Exercise 25

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

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



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



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

➡ Defining Step

We assume that load is applied slowly such that inertia effects can be neglected. So analysis will be performed using the Static, General procedure.

Change to Step module.



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

Create Step		×
Name: Step-1		
Insert new step a	fter	
Initial		
Step-1		
Procedure type:	General	\sim
Coupled thermal	-electrical-structural	^
Direct cyclic		
Dynamic, Implicit	t	
Geostatic		
Soils		
Static, General		
Static, Riks		
Visco		~
Continue	Cancel	

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

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

Fait Step					×
Name: Step-1					
Type: Static, Gei	neral				
Basic Increme	ntation	Other			
Type: Autom	atic O Fix	ked			
	har of in	rements: 100		T	
Maximum num	per or inc	rements. Too		-	
Maximum num	Initial	Minimum	Maximum		

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

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

Defining Contact Interaction

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

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



Pick to create a new interaction.

Enter Plate-Indenter as the name of the interaction.

Pick **Initial** in the Step field.

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

Create Interaction	×
Name: Plate-Indenter	
Step: Initial	
Procedure:	
Types for Selected Step	
General contact (Standard)	
Surface-to-surface contact (Standard)
Self-contact (Standard)	
Fluid cavity	
Fluid exchange	
XFEM crack growth	
Cyclic symmetry (Standard)	
Elastic foundation	
Actuator/sensor	
Continue	Cancel

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

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



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

For the slave type, pick Surface.

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

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

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

Eligible Surfaces			
Surfaces below may cont Name filter:	tain faces.		
Name	Туре		
Indenter-1.Surf-1	Surface	- 10	
☑ Highlight selections in	viewport		

Pick Continue and Edit Interaction dialog box will appear.

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

To define the contact interaction property, pick ¹/₂ in the Edit Interaction dialog box.

Contact interaction property:	✓ B
Options: Interference Fit	
Contact controls: (Default)	
☑ Active in this step	
ОК	Cancel

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

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

Name: DefaultProp	
Туре	
Contact	^
Film condition	
Cavity radiation	
Fluid cavity	
Fluid exchange	
Acoustic impedance	

Pick Continue and Edit Contact Property dialog box will appear.

Handle state of the second state of the	~
Name: IntProp-1	
Contact Property Options	
<u>M</u> echanical <u>T</u> hermal <u>E</u> lectrical	4
Mechanical Thermal Electrical	<i>y</i>

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

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

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

Contact interaction property: DefaultProp	~ 7
Options: Interference Fit	
Contact controls: (Default)	
Active in this step	
OK	Cancel

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

🔿 Defining Load

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

So change to Load module.

Module: Load	Model: Model-1	Step: 🗧 Initial 🗠

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

Select Pressure in the Types field and pick Continue.



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



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

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

Boundary Conditions

Boundary conditions required for the analysis have already been defined. To review all the boundary conditions pick while the Load module is active as shown in the figure below.

Module: 🗄 Load	✓ Model: - Model-1	✓ Step: Step-1 ✓
ц 💼		

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

The boundary condition "Indenter_Symm" applies symmetry boundary condition to the edge of the indenter lying on symmetry plane.

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

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

Pick **Dismiss** to close the manager.

➡ Job Submission

All the information required for analysis has been set up in the model. Now we can submit the job for analysis.

So change to **Job** module and pick **III** to open the job manager.

Module: Job	✓ Model: ♥ Model-1	Step: Step-1	~
2			
€∎ 🚍			

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

Create Job	×
Name: Indent	ation
Source: Mode	I ~
Model-1	
Continue	Cancel

Pick Continue and then OK.

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

Name	Model	Туре	Status	Write Input
Indentation	Model-1	Full Analysis	None	Data Check
				Submit
				Continue
				Monitor
				Results
				Kill

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

Postprocessing

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



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

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

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

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

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

h	×
la <mark>n</mark>	
Cancel	Tin
	Cancel

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

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

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

Pick by shortest distance to pick the desired edges.

 by shortest distance
individually by feature edge
by shortest distance

Pick the edge shown in the figure below.



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



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

Pick to access the Create XY Data dialog box.

Pick Path > Continue

💠 Create XY Da	ata X
Source	
O ODB history	output
O ODB field ou	utput
O Thickness	
O Free body	
O Operate on 2	XY data
O ASCII file	
O Keyboard	
Path	
Continue	Cancel

In the "XY Data from Path" dialog box, pick Field Output.

