

ME 24-688 – Week 10

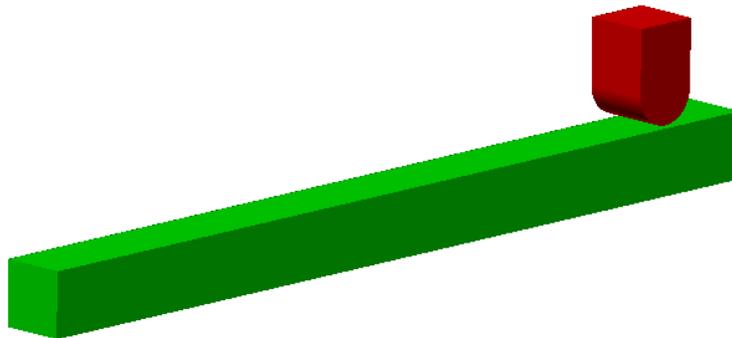
Nonlinear Static Stress

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.

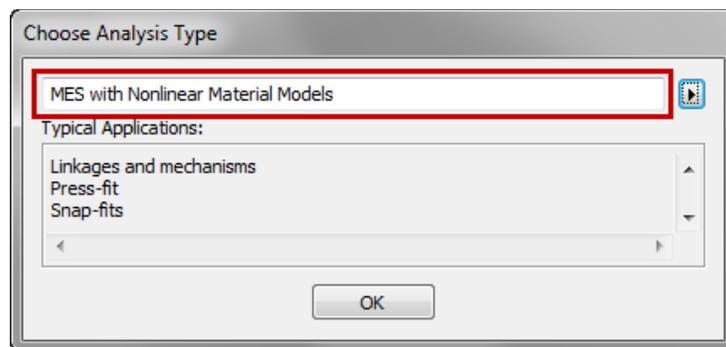


1. Click on the **Open** command from the Launch panel of Autodesk Simulation 2012.
2. Set the **Files of Type** setting to *STEP* to only display STEP files. STEP files are neutral 3D CAD formats exported from most modern CAD systems.

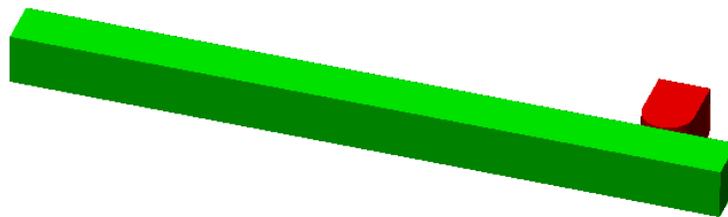
ME 24-688 – Week 10

Nonlinear Static Stress

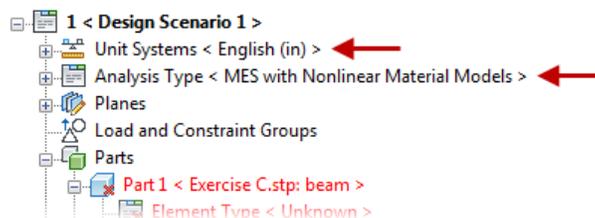
3. Select the “*Beam and Ram Assembly.step*” file and click **Open**.
4. Chose the option to “*Use STEP file units*” to set the length unit value to match the value in the STEP file.
5. Select “*MES with Nonlinear Material Models*” as the **Analysis Type** and click **Ok**. This will be used to conduct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



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.



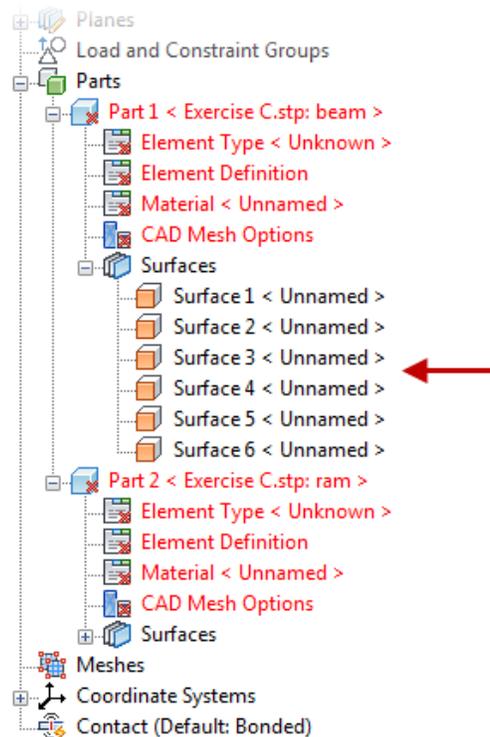
7. Within the **FEA Editor Browser** you will notice that the units are set to “*English (in)*” and the **Analysis Type** is set. You can change these at any time within the browser if required.



