## **Hyperelastic Seal Analysis**

The following tutorial will set up the analysis for modeling a hyperelastic seal. The analysis will be set up as a plane strain problem. The material properties for the hyperelastic material were obtained from a uniaxial tensile test, a biaxial tensile test, and a shear test. Additional key features of this analysis is the set up of a contact analysis.



The material models well be first input as tables from the data collected. Secondly, the data will be fit to the two-parameter Mooney-Rivlin hyperelastic model. By contrast, a linearly elastic material will only require the modulus of elasticity to be entered since the stress-strain relation is linear and the slope is constant.

An example of the Stress-Strain plot for the tabulated data of a hyperelastic material is shown in the graph below





1. Prepare the geometry described in the previous tutorial.

2. Prepare the material properties for the hyperelastic material.

1. Double click on Engineering Data and select the units: US Customary. 2. Verify that there is a check mark next to "Display Values in Project Units".

Note that in the project schematic a green checkmark will appear when that module has been completed. A green checkmark means that you can go on to the next step.



3. Click to add a new material and name it Rubber by typing in the space. 4. Expand Hyperelastic Experimental Data and double-click Uniaxial Test Data, Biaxial Test Data, and Shear Test Data.

Toolbox 👻 🗸	× Outine	of Schematic A2: Engineering Data						- <b>q</b>	L X
Physical Properties		A	В	С	D		E		
Elastic	1	Contents of Engineering Data 🗦	0	0	Source	De	scription		
Hyperelastic Experimental Data	2	= Material							
🚰 Uniaxial Test Data	3	2 Rubber							_
Plaxial Test Data Para Shear Test Data Volumetric Test Data	4	Structural Steel	·		General	Fatigue Data at ze from 1998 ASME B 2, Table 5-110.1	ro mean stres PV Code, Sect	tion 8, [	s Div
Simple Shear Test Data Uniaxial Tension Test Data Uniaxial Compression Test Data	* Propert	Click here to add a new material ties of Outline Row 4: Rubber						• q	ιx
Hyperelastic		А			В		с	D	Е
🗄 Chaboche Test Data	1	Property			Valu	e	Unit	8	(¢7
Plasticity	2	2 Uniaxial Test Data	1	Та	abular				
E Creep	3	🗄 🥐 Biaxial Test Data		Та	abular				
🖽 Life	5	🗄 🤿 🚰 Shear Test Data		Та	abular				

5. Click the Tabular box and enter the corresponding property test data. You can copy the data from the Excel file that was shared with you in class. Repeat for Uniaxial, Biaxial, and Shear Test data.

Note: If we were dealing with a linearly elastic, isotropic material it would not be necessary to enter the data in the form of a table. In that case the slope of the stress-strain relation is constant. Therefore, we would just enter the slope (which is E, the modulus of elasticity). However, in the case of the hyperelastic material the slope is not constant and we would either need to input the actual data in tabular form, or we can select a non-linear model that fits the data. This alternate method will be shown below.

Proper	ties of Outline Row 4: Rubber			• 9	L X
	A	В	с	D	Е
1	Property	Value	Unit	8	(p)
2	🥐 🔀 Uniaxial Test Data	Tabular			
3	🗄 🤰 🔁 Biaxial Test Data	Tabular			
5	🗄 🤋 🔀 Shear Test Data	Tabular			

Type or copy/paste the data from the Excel sheet

				В	с			
			1	Strain (in in ^-1) 🗦	Stress (psi) 💌			
			2	0	0			
			3	0.0056	17.94			
			4	0.0124	29.12			
			5	0.0192	38.22			
			6	0.0264	46.45			
			7	0.0337	53.81			
	1000		8	0.041	60.37			
	В	С	9	0.0481	66.18			
1	Strain (in in^-1) 📮	Stress (psi) 💌	10	0.0555	71.76			
2	0	0	11	0.0628	77.06			
3	0.0116	12.34	12	0.0702	81.97			-
4	0.0227	20.35	13	0.0775	86.67		В	С
5	0.0339	27.28	14	0.0851	91.21	 1	Strain (in in ^-1) 📮	Stress (pr
6	0.0451	33.06	15	0.0924	95.29	2	0	0
7	0.0557	38.29	16	0.1	99.56	3	0.012	20.22
8	0.0664	43.45	17	0.1078	103.8	4	0.0252	33.06
9	0.0773	47.86	18	0.1154	107.33	5	0.0376	43.07
10	0.088	52.12	19	0.1231	110.94	6	0.0499	51.64
11	0.0989	56.3	20	0.1312	114.72	 7	0.0624	59.27

6. In Table of Properties, type 70 (for °F) for Temperature. If Table of Properties is not visible, turn on View/Table. Note that the data from Excel can also be imported.

