### **Exercise 26**

In this exercise we will simulate the bending of a staple using both Abaqus/Standard and Abaqus/Explicit.

A staple is a short piece of wire driven through and bent to bind together papers etc. The bending process takes place very slowly such that inertial effects are negligible and it can be termed as quasi-static. Quasi-static problems can be simulated well with either Abaqus/Standard or Abaqus/Explicit. Typically, quasi static problems are solved with Abaqus/Standard but may face convergence difficulties due to contact or other nonlinearities. Such problems are a good candidate to be solved with Abaqus/Explicit as explicit procedure can resolve complicated contact problems and other discontinuous nonlinearities more easily than Abaqus/Standard.

We will simulate the bending of a staple using both Abaqus/Standard and Abaqus/Explicit and compare. By taking advantage of the symmetry, only half of the problem will be simulated.

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

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



The model consists of three parts: punch, cavity and staple. The punch and the cavity are modeled as discrete rigid parts. The staple is assumed to be made from an alloy with a Young's modulus of 90 GPa and a Poisson's ratio of 0.3. A coefficient of friction of 0.05 is assumed between contacting surfaces. The staple is meshed with bilinear plane stress quadrilateral elements with reduced integration (CPS4R).

In the model tree, it can be seen that there are two models. "Staple\_Implicit" model will be used to perform analysis using the Abaqus/Standard and "Staple\_Explicit" model will be used to perform analysis using the Abaqus/Explicit.

■ ▲ Models (2)
 ■ Staple\_Explicit
 ■ Staple\_Implicit

# ➡ Solution with Abagus/Standard

As the inertia effects can be ignored, we can perform the analysis using Abaqus/Standard.

# 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.

# Boundary Conditions

Boundary conditions required for the analysis have already been defined. The punch is moved by a magnitude of 0.0089 in the y-direction with the "Punch" boundary condition. By taking advantage of symmetry, only half of the problem is modeled. Symmetry boundary condition has been defined accordingly for the staple.

### Contact Interactions

Two contact pairs interactions, "Cavity-Staple" and "Punch-Staple", have already been defined. "Cavity-Staple" defines contact between the cavity and the staple and "Punch-Staple" defines contact between the punch and the staple. A contact interaction property, named "Friction", is specified for both contact interactions. In this contact interaction property, a friction coefficient of 0.05 has been specified.

#### → Job Submission

All the information required for analysis has already 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 BendImplicit or any other suitable name for the Staple Implicit 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

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

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

Pick successfully and job runs till 30<sup>th</sup> increment completing the 0.1424 of step time.



To investigate the cause of analysis termination, pick **Tools** > **Job Diagnostics** and Job Diagnostics dialog box will appear. It can be seen that during the  $30^{th}$  increment, system makes 3 attempts before aborting the analysis. During the  $2^{nd}$  attempt, there have been total 12 iterations and all of them are severe discontinuity iterations as shown in the figure below.

History	Summary	Warnings	Residuals	Contact	Elements
<ul> <li>Increment 30</li> <li>Attempt 1</li> <li>Attempt 2</li> <li>Iteration 1 (SDI)</li> <li>Iteration 2 (SDI)</li> <li>Iteration 3 (SDI)</li> <li>Iteration 4 (SDI)</li> </ul>	Summary Overclosu Maximum Maximum Details	res: 1 contact for penetration	ce error: 1 n error: 1	Descript Overclos Maximur Maximur	tion sures m contact forc m penetration
Iteration 5 (SDI)	Nede	Over	lagura	Claura	
<ul> <li>Iteration 6 (SDI)</li> <li>Iteration 7 (SDI)</li> <li>Iteration 8 (SDI)</li> <li>Iteration 9 (SDI)</li> <li>Iteration 10 (SDI)</li> <li>Iteration 11 (SDI)</li> <li>Iteration 12 (SDI)</li> <li>Attempt 3 v</li> </ul>	STAPLE-1	.9 1.2074	14e-08	ASSEMBL	Y_STAPLE-1_S

In the above figure, it can be seen that in the 1<sup>ST</sup> iteration, contact status for node "9" changes from open to closed. In the next iteration, contact status for the same node changes from closed to open.

If contact status for a node changes from closed to open, it is recorded as an "opening." If contact status changes from open to closed, it is recorded as an "overclosure."

When a node's status changes from closed to open in one iteration and in the next iteration its contact status changes from open to closed; it is said to be chattering and is an indication of instability in the contact.

It can be observed that bottom end of staple has a sharp corner which is penetrating the cavity surface as shown in the figure below.



These sharp corners make the resolution of contact a very difficult task. We will introduce the "softened" pressure-overclosure relationship to overcome the convergence problems at the contact interface.

The "softened" contact pressure-overclosure relationships are usually used to model a soft, thin layer on surfaces. In Abaqus/Standard they are also sometimes used because they can make it easier to resolve the contact difficulties.

### Defining Contact Interaction Property

We will define a new contact interaction property using "Tabular" pressure-overclosure relationship. This contact property will be used for the "Cavity-Staple" interaction.

So change to Interaction module and make sure that "Staple\_Implicit" model is active.

Pick to create a new contact interaction property.

Select **Contact** as type and enter **Tabular-Friction** as name of the property.

Pick Continue and Edit Contact Property dialog box will appear.

Pick Mechanical > Tangential Behavior to specify tangential behavior (friction).

Pick Penalty as friction formulation and enter 0.05 as friction coefficient.

Pick **Mechanical > Normal Behavior** to specify normal behavior.

In the Pressure-Overclosure field, select Tabular.

Tabular law is used to define a pressure-overclosure relationship in tabular form in which the contact pressure is a piecewise linear function of the overclosure between the surfaces. To define a piecewise-linear relationship in tabular form, data pairs  $(p_i,h_i)$  of pressure versus overclosure (clearance corresponds to negative overclosure) are specified as shown in the figure below.



In this relationship the surfaces transmit contact pressure when the overclosure between them, measured in the contact (normal) direction, is greater than  $h_1$ , where  $h_1$  is the overclosure at zero pressure. So the surfaces can transmit pressure even there is a clearance between them.

Enter -1E-5 as the overclosure at 0 contact pressure and 0 as the overclosure for 10E6 contact pressure and 1E-5 as the overclosure for 20E6 contact pressure as shown in the figure below.

Press	ure-Overclosure:	Tabular	~
Const	traint enforcement	t method: Default	
Provi	de data in order o	f ascending overclosure	
Note	: A negative over	closure is a positive clearance.	
Note	A negative over Pressure	closure is a positive clearance.	
Note	A negative over Pressure 0	closure is a positive clearance. Overclosure -1E-005	
Note	A negative over Pressure 0 10E6	closure is a positive clearance. Overclosure -1E-005 0	

The data must be specified as an increasing function of pressure and overclosure. The data table must begin with a zero pressure. The pressure-overclosure relationship is extrapolated beyond the last overclosure point by continuing the same slope.

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



Open the Interaction Manager by picking .

Pick the "Cavity-Staple" interaction under column "Initial" and pick Edit.

\$	Interaction Mar	nager				×
	Name	Initial	Step-1			Edit
~	Cavity-Staple	Created	Propagat	tec		Mouniloff
~	Punch-Staple	Created	Propagat	tec		IVIOVE LEIL
						Move Right
						Activate
						Deactivate
Ste	p procedure:					
Inte	eraction type: S	Surface-to-su	urface conta	act (Standard)		
Inte	eraction status: (	reated in th	is step			
				1	Trans Desc	
	Create	Copy	/	Rename	Delete	Dismiss

Pick Tabular-Friction as the contact interaction property.

Contact interaction property: Tabular-Friction	≚ <sup>™</sup>
Options: Interference Fit	
Contact controls: (Default)	
Active in this step	
ОК	Cancel

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

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 BendImplicit and pick **Submit > OK** and notice that job is aborted due to errors.

