Nonlinear Static Stress

Nonlinear Static Stress

This project will explore the requirements and different requirements of a nonlinear static stress analysis within Autodesk Simulation. The basic principles covered will provide a foundation of completing more complex nonlinear studies.

1.1 Project 2 – Beam and Ram Analysis

During this project we will conduct a mechanical event simulation with nonlinear materials analysis of a ram part contacting an aluminum cantilever beam part. The ram gradually moves downward to contact the beam and returns back upward in a gradual motion. The ram forces the beams displacement enough taking the material beyond the yield stress level.



- 1. Click on the **Open** command from the Launch panel of Autodesk Simulation 2012.
- 2. Set the **Files of Type** setting to *STEP* to only display STEP files. STEP files are neutral 3D CAD formats exported from most modern CAD systems.

Nonlinear Static Stress

- 3. Select the "Beam and Ram Assembly.step" file and click **Open**.
- 4. Chose the option to "*Use STEP file units*" to set the length unit value to match the value in the STEP file.
- 5. Select "*MES with Nonlinear Material Models*" as the **Analysis Type** and click **Ok**. This will be used to conduct a mechanical event simulation with nonlinear materials.

0	Choose Analysis Type				
	MES with Nonlinear Material Models				
ł.	Typical Applications:				
	Linkages and mechanisms Press-fit Snap-fits	* +			
	< >				
	ОК				

6. The 3D CAD model is now open within Autodesk Simulation and ready to be meshed and setup for the linear analysis. **Orbit** the model to review the parts before meshing.



7. Within the **FEA Editor Browser** you will notice that the units are set to "*English (in)*" and the **Analysis Type** is set. You can change these at any time within the browser if required.



Nonlinear Static Stress

8. Expand the **Surfaces** group of *Part 1* in the browser as shown below. You will notice that there is a surface for each exterior surface of the CAD model.



- 9. One of the first things you always do is mesh the model. Click on **Mesh tab | Mesh panel | 3D Mesh Settings** to open the **Model Mesh Settings** dialog.
- 10. Click on the Options button
- 11. Select *Absolute Mesh Size* for the surface mesh **Type**. Then enter 0.3 as the **Mesh Size** within the **General** tab.



- 12. Click **OK** button to close the options. Then press **Mesh Model** to start the meshing process of the model. Autodesk Simulation first meshes the exterior surfaces then checks to ensure the volume is closed of a 3D solid model. Then proceeds and meshes the interior areas by adding elements.
- 13. Press **Yes** to new the mesh results. You will notice that there are around 647 elements now in the model. Click **Close** to exit the **Meshing Results** dialog.
- 14. Now that the model has been meshed you will notice that Brick elements were used for the parts in the browser. Right-Click on the **Element Definition** heading of *Part 1* and select **Edit Element Definition**.
- 15. Scroll down in the **General Settings** and select "*von Mises with Isotropic Hardening*" for the **Material Model**. This will allow for analysis of the stress and strain of a part as they enter the plastic region of the Stress-Strain curve. A bilinear curve will be defined to control the stress strain relationship. This is only needed if the material properties dictate that the part will have stain hardening.
- 16. Then ensure that *Large Displacement* is selected as the **Analysis Type** as the part will have large deformations in this nonlinear analysis.

Element Definition - Brick/Tetrahedral						
General Thermal Othotropic Advanced Soil General Settings Material model: von Mises with Isotropic Hardening Viscoelastic Yeoh Viscoelastic Van der Waals Plasticty Plastict						
Midside Nodes Not Included Analysis Type Large Displacement						
OK Cancel Help	Reset From Model					

- 17. Now we will assign the material to *Part 1*. Right-Click on the **Material** heading of *Part 1* and select **Edit Material**. This will open the **Element Material Selection** dialog.
- 18. Expand the Aluminum folder and select Aluminum 6061-T6-T651 as the material. Notice that the Yield Strength is 40,030 psi. This aluminum material is a softer material then a steel material.