		Table o	f Properties Row 2: Un	iaxial Test Da	ata			۳	<b>p</b> >
			А			В	С		
		1	Temperature (F)		1 S	train (in in^-1) 🏓	Stress (psi)	-	
_		*			*			_	]
	A					Paste		٦	
1	Temperature (F) 🗦				1	Import Delimited	Data		
2	70				<b></b>	Engineering Data	Sources		
8									

nport File	E:\ANSYS\Book18\Sec	14-3\UniaxialTestData.c	SV.		_
Data					
• Previe	w C Full File				
	A	В		-	
	V Import	V Import	-		
Variable	Strain 💌	Stress	-		
Unit	in in^-1 💌	psi	-		
File					
1	0	0			
2	0.0116	12.34			
3	0.0227	20.35			
4	0.0339	27.28			
5	0.0451	33.06			
6	0.0557	38.29			
7	0.0664	43.45			
8	0.0773	47.86			
9	0.088	52.12			
10	0.0989	56.3			
11	0.1095	60.06			
12	0.1203	63.74			
13	0.131	67.28			
14	0.1415	70.67		-	

7. Expand Hyperelastic and double-click Mooney-Rivlin 2 Parameter. Then Expand the material model in the properties box. Right click **Curve Fitting** and select **Solve Curve Fit**.



The calculated parameters, material constants C01 and C10, appear in the upper right corner above the plot. Review the constants. The fitted data and the test data are plotted for comparison.

8. Right click **Curve Fitting** again and select **Copy Calculated Values to Property** from the popup menu window. The material properties are formally entered in the rows above Curve fitting. The material parameters C10 and C01 could have been entered without any test data if you know already know the values for these parameters. For example, more common materials may have published values. However, these are usual not known and they must be calculated using the experimental data.

Propert	ies of Chart : Uniaxial Test Data Rub	ber	Ŧ	ą	×
	A	В	с	D	Е
1	Property	Value L	Jnit (	8	(p)
2	🗉 🔀 Uniaxial Test Data	III Tabular	[	1	
5	🗉 🔀 Biaxial Test Data	III Tabular	[		
9	🗉 🔀 Shear Test Data	III Tabular	[		
13	Mooney-Rivlin 2     Parameter		[		
14	Material Constant C10	103.17 ps	á 🛛		
15	Material Constant C01	-4.8079 ps	á 👘		
16	Incompressibility Parameter D1	0 ps	i^-1		
17	🗉 📓 Curve Fitting	Fit Type: Mooney-Rivlin 2 Parame	ter		
18	Error Norm for Fit	Normalized Error			
19	Uniaxial Test Data	III Tabular	[		
20	🔀 Biaxial Test Data	Tabular	[		
21	🔀 Shear Test Data	Tabular	1		
22	Volumetric Test Data	Add this experimental data, to inc curve fitting.	lude it i	n th	e

9. Return to the project schematic. Right click on Geometry and select Properties from the menu. In the pop-up menu scroll down to Advanced Geometry Options and select "2D" if not already selected in the Analysis Type row.



operti	es of Schematic A3: Geometry		- <b> </b>
	A	В	
1	Property	Value	
13	Parameters	Independent	-
14	Parameter Key	ANS;DS	
15	Attributes		
16	Named Selections		
17	Material Properties		
18	<ul> <li>Advanced Geometry Options</li> </ul>		
19	Analysis Type	2D	-
20	Use Associativity	<b>V</b>	
21	Import Coordinate Systems		
22	Import Work Points		
23	Reader Mode Saves Updated File		
24	Import Using Instances	<b>V</b>	
25	Smart CAD Update	<b>V</b>	
26	Compare Parts On Update	No	-
27	Enclosure and Symmetry Processing	<b>V</b>	
28	Decompose Disjoint Geometry	$\checkmark$	
29	Clean Geometry On Import		

Set Up The simulation.

