Linear Static Stress

Linear Static Stress

During this project you will learn the general information about performing a linear static stress analysis in Autodesk Simulation Multiphysics. The goal of these projects is to provide an introduction to the Autodesk Simulation Multiphysics application and user interface before advancing to nonlinear simulations.

1.1 Project 1 – Yoke and Clevis Assembly

During this project we will conduct a static stress with linear materials analysis on the yoke and clevis assembly. The constraint methods will be fixing the location of the clevis and adding a bolted connection to hold the yoke part to the clevis part. The yoke will then have an 800 lbf load added to the inside of the large diameter hole to complete the setup. Once the analysis is run we will review the results of this linear analysis to complete an introduction to Autodesk Simulation Multiphysics.



Linear Static Stress

- 1. Click on the **Open** command from the Launch panel.
- 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.

Se Open				X
	Select a file to open			
Open	Look in: 🎉 Projects	- 🕝 🏂 📂 🖽 -		Preview
	Name	Date modified	ту	
	Project 1	3/10/2012 11:43 AM	Fi	
New	Project 2	3/10/2012 11:44 AM	Fi	
	Project 3	3/8/2012 6:22 PM	Fi	No preview available
3				
Recent Files				
	•		۲	
	File name:		•	
	Files of type: STEP (*.stp, *.ste, *.step)		•	
Options	Show preview			Open Cancel

- 3. Select the "Yoke and Clevis 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 "Static Stress with Linear Material Models" as the Analysis Type and click Ok.

C	hoose Analysis Type	
Γ	Static Stress with Linear Material Models	
Ŀ	Typical Applications:	
	Structures Buildings; Car frames; Truss systems Bodies	*
L	٠	Þ
	ОК	

Linear Static Stress

6. The 3D CAD model is now open within Autodesk Simulation and ready to be meshed and setup for the linear analysis.



- One of the first things to complete is the meshing of the model as most other items are established from the mesh instead of the CAD model geometry. Click on the Mesh tab | Mesh panel | 3D Mesh Settings to open the Model Mesh Setting dialog.
- 8. Move the **Mesh Size** slider to **85%** to use a finer mesh and ensure the **Mesh Type** is set to **Solid** to use brick elements. The overall model size and specified mesh size value is used to calculate the mesh size.

Model Mesh Settings		X
Mesh type	Mesh size	
Solid	Coarse 🔻	Fine
Midplane		
Plate/shell		
Options		
Defaults	OK Cancel	Mesh model

 Click Mesh Model and this will create the mesh on the model and close the Model Mesh Setting dialog. When prompted to View Mesh Results at this time click Yes and you will notice there are over 3500 elements. Click Close to continue.

Linear Static Stress

10. Use the **Orbit** command located number the **Navigate panel** within the **View tab** to rotate the model around and review the mesh results. You will notice that the general mesh size is fairly good but is maybe to coarse in some areas around the fillets of the yoke part.



11. To add several refinement points to the model to increase the mesh around key smaller features click **Mesh tab | Refinement Points tab | Automatic** to open the **Automatic Refinement Points** dialog. Move the slider to the right so it is two tick marks away from the right fine end and then click **Generate**.

Automatic	Refine	ment P	oints					×
Density	ofrefin	ement p	oints					
Coarse	2							Fine
1				1	i.		Ų	1
Genera	ate					Clos	e	Help

Linear Static Stress

12. This will add 30+ points to the model as shown by block dots in the areas that require additional refinement. These refinement points will be used to create a finer mesh in these areas to ensure more accurate results. Click **Close** to exit the dialog.



13. To recreate the mesh on the model using these refinement points click Mesh tab | Mesh panel | Generate 3D Mesh. This will create a new more refined mesh and click Yes to View Mesh Results. This will display the Meshing Result dialog showing the number of elements in the model which now should have over 4000 elements. Once complete click the Close button to exit.



Linear Static Stress

14. Now that the model has been meshed you will notice that the Element Type and Element Definition have been established in the FEA Editor Browser for each part. These can be edited by right-clicking on them from the Browser if needed.



