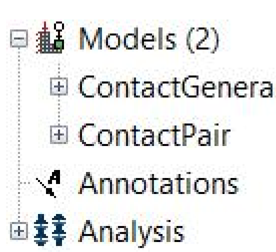
Pick **File > Set Work Directory** and set the work directory to Sandwich folder

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



The assembly consists of three three-dimensional, deformable parts: punch, die and blank. The punch presses the blank against the die. The punch and die are assumed to be made from steel with a Young's modulus of 200 GPa and a Poisson's ratio of 0.3. The blank is assumed to be made from aluminum with a Young's modulus of 70 GPa and a Poisson's ratio of 0.3. The blank is meshed with linear, fully integrated, quadrilateral shell elements (S4). The punch and die are meshed with 8-node, fully integrated, brick elements (C3D8).

In the model tree, it can be seen that there are two models. In the "ContactPair" model, contact has been defined by creating contact pair interactions. We will define a general contact interaction in the "ContactGeneral" model in this exercise.



Analysis Steps

The analysis will be performed in one step. The step has already been defined with a total time period set to **1.0** and the initial time increment to **0.01**.

Load and Boundary Conditions

Boundary conditions required for the analysis have already been defined. The punch is moved by a magnitude of 0.0016 in the y-direction with the "Punch" boundary condition. By taking advantage of symmetry, only a small portion of the assembly is modeled. Symmetry boundary conditions have been defined accordingly.

Defining Contact Interactions

Now we will create a general contact interaction to define contact for the entire model..

First Change to **Interaction** module and make sure that ContactGeneral model is active.

Open the Interaction Manager by picking .

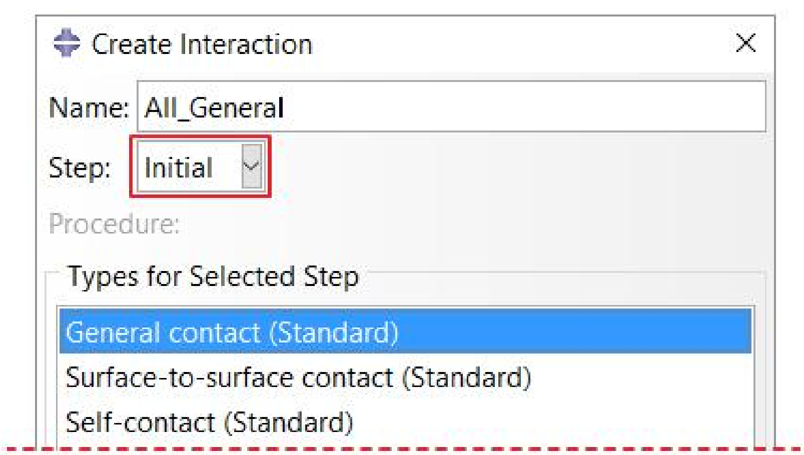
Pick **Create** to define a new interaction.

Enter **All_General** as the name of the interaction.

Pick **Initial** in the Step field.

In Abaqus/Standard general contact can be defined only in the initial step.

Select **General contact** and pick **Continue** to proceed.

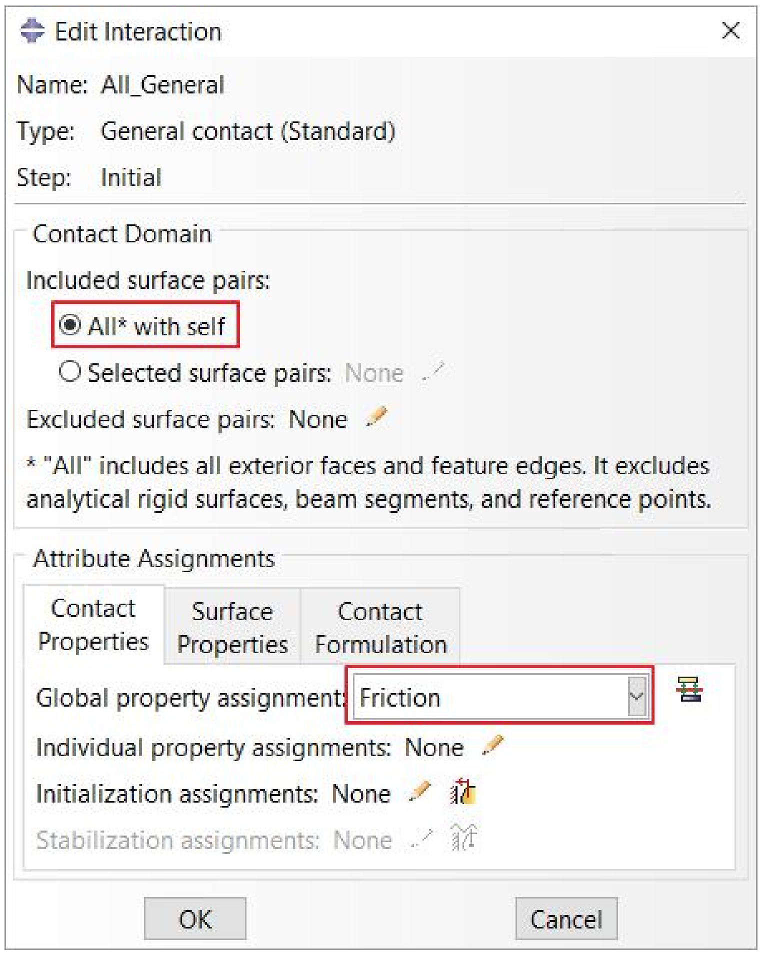


In the Edit Interaction dialog box, pick **Friction** as the global contact interaction property.

This contact property is assigned globally to the general contact interaction. It is also possible to assign a contact property individually to particular regions within a general contact domain.

Notice that **All* with self** is selected by default. This is the simplest way to define the contact domain.

When "All* with self" option is used, Abaqus/Standard generates a surface that contains all exterior element faces (with some exceptions) in the model and solves a self-contact problem for this all-inclusive surface. The surface can span many disconnected regions/bodies in the model. Self-contact for the surface that spans multiple bodies implies self-contact for each body as well as contact between the bodies. It is not required to select any region/surface for a general contact interaction when "All* with self" option is used.



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

Minimum initial gap between the blank and other parts is equal to the half of shell thickness. The general contact interaction will account for the initial thickness of the blank by default.

General contact automatically accounts for thicknesses and offsets associated with shell surfaces.

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 SandwichGeneral or any other suitable name.

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

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

Notice that job completes successfully.

An analysis job, named SandwichPair, has already been created for the ContactPair model. Also submit this job so that a comparison could be made.

Postprocessing

Now we will compare the simulation results obtained by using contact pair approach with general contact approach.

Select SandwichGeneral job in the manager and pick **Results**. Similarly open the SandwichPair job in the visualization module.

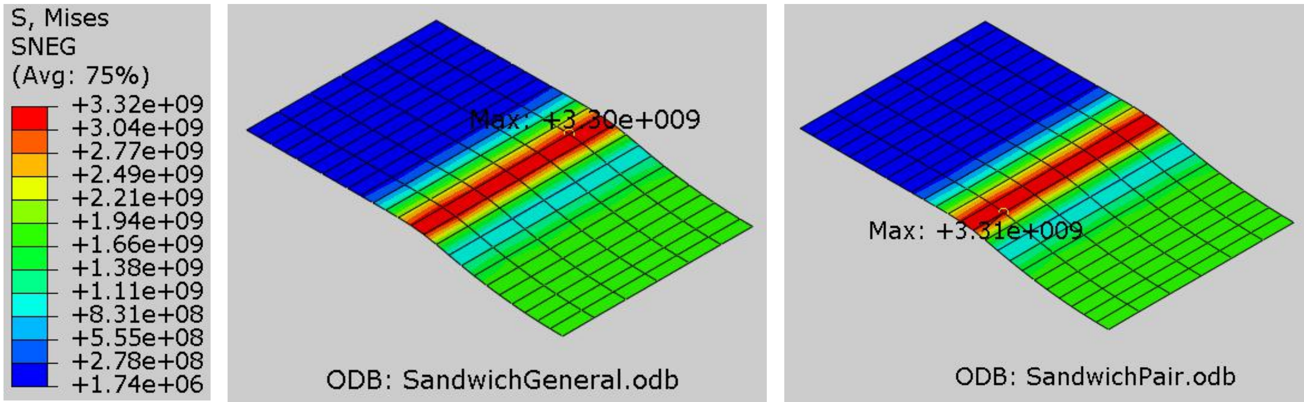
First we will compare the stress distribution in the blank. So we will remove other parts from display.

So pick  and select **Part instances** as the entities to remove.

Now select the blank in the viewport and pick **Done**.

Pick  to plot the contours on deformed shape.

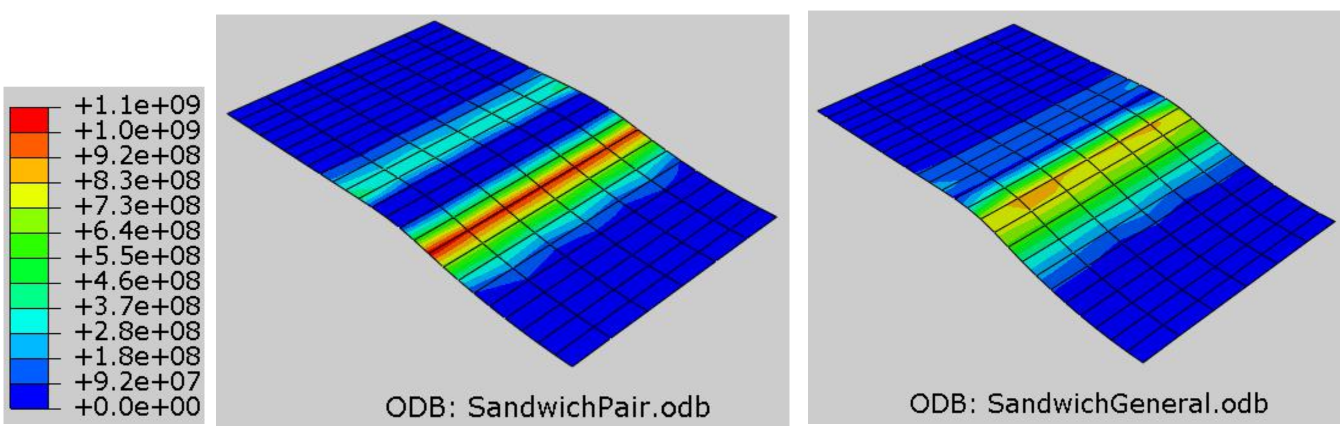
The following figure compares the distribution of von Mises stresses on the SNEG face of the shell for both cases. (Note: Legend is same for both plots).



It can be seen that stress distribution is almost similar for both cases.

Now we will plot the contours of contact pressure.

Select **CPRESS** as output variable for SandwichGeneral and **CPRESS SNEG** as output variable in the Field Output toolbar for SandwichPair. The contact pressure distribution will appear as shown in the figure below.



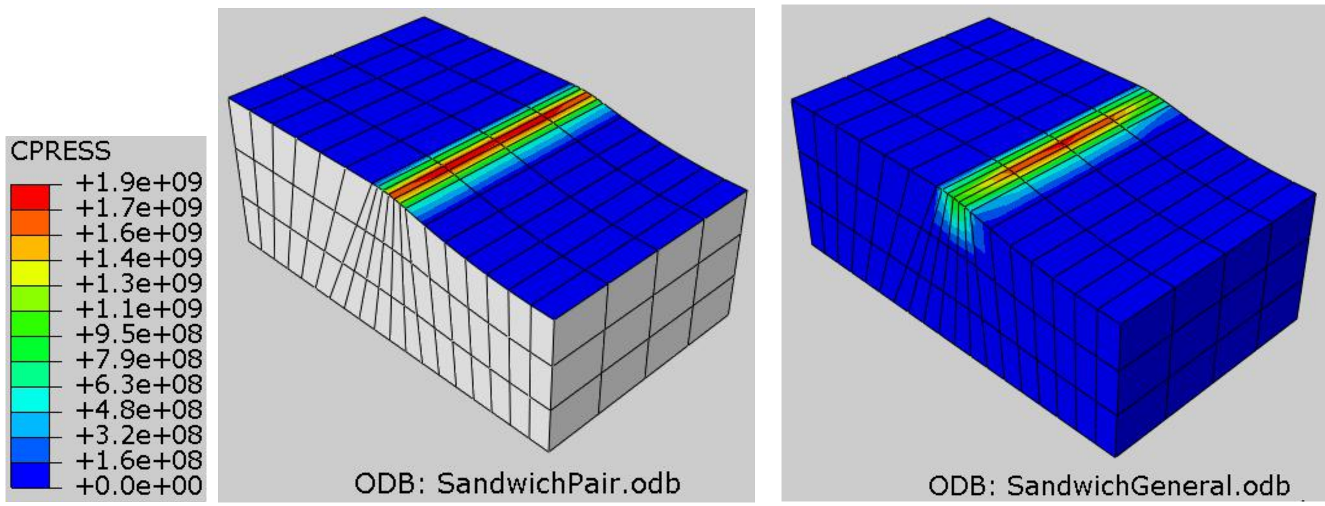
When general contact approach is used, the convention for reporting contact pressure is different from the convention used for contact pairs. If contact output is requested for shell elements modeled with general contact, the data for the SPOS and SNEG surfaces is not reported in separate data sets. Therefore the comparison is not very helpful.

Now we will compare the contact pressure distribution on the die surface.

Pick  to display all the parts on screen.