1. Start the mechanical setup by selecting Model from the schematic tree. Note the green check marks for Engeneering Data and Geometry before proceeding. When Mechanical opens, if prompted, select the units: in-lbm-lbf-s (inches, pound mass, pound force, seconds). Be patient, it will take a minute to open. You can verify that it is opening by looking at the alert in the lower left corner of the window. Verify the units.

					Home Environment Display Selection Automation				
-		٨			A Cut X Delete     My Computer *      Corror 4     Solve     Corror 4     Solve     Gordmant     Gordmant     Solve     Gordmant     Solve     Gordmant     Gordmant     Solve     Gordmant     Gordmant     Solve     Gordmant     Gordmant     Solve     Gordmant     Gordmat     Gordmant     Gordmant     Gordmat     Gordmant     Gordmat	iection ∰ Commands ® Images * e System □ Comment 🖓 Section Plane bint ∰ Chart ➡ Annotation Insert	<b>t</b> Uni	t Worksheet Keyfr	Tags ( Wizard ) ame ation Show Errors
•	-	~		_	- † □ ×	ର୍ ର୍ 📦 📽 🕒 ୦ - 💠 ଓ	~	Metric (m, kg, N, s, )	/, A)
1	777	Static Structural			e v Search Outline v	A: Static Structural		Metric (cm, g, dyne,	s, V, A)
2	٢	Engineering Data	~	4	v S Materials ^ → V Structural Steel	Displacement Time: 0.36756 s		Metric (mm, kg, N, s Metric (mm, t, N, s, i	, mV, mA) nV, mA)
3	OM	Geometry	~	4	v i Coordnate Systems v i Connections ⊟ -v i Contacts	Components: Free,-0.85 m		Metric (mm, dat, N, Metric (µm, kg, µN,	s, mV, mA) s, V, mA)
4	0	Model	3		Prictoriess - Rubber To Upper Plate     Ventoriess - Rubber To Lower Plate     Weth			U.S. Customary (in, U.S. Customary (in,	bm, lbf, "F, s, V, A)
5	٢	Setup	P		Static Structural (AS)		1	Degrees	Set the units to this t
6		Solution	2		- v@, Fixed Support - v@, Displacement			Rotational Velocity	
7	1	Results	2	1	B-VE Solution (Ap) Solution Information VE Equivalent Stress		~	RPM	
		Static Structural		1	"Displacement" + 🛛 🗸		<b>v</b>	Temperature Celsius (For Metric S	Systems)
		Score Scructural			n Mathod Geometri Salartion		_	Kelvin (For Metric Sy	(stems)

2. Select Geometry from the menu tree. For the 2D behavior select Plane Strain. (Do you recall discussing plane stress and plane strain in class?). Select 2D behavior in the left column and select Plane Strain in the pull-down menu. A 2D model is often used to model 3D geometry. This makes the setup and analysis much easier and requires far less computing time.