Step:	1, Step-1		
Frame:	7 Step/Frame		
Fi <mark>eld</mark> ou	tput variable: S,	Mises (Avg: 75%)	Field Output
Note: R	esult option set calculate resul	tings will be applied t values for the curr	l rent

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

Step: 1,	Step-1
Frame: 7	Step/Frame
Field outp	out variable: CPRESS (Not averaged) Field Outpu
Note: Re to	sult option settings will be applied calculate result values for the current

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



Comparing to Analytical Results

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

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

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

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

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

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

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

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

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



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

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

Exercise 26

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

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

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

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



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

➡ Defining Steps

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

Change to Step module.

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

	Name	Proce	edure	- 1	ligeom	Time	
V	Initial	(Initia	al)	1	N/A	N/A	

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

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

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

🜩 Edit Step			×
Name: Step-1 Type: Static, General			
Basic Incrementation	Other		
Type: Automatic Fix	red		
Maximum number of incr	ements: 100		
Initial	Minimum	Maximum	

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

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

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

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

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

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

Defining Contact Interaction

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

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

Pick Create to define a new interaction.

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

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

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

💠 Cre	ate Interaction	×
Name:	Fuse-Holder	
Step:	Initial 🖌	
Proced	ure:	
Types	for Selected Step	
Gener	al contact (Standard)	
Surfac	ce-to-surface contact (Standard)	
Self-co	ontact (Standard)	
Fluid o	cavity	
Fluid e	exchange	

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

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

Pick Continue and select the rigid surface as master.

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



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

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

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

Eligible Surfaces	
Conference bedress and a starting for some	
Surfaces below may contain faces.	
Name filter:	
Name Type	
Holder-1.Surf-1 Surface	

Pick Continue and Edit Interaction dialog box will appear.

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

Contact interaction property: NoFric	¥ #
Options: Interference Fit	
Contact controls: (Default)	
☑ Active in this step	
ОК	Cancel

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

Edit Interaction	×
Name: Fuse-Holder	
Type: Surface-to-surface contact (Standard)	
Step: Initial	
Master surface: (Picked)	
Sliding formulation: Finite sliding Small sliding	
Discretization method: Surface to surface	

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

			1	
Name	Initial	Step-1	Step-2	Edit
✓ Fuse-Holde	r Created	Propagated	Propagated	Move Left
				Move Righ
				Activate
				Deactivate
Step procedure Interaction typ	e: Surface- tus: Created	to-surface con	tact (Standard)	
Interaction stat	cubi cicatea			

Pick **Dismiss** to close the manager.

Boundary Conditions

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

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

Module: 🗄 Load	✓ Model: [‡] Model-1	✓ Step: Step-1	~
ц 💼			

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

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

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

1		minuter	Step-1	Step-2	Edit
~	Fuse	Created	Propagated	Propagated	Moundate
~	Holder-Fasten	Created	Propagated	Propagated	IVIOVE LEI
Y	Holder-Symm	Created	Propagated	Propagated	Move Righ
					Activate
					Deactivate
Step	procedure:	St	atic, General		
Bou	ndary conditio	n type: Di	splacement/Ro	tation	
Bou	ndary conditio	n status: Pr	opagated from	a previous step	

Enter -0.0211 in the U2 field as shown below.

💠 Edit Bound	ary Condition	×
Name: Fuse		
Type: Displ	acement/Rotation	n
Step: Step-	1 (Static, General))
Region: (Picke	d)	
CSYS: (Globa	1)	
Distribution:	Iniform	
☑ U1:	0	
* ☑ U2:	-0.021 1	
⊡ UR3:	0	radians
Amplitude:	(Ramp)	~ Pv
* Modified in t Note: The dis mainta	his step placement value ned in subsequer	will be ht steps.
OK		Cancel

Pick **OK** to apply and exit.