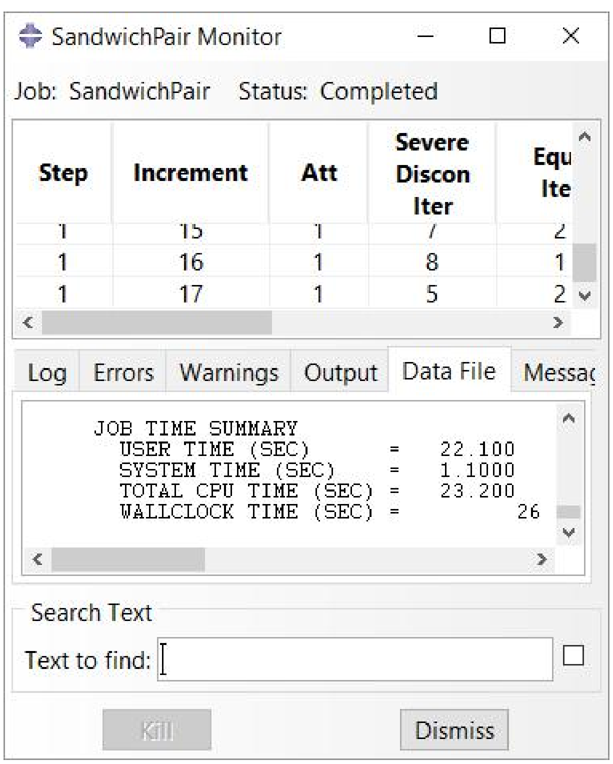
Pick  and select the die in the viewport and pick **Done**.

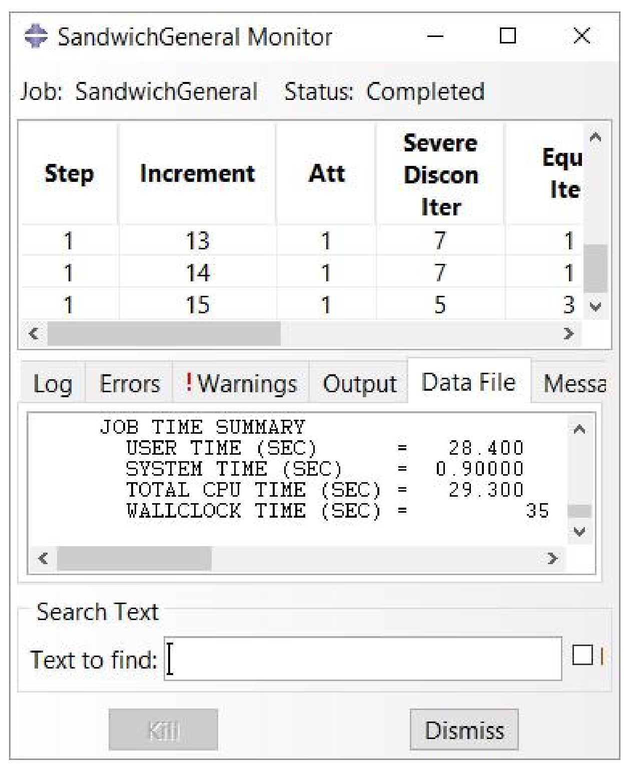
Select **CPRESS** as output variable for both analysis jobs. The contact pressure distribution will appear as shown in the figure below.



It can be seen that contact pressure distribution is different for both cases. It can also be observed that the contact pressure extends beyond the actual contact region when contact is modeled with general contact approach. It is because the contour plots are constructed by interpolating nodal values, which can cause nonzero values to appear outside of the contact region.

In the Job Monitor it can be seen on the **Data File** tab that it takes 26 seconds (wallclock time) to complete the job for the model with contact pairs and 35 seconds for the model with general contact as shown below.





Contact pairs often result in more efficient analyses as compared to general contact approach. It is because Abaqus/Standard generates a surface that contains all exterior element faces in the model (when "All* with self" option is used) and solves a self-contact problem for this all-inclusive surface as compared to contact pair approach where typically only those surfaces are selected where contact is anticipated. Inclusion of surfaces that never come into contact takes additional computation resources and hence general contact is often less computational efficient.

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

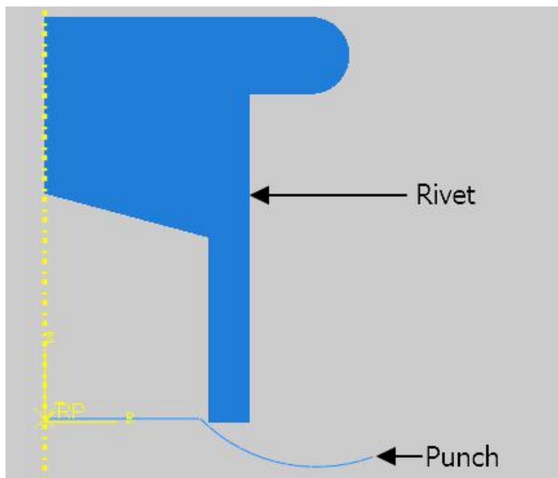
Exercise 22

In this exercise we will simulate the upsetting of a tubular rivet using general contact approach to define contact interaction. We will also compare the results with contact pair approach.

Tubular rivets are similar to solid rivets, except they have a partial hole at the tail end. The purpose of this hole is to reduce the amount of force required to upset the rivet.

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

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



The assembly consists of two axisymmetric parts: rivet and punch. The punch is modeled as discrete rigid part. The staple is assumed to be made from an alloy with a Young's modulus of 200 GPa and a Poisson's ratio of 0.3. A coefficient of friction of 0.05 is assumed between contacting surfaces. The rivet is meshed with bilinear axisymmetric quadrilateral elements with reduced integration (CAX4R).

In the model tree, it can be seen that there are two models. In the "ContactPairs" model, contact has been defined by creating contact pair interaction. In the "ContactGeneral" model, contact will be defined by using general contact interaction.

Analysis Steps

The analysis will be performed in one step. The step has already been defined with a total time period set to **1.0** and the initial time increment to **0.01**. Due to large relative sliding of surfaces and friction, it is expected that the magnitude and influence of unsymmetric terms would be significant. So the unsymmetric solver has been specified for this step. As the rivet is upset by the punch, large deformations take place. To take into account the large deformations, NLgeom option has been toggled on.

Boundary Conditions

Boundary conditions required for the analysis have already been defined. The punch is moved by a magnitude of 0.004 in the y-direction with the “Punch” boundary condition.

Defining Contact Interaction

Now we will create a general contact interaction to define contact for the entire model..

First Change to **Interaction** module and make sure that ContactGeneral model is active.

Open the Interaction Manager by picking .

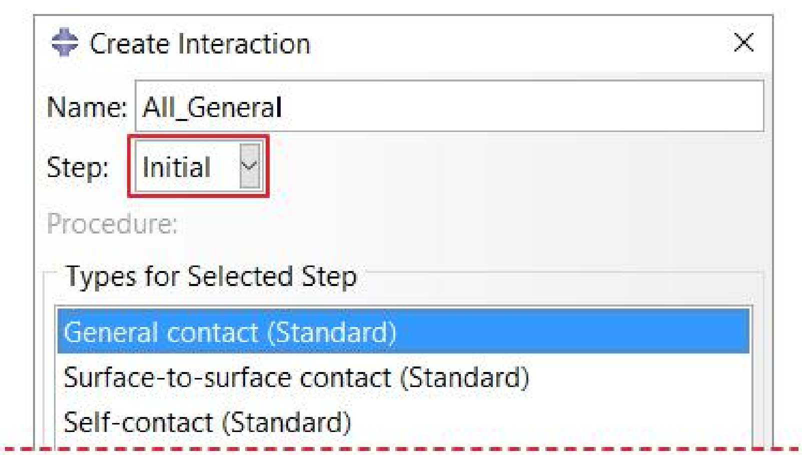
Pick **Create** to define a new interaction.

Enter **All_General** as the name of the interaction.

Pick **Initial** in the Step field.

In Abaqus/Standard general contact can be defined only in the initial step.

Select **General contact** and pick **Continue** to proceed.



In the Edit Interaction dialog box, pick **Friction** as the global contact interaction property.

Notice that **All* with self** is selected by default. This is the simplest way to define the contact domain.

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

Pick **Dismiss** to close the manager.

Field Output Requests

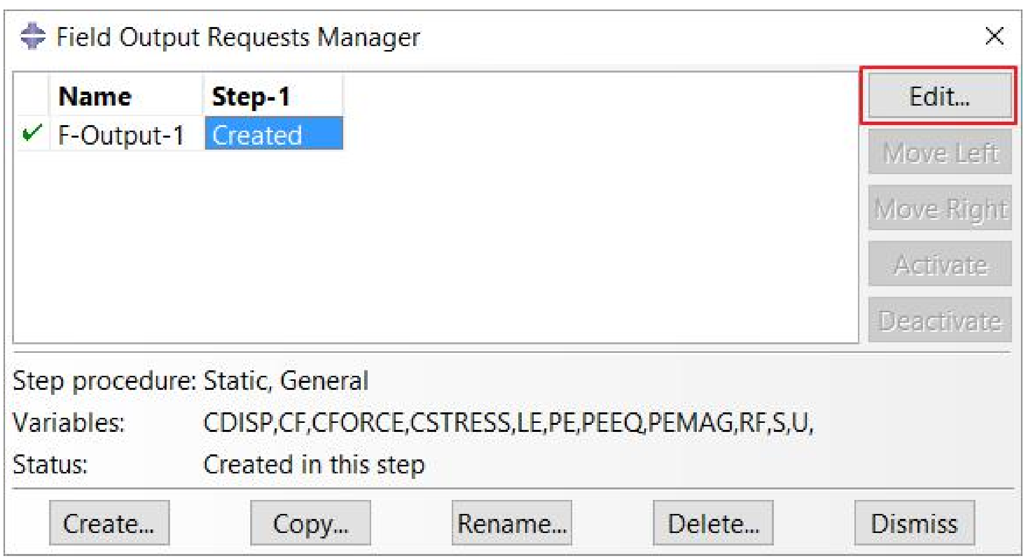
We will plot contact normal force (CNORMF) for comparison purposes. This information is not included by default in the ODB file. We will modify the existing field and request CFORCE variable.

If CFORCE is requested, the variables CNORMF (normal contact force) and CSHEARF (shear contact force) become available in the output database.

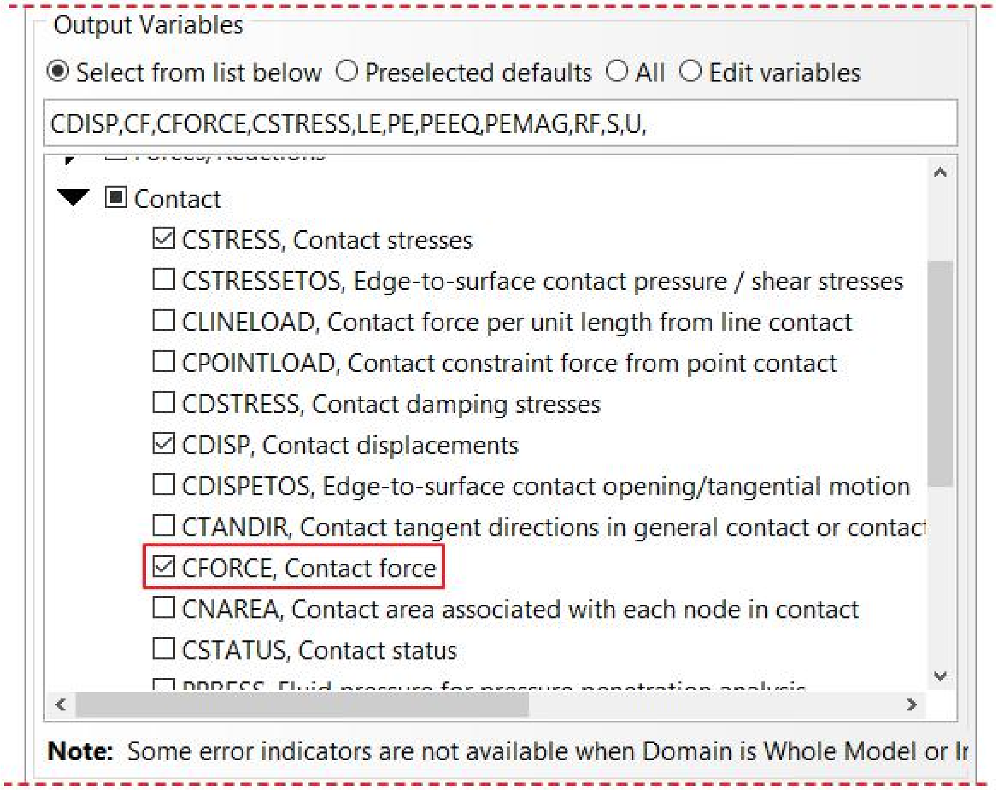
Change to **Step** module.

Open the Field Output Manager by picking  .

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



Check the **CFORCE** variable (located under the Contact container).



Pick **OK** to apply and exit.

Pick **Dismiss** to close the manager.

Job Submission

All the information required for analysis for the ContactPairs model has already been set up. We will submit an analysis job for the model.

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

Pick **Create** and create a job named UpsetPairs (or any other suitable name) for the ContactPairs model .

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

Pick **Submit** to submit the job for analysis and notice that job completes successfully.

All the information required for analysis of the ContactGeneral model has now been set up. So we can submit a job for analysis.

Pick **Create** and create a job named UpsetGeneral (or any other suitable name) for the ContactGeneral model.

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

Pick **Submit** to submit the job for analysis and notice that job is aborted due to errors.

Diagnosing the error

Both the ContactPairs and the ContactGeneral models have the same parameters set for analysis except the different approaches to define contact. We know that when general contact approach is used, Abaqus/Standard automatically generates a surface and assigns master and slave roles. So we will investigate how general contact approach is handling the surface definition and master-slave roles.

We will try to find the cause of the errors in UpsetGeneral job in the Visualization module.