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

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

Pick  $\searrow$  to plot the contours on deformed shape. The analysis job runs till 127<sup>th</sup> increment completing the 0.5212 of step time. It can be observed that bottom end of staple is penetrating the cavity surface as shown in the figure below.



To fix this problem, we will modify the "Tabular-Friction" contact property such that a higher contact pressure is generated at the contact interface. It is assumed that the higher contact pressure will push the staple away from the cavity thus reducing or eliminating the penetration.

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

Select Tabular-Friction and then pick Edit to modify the selected interaction property.

Friction Contact Tabular-Friction Contact
Tabular-Friction Contact
es anne fille de la constante parte

For the normal behavior option, enter 20E6 as contact pressure for the 0 overclosure and 40E6 as contact pressure for the 1E-5 overclosure as shown in the figure below.

ress	ure-Overclosure:	Tabular	$\sim$
Const	raint enforcement	t method: Default	
rove	de data in order o	t accending overclosure	
Vote	: A negative over	closure is a positive clearance.	
Vote	A negative over Pressure	closure is a positive clearance. Overclosure	
Note	A negative over Pressure 0	closure is a positive clearance. Overclosure -1E-005	
Note	A negative over Pressure 0 20E6	closure is a positive clearance. Overclosure -1E-005 0	

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

Now we will resubmit the job.

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

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

# Solution with Abaqus/Explicit

For the explicit solution of the problem, we will use the "Staple\_Explicit" model so make sure that it is active as shown below.



# Analysis Steps

The analysis will be performed in one step. The step has already been defined. To reduce the time required to complete the analysis, the simulation will be performed at an artificially high speed. To find out an approximate lower bound on step time duration, a frequency analysis was performed in Abaqus/Standard. The lowest natural frequency of the staple was found to be 1631.8 Hz, corresponding to a time period of 0.00061 s. To make sure that inertial forces remain insignificant and quasi-static results are obtained, a time period of 0.005 s, a factor of approximately 10 times more than the time period corresponding to the lowest frequency, has been specified for the analysis.

### riangleright Soundary Conditions

Boundary conditions required for the analysis have already been defined. The punch is moved by a magnitude of 0.0089 in the y-direction with the "Punch" boundary condition. A smooth amplitude curve has been specified for this boundary condition so that displacement loading takes place with smooth acceleration. This makes sure that the quasi-static analysis is performed without generating waves. By

taking advantage of symmetry, only half of the problem is modeled. Symmetry boundary condition has been defined accordingly for the staple.

# Contact Interactions

A contact pair interaction, "Punch-Staple", has already been created. This defines contact between the punch and the staple. A contact interaction property, named "Friction", is specified for the contact interaction. In this contact interaction property, a friction coefficient of 0.05 has been specified.

We will create another interaction to define contact between cavity and the staple.

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

Pick Create to define a new interaction.

Enter Cavity-Staple as the name of the interaction.

Pick Initial in the Step field.

Select Surface-to-surface contact as type.

Create Interaction	×
Name: Cavity-Staple	
Step: Initial	
Procedure:	
Types for Selected Step	
General contact (Explicit)	
Surface-to-surface contact (Explicit)	
Self-contact (Explicit)	5.
Fluid cavity	
Fluid exchange	

We can not define a general contact interaction in the current model as the general contact algorithm in Abaqus/Explicit can only be used with three-dimensional surfaces.

#### Pick Continue to proceed.

We have already defined a surface so pick **Surfaces** on the right side of the prompt area and select the Cavity-1.Surf-Forming.

	<ul> <li>Region Selection</li> <li>Eligible Surfaces</li> <li>Surfaces below may contain factoria</li> <li>Name filter:</li> </ul>	ces.
	Name	Туре
	Cavity-1.Surf-Forming	Surface
N I	Staple-1.Surf-Bottom	Surface
	Staple-1.Surf-Top	Surface
	Highlight selections in viewp	port
	Continue	Dismiss

#### Pick Continue to proceed.

For the slave type, pick **Surface** and select the Staple-1.Surf-Bottom.

Region Selection		×	
Eligible Surfaces			
Surfaces below may contain fa	ces.		
Name filter:	÷ <b>ġ</b> •		
Name Cavity-1.Surf-Forming	<b>Type</b> Surface		
Staple-1.Surf-Bottom	Surface		
Staple-1.Surf-Top	Surface		
Highlight selections in viewp	port		
Continue	Dismiss		

Pick Continue to proceed.

Pick Friction as contact interaction property.

Edit Interaction	×
Name: Cavity-Staple	
Type: Surface-to-surface contact (Explicit)	
Step: Initial	
First surface: Cavity-1.Surf-Forming	
Second surface: Staple-1.Surf-Bottom	
Mechanical constraint formulation: Kinematic contact method	~
Sliding formulation:	
Clearance	
Contact interaction property: Friction 모 뮬	
Weighting factor   Use analysis default   Specify	
Contact controls: (Default)	
Active in this step	
OK Cancel	

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

Pick **Dismiss** to close the manager.

# ➡ Job Submission

All the information required for analysis has already 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 BendExplicit or any other suitable name for the Staple\_Explicit model.

Pick Continue and then OK.

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

After the job is completed, pick **Results** to view the results in the Visualization module.

# Comparing the Results

First we need to determine whether or not the solution obtained with Abaqus/Explicit is quasi-static. For that purpose we will compare the kinetic energy history to the internal energy history.

Make sure that BendExplicit is active in the Visualization module and pick

In the XY Data dialog box, pick **ODB history output > Continue** 

In the History Output dialog box, select ALLKE to plot the kinetic energy history and ALLIE to plot the internal energy history.

History	Output		)
Variables	Steps/Frames		
Output V	ariables		
Name filt	er:	÷ <b>ġ</b> -	
External	work: ALLWK for W	hole Model	^
Frictional	dissipation: ALLFD	for Whole Model	
Internal e	nergy: ALLIE for Wi	hole Model	_
Kinetic e	hergy: ALLKE for WI	hole Model	
Plastic di	ssipation: ALLPD fo	r Whole Model	
Strain on	ergy: ALLSE for Who	ole Model	
Suamen			
Total ene	rgy of the output s	et: ETOTAL for Whole Model	

Pick **Plot** and the graph will appear. The following figure shows the graphs for both energy histories.



It can be seen that kinetic energy is less that 1% of the internal energy of the staple. Furthermore both histories seem appropriate and reasonable (no considerable oscillations). So we can conclude that solution obtained is indeed quasi-static.

Now we will compare the simulation results obtained by using Abaqus/Standard with those obtained with Abaqus/Explicit.

Pick stoplot the contours on deformed shape.

Select Mises in the Field Output toolbar if not selected by default.

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



The contour plots show that the stress distribution is almost similar for both cases and peak stresses are also very close to each other.

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

### Exercise 27

In this exercise we will simulate the deep drawing of a cup using both Abaqus/Standard and Abaqus/Explicit.

Deep drawing is a forming process in which a blank of sheet metal is drawn into a die by a moving punch. The blank is clamped by a blank holder against the die. The deep drawing process takes place slowly such that inertial effects are negligible and it can be termed as quasi-static. Quasi-static problems can be simulated well with either Abaqus/Standard or Abaqus/Explicit. Typically, quasi static problems are solved with Abaqus/Standard but may face convergence difficulties due to contact or other nonlinearities. Such problems are a good candidate to be solved with Abaqus/Explicit as explicit procedure can resolve complicated contact problems and other discontinuous nonlinearities more easily than Abaqus/Standard.

We will simulate the forming of the cup using both Abaqus/Standard and Abaqus/Explicit and compare. By taking advantage of the symmetry, only a quarter of the blank will be considered for simulation.