ME 24-688 – Week 10

Nonlinear Static Stress

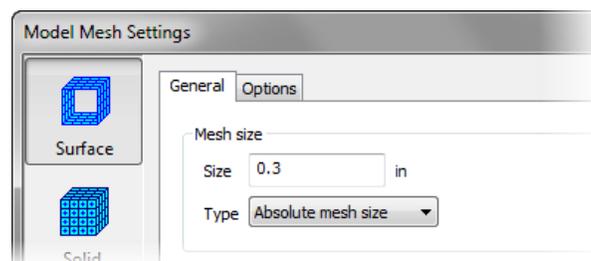
- Expand the **Surfaces** group of *Part 1* in the browser as shown below. You will notice that there is a surface for each exterior surface of the CAD model.



- 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.

- Click on the **Options** button

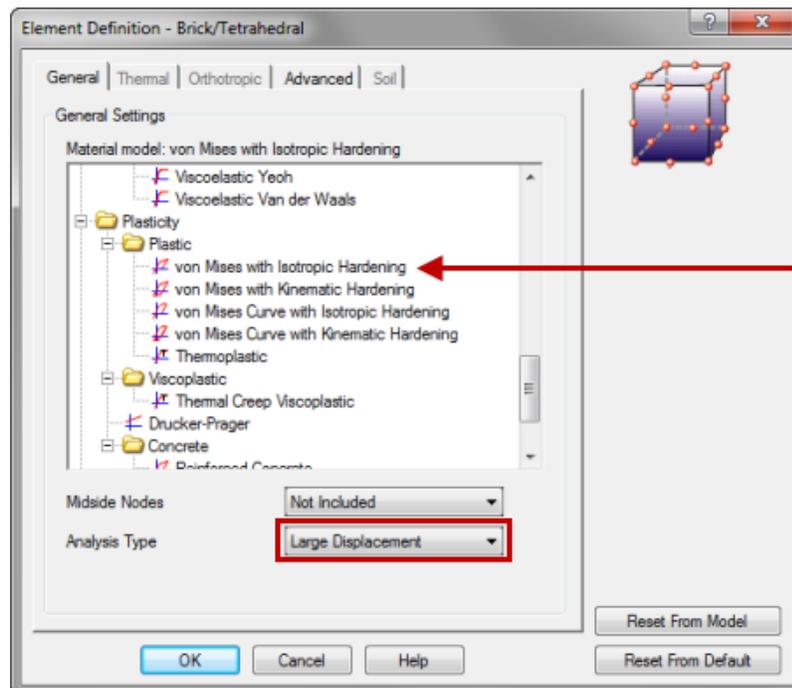
- Select *Absolute Mesh Size* for the surface mesh **Type**. Then enter **0.3** as the **Mesh Size** within the **General** tab.