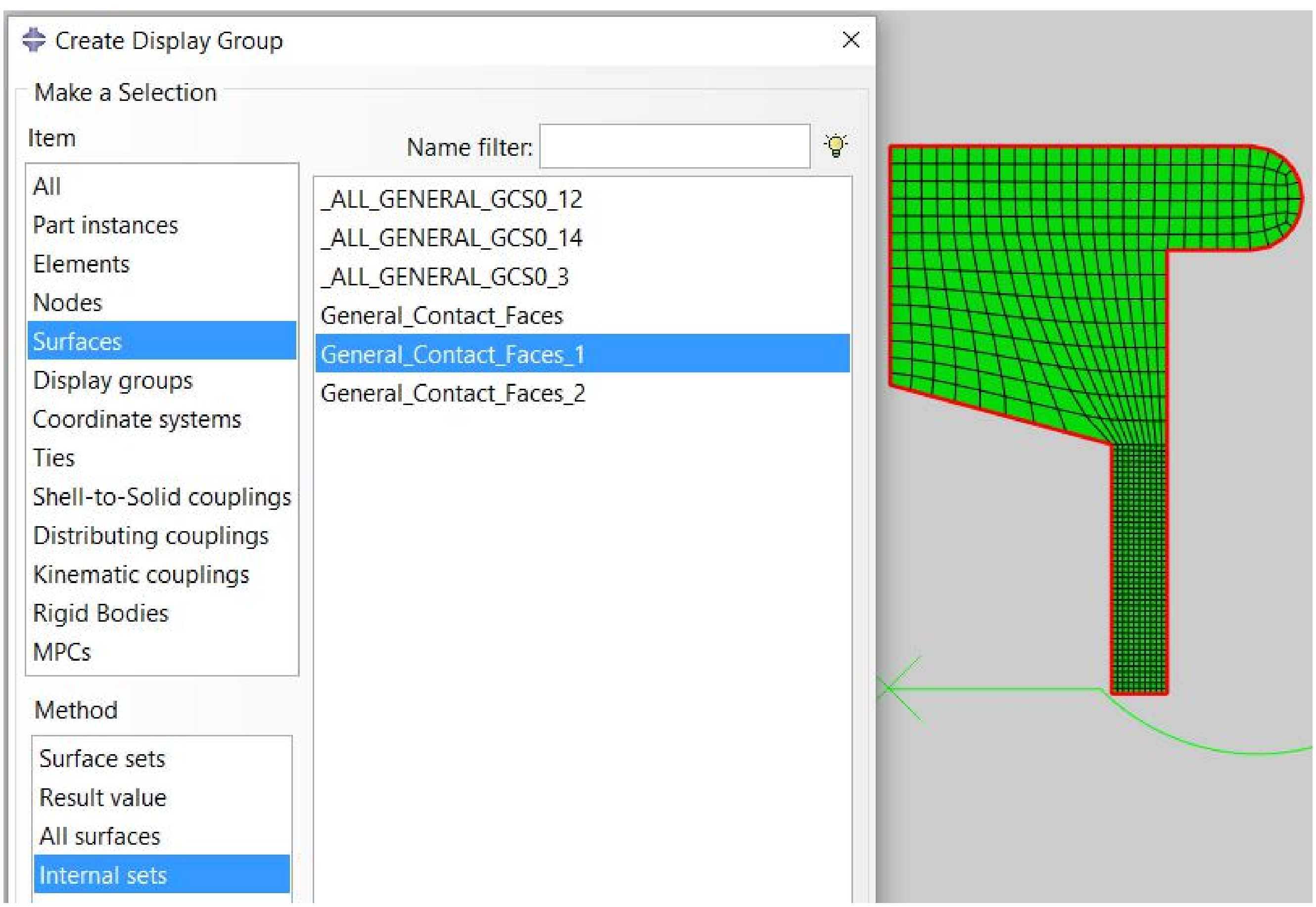
So pick **Results** to view the results in the Visualization module.

Pick to open the Create Display Group dialog box. (alternatively pick Tools > Display Group > Create).

Pick **Surface** as the Item and **Internal sets** as the Method.

Check **Highlight items in viewport**

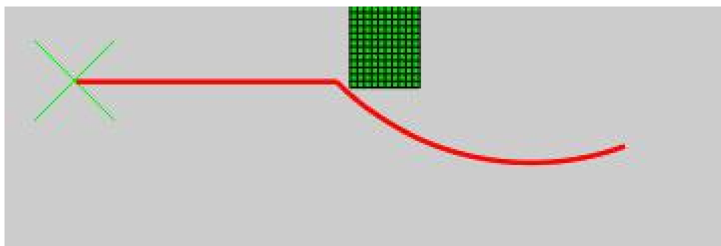
Pick General_Contact_Faces_1 and system will highlight the automatically generated internal surface as shown below.



Abaqus automatically generates internal surfaces using the naming convention *General_Contact_Faces_k*, where *k* corresponds to an automatically assigned component number. By default, the lowered-number component surfaces will act as master surfaces to the higher-numbered component surfaces. Abaqus/Standard assigns default pure master-slave roles for contact involving disconnected bodies within the general contact domain.

As there are two disconnected bodies in the general contact domain, two internal component-surfaces, i.e. General_Contact_Faces_1 and General_Contact_Faces_2, are generated. The internal surface without a component number, i.e. General_Contact_Faces, contains all surface faces included in the general contact domain. The General_Contact_Faces_1 will act as master surface to the General_Contact_Faces_2.

Next pick General_Contact_Faces_2 and system will highlight the automatically generated internal surface as shown below.

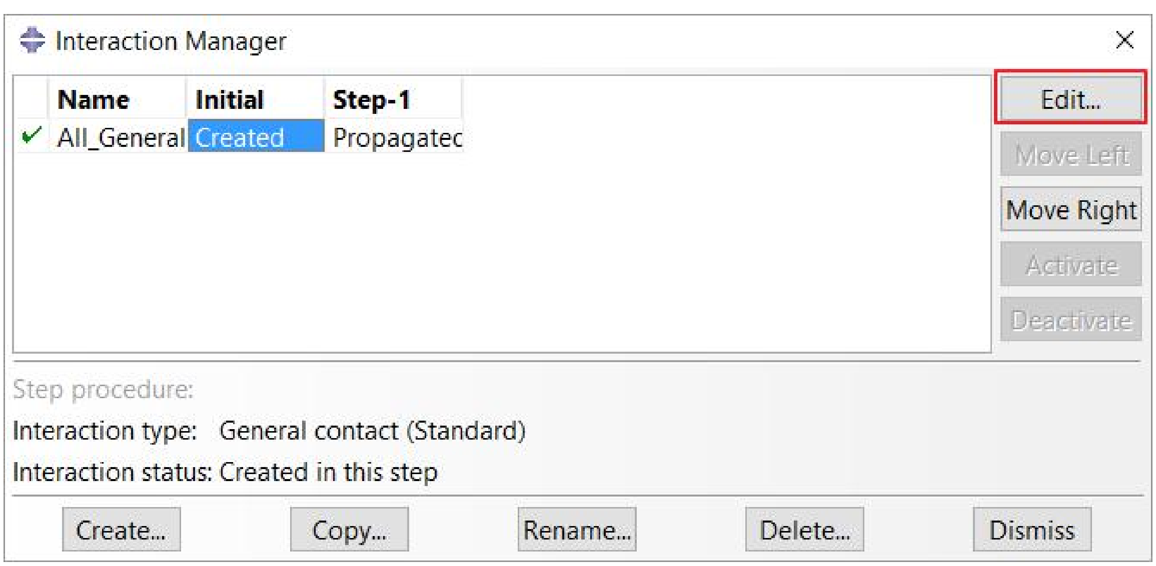


We can conclude that the internal surface containing all exterior element faces of the rivet is acting as master surface. This is in contrast to the ContactPairs model, where a surface defined on the exterior faces of the rivet acts as slave surface. So we will modify the general contact interaction to override the default master-slave assignment. (Note: Although there are other ways to fix the error, our aim is to make the changes in a way so that the solution is comparable to the solution obtained with contact pair approach.)

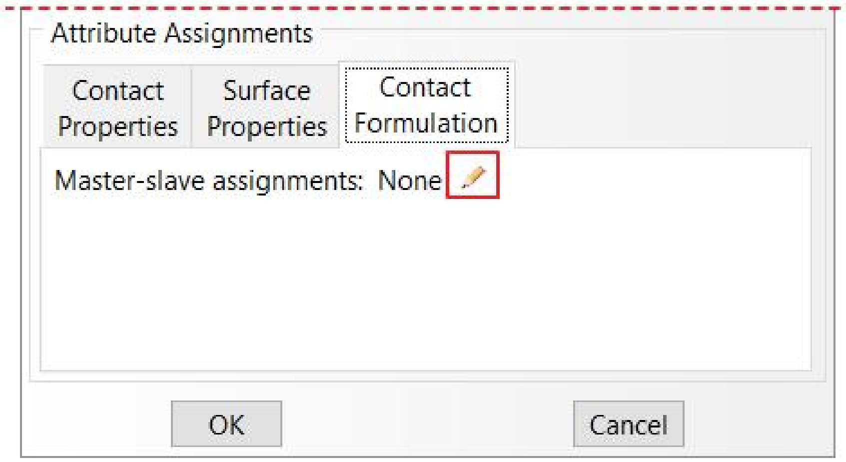
Modifying Contact Interaction

Open the Interaction Manager by picking .

Pick the "All_General" interaction under column "Initial" and pick **Edit**.

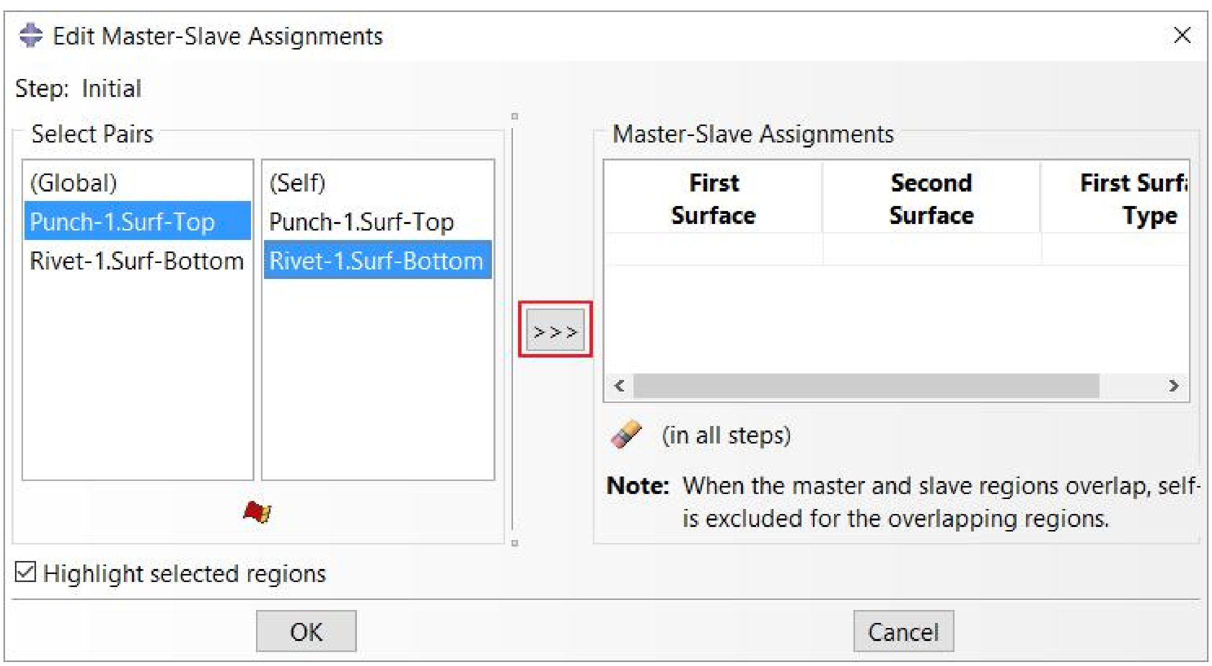


Pick under the Contact Formulation tab to specify master-slave assignments.



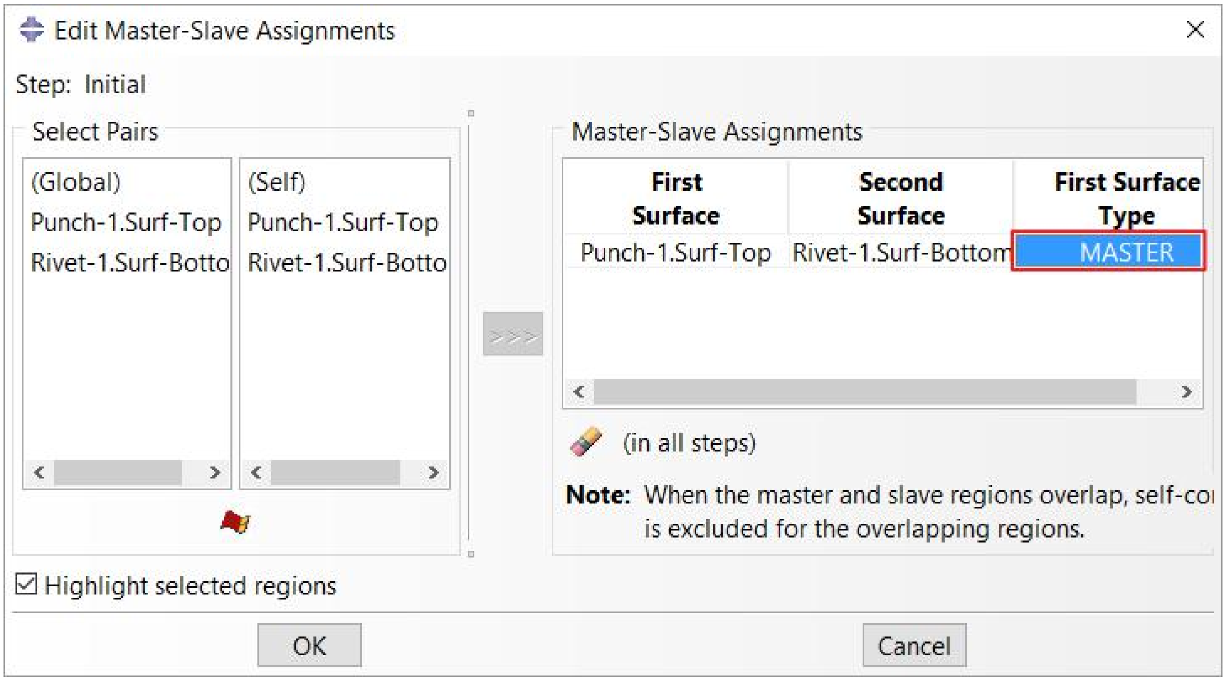
The surfaces required to assign master-slave roles have already been defined and therefore appear in the Edit Master-Slave Assignments dialog box.

