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.

Module: Interaction	Vodel: ContactGeneral	✓ Step: ↓ Initial ✓
Open the Interaction Manager by picking	E	
Pick Create to define a new interaction.		
Enter All_General as the name of the inter	raction.	
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.

.....

💠 Crea	ate Interaction	×
Name:	All_General	
Step:	Initial 🖌	
Proced	ure:	
Types	for Selected Step	
Gener	al contact (Standard)	
Surfac	ce-to-surface contact (Standard)	
Self-c	ontact (Standard)	

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.

💠 Edit Intera	action		×
Name: All_Ge	eneral		
Type: Gener	ral contact (S	tandard)	
Step: Initial			
Contact Dor	main		
Included sur	face pairs:		
All* with	th self		
O Selecte	ed surface pa	irs: None 🦯	
Excluded sur	face pairs: N	Jone 🥖	
* "All" includ	loc all ovtorio	r faces and feature es	laes It evoludes
analytical ric	id surfaces, l	beam segments, and r	eference points.
Attribute As	signments		
Contact	Surface	Contact	
Properties	Properties	Formulation	
Global prop	perty assignm	nent: Friction	~ 뀸
	property assig	nments: None 🧷	
Individual p			
Individual p	n assignment	ts: None 🥕 👔	
Individual p Initialization Stabilizatio	n assignment n assignment	ts: None 🥒 👬 ts: None 🦯 វ៉ាវិ	
Individual p Initialization Stabilizatio	n assignment n assignment	ts: None 🥒 👬 ts: None 🦯 🏦	

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 **III**.

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.

Select entities to replace	Part instances	🛛 Done Unc	lo Redo
----------------------------	----------------	------------	---------

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 low to display all the parts on screen.

Pick ⁽¹⁾ 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.

💠 Sa	ndwichPair Monite	or		×	💠 Sand	dwichG	eneral Mo	o <mark>nit</mark> or	3 <u>375</u>	
Job: S	andwichPair St	atus: Comp	leted		Job: Sa	ndwich	General	Status:	Completed	l.
Step	Increment	Att	Severe Discon Iter	Equ [^] Ite	Step	Inc	rement	Att	Severe Discon Iter	Equ [^] Ite
1	15	1	1	2	1		13	1	7	1
1	16	1	8	1	1		14	1	7	1
1	17	1	5	2 🗸	1		15	1	5	3 ~
<				>	<					>
Log	Errors Warning	s Output	Data File	Messag	Log	Errors	!Warnin	gs Outp	out Data	File Messa
	JOB TIME SUMM USER TIME (1 SYSTEM TIME TOTAL CPU T WALLCLOCK T	ARY (SEC) (ME (SEC) IME (SEC) IME (SEC)	= 22.10 = 1.100 = 23.20		,	JOB TI USER SYST TOTA WALL	ME SUMMA TIME (S EM TIME L CPU TI CLOCK TI	ARY (SEC) (SEC) ME (SEC) ME (SEC	= 28. = 0.90 () = 29. () =	400 000 300 35
<				>	<					>
Sear	h Text				Search	n Text				
Text t	o find: [Text to	find:				
	Kill		Dismiss			Kil	1		Dismi	iss

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.

■ Models (2)
 ■ ContactGeneral
 ■ ContactPairs

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.

Module: Interaction	Model: 🗘 ContactGeneral 🗸	Step: 🗧 Initial 🗸
Open the Interaction Manager by picking		
Pick Create to define a new interaction.		
Enter All_General as the name of the inte	eraction.	

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.

Name:	All_General	
Step:	Initial 🖌	
Proced	lure:	
Туре	s for Selected Step	
and the second se	ral contact (Standard)	
Gene	rar contact (standard)	
Gene Surfa	ce-to-surface contact (Standard)	

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.

	Name	Sten-1			Edit
V	F-Output-1	Created			Move Left
					Move Righ
					Activate
					-
					Deactivate
Ste	p procedure	: Static, General			Deactivate
Ste Va	ep procedure riables:	: Static, General CDISP,CF,CFORC	E,CSTRESS,LE,PE,PEE	Q,PEMAG,RF,S,U,	Deactivate
Ste Vai Sta	ep procedure riables: atus:	: Static, General CDISP,CF,CFORC Created in this s	E,CSTRESS,LE,PE,PEE ep	Q,PEMAG,RF,S,U,	Deactivate

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

CDISP	.CF.CFORCE.CSTRESS.LE.PE.PEEO.PEMAG.RF.S.U.	
-		
▼ 🗉	Contact	
	CSTRESS, Contact stresses	
	CSTRESSETOS, Edge-to-surface contact pressure / shear stresses	: 1
	CLINELOAD, Contact force per unit length from line contact	
	CPOINTLOAD, Contact constraint force from point contact	
	CDSTRESS, Contact damping stresses	
	CDISP, Contact displacements	
	CDISPETOS, Edge-to-surface contact opening/tangential motio	n
	CTANDIR, Contact tangent directions in general contact or cont	act
	CFORCE, Contact force	
	CNAREA, Contact area associated with each node in contact	
	CSTATUS, Contact status	

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 **III**.