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

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



The model consists of four three-dimensional parts: punch, die, holder and blank. The punch, die and holder are modeled as analytical rigid parts. The blank is assumed to be made from steel with a Young's modulus of 200 GPa and a Poisson's ratio of 0.3. A coefficient of friction of 0.1 is assumed between contacting surfaces. The blank is meshed with linear, quadrilateral shell elements with reduced integration (S4R).

In the model tree, it can be seen that there are two models. "Cup\_Implicit" model will be used to perform analysis using the Abaqus/Standard and "Cup\_Explicit" model will be used to perform analysis using the Abaqus/Explicit.



# Solution with Abaqus/Standard

As the inertia effects can be ignored, we can perform the analysis using Abaqus/Standard.

# Analysis Steps

The analysis is completed in two steps. In the first step, a load is applied on the holder and in the second step the punch is moved downwards by applying a boundary condition. 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 step. Due to large relative sliding of surfaces and friction, it is expected that the magnitude and influence of unsymmetric terms would be significant during second step. So the unsymmetric solver has been specified.

#### Boundary Conditions

Boundary conditions required for the analysis have already been defined. The punch is moved by a magnitude of 0.03 in the y-direction with the "Punch" boundary condition. By taking advantage of symmetry, only a quarter of the blank is modeled. Symmetry boundary conditions have been defined accordingly for the blank. The holder can move only in the y-direction which allows the holder to accommodate changes in the blank thickness during deep drawing process.

### Contact Interactions

Three contact pair interactions, "Die-Blank", "Punch-Blank" and "Holder-Blank" have already been defined. "Die-Blank" defines contact between the die and the blank. "Punch-Blank" defines contact between the punch and the blank. "Holder-Blank" defines contact between the holder and the blank. A contact interaction property, named "Friction", is specified for each contact interaction. In this contact interaction property, a friction coefficient of 0.1 has been specified.

### ➡ Job Submission

All the information required for analysis has already 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 Continue and then OK.

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

# Solution with Abaqus/Explicit

For the explicit solution of the problem, make sure that "Cup\_Explicit" model is active as shown below.



# Analysis Steps

The analysis will be performed in two steps. In the first step, a load is applied on the holder in a time period of 0.0001 s. This time period is considered to be long enough to avoid any dynamic effects but short enough so that it does not cause a significant impact on the run time of the analysis job. In the second step the punch is moved downwards by applying a boundary condition. For forming processes, typically, the punch speed is on the order of 1 m/s. Usually speed of forming events is increased artificially to obtain an economical solution. A time period of 0.006 s has been specified for the second step in which punch is moved by a magnitude of 0.03 in the y-direction. This gives average speed of 5 m/s to the punch.

### Boundary Conditions

Boundary conditions required for the analysis have already been defined. The punch is moved by a magnitude of 0.03 in the y-direction with the "Punch" boundary condition. A smooth amplitude curve has been specified for this boundary condition so that displacement loading takes place with smooth acceleration. This makes sure that there is no sudden impact load onto the blank at the start of the analysis. By taking advantage of symmetry, only a quarter of the blank is modeled. Symmetry boundary conditions have been defined accordingly for the blank. The holder can move only in the y-direction which allows the holder to accommodate changes in the blank thickness during deep drawing process.

### 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. The following message will appear on screen.



General contact interaction requires an analytical part to be referred by a surface for it to be considered for a contact interaction. So we need to define a surface referring the analytical rigid part. We will continue to define the contact interaction and later define a surface referring the die. We have already defined surfaces for other rigid parts therefore there is no error message for them.

Pick Yes and Edit Interaction dialog box will appear.

Pick Friction as the global contact interaction property.

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

Pick **Dismiss** to close the manager.

Now we will define a surface referring the die.

Change to **Part** module and select the Die part.

Module: Part Model: Cup_Explicit Part: Die	Die 🗸 🗸
--	---------

Open the Surface Manager by picking **Tools > Surface > Manager**.

Pick Create to define a new surface.

Pick Continue to create the surface with default name.

Pick the following surface as reference.



Pick **Done** to proceed.

Pick Brown for the side of shell 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.

Pick **Dismiss** to close the manager.

# Field Output Requests

In sheet metal forming processes, changes in the thickness of blank are of great interest. The thickness for the shell elements can be monitored with output variable STH. This information is not included by default in the ODB file. We will modify the existing field and request STH variable.

Change to Step module.

Open the Field Output Manager by picking 🛄 .

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

	Name	Step-1	Step-2		Edit
~	F-Output-1	Created Propagatec		atec	Møve Left
					Move Right
					Activate
-					Deactivate
Ste	p procedure	: Dynamic,	Explicit		
Var	riables:	Preselect	ed default	s	
Sta	tus:	Created i	n this step		

Check the STH variable (located under the Volume/Thickness/Coordinates container).

S,SVAVG,PE,PEVAVG,PEEQ,PEEQVAVG,LE,U,V,A,RF,CSTRESS,EVF,STH	
Volume/Thickness/Coordinates	'
SVOL, Integrated section volume	
EVOL, Element volume	
EVF, Void/Material volume fraction in elements (Eulerian only)	
$\Box$ ESOL, Amount of solute summed over integration points	ł
IVOL, Integration point volume	
STH, Section thickness	
COORD, Current nodal coordinates	
State/Field/User/Time	
<	

Pick **OK** to apply and exit.

Pick **Dismiss** to close the manager.

# ➡ Job Submission

All the information required for analysis has already 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 DrawExp or any other suitable name for the Cup\_Explicit model .

Pick Continue and then OK.

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

# Comparing the Results

We have simulated only quarter of the problem by taking advantage of the symmetry. Now we will mirror the simulation results about symmetry planes so that complete shape of the cup could be seen on screen.

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

General	Entity Display	Constraints	Sweep/Extrude	Mirror/Pattern
Mirror				
	and Free or to t			
Mirror C	SYS: (Global)			~
Mirror p	lanes: 🛛 XY 🗆 🛛	XZ 🛛 YZ		
	and the second second second			

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

Now we will compare the simulation results obtained by using Abaqus/Standard with those obtained with Abaqus/Explicit.

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 almost similar for both cases.

Select STH in the Field Output toolbar to show contours of shell thickness in the blank.

The following figure compares the changes in the thickness of blank. (Note: Legend is same for both plots).



The predicted results are very similar. The contour plots show that in some areas thickness of blank has increased while in other areas thickness has reduced. The changes in thickness are significant, however general contact algorithm does not account for changes in shell thickness by default. In a sheet forming analysis, thinning of a sheet significantly influences contact. We will modify the general contact interaction so that thickness changes are considered.

# Antipying Contact Interaction

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

Pick the "All\_General" interaction under column "Initial" and pick Edit.

Pick *vinder the Surface Properties tab to specify surface thickness assignments.* 

Properties	Properties	Formulation	
Surface thick	mess assignr	ments: None 📝	1
hell/Memb	rane offset a	assignments: Nor	ne 🧷
eature edge	e criteria ass	ignments: None	1
Surface smo	othing assig	nments: None 🍃	1

Pick **Blank-1.Surf-1** and pick >>> in the dialog box to transfer the selection to the list of surface thickness assignments.

Enter **THINNING** in the thickness field. (Tip: Enter T and hit enter key and system will write THINNING in the thickness field.

	Thickness		
Surface	(ORIGINAL/ THINNING/ magnitude)	Scale Factor	
Blank-1.Surf-1	THINNING	1	
<ul> <li>(in all steps)</li> <li>Note: When assignments</li> </ul>	ments overlap, more override earlier assig	recent Inments.	
	Blank-1.Surf-1 (in all steps) Note: When assignments	Blank-1.Surf-1       THINNING/ magnitude)         Blank-1.Surf-1       THINNING         Imagnitude       Imagnitude         Imagnitude<	

THINNING option is used to consider the current shell thickness instead of the original shell thickness throughout the analysis.