ME 24-688 – Week 10

Nonlinear Static Stress

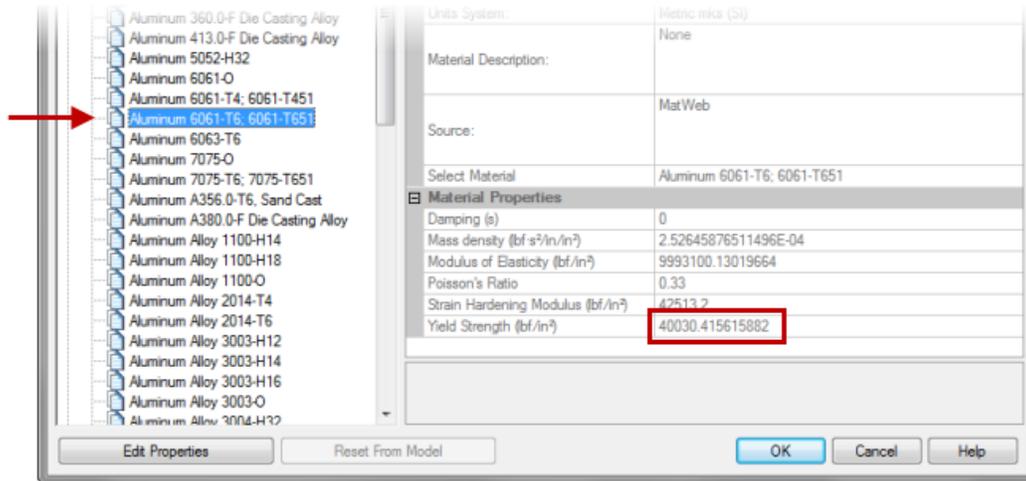
- Click **OK** button to close the options. Then press **Mesh Model** to start the meshing process of the model. Autodesk Simulation first meshes the exterior surfaces then checks to ensure the volume is closed of a 3D solid model. Then proceeds and meshes the interior areas by adding elements.
- Press **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.
- 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**.
- 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.
- Then ensure that *Large Displacement* is selected as the **Analysis Type** as the part will have large deformations in this nonlinear analysis.



ME 24-688 – Week 10

Nonlinear Static Stress

- 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.
- 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 than a steel material.

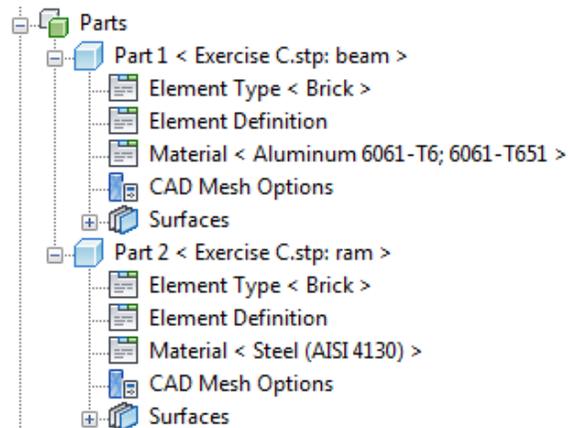


- Click **OK** to exit the **Element Material Selection** dialog and assign the material.
- Right-Click on the **Element Definition** heading of *Part 2* and select **Edit Element Definition**.
- Then ensure that *Isotropic* is selected as the **Material Model**. Then also ensure that *Large Displacement*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

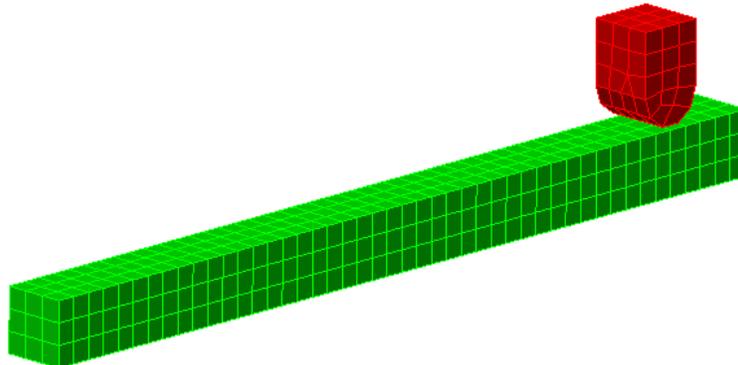
ME 24-688 – Week 10

Nonlinear Static Stress

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

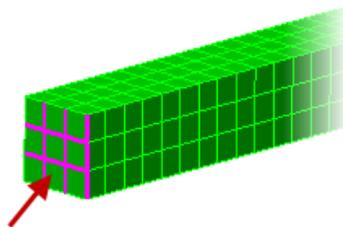


25. **Orbit** to position the model in a position as shown below.



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.