- Auminum 360.0-F Die Casting Aloy - Auminum 413.0-F Die Casting Aloy - Auminum 5052-H32 - Auminum 6061-0	1		Units System: Material Description:	Metric mics (SI) None
Auminum 6061-T4; 5061-T451 Auminum 6061-T6; 6061-T651 Auminum 6063-T6 Auminum 7075-0			Source:	MatWeb
			Select Material	Aluminum 6061-T6; 6061-T651
Aluminum A356.0-T6, Sand Cast		Ξ	Material Properties	
Aluminum A380.0-F Die Casting Alloy			Damping (s)	0
Aluminum Alloy 1100-H14			Mass density (bf·s²/in/in²)	2.52645876511496E-04
Aluminum Alloy 1100-H18			Modulus of Elasticity (bf/in²)	9993100.13019664
Aluminum Alloy 1100-0			Poisson's Ratio	0.33
Aluminum Alloy 2014-T4			Strain Hardening Modulus (bf/in²)	42513.2
Aluminum Alloy 2014-T6			Yield Strength (bf/in?)	40030 415615882
Aluminum Alloy 3003-H12				
Aluminum Alloy 3003-H14				
- Aluminum Alloy 3003-H16				
- Aluminum Alloy 3003-O				
Aluminum Allov 3004-H32		L		
Edit Properties Rese	t From	Мо	del	OK Cancel Help

- 19. Click **OK** to exit the **Element Material Selection** dialog and assign the material.
- 20. Right-Click on the Element Definition heading of Part 2 and select Edit Element Definition.
- 21. Then ensure that *Isotropic* is selected as the **Material Model**. Then also ensure that *Large Displacement* is selected as the **Analysis Type**. This defines that the material properties are the same in all directions.
- 22. Click **OK** to exit the Element Definition dialog.
- 23. Assign Steel (AISI 4130) as the Material type to Part 2 now.

Nonlinear Static Stress

24. When complete the past several steps the **Part** tree in the **FEA Editor** browser should look like the following.



25. Orbit to position the model in a position as shown below.



- 26. Change the selection settings to use **Point** as the **Shape** and **Surfaces** as the **Select**.
- 27. Click to select the end face as shown below of the beam part.



Nonlinear Static Stress

- 28. Click **Setup tab | Constraints panel | General Constraint** then select **Fixed** to lock all DOFs for the nodes on this surface. Click **OK** to complete the assignment.
- 29. Click on the top surface of the ram part as shown below.



30. To ensure the ram part stays in position and only moves along the Y axis we will add a new constraint. Click **Setup tab | Constraints panel | General Constraint** then select **Tx** and **Tz** to lock the translational movement in the X and Z axis of the nodes. Click **OK** to complete the assignment.

Constrained DOFs	Predefined		
Tx	Fixed	X Symmetry	X Antisymmetric
Tz	Free	Y Symmetry	Y Antisymmetric
Rx	No Translation	Z Symmetry	Z Antisymmetric
Rz	No Rotation		
Coordinate System:	Global (Default)		
Description			

Nonlinear Static Stress

31. Zoom in and select the top surface of the ram part again. Once selected right-click the graphics area and select **Select Subentities | Vertices**. This will select all of the nodes on that surface instead.



- 32. With the nodes selected we will assign a displacement movement. Click **Setup tab** | **Constraints panel | Prescribed Displacement** to open the **Prescribed Displacement** dialog. We will use a translation displacement and enter -0.6 **Magnitude** value. Then select **Scalar Y** as the **Direction** value. This is going to move the downs down 0.6 of an inch just as a displacement so no force value is used.
- 33. Notice the **Load Case** is set as **1** then click the **Curve** button to open the **Multiplier Table Editor** dialog.
- 34. Click the **Add Row** button. Viewing the graph you will notice the first column is time and the second is the multiplier value for the magnitude. So the ram part will move down 0.6" in the first second then return back 0.6" in the last second. Click **OK** to close.