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

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

Pick **Dismiss** to close the manager.

Now we will submit the job. So change to **Job** module and open the Job Manager by picking **III**.

Pick **Create** and create a job named DrawExp\_Thickness or any other suitable name for the Cup\_Explicit model .

Pick Continue and then OK.

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

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



The contour plots show that the distribution of equivalent plastic strain is slightly different when changes in thickness of blank are considered.

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

#### Exercise 28

In this exercise we will simulate the v-bending of a sheet metal blank using the Abaqus/Explicit. We will compare the results obtained with contact pair approach and general contact approach.

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

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



The model consists of three three-dimensional parts: punch, die and blank. The punch and die are modeled as analytical rigid parts. The blank is assumed to be made from steel with a Young's modulus of 180 GPa and a Poisson's ratio of 0.3. A coefficient of friction of 0.1 is assumed between contacting surfaces. The blank is meshed with linear, quadrilateral shell elements with reduced integration (S4R). By taking advantage of the symmetry, only a half of the blank will be considered for simulation.

In the model tree, it can be seen that there are two models. "Bend\_General" model will be used to perform analysis using the general contact approach and "Bend\_Pairs" model will be used to perform analysis using the contact pairs.



# Analysis Steps

The analysis will be performed in one step. In this step the punch is moved downwards by applying a boundary condition. For forming processes, typically, the punch speed is on the order of 1 m/s. Usually speed of forming events is increased artificially to obtain an economical solution. A time period of 0.004 s has been specified for the step while the punch is moved by a magnitude of 0.016 in the y-direction. This gives an average speed of 4 m/s to the punch.

### Boundary Conditions

Boundary conditions required for the analysis have already been defined. The punch is moved by a magnitude of 0.016 in the y-direction with the "Punch" boundary condition. A smooth amplitude curve has been specified for this boundary condition so that displacement loading takes place with smooth acceleration. This makes sure that there is no sudden impact load onto the blank at the start of the analysis. By taking advantage of symmetry, only a half of the blank is modeled. Symmetry boundary conditions have been defined accordingly for the blank.

# Contact Interaction

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

First make sure that "Bend General" model is active as shown below.

Module: Interaction Model: Bend_General Step: Initial
Change to <b>Interaction</b> 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.
Pick Friction as the global contact interaction property.
Pick <b>OK</b> and it completes the definition of interaction.
Pick <b>Dismiss</b> to close the manager.

### ➡ Job Submission

All the information required for analysis has already 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 VBendGen or any other suitable name for the Bend\_General model .

Pick Continue and then OK.

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

Also submit a job for the Bend Pairs model, named VBendPairs, so that a comparison could be made.

# Comparing the Results

We have simulated only half of the problem by taking advantage of the symmetry. Now we will mirror the simulation results about symmetry plane.

While in the Visualization module, pick View > ODB Display Options and check the YZ option located under the Mirror/Pattern tab. This will mirror the analysis results about the YZ datum plane of the model.

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



Pick to plot the deformed shape.

The following figure compares the deformed shapes of both parts.



The deformed shape plots show that results are not similar for both cases. For clarity, we will visualize the shell thickness.

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

ODB D	isplay Options			×
General	Entity Display	Constraints	Sweep/Extrude	Mirror/Pattern
Curved Refinem	Lines & Faces ent level: Coars	e 🖌		
Elemen Color: 🗌	ts with No Resul	ts		
Feature	Angle 20			
(Used w	hen Visible Edge	es = Feature e	dges)	
Idealiza	ntions er beam profiles factor: 1			
Rende	er shell thickness factor: 1			

The thickness representation uses the thickness defined in the section assignment properties to render the shell thickness.

Pick **OK** to apply and exit.

The following figure compares the deformed shape of blank for both cases while visualizing the shell thickness.



It appears that blank is penetrating the die in the solution obtained with general contact approach. So we need to investigate it further. We will start the investigation by using job monitor.

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

Now select VBendGen job in the manager and pick Monitor

No error message is reported in the Errors tab.

Pick the Warnings tab and notice the warning about thickness reduction as highlighted below.

14659		Time	Time	Time Inc	Energy	Energy
11055	0.0036001	5.7	0.0036001	2.42552e-07	0.0113172	0.00012291
15483	0.0038001	6	0.0038001	2.4276e-07	0.006322	0.00012984
16307	0.004	6.3	0.004	2.42547e-07	0.00111638	0.00012994
						>
ors !Warning	output I	Data File	Message File	Status File		
ness used in th nGContThickR	e general con educe has bee	tact algor	ithm for eleme I. See the statu	nts in element set s file for further de	tails.	
	16307 ors !Warning 6 warning me ess used in th GContThickR	16307 0.004 ors !Warnings Output 1 6 warning messages in the ness used in the general con nGContThickReduce has been xt	16307       0.004       6.3         ors       !Warnings       Output       Data File         6 warning messages in the data (.dat)         ness used in the general contact algor         nGContThickReduce has been reduced         xt	16307       0.004       6.3       0.004         ors       !Warnings       Output       Data File       Message File         6 warning messages in the data (.dat) file.       Please ch         ness used in the general contact algorithm for element         nGContThickReduce has been reduced.       See the statuent         xt	16307       0.004       6.3       0.004       2.42547e-07         ors       !Warnings       Output       Data File       Message File       Status File         6 warning messages in the data (.dat) file.       Please check the data file f         ness used in the general contact algorithm for elements in element set         ngContThickReduce has been reduced. See the status file for further de         xt	16307       0.004       6.3       0.004       2.42547e-07       0.00111638         ors       I Warnings       Output       Data File       Message File       Status File         6 warning messages in the data (.dat) file.       Please check the data file for possible errors         ness used in the general contact algorithm for elements in element set         nGContThickReduce has been reduced. See the status file for further details.         xt

We can view the mentioned element set on screen in the Visualization module.

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

In the Results Tree, expand the Element Sets container underneath the VBendGen.odb

Pick on the WarnElemGContThickReduce and corresponding element set will highlight on the screen as shown below.



So Abaqus has reduced the thickness of all the elements of the blank.

By default, the general contact algorithm requires that the contact thickness does not exceed a certain fraction of the surface facet edge lengths or diagonal lengths. This fraction generally varies from 20% to 60% based on the geometry of the element and whether the element is near a shell perimeter. The general contact algorithm will scale back the contact thickness automatically where necessary.

To view the reduction in shell thickness in a general contact interaction, Abaqus provides CTHICK output variable. So we will request the the CTHICK variable and resubmit the job.

# ➡ Field Output Requests

CTHICK variable is not included by default in the ODB file. We will modify the existing field and request CTHICK variable.

Change to Step module.

Open the Field Output Manager by picking

Select the F-Output-1 field and pick Edit.

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

A,CSTRESS,CTHIC	K,EVF,LE,PE,PEEQ,PEEQVAVG,PEVAVG,RF,S,SVAVG,U,V,	
▼ ■ Contact		^
CSTRE	SS, Contact stresses	
	, Contact displacements	
	R, Contact slip rates in general contact	
CTAN	DIR, Contact tangent directions in general contact or c	ont
	CE, Contact force	
☐ СТНІС	K, Contact thickness in general contact or contact pair	s
	R, Slip velocity magnitude	_
□ FSLIP,	Accumulated slip displacement	
PPRES	S, Fluid pressure for pressure penetration analysis	
<		>

Pick **OK** to apply and exit.

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 VBendGen and pick **Submit > OK.** 

After the job is completed, pick **Results** to view the results in the Visualization module.

Select the CTHICK variable in the Field Output toolbar .

Now the simulation results will appear as shown below.