Pick **Punch-1.Surf-Top** and **Rivet-1.Surf-Bottom** in the first and second columns respectively to define the surface pairings.



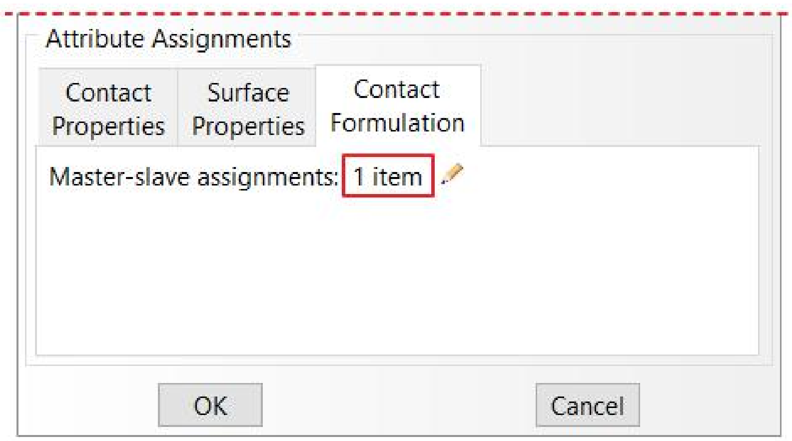
Pick >>> in the dialog box to transfer the selection to the list of Master-Slave Assignments.

Select **MASTER** in the third column to specify that the first surface is the master surface



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

Notice that Edit Interaction dialog box shows the number of Master-slave assignments as shown below.



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

Pick **Dismiss** to close the manager.

Now we will resubmit the job.

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

Select UpsetGeneral and pick **Submit > OK** and notice that job completes successfully.

⇨ Comparing the Results

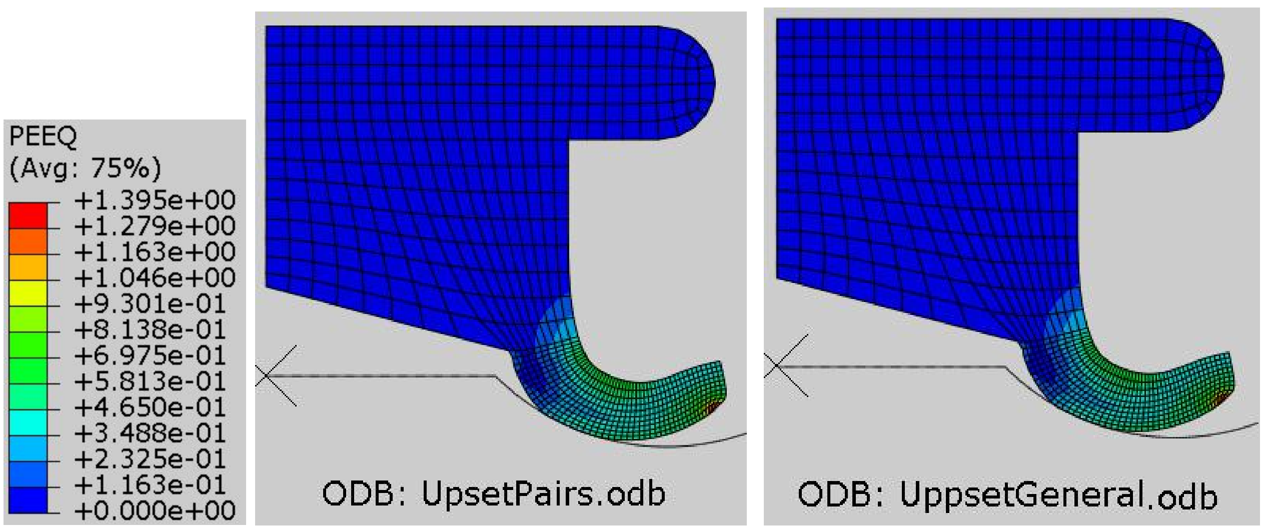
Now we will compare the results obtained from both models.

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

Pick  to plot the contours on deformed shape.

Select PEEQ in the Field Output toolbar.

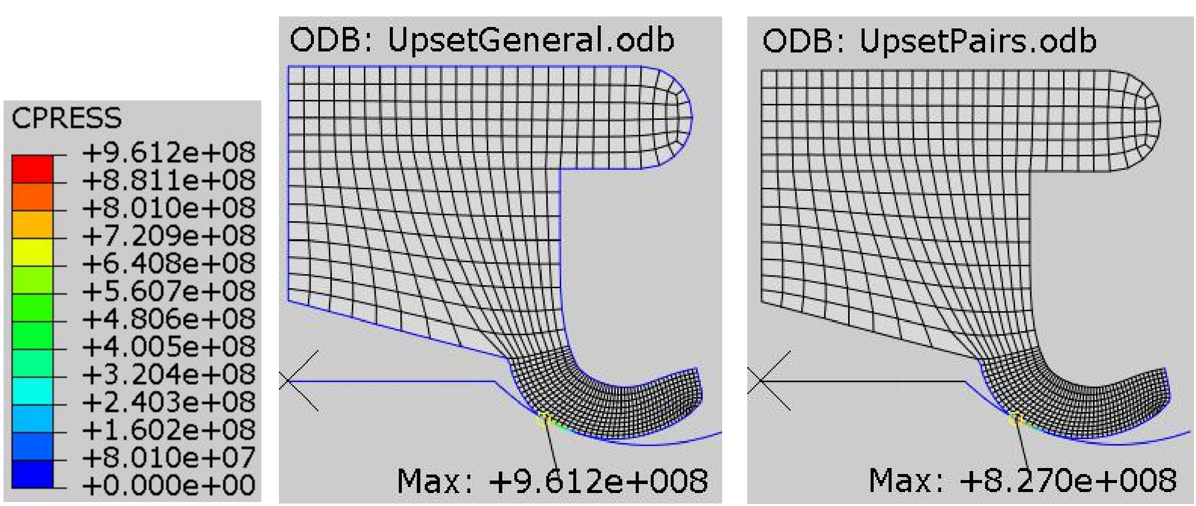
The following figure compares the distribution of equivalent plastic strain. (Note: Legend is same for both plots).



The contour plots show that the equivalent plastic strain distribution is similar for both cases.

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

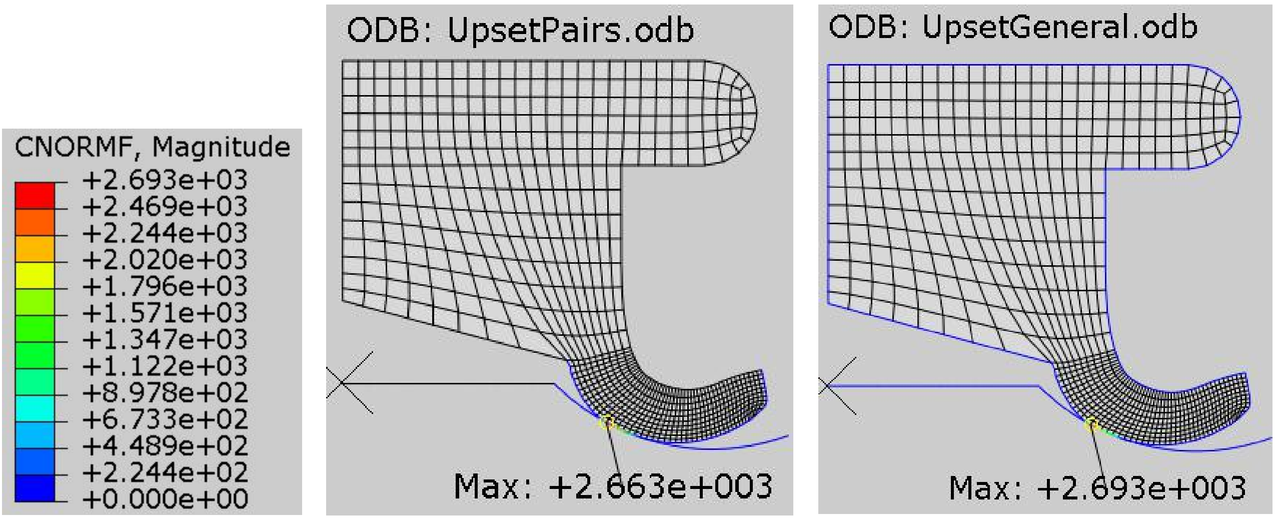
The contact pressure distribution will appear as shown in the figure below.



It can be seen that maximum contact pressure value is quite different for both cases. The convention for computing contact pressure is different for the general contact and contact pair approaches. CPRESS is computed from the contact normal force (the CNORMF vector, which is reported at the element nodal positions), therefore we will compare the contact normal force distribution instead.

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

The contact normal force distribution will appear as shown in the figure below.



It can be seen that maximum values of contact normal force are different for both cases. This difference could be due to automatic surface smoothing.

By default, Abaqus/CAE automatically detects all circumferential, spherical, and toroidal surfaces in the general contact domain that can be smoothed and applies the appropriate smoothing.

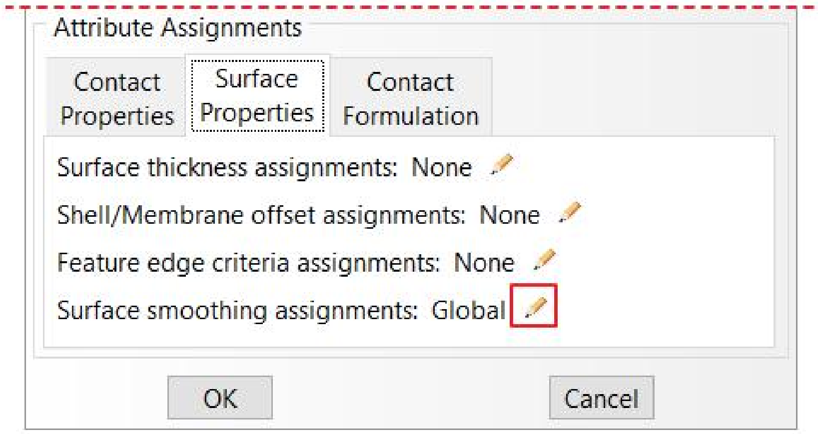
So for comparison purpose, we will modify the general contact interaction to prevent automatic surface smoothing of the model.

Modifying Contact Interaction

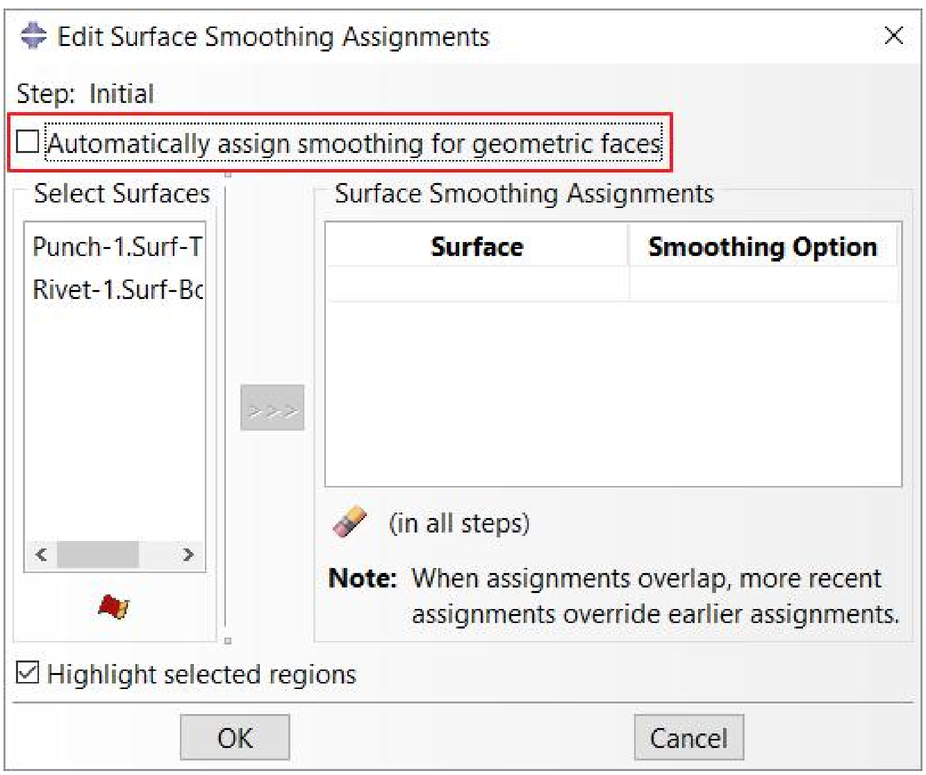
Open the Interaction Manager by picking .

Pick the "All_General" interaction under column "Initial" and pick **Edit**.

Pick  under the Surface Properties tab.

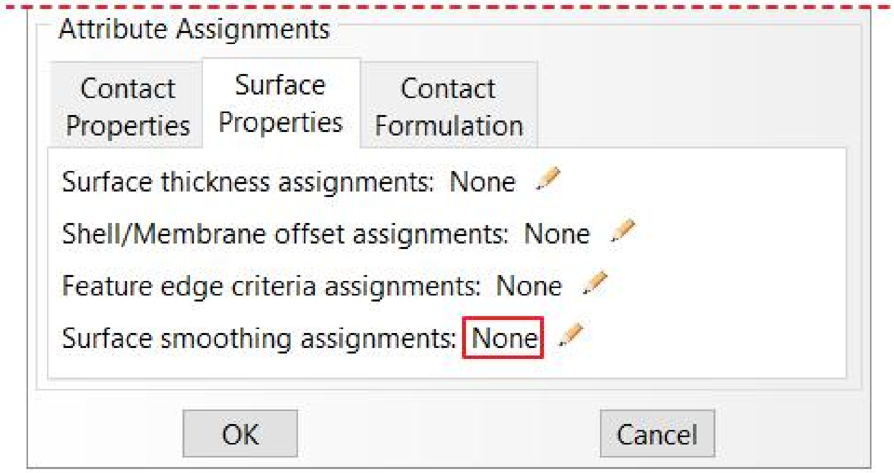


Toggle off the **Automatically assign smoothing for geometric faces** option.