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.

💠 Create Display Group		×	
Make a Selection			
Item	Name filter:	÷ Ö t	
All Part instances Elements Nodes	_ALL_GENERAL_GCS0_12 _ALL_GENERAL_GCS0_14 _ALL_GENERAL_GCS0_3 General_Contact_Eaces		
Surfaces	General Contact Faces 1		
Display groups Coordinate systems Ties Shell-to-Solid couplings Distributing couplings Kinematic couplings Rigid Bodies MPCs	General_Contact_Faces_2		
Method			×
Surface sets Result value All surfaces Internal sets			

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.)



Open the Interaction Manager by picking 🛄.

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

🜩 In	nteraction	Manager				×
N	lame	Initial	Step-1			Edit
¥ A	II_General	Created	Propagate	c		Move Left
						Move Right
						Activate
						Deactivate
Step	procedure	λ ⁴				
Intera	action type	e: General	contact (Sta	ndard)		
Intera	action stat	us: Created	in this step			
	Create		Copy	Rename	Delete	Dismiss

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

Contact Properties	Surface Properties	Contact Formulation	
Master class	e assignmen	ts: None	
viaster-slav	c assignment	LS, INUTIC /	
waster-stav	e assignmen	a. None	
waster-stav	e assignmen		
waster-slav	e assignmen		

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.

Edit Master-Slave	Assignments			×
Step: Initial				
Select Pairs		Master-Slave Assign	nments	
(Global) Punch-1.Surf-Top	(Self) Punch-1.Surf-Top	First Surface	Second Surface	First Surf: Type
Rivet-1.Surf-Bottom	Rivet-1.Surf-Bottom			22444
		>>>		
		(in all steps)		<u> </u>
	↓↓ ♥	Note: When the ma	aster and slave region or the overlapping	ons overlap, self- regions.
Highlight selected	regions			
	OK		Cancel	

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

💠 Edit Master-Slav	ve Assignments			×
Step: Initial				
Select Pairs		Master-Slave Assig	nments	
(Global) Punch-1 Surf-Top	(Self) Punch-1 Surf-Top	First Surface	Second Surface	First Surface Type
Rivet-1.Surf-Botto	Rivet-1.Surf-Botto	Punch-1.Surf-Top	Rivet-1.Surf-Bottom	MASTER
		>>		
		<		>
		🤣 (in all steps)		
< >		Note: When the m is excluded t	aster and slave region for the overlapping re	ns overlap, self-con gions.
Highlight selecte	d regions			
	ОК		Cancel	

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.

Contact Properties	Surface Properties	Contact Formulation	
Master-slav	e assignmen	ts: 1 item 🥒	

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

Pick **Dismiss** to close the manager.

Now we will resubmit the job.



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 stoplot 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.

And the second s

Open the Interaction Manager by picking 📖

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

Pick [✓] under the Surface Properties tab.

properties	Properties	Formulation	
urface thic	kness assignr	nents: None	1
hell/Memb	rane offset a	ssignments: N	one 🥒
eature edg	e criteria ass	ignments: Nor	ne 🥒
urface smo	othing assig	nments: Globa	

Toggle off the Automatically assign smoothing for geometric faces option.

	sign smoothing for geometr	ic faces
Select Surfaces	Surface Smoothing A	Assignments
Punch-1.Surf-T	Surface	Smoothing Option
Rivet-1.Surf-Bc		
	>>>	
	>>>	
	(in all steps)	

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.

Contact Properties	Surface Properties	Contact Formulation	
Surface thic	kness assign	ments: None 🥕	
hell/Mem	orane offset a	assignments: Nor	ne 🥕
aatura ada	e criteria ass	signments: None	1
-eature eug		2	

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.

	Luon	^
Name: All_Gener	ral	
Step: Initial 🗠		
Procedure:		
Types for Select	ted Step	
General contact	t (Standard)	
Surface-to-surfa	ace contact (Standard)	
Calf contact (Ct	andard)	

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 ¹/₁ to create a new contact initialization

Contact Properties	Surface Properties	Contact Formulation	
Global prop	perty assignm	nent: Friction	<u>√</u> 뮬
Initializatio	n assignment	s: None 🥒 👔	

Enter *Resolve_Interference* as the name of contact initialization.