It can be seen that the Abaqus has reduced the shell thickness to 0.0006364 (in the section definition of the shell a thickness of 0.001 is specified.)

There are two ways to fix this issue: one is to use a coarser mesh and other is to exclude self-contact for the blank. In this exercise we will use the second method to avoid the reduction in shell thickness.

# An antipying Contact Interaction

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

Pick the "All\_General" interaction under column "Initial" and pick Edit.

Pick <sup>to</sup> specify surface exclusions.

-

💠 Edit	Interaction	×
Name:	All_General	
Type:	General contact (Explicit)	
Step:	Initial	
Conta	ct Domain	
Include	ed surface pairs:	
۹	All* with self	
OS	elected surface pairs: None	
Exclude	ed surface pairs: None 📝	
* "All" and an	includes all exterior faces, feature edges, beam segm nalytical rigid surfaces. It excludes reference points.	nents,

In the Edit Excluded Pairs dialog box it can be seen that there is no surface representing the blank. So pick  $\stackrel{\frown}{}$  to define a new surface.

Step: Initial * "All" includes all surfaces. It exclude Select Pairs	exterior faces, shell edge es reference points.	s, beam segments, and analytical i	rigid
(All*) Die-1.Surf-1 Punch-1.Surf-1	(Self) Die-1.Surf-1 Punch-1.Surf-1	First Surface	Second Surface
☐ Highlight select	ed regions	Excluded domains w included domains.	vill always override

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

Pick the blank surface as shown below.



Pick **Done** to proceed.

Pick **Both sides** to include both sides of shell in the surface definition.

Pick Surf-Blank both in the first and second columns.

<ul> <li>Edit Excluded F</li> <li>Step: Initial</li> <li>"All" includes all surfaces. It exclude</li> <li>Select Pairs</li> </ul>	Pairs exterior faces, shell e es reference points.	dges, be	am segments, and analytica Excluded Pairs	× Il rigid
(AII*)	(Self)		First	Second
Surf-Blank	Surf-Blank		Surface	Surface
Die-1.Surf-1 Punch-1.Surf-1	Surf-1 Die-1.Surf-1 -1.Surf-1 Punch-1.Surf-1		Note: Duplicate assignm Excluded domains	ents will be ignored.
☑ Highlight select	ed regions		included domains.	Cancel

Pick >>> in the dialog box to transfer the selection to the list of excluded pairs.

ep: Initial "All" includes all Irfaces. It exclude	exterior faces, shell edge es reference points.	s, beam segments, and analytica	l rigid
Select Pairs		Excluded Pairs	
(All*) Surf-Blank	First Surface	Second Surface	
Die-1.Surf-1	Die-1.Surf-1	Surf-Blank	(Self)
Punch-1.Surf-1	Punch-1.Surf-1		
		Note: Duplicate assignm Excluded domains	ents will be ignored. will always override
	<b>.</b>	included domains.	
Highlight select	ed regions		

So the selected surface is excluded from self-contact consideration.

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

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

Pick **Dismiss** to close the manager.

Now we will add keywords in the Keywords Editor to eliminate thickness reductions in regions of the model that are excluded from self-contact.

Pick Model > Edit Keywords > Bend General from the main menu and enter the following keywords after the contact property assignment.

\*CONTACT CONTROLS ASSIGNMENT, CONTACT THICKNESS REDUCTION=NOPERIMSELF

The keyword property editor is shown below.

💠 Edit keywords, Model: Ber	nd_General	×
*Contact, op=NEW		^
*Contact Inclusions, ALL EXT	ERIOR	
*Contact Exclusions Surf-Blank ,		
*Contact Property Assignme , , Friction	nt	
*CONTACT CONTROLS ASSI	GNMENT, CONTACT THICKNESS REDUCTION	N=NOPERIMSELF
**		
**		
** STEP: Step-1		
*Step, name=Step-1, nlgeor	n=YES	
*Dynamic, Explicit , 0.004		
*Rulk Viscosity		~
Block: Add After Rem	ove Discard Edits	
OK	Discard All Edits	Cancel

These keywords eliminate thickness reductions in regions of the model that are excluded from self-contact and at all shell perimeters.

Pick **OK** to apply and exit.

Now we will resubmit the job.

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



Select VBendGen and pick Submit > OK.

After the job is completed, pick **Results** to view the results in the Visualization module.

Select the CTHICK variable in the Field Output toolbar .

Now the simulation results will appear as shown below.



It can be seen that thickness of the blank remains unaffected.

The following figure compares the deformed shape of blank obtained by using contact pair approach with general contact approach while visualizing the shell thickness.



It can be seen that result are very similar when thickness reduction does not take place in general contact approach.

Note: The general contact and contact pairs algorithms in Abaqus/Explicit use completely separate implementations with many key differences in the designs of the numerical algorithms.

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

#### **Exercise 29**

In this exercise we will simulate the bending of an extrusion using the Abaqus/Explicit. We will use both the contact pair and the general contact approach.

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

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



The model consists of four three-dimensional parts. Two cylinder located at the bottom of extrusion act as supports while the third cylinder, located at top of the extrusion, acts as punch. The punch and supports are modeled as analytical rigid parts. The extrusion is assumed to be made from an alloy with a Young's modulus of 70 GPa and a Poisson's ratio of 0.31. A coefficient of friction of 0.2 is assumed for the self-contact of extrusion and 0.05 for the contact between the rigid cylinders and the extrusion. The extrusion is meshed with linear, quadrilateral shell elements with reduced integration (S4R).

In the model tree, it can be seen that there are two models. "Bend\_General" model will be used to perform analysis using the general contact approach and "Bend\_Pairs" model will be used to perform analysis using the contact pairs.

# Analysis Steps

The analysis will be performed in one step under quasi-static loading conditions. In this step the punch is moved downwards by applying a boundary condition. A time period of 0.05 s has been specified for this step while the punch is moved by a magnitude of 0.125 in the y-direction.

# Boundary Conditions

Boundary conditions required for the analysis have already been defined. The punch is moved by a magnitude of 0.125 in the y-direction with the "Punch" boundary condition. A smooth amplitude curve has been specified for this boundary condition so that displacement loading takes place with smooth acceleration. This makes sure that there is no sudden impact load onto the extrusion at the start of the analysis.



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

First make sure that "Bend General" model is active as shown below.

	Module: Interaction	Model:	Bend_General Y	Step: 🔄 Initial	~
Change to	Interaction module and op	pen the Intera	nction Manager by pi	icking 🛄.	
Pick Creat	te to define a new interaction	on.			
Enter All_	General as the name of the	e interaction.			
Pick Initia	l in the Step field.				

Select General contact and pick Continue to proceed.

Pick Friction\_05 as the global contact interaction property.

"Friction\_05" interaction property defines a friction coefficient of 0.05. As we intend to specify a friction coefficient of 0.2 for the self contacting surfaces of extrusion, we will define an individual property assignment. Individual property assignments are used to assign different contact properties to individual surface pairs.

Pick *for the create individual property assignment.* 

Contact Properties	Surface Contact Properties Formulation		t tion		
Global prop	perty assignme	ent: Frict	ion_05		~ 묩
			_	-	100001
Individual p	property assign	nments:	None 🔎	2	Tanan
Individual p	property assign	nments:	None 🗾	2	

The surface required to specify the property assignment have been defined beforehand.

Pick Extrusion-1.Surf-All both in the first and second column to define the surface pairings.

In the third column, select the "Friction\_2" as the contact property.

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

tep: Initial Select Pairs and Cont	act Property		
(Global) Cylinder-1.Surf-1 Cylinder-2.Surf-1 Cylinder-3.Surf-1	(Self) Cylinder-1.Surf-1 Cylinder-2.Surf-1 Cylinder-3.Surf-1	Friction_05 Friction_2	
Extrusion-1.Surf-All	Extrusion-1.Surf-All		~
Highlight selected r	egions		

Notice that system has assigned the property for the self-contact of selected surface.