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

Notice that Edit Interaction dialog box shows that no surface smoothing assignment is applied in the model as shown below.



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

Pick **Dismiss** to close the manager.

Now we will resubmit the job.

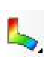
So change to **Job** module and open the Job Manager by picking .

Select UpsetGeneral and pick **Submit > OK** and notice that job completes successfully.

⇨ Comparing the Results

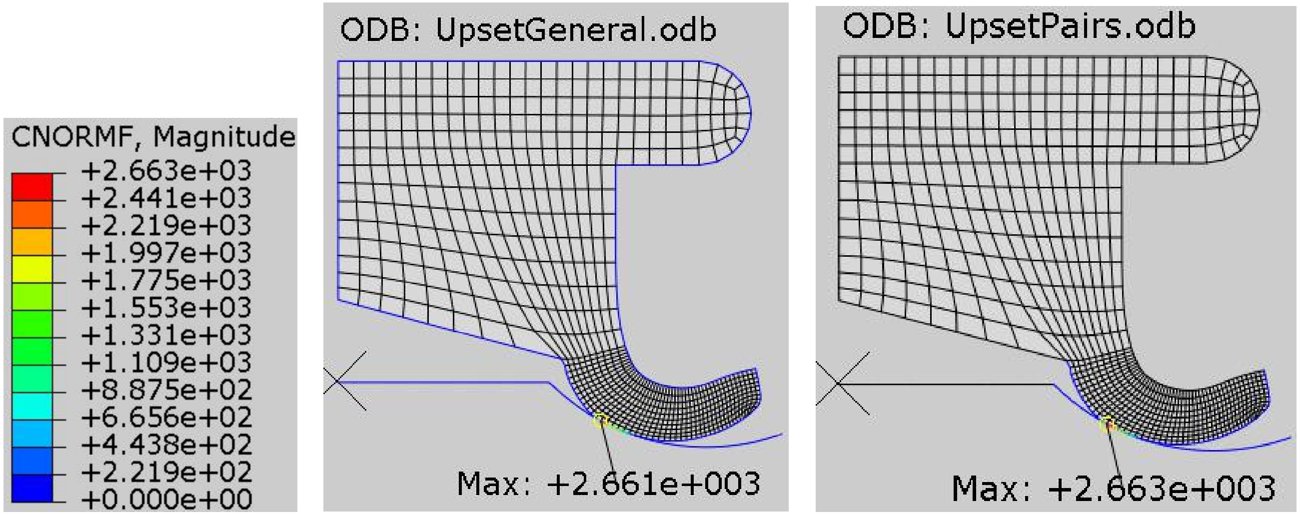
Now we will compare the results obtained from both models.

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

Pick  to plot the contours on deformed shape.

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

The contact normal force distribution will appear as shown in the figure below.



It can be seen that maximum values of contact normal force are almost same for both cases.

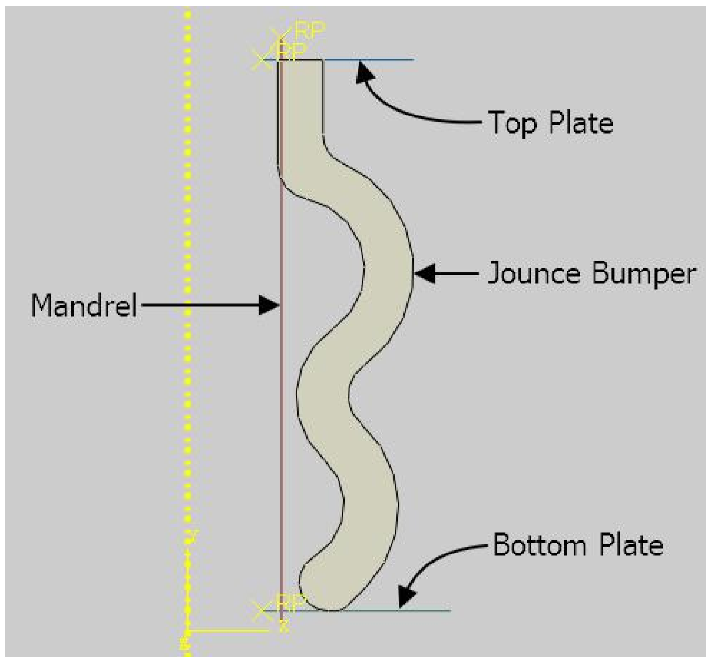
Select **File** > **Save** to save the changes we made so for.

Exercise 23

In this exercise we will perform the stress analysis of a jounce bumper. We will use general contact approach to define the contact interactions.

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

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



The model consists of four axisymmetric parts: top plate, bottom plate, mandrel and jounce bumper. All the parts except the jounce bumper are modeled as discrete rigid parts. There is an initial interference between jounce bumper and mandrel. The jounce bumper is assumed to be made of rubber, which is modeled as a Mooney-Rivlin material with C10= 5.2 MPa, C01 = 0.98 MPa and D=0.002 MPa-1. A coefficient of friction of 0.1 is assumed between contacting surfaces. The bumper is meshed with bilinear axisymmetric quadrilateral elements with reduced integration and hybrid formulation (CAX4RH).

⇨ Analysis Steps

The analysis will be performed in two steps. In the first step, interference between jounce bumper and mandrel is resolved. In the second step top plate is moved downwards which compresses the bumper between bottom and top plates. The steps have already been defined with a total time period set to **1.0** and the initial time increment to **0.01** for each.

As the top plate moves downward, large deformations and self-contact take place. To take into account the large deformations, NLgeom option has been toggled on in both steps.

Furthermore due to large relative sliding of surfaces and friction, it is expected that the magnitude and influence of unsymmetric terms would be significant. So the unsymmetric solver has been specified for this step.

Defining Contact Interactions

Now we will create a general contact interaction to define contact for the entire model..

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

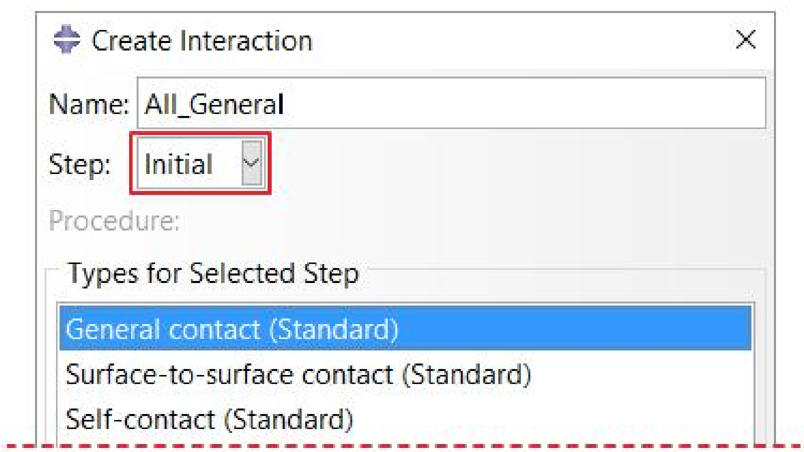
Pick **Create** to define a new interaction.

Enter **All_General** as the name of the interaction.

Pick **Initial** in the Step field.

In Abaqus/Standard general contact can be defined only in the initial step.

Select **General contact** and pick **Continue** to proceed.



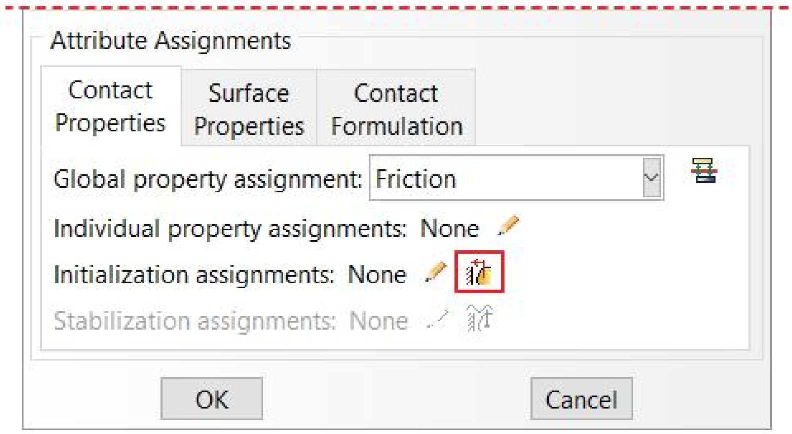
In the Edit Interaction dialog box, pick **Friction** as the global contact interaction property.

Notice that **All* with self** is selected by default.

When "All* with self" option is used, Abaqus/Standard generates a surface that contains all exterior element faces (with some exceptions) in the model and solves a self-contact problem for this all-inclusive surface. The surface can span many disconnected regions/bodies in the model. Self-contact for the surface that spans multiple bodies implies self-contact for each body as well as contact between the bodies.

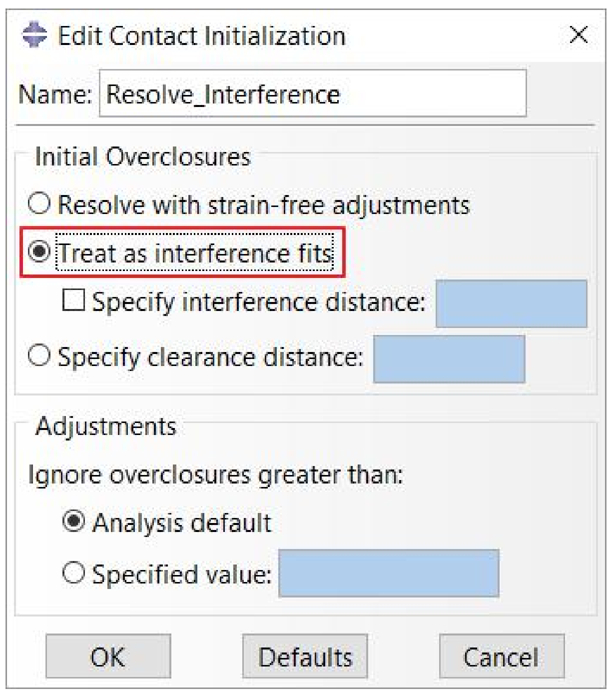
It can be seen that a small interference exists between two parts. By default in general contact interactions, small initial overclosures are resolved using strain-free adjustments to the positions of surface nodes. We will define a contact initialization method such that the initial overclosures are treated as interference fit.

Pick to create a new contact initialization



Enter *Resolve_Interference* as the name of contact initialization.

Pick **Treat as interference fits** in the Edit Contact Initialization dialog box.



When "Treat as interference fits" option is selected, Abaqus resolves contact overclosures gradually during the first step in the analysis. As these overclosures are resolved, it results in stresses and strains in the model.

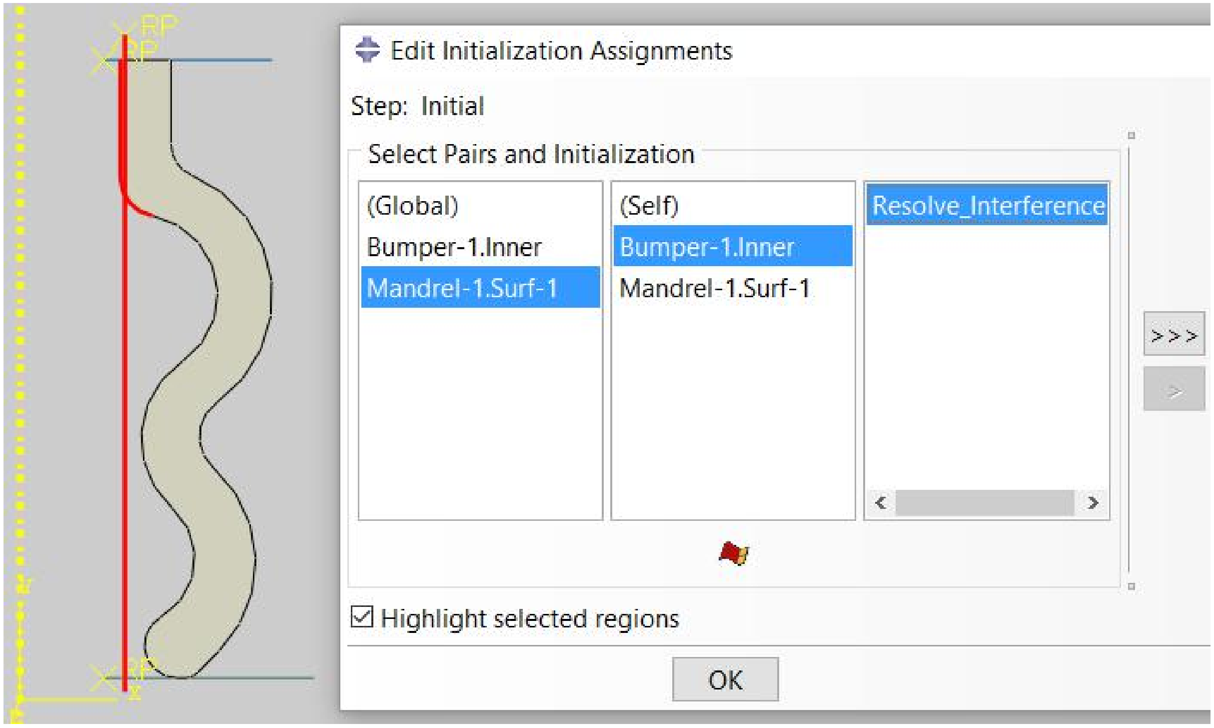
Pick **OK** and it completes definition of the contact initialization.

Pick  to assign this contact initialization to the desired regions.