- 35. Notice the **Active Range** is set as **1** then click the **Data** button to open the **Analysis Parameters** dialog.
- 36. This will control when the load case is used in the simulation. Enter 2 in the Death Time field. As you can see Index 1 which is the load case will start at 0 seconds and end at 2 seconds. Click OK to close.

Analysis Parameters - MES: Prescribed Motion	
Active Ranges	
Birth and Death Times	
Index Birth Time (s) Death Time (s) Rebirth Index	
Add Paur Doloto Paur	
Add Row Delete Row Import Exporte	
OK Cancel Help Reset From Model Reset From Default	1
	,

- 37. Click **OK** to close the **Prescribed Displacement** dialog.
- 38. The model should now look like the image below showing the constraints.



Nonlinear Static Stress

- 39. Now we need to start the process of assigning the contract relationship between the two parts. Set your selection type to **Point** for **Shape** and **Surfaces** for **Select**.
- 40. Click the top surface of the beam part and then hold down CTRL and select the bottom surface of the ram part as shown below.



41. Click Setup tab | Contacts panel | Surface-to-Surface Contact to assign a contract relationship between the two surfaces. Hit the Enter key to accept the default contact pair name in the browser to complete the command.



42. Select the *Surface* <1/4 *with* 2/6> contact in the browser then right-click and select **Settings** to open the **Controls and Parameters for Contact Pair** dialog.

Nonlinear Static Stress

43. This is where you can adjust the contact parameters and settings. Ensure **Automatic** is set as the **Contact Type** and **Frictionless Contact** as the **Contact Method**. Click **OK** to close.

Parameters		1
Contact problem type	Low Speed Contact (Press-Fit) 🔹	,
Contact method	Frictionless Contact 👻	Reset From Model
Contact type	Automatic 👻	Reset From Default
Modeling Friction		1
Static friction coefficient	0	Advanced
Sliding friction coefficient	0	
Tangential stiffness ratio	1	
Tied Contact Options	Slide / No Bounce Contact Options	
Tied contact initial interference	√ No bounce	
Tied contact tolerance	No slide	

44. Click Setup tab | Model Setup panel | Parameters to open the Analysis Parameters dialog. This provides settings for the overall simulation. Enter 2 for the Duration field and 50 for the Capture Rate. This will make the simulation event run for 2 seconds capturing 50 different segments per each second. There will be 100 different load cases at the conclusion. Click OK to close.

Analysis	Parameters - MES w	ith Nonlinear M	Material Mod	dels		? ×
Descript	tion of model					
MES					^	Reset From Model
					-	Reset From Default
Event						
۲	Duration	2	s	Capture rate	50	1/s
01	Number of time steps	100		Initial time-step size	0.0	2 s
Load (Curves Gravity/Acc	eleration Therm	al/Electrical	Output		

Nonlinear Static Stress

- 45. To run the simulation click **Analysis tab | Analysis panel | Run Simulation**. The complete mechanical event simulation will be solved now.
- 46. Click **View tab | Navigate panel | Orientation Top** to position the model where you can see the travel of the ram part.
- 47. To view the results as a step by step video click Results Contours tab | Captures panel | Start



48. As you watch the capture you will notice that is cycling through each of the 100 time steps and the Von Mises Stress Legend values are updating at each step.



49. Click **Stop** under the **Captures** panel when done.

Nonlinear Static Stress

- 50. To change the Legend chart to keep a constant maximum and minimum value throughout the two second simulation we need to change a property. Click on Results Contours tab | Settings Panel | Legend Properties Setup. Then click on the Range Settings tab of the Plot Settings dialog. Uncheck the Automatic calculate value range checkbox. This will set the low and high values to the max and min. Click OK to close.
- 51. To view the results as a step by step video click **Results Contours tab | Captures panel | Start.** Noticed the legend is not changing per each step.
- 52. Click Stop under the Captures panel when done.
- 53. Turn on a probe to display the maximum value at each load case on the model. Click on **Results** Inquire tab | Probes tab | Maximum.
- 54. To see a list of the max Von Mises Stress at each load case click **Results Inquire tab | Inquire panel | Maximum Results Summary**. This will open the dialog and allow you to review each load case. Write down the node numbers that are constant at these different load case ranges to be used later.