First Surface	Second Surface	Property Assigned
trusion-1.Surf-All	(Self)	Friction 2
<ul> <li>(in all steps)</li> <li>te: When assignment assignments over</li> </ul>	nts ove <mark>r</mark> lap, more erride earlier assig	recent nments

To assign a property for self-contacting surface, select either the same surface name or (Self) in the second column.

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.

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

Pick **Dismiss** to close the manager.

# Contact Pair Approach

Make sure that "Bend Pairs" model is active as shown below.

Module: Finteraction	Model:	Bend_Pairs	Step:	lnitial	~
	- Supervised by	And a second	a construction of		

Three contact pair interactions, "Cylinder1-Extrusion", "Cylinder2-Extrusion" and "Punch-Extrusion" have already been defined. These interactions define contact between the analytical rigid parts and the extrusion. Now we will create another interaction defining the self-contact of extrusion surfaces.

Change to Interaction module and open the Interaction Manager by picking

Pick Create to define a new interaction.

Enter Extrusion-Extrusion as the name of the interaction.

Pick **Initial** in the Step field.

Select Self-contact as type.

💠 Crea	ate Interaction	×
Name:	Extrusion-Extrusion	
Step:	Initial 🖌	
Proced	ure:	
Types	for Selected Step	
Gener	ral contact (Explicit)	
Surfac	ce-to-surface contact (Explicit)	
Self-c	ontact (Explicit)	
Fluid	cavity	
Fluid	exchange	

A self-contact interaction allows to define contact between different areas of a single surface.

Pick Continue to proceed.

A surface has already been defined, so pick **Surfaces** on the right side of the prompt area and select the Extrusion-1.Surf-All.

Self-contact models contact interaction between a single surface and itself by specifying only a single surface.

Pick Continue and Edit Interaction dialog box will appear.

Pick Friction\_05 as the contact interaction property.

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

Pick **Dismiss** to close the manager.

# ➡ Job Submission

All the information required for analysis has already 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 ExtrusionGen for the Bend\_General model.

Pick Continue and then OK.

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

Also submit a job for the Bend\_Pairs model, named ExtrusionPairs. Notice that job is aborted due to errors.

# ➡ Diagnosing the error

We will start the investigation of the error by using job monitor.

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

Now select ExtrusionPairs job in the manager and pick Monitor

An error message appears in the Errors tab about the contact definition as shown in the figure below.

Step	Increment	Att	Severe Discon Iter	Equil Iter	Total Iter	Total Time/Freq	Step Time/LPF	Time/LPF Inc
								>
og !E	rrors !Warnir	ngs Out	put   Data Fi	le Messag	ge File   Stat	tus File		8
.og !E Contact out this s	pair references surface/node-b	surface/ ased sur	put Data Fi node-based face/analytic	le Messag surface/ana al rigid surf	ge File   Sta Ilytical rigid ace cannot	surface assemb be used with *co	ly_extrusion-1_ ontact pair. Che	_surf-all
og !E Contact out this s previous	rrors Warnir pair references surface/node-b warning messa	ngs Out surface/ based sur ages for t	put Data Fi node-based face/analytic this surface to	le Messag surface/ana al rigid surf o find the ca	ge File Star Ilytical rigid ace cannot l ause.	tus File surface assemb be used with *co	ly_extrusion-1_ ontact pair. Che	_surf-all eck
og !E Contact out this : previous	rrors Warnir pair references surface/node-b warning mess pair references	ngs Out surface/ based sur ages for t surface/	put Data Fi node-based face/analytic this surface to node-based	le Messag surface/ana al rigid surf o find the ca surface/ana	ge File Star Ilytical rigid ace cannot l ause.	surface assemb be used with *co surface bu	ly_extrusion-1_ ontact pair. Che It this	surf-all eck
og !E Contact but this : previous Contact	rrors Warnin pair references surface/node-b warning mess pair references pair references	ngs Out surface/ based sur ages for t surface/	put Data Fi node-based face/analytic this surface to node-based	le Messag surface/ana al rigid surf o find the ca surface/ana	ge File Star Ilytical rigid acce cannot l ause.	surface assemb be used with *co surface bu	ly_extrusion-1_ ontact pair. Che ut this	surf-all eck

From this error message, it is not clear what is the problem in the definition of contact interaction. So we will look at the warning messages as suggested in the error message.

Pick the **Warnings** tab and notice the warning about t-intersections in the selected surface as highlighted below.

Step	Increment	Att	Severe Discon Iter	Equil Iter	Total Iter	Total Time/Freq	Step Time/LPF	Time/LP Inc
								3
	I Marine in		200 <u>20</u> 0 200					
og !E	rrors : warnin	ngs Out	put Data Fi	le Messag	je File   Sta	tus File		
og !E ure that	t there are no s	uch jump	put Data Fi is. All jumps i	le Messag n displacen	e File   Stat nents across	steps are ignor	ed	
og !E ure that	t there are no s	uch jump	put Data Fi is. All jumps i	le Messag n displacen	e File Stat	steps are ignor	ed	
og <b>!</b> E ure that Oouble-	t there are no s	uch jump	put Data Fi s. All jumps i extrusion-1_s	le Messag n displacen surf-all cont	e File Stan	steps are ignor t edges which a	ed are shared by n	nore than
og !E ure that ouble- wo surfa	t there are no s sided surface a ace facets. This	uch jump	put Data Fi s. All jumps i extrusion-1_s cannot be use	le Messag n displacen surf-all cont ed with *cor	e File Sta nents across ains elemen ntact pair or	steps are ignor steps are ignor nt edges which a *tie unless all t-	ed are shared by n intersections a	nore than ire
og !E ure that oouble- wo surfa emoveo	sided surface a ace facets. This J. There are 199	uch jump assembly surface o 9 nodes o	put Data Fi s. All jumps i extrusion-1_: cannot be use on the edges	le Messag n displacen surf-all cont ed with *cor forming the	ne File Star nents across cains elemer ntact pair or t-intersecti	steps are ignor steps are ignor at edges which a *tie unless all t- ons.	ed are shared by n intersections a	nore than Ire
og !E ure that ouble- wo surfa emoved	sided surface a ace facets. This J. There are 199	igs Out uch jump assembly surface o 9 nodes o	put Data Fi s. All jumps i extrusion-1_s cannot be use on the edges	le Messag n displacen surf-all cont ed with *cor forming the	re File Star nents across ains elemen ntact pair or t-intersecti	steps are ignor at edges which a *tie unless all t- ons.	ed are shared by n intersections a	nore than ire

So the analysis is aborted due to presence of t-intersections in the Extrusion-1.Surf-All surface.

The contact pair algorithm in Abaqus/Explicit does not allow T-intersections in shells. In a self-contact interaction, self-contacting surface act as both master and slave surfaces, therefore, if a restriction applies to contact pair, it also applies to self-contact.



# An antipying Contact Interaction

To avoid this error, we will not use a self-contact interaction. Instead contact pair interactions will be defined in such a way that no t-intersection occurs in the surface definition.

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

Select Extrusion-Extrusion and pick Delete.

To create a new interaction, pick Create.

Enter **Top-Right** as the name of the interaction.

Pick **Initial** in the Step field.

Select Surface-to-surface contact as type.

Pick Continue to proceed.

We have already defined a surface so pick **Surfaces** on the right side of the prompt area and select the **Extrusion-1.Surf-Top**.

Pick Continue to proceed.

For the slave type, pick Surface and select the Extrusion-1.Surf-Right.