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

	Name	Initial	Step-1	Step-2	Edit
~	Fuse	Created	Modified	Propagated	Moundate
~	Holder-Fasten	Created	Propagated	Propagated	IVIOVE Lei
1	Holder-Symm	Croated	Dropagated	D 1 1	1 A
~	Holder-Symm	Created	Propagated	Propagated	Move Righ
V	rioider-Symm	Created	Propagated	Propagated	Activate
V	Holder-Symm	Created	Propagated	Propagated	Activate
Ste	p procedure:	St	atic, General	Propagated	Activate
Ste Bou	p procedure: undary conditio	St n type: Di	atic, General	propagated	Activate

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

💠 Edit Bound	dary Condition	×
Name: Fuse		
Type: Displ	acement/Rotation	
Step: Step	-2 (Static, General)	
Region: (Pick	ed)	
CSYS: (Globa	1)	
Method:	Specify Constraints	
Distribution:	Uniform	
☑ U1:	0	
* 🗆 U2:		
UR3:	0	radians
Amplitude:	(Ramp)	~ 14
* Modified in Note: The dis mainta	this step splacement value will b ined in subsequent ste	e ps.
ОК	Ca	ncel

Pick **OK** to apply and exit.

Pick **Dismiss** to close the manager.

→ Job Submission

All the information required for analysis has been set up in the model. Now we can submit the job for analysis.

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

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

Pick Continue and then OK.

Pick Submit to submit the job for analysis.

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

Postprocessing

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



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

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

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

ODB C	isplay Options			>
General	Entity Display	Constraints	Sweep/Extrude	Mirror/Pattern
General Note: N ir	Sweep lo sweepable el n the current mo	ements exist del.		
Extrude				
Extrue Dept	de elements h: 0.01			

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

Solving Nonlinear Problems

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

General	Entity Display	Constraints	Sweep/Extrude	Mirror/Pattern
Mirror				
Mirror C	SYS: (Global)			~
		VZ		

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

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

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



The simulation results will appear as shown below.



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

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

Basic	Color & Style	Labels	Normals	Other
Rend O Wir O Fill Defo O Aur © Uni Value	er Style reframe O Hido ed Shao rmation Scale Fa to-compute (0.1 iform O Nonuni	len led actor 192957) iform	Visible O All G O Fear O Free O No	e Edges edges erior edges ture edges e edges edges ŵ

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

Exercise 27

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

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

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



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

An antipying Contact Interaction

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

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

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

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

	Name	Initial	Step-1	Step-2	Edit
~	Fuse-Holder	Created	Propagate	d Propagated	Move Left
					Move Right
					Activate
					Deactivate
Ste	p procedure: eraction type:	Static, Ge Surface-t	eneral o-surface co	ntact (Standard)	

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

Slave Adjustment	Surface Smoothing	Clearance	Bonding	
No adjustment				
O Adjust only to re	emove overclosure			
O Specify tolerance	e for adjustment zone	e: 0		
O Adjust slave nod	les in set:		Y	
Contact interaction	property: NoFric			¥
Contact interaction Options: Interferer	property: NoFric			7
Contact interaction Options: Interferer Contact controls: ((property: NoFric nce Fit Default)			→ 1
Contact interaction Options: Interferer Contact controls: ((Z Active in this step	property: NoFric nce Fit Default)			∑ ₹

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

Interference Fit Options	×
○ No allowable interference	
Gradually remove slave node overclo	osure during the step
Overclosure Adjustment	
Automatic shrink fit (first general ar	alysis step only)
O Uniform allowable interference	
Amplitude: (Ramp)	J
Magnitude at start of step:	
Interference Direction	
Automatically determined	
O Along direction:	
X	
γ	
Z	
OK	Cancol
UN	Cancer

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

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

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

Pick Dismiss to close the Interaction Manager.

Boundary Conditions

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

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

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

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

	Name	Initial	Step-1	Step-2	Edit
V	Fuse	Created	Propagated	Propagated	A dama hadd
V	Holder-Fasten	Created	Propagated	Propagated	IVIOVE Len
~	Holder-Symm	Created	Propagated	Propagated	Move Righ
					Activisto
					Activate
					Deactivate
ste	p procedure:	S n turnoi	tatic, General	otation	Deactivate
ite	p procedure: undary conditio	S n type: D	tatic, General isplacement/R	otation	Deactivate
Ste 30	p procedure: undary conditio undary conditio	S n type: D n status: P	tatic, General isplacement/R ropagated fror	otation n a previous step	Deactivate

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

💠 Edit Boun	dary Condition	×	
Name: Fuse			
Type: Displ	acement/Rotation		
Step: Step-	-2 (Static, General)		
Region: (Picke	ed)		
CSYS: (Globa	1)		
Method:	Specify Constraints		
Distribution:	Uniform		
⊠ U1:	0		
* 🗆 U2:			
Ø UR3:	0	radians	
Amplitude:	(Ramp)	►	
* Modified in Note: The dis mainta	this step placement value will ined in subsequent s	be teps.	
OK	Ca	ancel	

Pick **OK** to apply and exit.