2							
Project*							
🖃 🐻 Model (A4)							
E-v Geometry							
~~~~	Lower Plate						
	j Seal						
	Upper Plate						
E Mati	erials						
	Rubber Structural Steel						
	rdinate Systems						
	nections						
- Kin Mes	h						
E 2 5ta	tic Structural (A5)						
	Analysis Settings						
÷	Solution (A6)						
Details of "Geome	try ♥ # Ll ×						
<ul> <li>Definition</li> </ul>							
Source	C:\Users\Bilbo Baggins\Documents\Courses\B						
Туре	DesignModeler						
Length Unit	Meters						
Element Control Program Controlled							
2D Behavior Plane Strain							
Display Style	Body Color						
E Bounding Box							
Properties							

3. Select Rubber from the tree and change the assignment to Rubber.



4. Hold the control key and select both the upper and lower plate and make sure the assignment is structural steel. Note that we did not set up the material properties for structural steel. We will use the default properties for structural steel in the built-in material library. If we wanted another material we could set up the properties and then select that material here in assignments.



Next, we will set up the contact behavior between our seal and the upper and bottom plate. This is necessary since the seal is not connected (bonded or glued) to the plates.

5. Highlight Connections/Contacts/Contact Region. Right click the first contact region. In the pop-up menu, scroll down to the bottom and rename the contact to "Frictionless rubber to Top Plate". Rename the second contact to "Frictionless Rubber to Bottom Plate.



Select the contact edges. This is a total of 7 edges. Select the edges in red shown in the right figure below. Hold the control key while select. In Scope, select Contact, then "Apply". Repeat for the Target and select the line shown in blue. This is the bottom line of the upper plate in contact with the seal. Note that the red and blue boxes in the right column correspond to the contact and target bodies in the model window. In **Definition**, set the Type to **Frictionless** in the pull-down menu. For **Behavior** select **Asymmetric.** 

	Contact Region 2							
	E → ? Totatic Structural (A5) Analysis Settings Solution (A6)							
D	etails of "Frictionless - Rubber	To Rubber"	🕈 🗖 🗖	×				
-	Scope			^				
	Scoping Method	Geometry Selection						
	Contact	1 Edge						
	Target	Apply	Cancel					
	Contact Bodies	Rubber						
	Target Bodies	Rubber						
	Shell Thickness Effect	No						
	Protected	No						
-	Definition							
	Туре	Frictionless						

)e	tails of "Frictionless - Rubbe	er To Upper Plate" 4
-	Scope	
	Scoping Method	Geometry Selection
	Contact	7 Edges
	Tauget	1 Edge
	Contact Bodies	Rubber
	Tauget Bodies	Upper Plate
	Shell Thickness Effect	No
-	Definition	
	Туре	Frictionless
	Scope Mode	Menuel
	Behavior	Asymmetric
	Trim Contact	Program Controlled
	Suppressed	No
Ξ	Advanced	
	Formulation	Program Controlled
	Detection Method	Piogram Controlled
	Penetration Tolerance	Program Controlled
	Normal Stiffness	Piogram Controlled
	Update Stiffness	Piogram Controlled
	Stabilization Damping Factor	0.
	Finball Region	Piogram Controlled
	Time Step Controls	None
=	Geometric Modification	
	Interface Treatment	Add Offset, No Ramping
	Offset	0. mm
	Contact Geometry Conjection	None
	Tauget Geometry Conjection	None

6. Repeat for the bottom plate. Select the appropriate edges as shown below. A total of 6 edges make up the Contact. Select the edges making up the red line in the figure below on right. Select Target and set the target. The target body is the upper line (in blue) of the lower plate.

De	etails of "Frictionless - Rubber To Lower Plate" 🛛 🐥						
Ξ	Scope						
	Scoping Method	Geometry Selection					
	Contact	6 Edges					
	Target	1 Edge					
	Contact Bodies	Rubber					
	Target Bodies	Lower Plate					
	Shell Thickness Effect	No					
Ξ	Definition						
	Туре	Frictionless					
	Scope Mode	Manual					
	Behavior	Asymmetric					
	Trim Contact	Program Controlled					
	Suppressed	No					
-	Advanced						
	Formulation	Piogram Controlled					
	Detection Method	Program Controlled					
	Penetration Tolerance	Program Controlled					