Pick **Continue** to proceed and Edit Interaction dialog box will appear. In this dialog box it can be seen that Kinematic contact method is selected by default

+ Edit Interaction	×
Name: Top-Right	
Type: Surface-to-surface contact (Explicit)	
Step: Initial	
<ul> <li>First surface: Extrusion-1.Surf-Top </li> <li>Second surface: Extrusion-1.Surf-Right </li> </ul>	
Mechanical constraint formulation: Kinematic contact method	>
Sliding formulation:      Finite sliding      Small sliding	

The kinematic contact algorithm uses a kinematic predictor/corrector contact algorithm to strictly enforce contact constraints and no penetrations are allowed. In addition to kinematic contact method, Abaqus/Explicit provides penalty contact method which enforces contact constraints weakly. The penalty contact algorithm results in less stringent enforcement of contact constraints than the kinematic contact algorithm.

Pick **Friction\_2** as contact interaction property.

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

Similarly create Top-Left, Top-Bottom, Bottom-Right and Bottom-Left interactions where corresponding surfaces have already been defined.

After creating all these interactions, we can resubmit the job.

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

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

Pick **Dismiss** to close the manager.

# ➡ Comparing the Results

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

Pick stoplot the contours on deformed shape.

Select Mises in the Field Output toolbar if not selected by default.

The following figure compares the distribution of von Mises stresses.



The figures show that results are not similar for both cases. This is due to the fact that the general contact algorithm in Abaqus/Explicit uses penalty enforcement method while the contact pair interactions uses kinematic contact method by default.

General contact does not support kinematic constraint enforcement so for comparison purposes, we will change the contact constraints enforcement method to "Penalty contact method".

The following figure shows the distribution of von Mises stresses when penalty contact method is used for all the contact pair interactions in the Bend\_Pairs model.



It can be seen that results are very close to those obtained with general contact interaction.

It can be concluded that general contact greatly simplifies the process of contact definition and has only few limitations.

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

# Exercise 30

In this exercise we will simulate the metal cutting using the contact pair approach in Abaqus/Explicit.

Abaqus provides element deletion functionality to model erosion due to material failure. Material failure refers to the complete loss of load-carrying capacity that results from progressive degradation of the material stiffness. A material model can contain multiple failure mechanism acting simultaneously on the specified elements. Once the material stiffness is fully degraded, elements can be removed from the calculations. We will use this functionality to simulate metal cutting.

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

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



The model consists of two two-dimensional parts: tool and workpiece . The tool is modeled as an analytical rigid part. The workpiece is assumed to be made from steel with a Young's modulus of 205 GPa and a Poisson's ratio of 0.3. The material definition also includes a failure model. Abaqus/Explicit removes the elements from the mesh as they fail. A coefficient of friction of 0.1 is assumed for the contacting surfaces. The workpiece is meshed with bilinear plane strain quadrilateral elements with reduced integration (CPE4R).

# Analysis Steps

The analysis will be performed in a dynamic, explicit step under quasi-static loading conditions. In this step the tool is moved into the workpiece material by applying a boundary condition. A time period of 0.004 s has been specified for this step while the punch is moved by a magnitude of 0.004 in the x-direction.



Boundary conditions required for the analysis have already been defined. The punch is moved by a magnitude of 0.004 in the x-direction with the "Tool" boundary condition. A smooth amplitude curve has been specified for this boundary condition so that displacement loading takes place with smooth acceleration.

# Contact Interaction

We will use a node-based surface as slave to model surface erosion. We can use either the general contact or contact pair algorithm for that purpose. It is quite easy to use contact pair interaction for a node-based surface so we will create a contact pair interaction to define the contact between tool and workpiece.

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

Pick Create to define a new interaction.

Enter **Tool-Workpiece** as the name of the interaction.

Pick Initial in the Step field.

Select Surface-to-surface contact as type.

Pick Continue and select the rigid surface as master.

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



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

For the slave type, pick Node Region and select the region highlighted in the figure below.



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

Pick Friction as the contact interaction property.

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

Pick **Dismiss** to close the manager.

# ➡ Job Submission

All the information required for analysis has already 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 MetalCutting.

Pick Continue and then OK.

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

# Postprocessing

Select MetalCutting job in the manager and pick Results

Pick 🤽 to plot the contours on deformed shape.

Select Mises in the Field Output toolbar if not selected by default.

The following figure shows the distribution of von Mises stresses after completion of 0.0016 s of step time.



It appears that some of the elements are distorting extensively. In reality these are the failed elements which have not been removed from the display although they have been removed for computation purposes. We will request the STATUS output variable so that failed elements can be excluded from the display.

# ➡ Field Output Request

Change to Step module.

Open the Field Output Manager by picking 🛄 .

Select the F-Output-1 field and pick Edit.

Check the **STATUS** variable (located under the State/Field/User/Time container).

S,SVAVO	5,PE,PEVAVG,PEEQ,PEEQVAVG,LE,U,V,A,RF,CSTRESS,EVF,STATUS	
,	State/Field/User/Time	
	SDV, Solution dependent state variables	
	FV, Predefined field variables	
	□ MFR, Predefined mass flow rates	ļ
	UVARM, User-defined output variables	
	EMSF, Element mass scaling factor	
	DENSITY, Material density	
	DENSITYVAVG, Volume-averaged material density (Eulerian on	
	STATUS, Status (some failure and plasticity models; VUMAT)	
<	¬ · · · · · · · · · · · · · · · ·	

Pick **OK** to apply and exit.

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 MetalCutting and pick **Submit > OK.** 

After the job is completed, pick **Results** to view the results in the Visualization module.

# Postprocessing

Pick stoplot the contours on deformed shape.

Select Mises in the Field Output toolbar if not selected by default.

The following figure shows the distribution of von Mises stresses after completion of the simulation. It can be seen that no failed element appears in the display.



It is also of interest to know which elements have failed in the undeformed configuration. So first we will plot the stress contours on undeformed shape and then remove the failed elements from display.

Pick <sup>Leg</sup> to plot contours on undeformed shape.



From the main menu bar, select **Result > Field Output** and Field Output dialog box will appear.

Click the **Status Variable** tab and notice that system removes elements which have a STATUS value of less than 0.5 but this is not applied to undeformed state by default.

Check the **Apply to undeformed state** option as shown below.

💠 Field Output				×
Step/Frame				
Step: 1, Step-1				
Frame: 18 📲				
Primary Variable D	eformed Variable	Symbol Variable	Status Variable	Stream Variable
Output Variable				
Use status variab	le Apply to und	eformed state		
	s with results:			
	s with results.			
Name Desc	ription (* indicates	complex)		^
CPRESS ASS CO	ontact stress variable	es at nodes		
LE LO	aprithmic strain con	nonents at integr	ation points	
DE DIS	astic strain compone	ants at integration	noints	
PEEO Eo	uivalent plastic stra	in at integration of	points	
RF Re	action force at nod		511125	
RM3 Re	action moment at n	lodes		
S Str	ress components at	integration points		
STATUS Sta	atus of the element	for whole element		<b>v</b>
Invariant		Component		
	March 1999 (1997)			
Section Points	Remove elements:	← Min: 3.402	82E+038 Max:	0.5
OK		Apply		Cancel

Pick **OK** to apply and exit.

The following figure shows the distribution of von Mises stresses in the undeformed shape at the end of the simulation. It can be seen that failed elements do not appear in the display.



# ALE Adaptive Meshing

In the contour plots on the deformed shape, it can be seen that mesh quality is not satisfactory for the analysis. To maintain a high-quality mesh throughout the analysis, we will introduce ALE adaptive meshing technique.

So change to **Step** module.

From the main menu bar, select **Other > ALE Adaptive Mesh Domain > Edit > Step-1** and Edit ALE Adaptive Mesh Domain dialog box will appear.

Select the Use the ALE adaptive mesh domain below option.