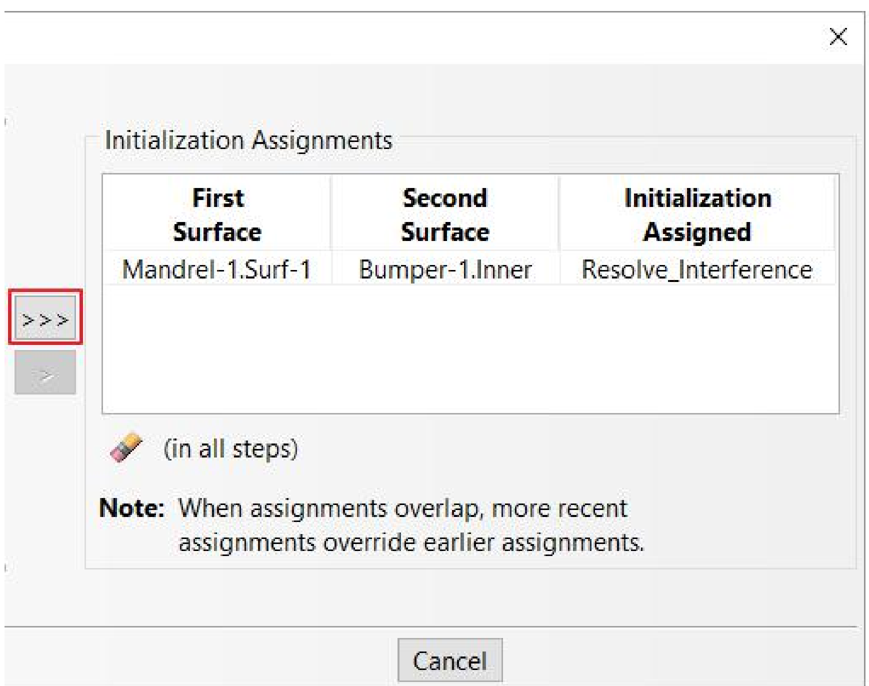
The surfaces required to specify the initialization assignment have already been defined and therefore appear in the Edit Initialization Assignment dialog box.

Pick **Mandrel-1.Surf-1** and **Bumper-1.Inner** in the first and second columns respectively to define the surface pairings.

In the third column, select the "Resolve_Interference" as the contact initialization.

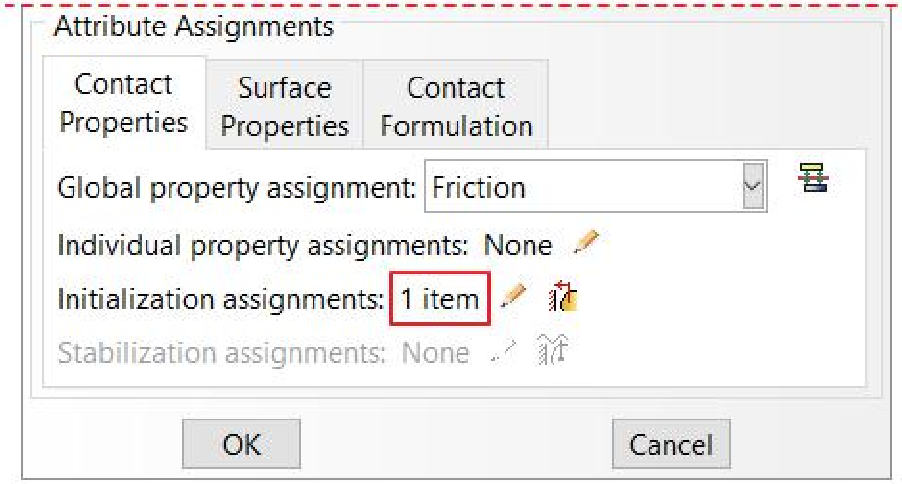


Pick >>> in the dialog box to transfer the selection to the list of initialization assignments.



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

Notice that Edit Interaction dialog box shows the number of initialization assignments as shown below.



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

Pick **Dismiss** to close the manager.

Boundary Conditions

Boundary conditions required for the analysis have already been defined. The boundary conditions “BottomPlate” and “TopPlate” and “Mandrel” constrain the the motion of the respective parts. Top plate is moved by a magnitude of 0.035 in the y-direction during Step-2.

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 BumperCompress or any other suitable name.

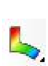
Pick **Continue** and then **OK**.

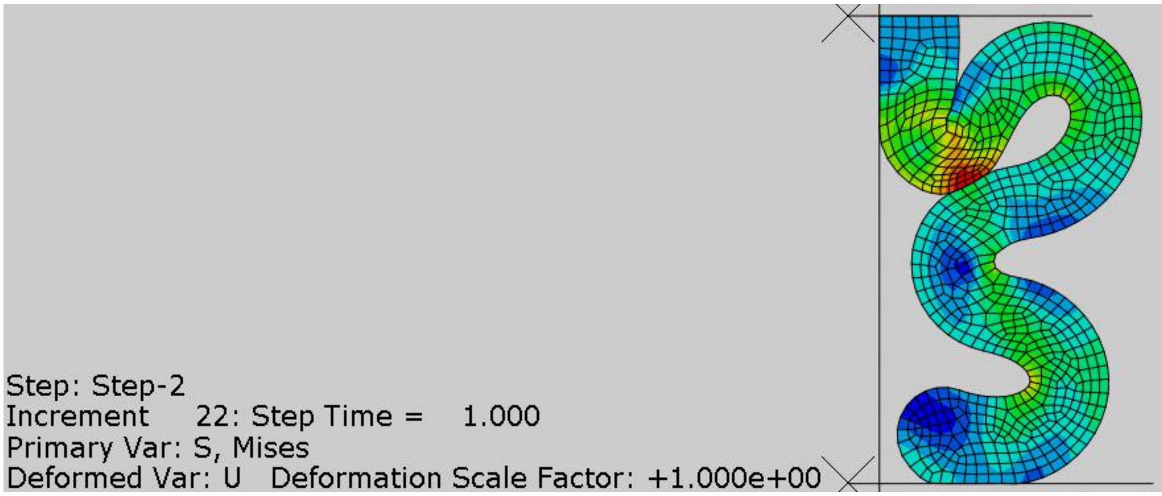
Pick **Submit** to submit the job for analysis.

Notice that analysis job completes successfully.

Postprocessing

Select BumperCompress job in the manager and pick **Results**

Pick  to plot the contours on deformed shape. The final deformed shape of the bumper will appear as shown in the figure below.



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

Note: In this example a coarse mesh has been used to limit the problem size to less than 1000 nodes (a limitation of student version of Abaqus). It is recommended to use a fine mesh of linear, reduced-integration elements (CAX4R, CPE4R, CPS4R, C3D8R, etc.) for simulations involving very large mesh distortions

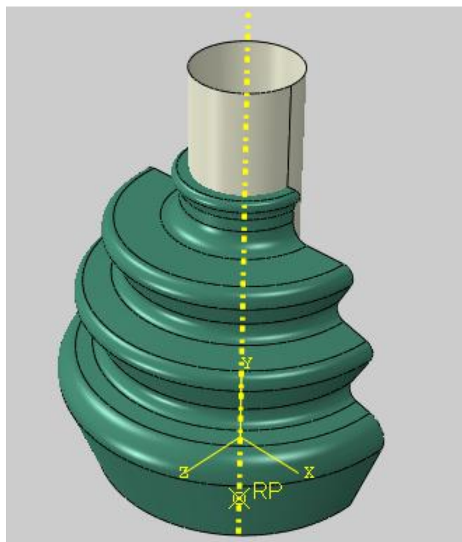
Exercise 24

In this exercise we will perform the stress analysis of a boot seal due to angular movement of the shaft. We will use both contact pairs and general contact approach to define the contact interactions.

Boot seals are used to protect steering mechanisms in automobiles. As the seal is symmetric about a plane passing through the axis, only half of the assembly is considered for the analysis.

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

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



The model consists of two three-dimensional parts: shaft and seal. The shaft is modeled as an analytical rigid part. There is an initial interference between the seal and the mandrel. The seal is assumed to be made of rubber, which is modeled as a Mooney-Rivlin material with C10= 5.2 MPa, C01 = 0.98 MPa and D=0.002 MPa-1. A coefficient of friction of 0.15 is assumed between contacting surfaces. The seal is meshed with 8-node brick elements with reduced integration and hybrid formulation (C3D8RH).

Analysis Steps

The analysis will be performed in two steps. In the first step, interference between the seal and the shaft is resolved. In the second step the shaft is rotated. The steps have already been defined with a total time period set to **1.0** and the initial time increment to **0.01** for each.

As the shaft rotates, large deformations and self-contact take place. To take into account the large deformations, NLgeom option has been toggled on.

Defining Contact Interaction

The shaft is modeled as an analytical rigid part. As the general contact algorithm does not consider analytical rigid surfaces in Abaqus/Standard, we need to define contact between the shaft and the seal with contact pairs approach. A contact pairs interaction has already been defined for that purpose. This interaction resolves interference between the shaft and the seal during Step-1.

Now we will create a general contact interaction to define contact for the entire model.

General contact can be used simultaneously with the contact pair algorithm. Abaqus automatically excludes interactions that are defined with the contact pair algorithm.

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

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

Enter **All_General** as the name of the interaction.

Pick **Initial** in the Step field.

In Abaqus/Standard general contact can be defined only in the initial step.

Select **General contact** and pick **Continue** to proceed.

In the Edit Interaction dialog box, pick **Friction** as the global contact interaction property.

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

Pick **Dismiss** to close the manager.

Boundary Conditions

Boundary conditions required for the analysis have already been defined. “SealBottom” boundary condition constrains the the motion of the bottom surface of the seal. “SealSymm” applies a symmetry boundary condition to symmetry surface of the seal. “ShaftRP” boundary condition constrain the the motion of the shaft. An angular movement of the shaft is specified during Step-2.

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 ShaftRotate or any other suitable name.

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

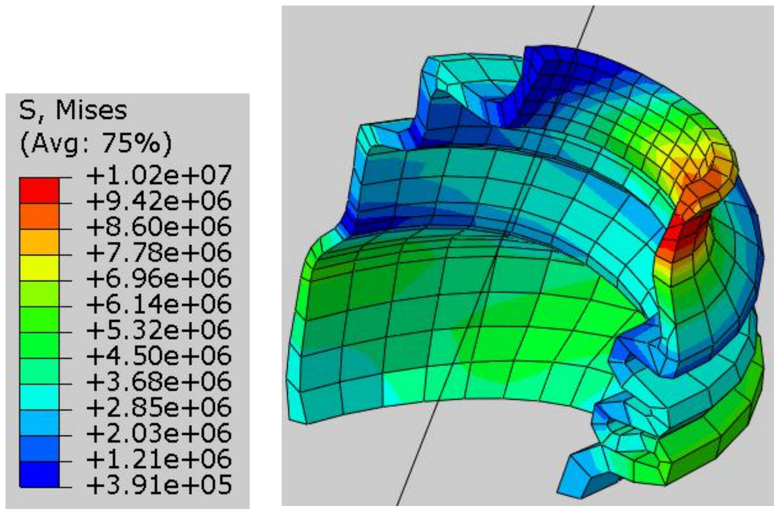
Pick **Submit** to submit the job for analysis.

Notice that analysis job completes successfully.

Postprocessing

Select ShaftRotate job in the manager and pick **Results**

Pick  to plot the contours on deformed shape. The final deformed shape of the bumper will appear as shown in the figure below.



It can be observed that the surfaces on compressed side of the seal have come into self-contact .

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

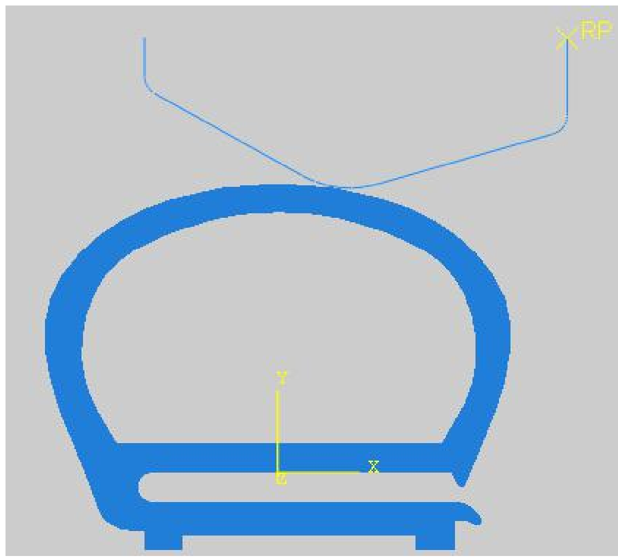
Note: In this example a coarse mesh has been used to limit the problem size to less than 1000 nodes (a limitation of student version of Abaqus). It is recommended to use a fine mesh of linear, reduced-integration elements (CAX4R, CPE4R, CPS4R, C3D8R, etc.) for simulations involving very large mesh distortions.

Exercise 25

In this exercise we will perform the stress analysis of a door seal. We will use general contact approach to define the contact interactions.

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

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



The model consists of two parts: seal and door. The door is modeled as discrete rigid part. The seal is assumed to be made of rubber, which is modeled as an incompressible Mooney-Rivlin material with C_{10}= 3.1 MPa and C_{01} = 0.85 MPa. A coefficient of friction of 0.1 is assumed between the door and the seal and a coefficient of friction of 0.15 is assumed for self contacting surfaces of the seal. The seal is meshed with bilinear plane strain quadrilateral elements with reduced integration and hybrid formulation (CPE4RH).

Analysis Steps

The analysis will be performed in one step. The step has already been defined with a total time period set to **1.0** and the initial time increment to **0.01.**

As seal is modeled as a hyperelastic material, it may undergo large deformations. To take into account these large deformations, NLgeom option has been toggled on in the step. Furthermore due to large relative sliding of surfaces and friction, it is expected that the magnitude and influence of unsymmetric terms would be significant. So the unsymmetric solver has been specified for this step.

Defining Contact Interaction

Now we will create a general contact interaction to define contact for the entire model.

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

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

Enter **All_General** as the name of the interaction.