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

💠 Edi	t Contact Ir	nitialization	×
Name:	Resolve_In	nterferenc e	
Initia	l Overclosu	res	
O Res	olve with s	train-free adjustmen	its
Tre	at as interf	erence fits	
	Specify inte	erference distance:	
⊖ Spe	e <mark>cify</mark> cleara	nce distance:	
Adjus	stments		
Ignore	e overclosu	res greater than:	
۲	Analysis de	fault	
0	Specified v	alue:	
	ОК	Defaults	Cancel

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 *f* 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.

Step: Initial Select Pairs and Ini	tialization	a
(Global)	(Self)	Resolve_Interference
Bumper-1.Inner	Bumper-1.Inner	
Mandrel-1.Surf-1	Mandrel-1.Surf-1	
		< >
	N	
Highlight selected	Aregions	

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

EU SI	Second	Initialization
Surface	Surface	Assigned
Mandrel-1.Surf-1	Bumper-1.Inner	Resolve Interference

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.

Properties	Surface Properties	Contact Formulation		
Global prop	perty assignm	nent: Friction	~	묩
ndividual p	oroperty assig	gnments: None	1	
nitializatio	n assignment	ts: 1 item 🥕 🥻		

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 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

Solving Contact Problems

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 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.

Abaqus 2016

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 30

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.*

Properties	Surface Properties	Contact Formulation		
Global prop Individual p Initializatio	perty assignm property assign n assignment	nent: Friction_15 gnments: None ts: None 🥓 👔		
			2	

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.

(Global)	(Self)	Friction_1 Friction_15	

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.

 Edit Individu Step: Initial Select Pairs an 	al Contact Property d Contact Property	/ Assignments	
(Global)	(Self)	Friction_1	
Surf-Door	Surf-Door	Friction_15	
Surt-Seal	Sun-Sear		>>>
	-		
Highlight sele	ected regions		
		ОК	

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

🜩 Edit Indiv	idual Contact	Property Assignme	nts		×
Step: Initial					
Select Pairs (Global)	and Contact F	Property Friction_1	Contact Property As Global property assig	signments gnment: Friction_15	
Surf-Door Surf-Seal	Surf-Door Surf-Seal	Friction_15	First Surface	Second Surface	Property Assigned
			Surf-Door	Surf-Seal	Friction_1
			 (in all steps) Note: When assign 	ments overlap, more	recent
			assignments of as well as the	override earlier assig global assignment.	nments
Highlight :	selected regio	ns			
		ОК		Cancel	

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.

Contact Properties	Surface Properties	Contact Formulation		
Global prop	perty assignm	nent: Friction_	15	~ 뮬
ndividual p	property assig	nments: 1 ite	em 🥒	
nitializatio	n assignment	ts: None 🧷	1 6	
			0.0	

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.

¢	Interaction	Manager			×
	Name	Initial	Step-1		Edit
~	All_General	Created	Propagated		Move Left
					Move Right
					Activate
					Deactivate
Ste Inte	p procedure eraction type eraction state	e: Static, C e: General us: Propag	General contact (Standard) ated from a previous	step	
	Create	Сору	Rename	Delete	Dismiss

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 *i* to create a new contact stabilization behavior.

-

Properties	Surface Properties	C Fori	ontact mulation		
Global prop	perty assignm	nent:	Friction_15		· 뀸
Individual p	property assig	gnme	nts: 1 item	1	
		and the second			
Initializatio	n assignment	ts: No	one an		

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.

Define new stabili	zation be	havior
Zero stabilization	distance	:
Analysis def	ault	
O Specify:		
Reduction factor:	1	
Scale factor:	0 .05	
Tangential factor:	0. 5	
Amplitude: (Ram	(a	~ A

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

Pick 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.

💠 Edit Stabiliz	ation Assignments					×
Step: Step-1						
Select Pairs ar	nd Stabilization			Stabilization	Assignments	
(Global)	(Self)	ConStabilization		First	Second	Stabilization
Surf-Door	Surf-Door			Surface	Surface	Assigned
Surf-Seal	Surf-Seal					
			>>>			
			>			
				🧳 (in this s	ten only)	
				(III UIIS 5	tep only)	
				Note: When a assignr	assignments o ments override	verlap, more recen e earlier assignmer
Highlight sel	ected regions					
	OK				Cancel	

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 **III**.

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 stopplot 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.