Pick **Dismiss** to close the manager.

➡ Job Submission

All the information required for analysis has been set up in the model. Now we can submit the job for analysis.

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

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

Pick Continue and then OK.

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

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

Postprocessing

For comparison, we will plot the contact pressure.

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

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

Now we will extrude the simulation results for better understanding.

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

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

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

Now the simulation results will appear as shown below.



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

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

Index

adaptive meshing, 64 adaptive remeshing, 65 adaptivity techniques, 63 analytical rigid surfaces, 60 Arbitrary Lagrangian-Eulerian, 64 arc length, 49 Arc-Length method, 45, 49 bifurcation, 20 Boltzmann superposition principle, 36 boundary condition imperfection, 21 boundary nonlinearity, 44 buckling, 20 buckling eigenmodes, 21 buckling mode shapes, 22 buckling problem, 22 Cauchy strain tensor, 14 collapse, 20 constitutive equation, 23 contact behavior, 51 contact pair algorithm, 51 contact property model, 51 Coulomb friction, 53 creep, 34 creep compliance, 34 damping, 50 dilatational strain energy, 30 displacement control, 20 displacement correction, 46 displacement-control, 50 distortional strain energy, 30 Drucker Prager yield criterion, 26 dynamic mechanical analysis, 42 dynamic test, 35, 42 eigenmodes, 23 eigenvalue buckling analysis, 22 element-based surfaces, 59 Engineering strain, 11 Eulerian meshes, 63 fabrication tolerances, 21 fields in solid mechanics, 10 finite strain theory, 13 first Piola-Kirchhoff stress, 29 force nonlinearity, 44 general contact algorithm, 51 geometric imperfections, 21 Geometric nonlinearity, 13

Green strain, 12 Green-Lagrange strain, 15, 16 hard contact, 52 hardening model, 26 hyperelasticity, 28 imperfections, 21 incremental-iterative methods, 45 infinitesimal strain, 13, 16 in-plane forces, 18 instantaneous elastic moduli, 39 internal forces, 46 isotropic hardening, 27 iteration, 47 Johnson-Cook hardening, 28 kinematic equations, 11 kinematic hardening, 27 Lagrangian meshes, 63 large rotations, 15 large shear strains, 15 large strain, 13 large translations, 15 limit point, 45 linear approximation, 46 linear perturbation procedure, 22 linear problems, 7 linear viscoelasticity, 35 linear-hardening, 28 load imperfections, 21 load proportionality factor, 49 load-control, 45 load-displacement analysis, 22 load-displacement response, 45 long term behavior, 38 loss modulus, 43 master surface, 56 material nonlinearity, 23 Maxwell model, 39 Mesh-to-mesh solution mapping, 66 Mooney-Rivlin model, 31 multilinear hardening, 28 negative stiffness, 19 negative stiffness matrix, 20 Neo-Hookean model, 30 Newton's method, 45 node-based surfaces, 59 node-to-surface, 56 nonlinear behavior, 10

nonlinear elastic materials, 28 Nonlinear Response, 9 nonlinear structure, 9 nonlinear viscoelastic behavior, 41 Ogden model, 33 parallel rheological framework, 41 perfect plastic deformation, 27 plastic deformation, 23 positive stiffness matrix, 19, 20 postbuckling behavior, 20 postbuckling response, 22 Power-law strain hardening, 42 principal stretches, 29 Prony series, 37 quasi-Newton method, 48 residual, 46 response curves, 9 rheological models, 39 rigid body rotation, 16 second Piola-Kirchhoff stress, 29 shear deformation, 14 Signorini, 51 slave surface, 56 small deformation, 8 small strain tensor, 14 smoothed particle hydrodynamic, 63 snap-through, 19 soft contact, 52 sources of nonlinearities, 10 storage modulus, 43 strain definitions, 11 strain energy density, 29 strain-displacement equations, 13 stress relaxation, 34 Stress stiffening, 18 Stretch ratio, 11 surface-to-surface, 56 system of equation, 48 transient tests, 35 true strain, 11 viscoelasticity, 33 volume ratio, 29 von Mises stress criterion, 24 Yeoh model, 32 vield criterion, 24 yield point, 23 yield strength, 23