Activate the ALE Adaptive Mesh Controls option as shown below.

🜩 Edit ALE Adaptive Mesh Domain	;	X
Step: Step-1		
O No ALE adaptive mesh domain fo	r this step	
• Use the ALE adaptive mesh doma	in below:	
Region: (Not selected)  🗟		
ALE Adaptive Mesh Controls:	Ada-1	+
Frequency:	10	
Remeshing sweeps per increment	1	
Initial remeshing sweeps: 🖲 Defa	ult O Value:	
ОК	Cancel	

To define the region for ALE adaptive mesh, pick  $\mathbb{R}$ .

Select the region highlighted in the figure below and pick **Done**.



Pick **OK** to apply and exit.

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 MetalCuttingALE.

Pick Continue and then OK.

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

After the job is completed, pick **Results** to view the results in the Visualization module.

The following figure shows the distribution of von Mises stresses after completion of the simulation.



The results obtained with ALE adaptive meshing show improved mesh quality.

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

### **Exercise 31**

In this exercise we will simulate the erosion of a plate due to impact of a projectile using the general contact approach in Abaqus/Explicit.

The material definition for the plate contains multiple failure models. So the Abaqus removes elements from the mesh as they fail due to the impact of the projectile.

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

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



The model consists of two three-dimensional parts: plate and projectile. The projectile is modeled as an analytical rigid part. The plate is assumed to be made from an alloy with a Young's modulus of 70 GPa and a Poisson's ratio of 0.33. The material definition also includes ductile and shear damage models. Abaqus/Explicit removes the elements from the mesh as they fail. The plate is meshed with 8-node, brick elements with reduced integration (C3D8R).

# Analysis Steps

The analysis will be performed in one step under dynamic loading conditions. In this step the projectile moves into the plate with an initial velocity. A time period of 0.000015 s has been specified for this step.

# Boundary Conditions

Boundary conditions required for the analysis have already been defined. The motion of projectile is restrained along all directions except the x-axis. A predefined field has been defined to specify an initial velocity of 700 m/s for the projectile along x-axis. Top and bottom faces of the plate are restrained to move in any direction.

# Contact Interaction

We will use an element-based surface as slave to model surface erosion. This surface will include all faces of the elements which could come into contact with the projectile. This is necessary so that the surface topology evolves to match the exterior of elements that have not failed. For element-based surface, exterior faces are initially active, and interior faces are initially inactive. Once an element fails, its faces are removed from the contact domain, and any interior faces that have been exposed are activated. Contact pair algorithm does not allow element-based surface so we will define a general contact interaction to model surface erosion.

An element set "ElemErode" has already been defined which contains all the elements in the plate that could come into contact with the impacting projectile. We will create a surface containing all the interior faces of these elements.

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.

Pick NoFric as the global contact interaction property.

Notice that **All\* with self** is selected by default. This is not sufficient because we also need to include the interior faces of the elements in the plate to define the contact domain. Due to limitations of Abaqus/CAE, we will do so by editing the input file.

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

Pick **Dismiss** to close the manager.

# ➡ Field Output Request

We will request the STATUS output variable so that failed elements can be excluded from the display.

Change to Step module.

Open the Field Output Manager by picking 🛄 .

Select the F-Output-1 field and pick Edit.

Check the STATUS variable (located under the State/Field/User/Time container).

S,SVA	/G,PE,PEVAVG,PEEQ,PEEQVAVG,LE,U,V,A,RF,CSTRESS,EVF,STATUS	
		1
▼ 🗉	J State/Field/User/Time	
	SDV, Solution dependent state variables	
	FV, Predefined field variables	
	MFR, Predefined mass flow rates	
	UVARM, User-defined output variables	
	EMSF, Element mass scaling factor	
	DENSITY, Material density	
	DENSITYVAVG, Volume-averaged material density (Eulerian on	

Pick **OK** to apply and exit.

Pick **Dismiss** to close the manager.

# ➡ Job Submission

Now we will create a new job and write input file for the analysis.

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

Pick Create and create a job named ProjectileImpact.

Pick Continue and then OK.

Pick Write Input to create the job input file for the analysis.

Name	Model	Туре	Status	Write Input
ProjectileImpact	Impact	Full Analysis	None	Data Check
				Submit
				Continue
				Monitor
				Results
				Kill
Consta	T all a	Comu		Palata Diamina

# ➡ Editing the Input File

Now we will edit the input file using Notepad++ although any other editor can be used for that purpose. Go to the working directory and open the ProjectileImpact.inp

Add the following lines before the definition of contact interaction.

#### \*SURFACE, NAME=SurfErode, TYPE=ELEMENT ElemErode, INTERIOR

These lines create a new surface, named "SurfErode", containing all the interior faces of the ElemErode element set. So the faces of the specified elements that are not on the exterior surface of the model will be included in the surface definition. This is shown schematically in the following figure.



After the addition of these lines, input file will appear as shown below.

🖹 *E:\Aba	aqus\ContactProblems\Projectile\ProjectileImpact.inp - Notepad++ -	×
File Edit	Search View Encoding Language Settings Macro Run Plugins Window ?	Х
6 🖻 🗎	🖻 🗟 🕞 😂   X 🛍 🛍   🤉 😋   8 🧤   🍳 🔍   💁 🖘 1 🎼 🗷 🔊	
🔚 Projectile Im	ipact inp 🗹	
1811	Projectile-1 Set-RP, 3, 0.	^
1812	**	
1813	*SURFACE, NAME=SurfErode, TYPE=ELEMENT	
1814	ElemErode, INTERIOR	
1815	** INTERACTIONS	
1816	**	
1817	** Interaction: All General	
1818	*Contact, op=NEW	
1819	*Contact Inclusions, ALL EXTERIOR	
1820	*Contact Property Assignment	
1821	, , NoFric	
1822	**	
<		>
length : 90,	211 lines : 1,8 Ln : 1,812 Col : 1 Sel : 0   0 Windows (CR LF ANSI	INS

Now we will add data lines for the contact interaction but it is not allowed after the ALL EXTERIOR. So we will delete this keyword and add the following three lines for the contact inclusions

, ,SurfErode SurfErode,

First line defines the self-contact for the default all-inclusive surface. In other words, it specifies contact between every exterior face in the model. The second line defines contact between the default all-inclusive surface and SurfErode. In other words, it specifies contact between every exterior face and SurfErode. The third line defines self-contact between the interior faces defined by SurfErode surface.

After the editing and addition of these lines, input file will appear as shown below.



Pick File > Save to save the changes in the input file.

Now open the Abaqus command window and set the directory to the folder where input file is located.

Enter following to run the analysis.

abaqus job=ProjectileImpact interactive

Notice that job completes successfully.

#### ➡ Postprocessing

Select ProjectileImpact job in the manager and pick Results

Pick stoplot the contours on deformed shape.

Select Mises in the Field Output toolbar if not selected by default.

The following figure shows the distribution of von Mises stresses after the impact of the projectile.



It is of interest to know the projectile's loss of kinetic energy in the impact analysis. So we will plot the history of kinetic energy.

So pick and Create XY Data dialog box will appear.

#### Pick **ODB history output > Continue**

In the History Output dialog box, select ALLKE.

History	Output		3
Variables	Steps/Frames		
Output V Name filt	ariables er:	÷	
External v Frictional Internal e	work: ALLWK for W dissipation: ALLFD energy: ALLIE for W	hole Model 9 for Whole Model hole Model	^
Kinetic er	nergy: ALLKE for W	hole Model	
Plastic di	ssipation: ALLPD fo	r Whole Model	
Strain en	ergy: ALLSE for Wh	ole Model	
Total ene	rgy of the output s	et: ETOTAL for Whole Model	
		a Mileste Mestel	

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



Approximately 47% of the initial kinetic is absorbed by the impact.

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