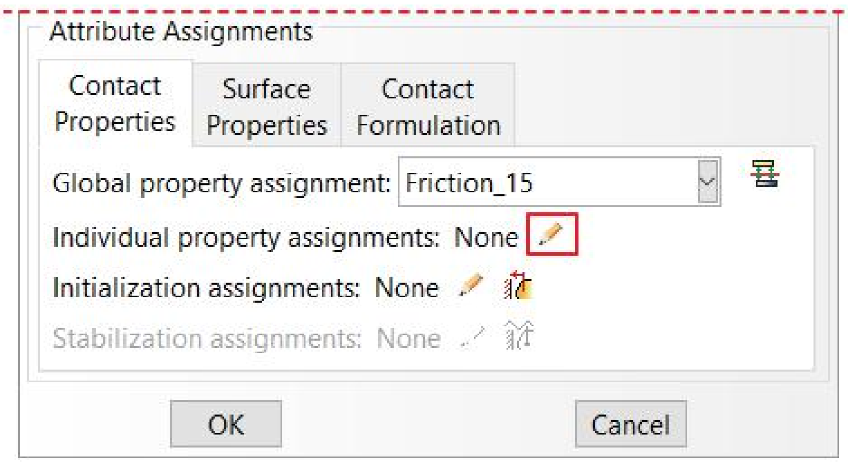
Pick **Initial** in the Step field.

Select **General contact** and pick **Continue** to proceed.

In the Edit Interaction dialog box, pick **Friction_15** as the global contact interaction property.

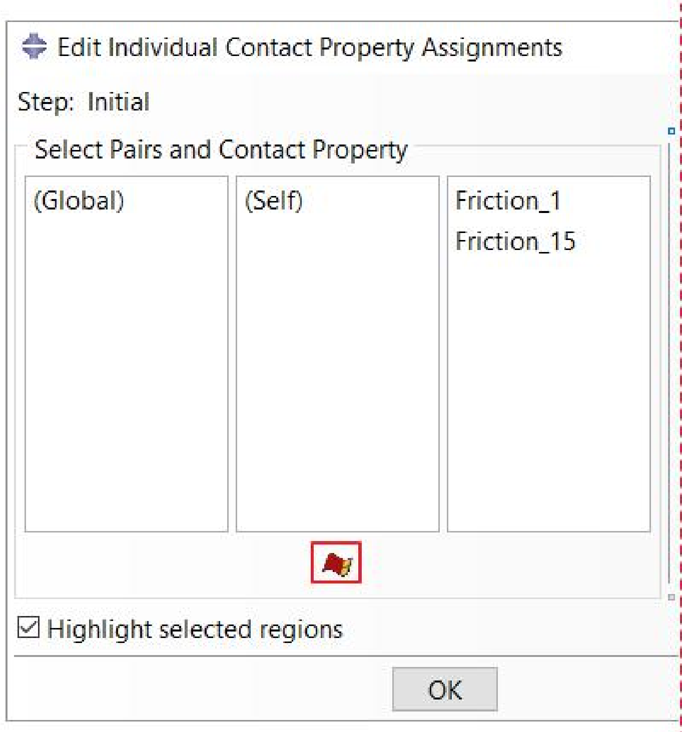
"Friction_15" interaction property defines a friction coefficient of 0.15. As we intend to specify a friction coefficient of 0.1 between the door and the seal, we will define an individual property assignment. Individual property assignments are used to assign different contact properties to individual surface pairs.

Pick  to create individual property assignment.



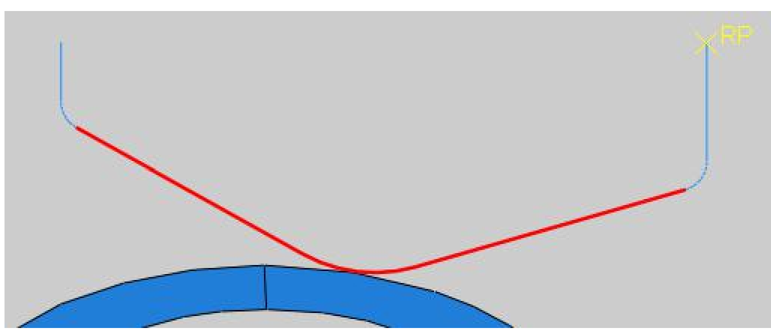
The surfaces required to specify the property assignment have not been defined beforehand, therefore we will define new surfaces.

Pick  to define a new surface.

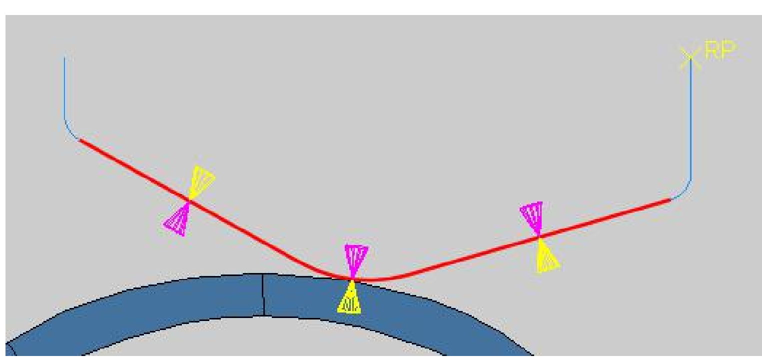


Enter **Surf-Door** as the name of the surface and pick **Continue**.

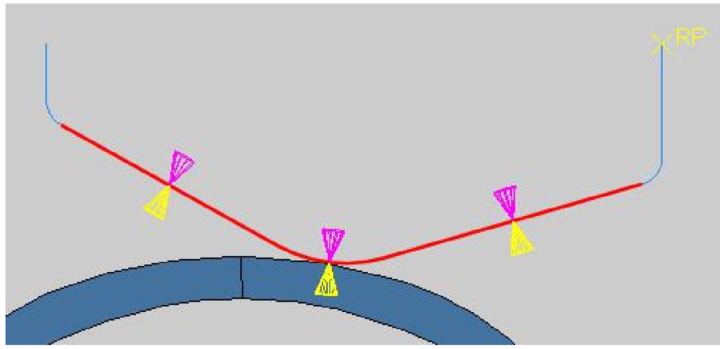
Pick the following edges while holding down Shift key.



Pick **Done** and notice that not all of the desired sides of the selected edges are of the same color as shown in the figure below.



So pick **Flip a surface** icon and select the left-most edge to reverse the orientation of this face. After flipping it will appear as shown below.



"Flip a surface" option allows to reverse the orientation of any individual face before creating the surface definition

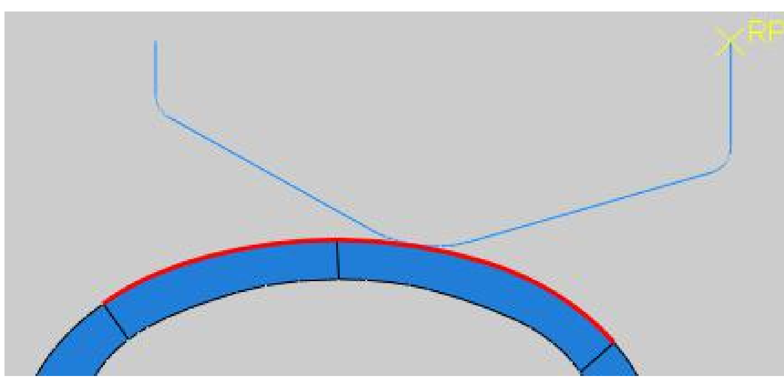
Pick **Yellow** for the side of surface in contact.

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.

Again pick  to define a new surface.

Enter **Surf-Seal** as the name of the surface and pick **Continue**.

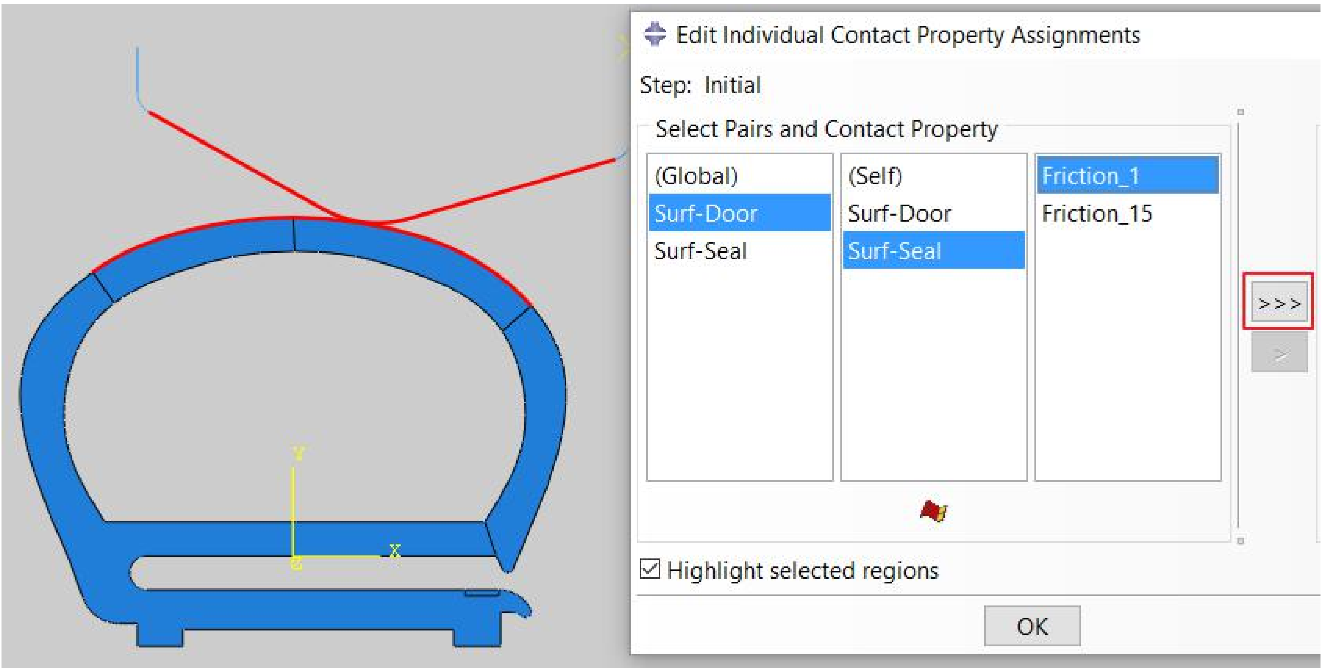
Pick the following edges while holding down Shift key.



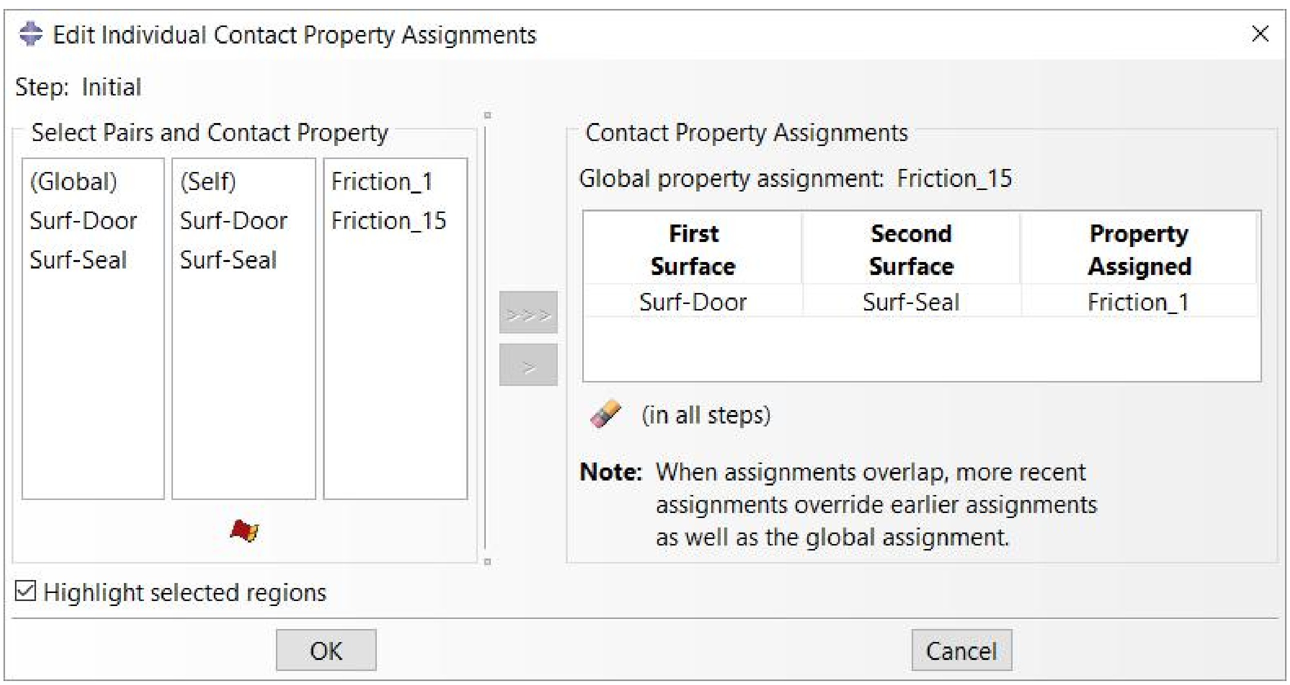
Pick **Done** to complete the definition.

Pick **Surf-Door** and **Surf-Seal** in the first and second columns respectively to define the surface pairings.

In the third column, select the "Friction_1" as the contact property.

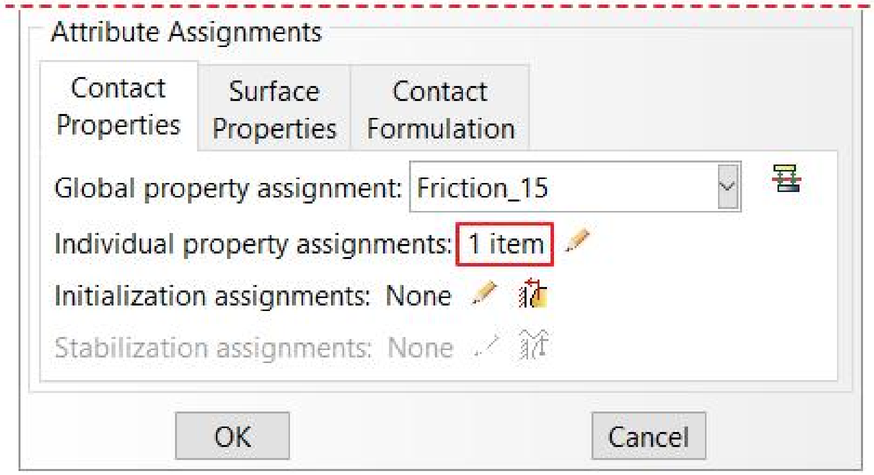


Pick >>> in the dialog box to transfer the selection to the list of contact property assignments.