15. To set the material for each part go to the **FEA Editor Browser** and right-click on the **Material** heading of *Part 1* and select Edit Material. This will open the **Element Material Selection** dialog where you can assign, create, and edit materials to parts.

Element Material Selection		? ×
Create New Library 🚹 Add Exisiting Library		
Select Library	[Customer Defined]	
Autodesk Simulation Material Library -	Current Material Information	
	Analysis Type:	Structural
Autodesk Simulation Material Library	Element Type:	Brick
[Customer Defined]	Material Model:	Standard
I I I I I I I I I I I I I I I I I I I	Material Specified:	[Customer Defined]
the Brass	Brass Material Source: Not Applicable Material Identification	Not Applicable
	In Library File:	Not Applicable
	Date Last Updated:	Not Available
	Units System:	English (in)
B ← Other B ← Steel	Material Description:	Customer defined material properties
	Not Available Source:	Not Available
	Select Material	[Customer Defined]
	Material Properties	
	Mass density (bf·s²/in/in³)	0
	Modulus of Elasticity (bf/in²)	0
	Poisson's Ratio	0
	Thermal Coefficient of Expansion (1/*	1 0
	Shear Modulus of Elasticity (bf/in²)	0
Edit Properties Reset From	Model	OK Cancel Help

Linear Static Stress

- 16. Expand the *Steel* folder and then expand the *ASTM* folder. Select the "*Steel* (*ASTM-A36*)" material by clicking on the name. Click the **OK** button to exit and assign this part to the part.
- 17. Repeat the process to assign the material "Steel (ASTM-A36)" to Part 2 of the model.
- 18. Once complete you will see the new material types assigned to the parts in the FEA Editor Browser as shown below. These material values will be used throughout the analysis and can be changed in the same manor if required.



19. Before assigning the constraints click on the Selection tab. Here you will see the Shape and Select panels. Located on these two panels are the selection settings that control how items are selected on the mesh model. Select Point from the Shape panel so single points on the mesh model. Then select Edges from the Select panel to only select the edges from the point clicked.



Linear Static Stress

20. **Orbit** the model so the bottom of the *Clevis* part can be viewed as shown below. Press ESC to exit **Orbit** when done.



21. Select one of the bottom edges of the *Clevis* part and then hold down the CTRL key to also select the three other edges as shown below.



22. Select Setup tab | Constraints panel | General Constraint to open the Boundary Condition Object dialog.

Linear Static Stress

23. Select Fixed as the Predefined setting. This will lock all six (6) degrees of freedom on the nodes of the four (4) selected edges preventing them from moving or rotating as shown in the Constrained DOFs values. Click OK to complete the assignment.

Creating 4 Edge Boundar	y Condition Objects	5	? ×
Constrained DOFs Tx Ty Ty Tz	Predefined Fixed Free	X Symmetry Y Symmetry	X Antisymmetric
Rx VRy VRy VRz	No Translation No Rotation	Z Symmetry	Z Antisymmetric
Coordinate System: Description	Global (Default)		
			÷
	ОК	Cancel	

- 24. The *Yoke* part is held assembled to the *Clevis* part by a bolt in this case. To simulate the bolted connection that is not in the CAD model we will use the **Bolt** command available within Autodesk Simulation.
- 25. Change your selection setting to have the **Shape** set to **Point** and the **Select** set to **Surfaces** to select a complete surface of the mesh model.



26. Start the bolted connection command by click Mesh tab | CAD Additions panel | Bolt.

Linear Static Stress

27. Select the outer surface of the *Clevis* part as shown below.



28. Click the **Add** button within the **Bolt Head** section of the **Generate Bolted Connection** dialog to add the selected surface as the bolt head surface.

Generate Bolte	d Connection			×
Part Number Bolt diameter	3	in	Type of Bolt Number of spokes	Bolt With Nut
Bolt head Contact sur	face(s)			
Head diame	ster 0		in	Remove
Totorior hole				

29. Select the opposite outer surface of the Clevis part.



Linear Static Stress

30. Click the **Add** button within the **Nut** section of the **Generate Bolted Connection** dialog to add the selected surface as the surface the nut would contact.