	Normal Stiffness	Program Controlled					
	Update Stiffness	Program Controlled					
	Stabilization Damping Factor	0.					
	Pinball Region	Program Controlled					
	Time Step Controls	None					
Ξ	Geometric Modification						
	Interface Treatment	Add Offset, No Ramping					
	Offiset	0. mm					
	Contact Geometry Conjection	None					
	Target Geometry Conection	None					



7. In Details of Analysis Settings, turn on Large Deflection



Next, we need to set up the supports.

8. From the top menu bar, select Frictionless from the Supports window. Insert a Frictionless Support for the 3 edges at right. We are taking advantage of symmetry and modelling only half the seal since the left and right sides are identical. At the location of symmetry (the center) we set up a frictionless support. Select the 3 edges where the arrows point in the figure below.

									D
ural 5									3
		- 🛠 Ø Ø I	ଷ <mark>ଷ</mark> ୍ଟ <sup>Select</sup>	Wode - H			副北南	Cit	
<ul> <li>•</li> </ul>					and the second s				
				Structural		10	100	P	- 1
on on	Inertial	Loads @ Pres		Enctionless     Displacement Structural	Conditions	Pirect N	File 1 700	k kastran F k	Э



▲			
-	_	_	
	De	etails of "Friction	less Support" 🛛 🧛
	Ξ	Scope	
		Scoping Method	Geometry Selection
		Geometry	3 Edges
	Ξ	Definition	
		Туре	Frictionless Support
		Suppressed	No

9. The bottom edge of the lower plate will be a fixed support. That is, this edge will not move and will constrain the model in the y-direction. In Scope select Geometry then select the bottle edge.

orce 1oment ressure	Supports	<ul> <li>Fixed</li> <li>Frictionless</li> <li>Displacemer</li> <li>Structural</li> </ul>	Conditions				
20	Q Select	K Mode∗ 👘					
				De	tails of "Fixed S	upport"	4
					Scope		
					Scoping Method	Geometry Selection	
					Geometry	1 Edge	
					Definition		
					Туре	Fixed Support	
					Suppressed	No	
			$\overline{}$				

10. Finally, we will apply a displacement of 0.85 inches to the top edge. The displacement will be in the negative y-direction. Select the edge. Then select y-component in the left column and enter the value in the right column.



De	tails of "Displacem	ent" 🗸
-	Scope	
	Scoping Method	Geometry Selection
	Geometry	1 Edge
-	Definition	
	Туре	Displacement
	Define By	Components
	Cocadinate System	Global Coordinate System
	X Component	Free
	🔄 Y Component	-0.85 in (namped)
	Suppressed	No

Note the displacement is visible in the window

sert				Structural	
×□	ତ୍ ତ୍ 📦 😜	) 🕞 🔿 - 💠	© • © ©	Select 🦎 Mode=	F. 🕞 🕞 🖻 🖫
^	A: Static Structural Displacement Time: 1. s Displacement Components: Free	,-0.85 m			<b>-</b>

Mesh the geometry

11. Select Mesh from the tree then select Solve from the top menu. Highlight Mesh from the model tree and in Details of Mesh in the lower left window select Relevance and enter 100 and in Relevance Center select Fine. Generate the mesh.



The default mesh settings produce a poor quality mesh. We will increase the number of elements by reducing the size of each element. Select Mesh again from the tree and in

Details of mesh reduce the element size. Slightly reduce the element size and solve the mesh again. For example, change 0.0015 to 0.001. Reduce the element size so that you end up with a mesh similar to the figure on the right.

	Prictionless - Rubbe     Participation	r To Lower Plate		\$		
D	etails of "Mesh"		<b>▼</b> ₽ 🗆	×		
Ξ	Display			^		A A A A A A A A A A A A A A A A A A A
	Display Style	Use Geometry Se	tting			
Ξ	Defaults					AN A
	Physics Preference	Mechanical				
	Element Order	Program Control	led			AH AN
	Element Size	1.e-003 m				LE LE
Ξ	Sizing					
	Use Adaptive Sizing	No				AH AN
	Growth Rate	Default (1.2)			1	
	Mesh Defeaturing	Yes				
	Defeature Size	Default (5.e-006	m)			
	Capture Curvature	No				Stand Street
	Capture Proximity	No				
	Enable Washers	No				
	Bounding Box Diagonal	4.9721e-002 m				

With the setup complete we are ready to solve for the solution. Select Solution from the Tree.