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

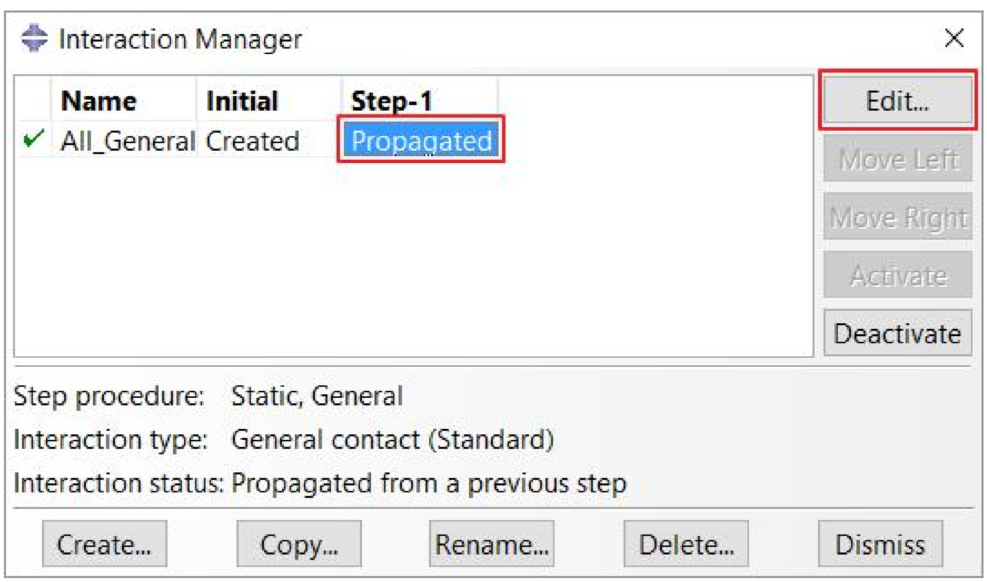
Notice that Edit Interaction dialog box shows the number of individual property assignments to 1 as shown below.



We will introduce the automatic contact stabilization to alleviate the convergence difficulties. It can be seen that stabilization assignments field is not active in the dialog box. It is because that contact stabilization cannot be assigned in the initial step. Therefore we will assign stabilization in the Step-1.

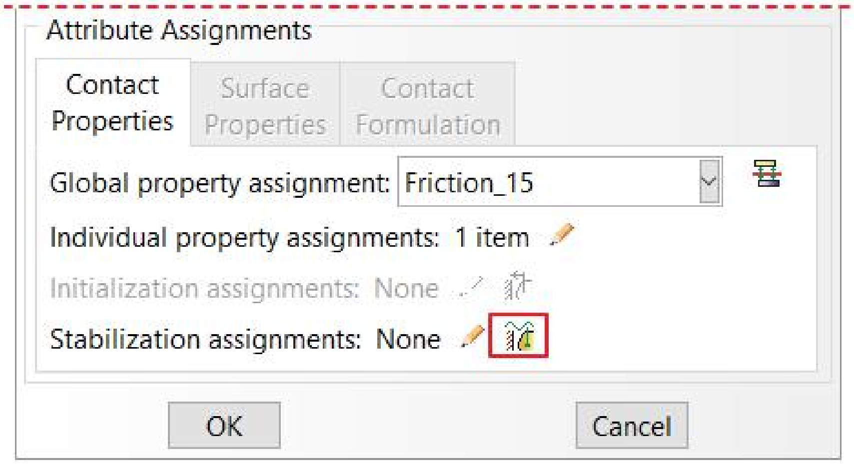
Pick **OK** to apply the changes and exit.

Pick the “All_General” interaction under column “Step-1” and pick **Edit**.



As general contact interaction can be created only in the initial step, stabilization assignment requires to edit a general contact interaction in a later step.

Pick  to create a new contact stabilization behavior.



Enter **ConStabilization** as the name of contact stabilization behavior.

Enter **1** as the reduction factor.

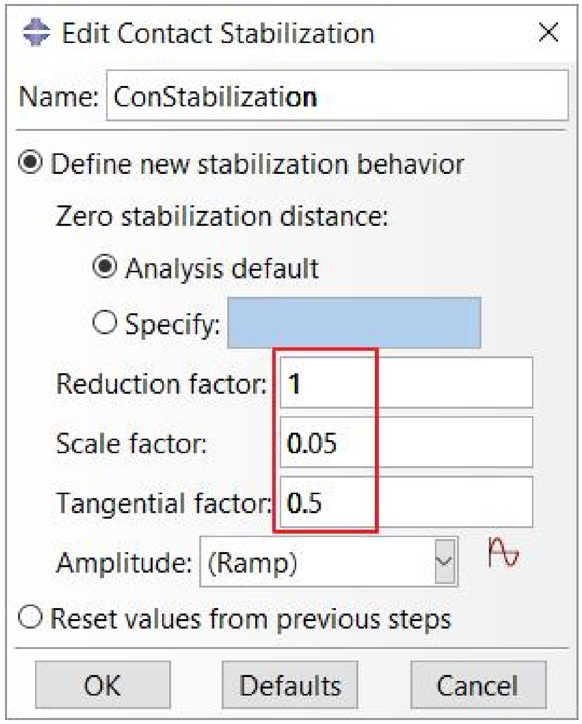
Reduction factor determines how the damping value changes in successive increments. A value less than one causes the damping to decrease with each increment. A value of one causes the damping to remain constant with each increment

Enter **0.05** as the scale factor to scale down the default damping factor.

Scale factor is used to scale the default damping coefficient by a specified factor to minimize the effects of stabilization on the solution.

Enter **0.5** as the tangential factor .

Tangential factor is used to control tangential stabilization. If it is set to zero then no stabilization is applied in tangential direction.



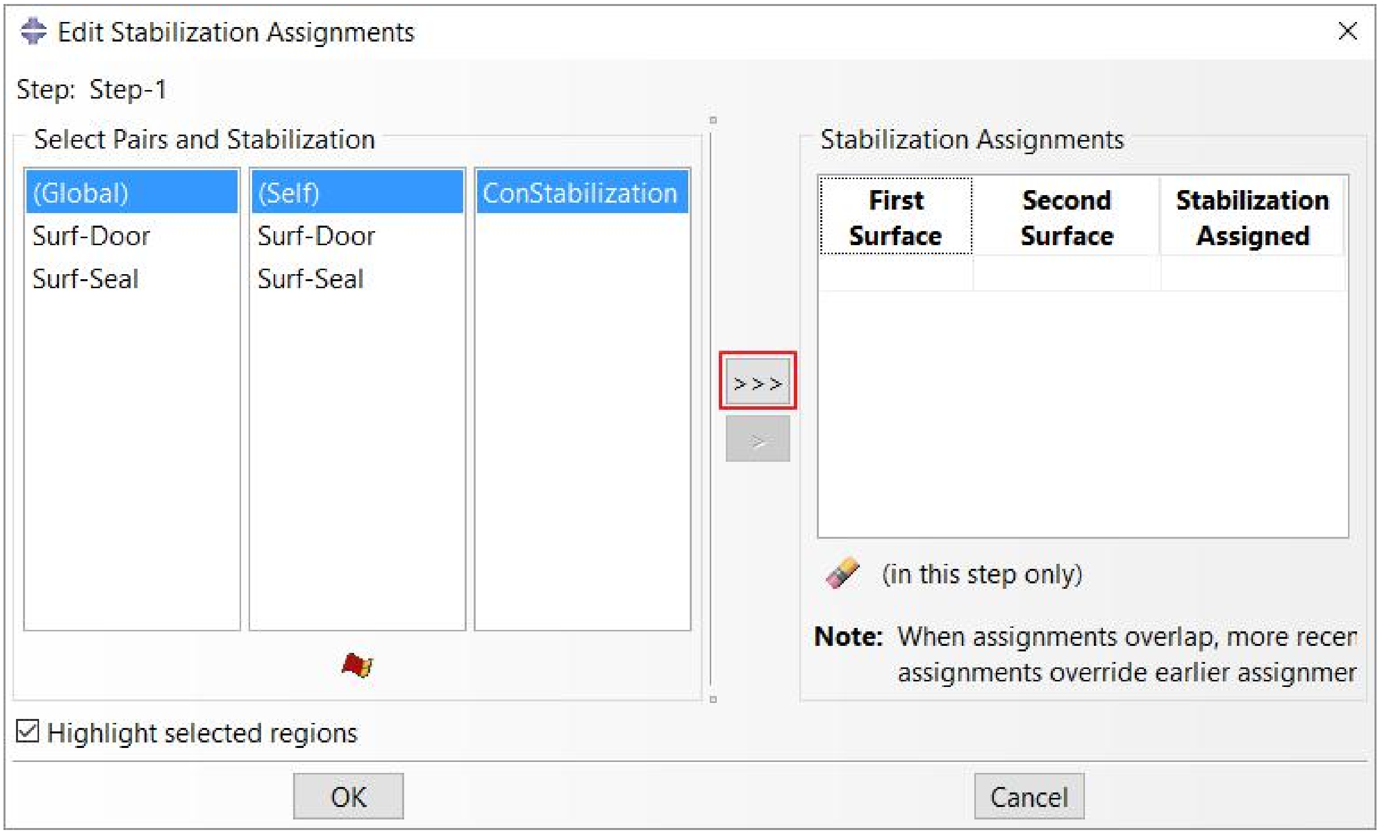
Pick **OK** and it completes definition of the contact stabilization behavior.

Pick to assign the contact stabilization to the desired regions.

Pick **Global** and **Self** in the first and second columns respectively to define the surface pairings.

In the third column, select the **ConStabilization** as the contact stabilization behavior.

Pick >>> in the dialog box to transfer the selection to the list of stabilization assignments.



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

Notice that Edit Interaction dialog box shows the number of stabilization assignments as 1.

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

Pick **Dismiss** to close the manager.

Boundary Conditions

Boundary conditions required for the analysis have already been defined. The boundary conditions "Door" and "Seal" constrain the the motion of the respective parts. Door is moved by a magnitude of 0.015 in the y-direction during Step-1.

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 DoorCloseGeneral or any other suitable name.

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

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

Notice that analysis job completes successfully.

Postprocessing

Select DoorCloseGeneral job in the manager and pick **Results**

Pick  to plot the contours on deformed shape.

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

For a three-dimensional visual effect, we will extrude the simulation results. This helps a lot to understand the contour plots.

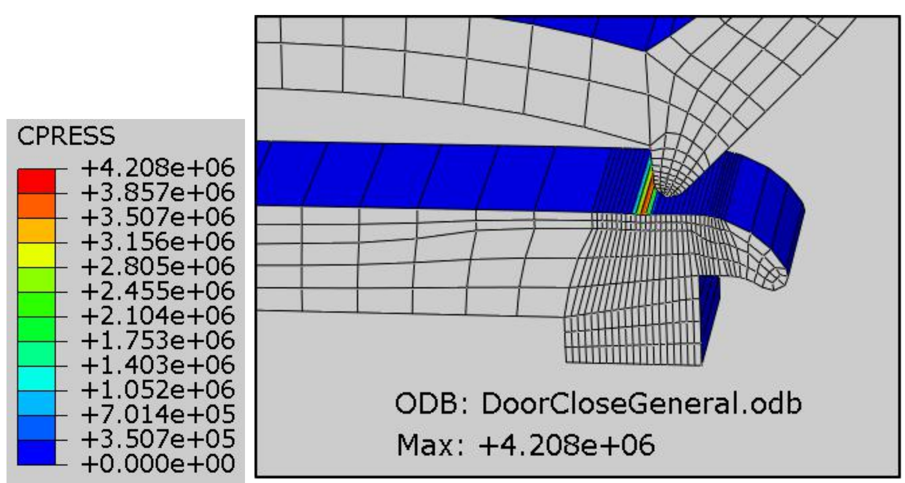
While in the Visualization module, pick **View > ODB Display Options** and check the "Extrude elements" option located under the Sweep/Extrude tab.

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

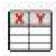
Pick **OK** to apply and exit the dialog box.

To see the contact pressure distribution in the self-contact area of the seal, pick  and set the scale factor to **0.6** in in the Common Plot Options dialog box.

Now the simulation results will appear as shown below.



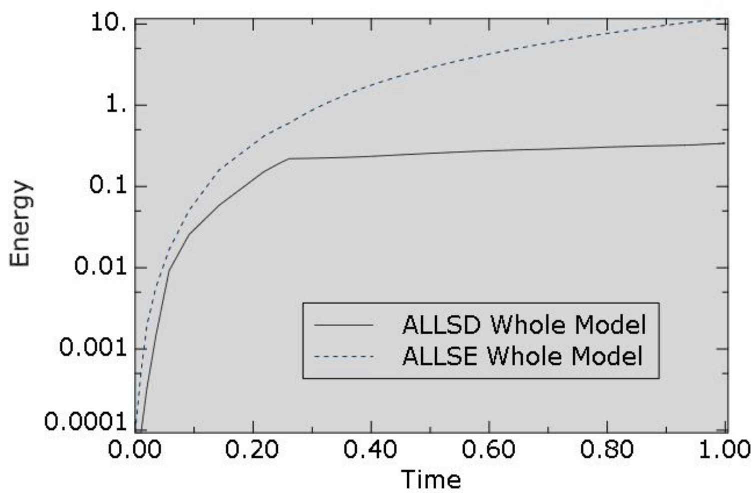
The introduction of contact stabilization in a problem can change the solution significantly. Although the automatically calculated damping coefficient typically provides enough damping to stabilize a problem, it is not certain that the value is unnecessarily high and distorting the solution. Therefore it is necessary to verify that the inclusion of contact 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: The y-axis is displayed in a base 10 logarithmic scale.)



It can be seen that the dissipation energy is less than 2% of the strain energy of the model. So it can be concluded that contact stabilization has a negligible effect on the solution.

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