31. Set the selection to use Circle as the Shape and Surfaces as the Select value.



32. Go to the View Cube and select the Bottom view to view the model.



Linear Static Stress

33. Position your cursor in the center of the clevis hole and drag a circle out just past the diameter of the hole to select all of the surfaces within this circle selection.



34. Click the **Add** button in the **Interior Hole Surfaces for One Hole** selection to add the six (6) selected surfaces. Then check the **Tight Fit** option next to each surface to indicate the bolt is a tight fit into the hole surfaces.

Tight Fit ♥ Tight Fit	1 <part 1=""> Surface 7 1 <part 1=""> Surface 7 2 <part 1=""> Surface 8 2 <part 2=""> Surface 1 2 <part 2=""> Surface 13 2 <part 2=""> Surface 2 2 <part 2=""> Surface 5</part></part></part></part></part></part></part>		Add
---	---	--	-----

- 35. Complete the following values to define the size of the bolt:
 - Enter 0.75 into the **Bolt Diameter** field
 - Enter 1.125 into the Head Diameter field
 - Enter 1.125 into the Nut Diameter field
 - Enter 500 into the Magnitude field for the Axial Force

Linear Static Stress

36. Click the **OK** button to complete the bolted connection. You will see several beam elements created within a new part that will represent the bolt and nut within the model. The blue dots are element nodes created at construction vertices that represent the bolt head and nut diameters.



37. Now we need to regenerate the mesh so that nodes will be created on the surface mesh of the clevis to connect to the new spoke nodes. The purpose of these construction vertices is to force the creation of a node at the same spot on the surface. To regenerate the mesh click **Mesh tab |** Mesh panel | Generate 3D Mesh.



Linear Static Stress

38. **Orbit** your model to a similar position as shown below so you can see the inside of the large diameter end of the *Yoke* part.



- 39. Change your selection settings to use **Point** as the **Shape** and **Surfaces** as the **Select** value.
- 40. Select the two (2) left side inner surfaces of the large diameter hole.



Linear Static Stress

41. Click the down arrow on the **Loads panel** of the **Setup tab**. Select the **Variable Pressure** load to start the assignment of a parabolic profile pressure load.



42. Select **Normal to Surface** for the **Load Orientation** then enter "*Parabolic Load*" as the **Active Function** value. For the **Expression** enter "-400*s^2+400".

Coordinate system	
⊿ ^f Global (Default)	
Load orientation	
Normal to surface	χ 0
Traction	Y 0
	Z 0
Function definition	
Active function	
If Parabolic Load	
Expression (use 'r', 's' and 't'	as variables)
-400*s^2+400	psi
r = X in s = Y in t = Z in	Available Primitives >> View
.oad case / load curve	
	× Curve
Description	

Linear Static Stress

43. Click the View button to open the Variable Load Viewer dialog. Then select S / Y as the Plot Direction.



- 44. You will notice the variable pressure load will ensure that the load magnitude will be 0 at the ends of the diameter. Click **Close** to exit the dialog.
- 45. Click **OK** to complete the load assignment.
- 46. Earlier when the bolt connection as added a third part was added to the model representing the bolt. Assign the material *Steel (AISI 4130)* to this *Part 3* component in the FEA Browser.

Linear Static Stress

47. The model is now ready to be solved.



48. The model is now ready to be solved. Click **Analysis tab | Analysis panel | Run Simulation** to start solving the model.



Linear Static Stress

- 49. Select Part 3 heading in the Browser and right-click Visibility off to hide the bolt.
- 50. Ensure that **Von Mises** is selected in the **Results Contours tab | Stress panel**. This will show the max stress of the model.
- 51. To display the maximum stress location click on **Results Inquire tab | Probes panel | Maximum**. This will place a probe at the maximum location of the model.



52. To display the displacement of the model click **Results Contours tab | Displacement panel | Displacement Magnitude**. The maximum probe will now show the new location which has a maximum displacement of 0.0008 in.