From the top menu bar select the following results: Maximum Principle Stress, Minimum Principle Stress, Shear Stress, Maximum Principle Elastic Strain, Minimum Principle Elastic Strain, and Shear Elastic Strain. The results will then be listed in the Solution tree. Finally, solve the analysis.



The convergence window will appear. Monitor the solution while it solves.



## **View The Results**

Select the result object from the tree.

1. Maximum Principle Stress: The Maximum tensile stress is shown in red in the contour map in the figure below.



The results can be animated by selecting the Play button in Animation.



2. Minimum Principle Stress: The maximum compressive stress is shown in blue in the figure below.



3. Maximum Shear Stress: Shown in blue.



4. Maximum Principle Elastic Strain: Maximum Tensile Strain shown in red.



5. Minimum Principle Elastic Strain: The maximum compressive strain shown in blue.



6. Shear Elastic Strain: The maximum shear strain shown in blue.



7. Animate the results. Select play animation.

Graph Animation		<b>)</b>		20 Frames		2 Sec (Auto)	-	<b>→</b> ‡   © <b>   </b>	□× » č
515.11	0.	0.125	0.25	0.375	0.5	0.625	0.75	0.875	1.
					<b>[5]</b> 1				

8. Right click on **Solution** in the tree. Insert a **Probe/Force Reaction**. In Details of Force Reaction Change the boundary condition to Displacement and select Y-Axis. Then solve.



	⊨	lution (A6) Solution Information Equivalent Stress Force Reaction	~	
D	etails of "Force React	ion" 🕶 🕂 🗖	×	
-	Definition		^	
	Туре	Force Reaction		
	Location Method	Boundary Condition		
	<b>Boundary Condition</b>	-	1	
	Orientation	Frictionless Support	1	
	Suppressed	Fixed Support		Context
Ξ	Options	Weak Springs		Solution Display Selection Automation
	Result Selection	All		🗙 Delete 🛛 My Computer 👻 🗲 🛛 🔤 Nam
	Display Time	End Time		Q Find
÷	Results			Cores 4 Solve Analysis Ø Rem
=	Maximum Value Ove	r Time		Solve Fa
	X Axis			<b>▼</b> ‡ □
	Y Axis			Search Outline 💙 –

Note that for a plane strain problem, Workbench assumes a unit depth (in this case, one inch). Therefore the force unit should be read as lbf/in (pound force per inch). For example, if the dept was 2 inches, multiply the values in the Tabular Data on the right times 2 to get the actual value.

												/
Graph										ą.	Tabular Data	/ +
Animation			<b>Q</b> 1	0 Frames		2 Sec (Au	to)	- 1	0	An 3 Cyc	Time [s] ▼ For	e Reaction (Y) [lbf]
										1.	2 0.4 -7.0309	e-002
-4.1	1292e-2							1			<u>3 0.7</u> -0.1145 4 14.1325	2
_	-1							1				
Ibf	-2.								1			
-	-3 -											
	-9,									1		
1	-4.1325 -	0 0	105	0.25	0.275	0.5	0.675	0.75	0.975	-		
		0. 0.	125	0.25	0.575	0.5	0.625	0.75	0.075	1.		
						[s]				_		
						1						

Create a Force vs Displacement Chart

1. With the Home Tab selected, select the Chart button. Select New Chart and Table icon on the toolbar. Note that Chart is added to the project tree at bottom of the list.

	Home	Result	Display	Selection	Automation					
] ate	Cut Copy Paste Outline	× Delete Q Find ₽₽ Tree ▼	My Computer Distributer Cores 4	d Solve	ve Analysis	i≌Named S ik Coordin Ø <sub>6</sub> Remote	selection ate System Point Ir	記 Commands つ Comment <mark> </mark>	Images ▼ Images ■ </th <th>tinits</th>	tinits
200					•••••••	• ₽ 🗆 ×	Q	२ 📦 🗣 🌍	≌ ○ • ♦	<b>Q Q</b>
ne		<ul> <li>Search Ou</li> <li>Subber</li> </ul>	utline 💙 🗸			•	A: Stati	c Structural		

2. Click on Chart in the project tree. Set Details of Chart as shown below.



Holding the control key, select Displacement from Static Structural and Force Reaction from Solution. Then next to Outline Selection select Apply. The left column will change to "2 Objects". Set the highlighted boxes below as follows:

<ul> <li>Definition</li> </ul>		A 17
Outline Selection	2 Objects	
- Chart Controls		
X Axis	Displacement (Y)	
Plot Style	Both	
Scale	Linear	
Gridlines	Both	
- Axis Labels		
X-Axis	Displacement	
Y-Axis	Force	
- Report		
Content	Chart And Tabular Data	
Caption		
<ul> <li>Input Quantities</li> </ul>		
Time	Omit	
Displacement (Y)	X Axis	
<ul> <li>Output Quantities</li> </ul>		1
TAT FILL DILATE OF	Disatau	~

A force-versus-displacement curve is generated. Note that as before the force unit is in lbf/in for a 2D problem.