Load Case $5-10 = Node \ 158$ in the image shown below Load Case $45-50 = Node \ 433$ in the image shown below Load Case $75-80 = Node \ 250$ in the image shown below

Inquire: Maximum Results Summary		E
Maximum results are computed from unsmoothed data Load Case Location Result	(lbf/(in^2))	A E
50 Part 1, Element 354, ElNode 7, Node 433 49 Part 1, Element 354, ElNode 7, Node 433 48 Part 1, Element 354, ElNode 7, Node 433 47 Part 1, Element 354, ElNode 7, Node 433 46 Part 1, Element 354, ElNode 7, Node 433 45 Part 1, Element 354, ElNode 7, Node 433 44 Part 1, Element 354, ElNode 7, Node 433 43 Part 1, Element 354, ElNode 7, Node 433 43 Part 1, Element 16, ElNode 0, Node 159 42 Part 1, Element 359, ElNode 3, Node 433	40486.8 40440.9 40435.3 40385.8 40374.2 40326.5 40259 40175.6 40172.8	Ŧ
4		F.
Clear Save Values Save To Report		Help

ME 24-688 – Week 10 Nonlinear Static Stress

- 55. Click back to the Results Contours tab and ensure Von Mises is selected in the Stress panel.
- 56. Click the **Next** and **Previous** buttons under the **Load Case Options** panel to move to the Time Step 10 of 100.
- 57. Select the node that is on the top edge of the bar that is one element in as shown below.



58. Right-Click in the graphics window and select **Graph Value(s)**. This will open a new window with a graph of the selected node showing the Von Mises Stress at each load case. You will notice that the material is Strain Harding from the 0.8 – 1.0 second range as the material has exceeded the yield strength of 40,000 psi. Also notice the stress levels at the end of the simulation that remain higher than the start of the simulation.



ME 24-688 – Week 10 Nonlinear Static Stress

- 59. Close the graph display window.
- 60. Now we will review the displacement of the model. Click on **Results Contours tab** | **Displacement panel** | **Displacement**.
- 61. To view the results as a step by step video click Results Contours tab | Captures panel | Start.
- 62. **Stop** the animation of the displacement.
- 63. Zoom in and select the right top corner node of the beam as shown below.



64. Right-Click in the graphics window and select **Graph Value(s)**. This will open a new window with a graph of the selected node showing the displacement at each load case.

Nonlinear Static Stress

65. Notice that the maximum displacement is about 0.7" but then when the ram return to a position above the beam at the two second mark the displacement of the node does not return to the zero position. This is because we past the yield stress levels of the aluminum material causing permanent deformation of the part. So in this case the beam part would remain bent after the event.



- 66. Close the displacement graph window.
- 67. Now that we know the beam part is our main concern we can turn off the ram part from the results. Select the *Part 2* in the **Results Browser** and right-click to turn the **Visibility** off.



- 68. Click Results Contours tab | Stress tab | Von Mises to active the stress result display again.
- 69. Now to measure the safety factor of the part click **Stress panel | Safety Factor**. This will open the allowable Stress Values dialog. The material yield stress values are used by default. Click **OK** to accept these values.

Allowable S	tress Values		X
Part		Allowable Stress (b	Load Vield Strees
1 < Exerc	ise C.stp:	40030.415615882	Load field Stress
2 < Exerc	ise C.stp: r	170000	
			Load Ultimate Stress
•	11	•	
ОК		Cancel	Help

- 70. Click on **Results Inquire tab | Probes panel | Minimum** to turn on the minimum value display probe. Then you can also turn off the **Maximum** probe.
- 71. Click on Load Case Options panel | Load Case Set and enter 50 as the load case and click OK. This will display the load case at the one second mark of the simulation. You will notice at this point the minimum Factor of Safety is below 1.0 confirming the part would fail.