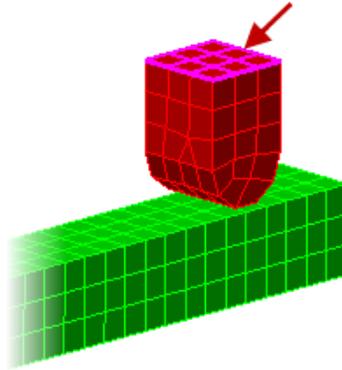


ME 24-688 – Week 10

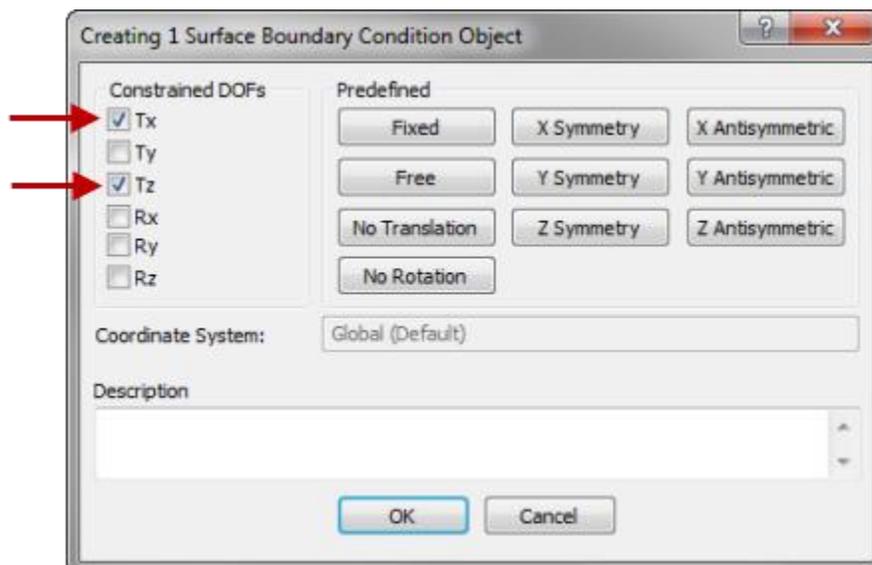
Nonlinear Static Stress

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.



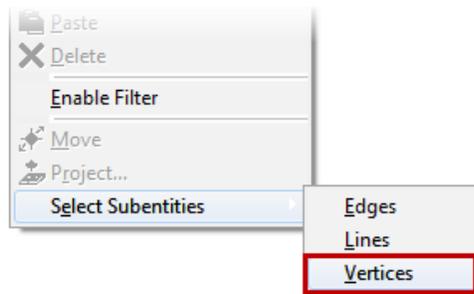
30. To ensure the ram part stays in position and only moves along the Y axis we will add a new constraint. Click **Setup tab | Constraints panel | General Constraint** then select **Tx** and **Tz** to lock the translational movement in the X and Z axis of the nodes. Click **OK** to complete the assignment.



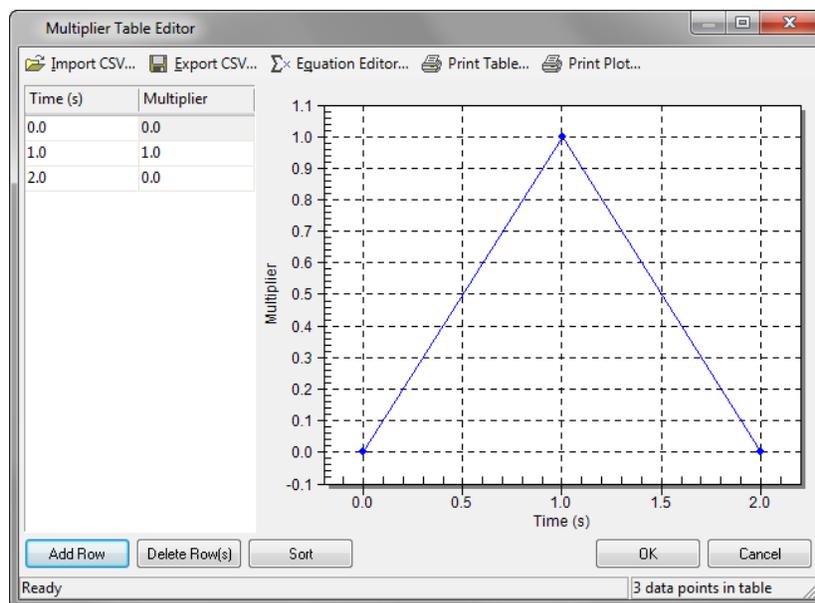
ME 24-688 – Week 10

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.



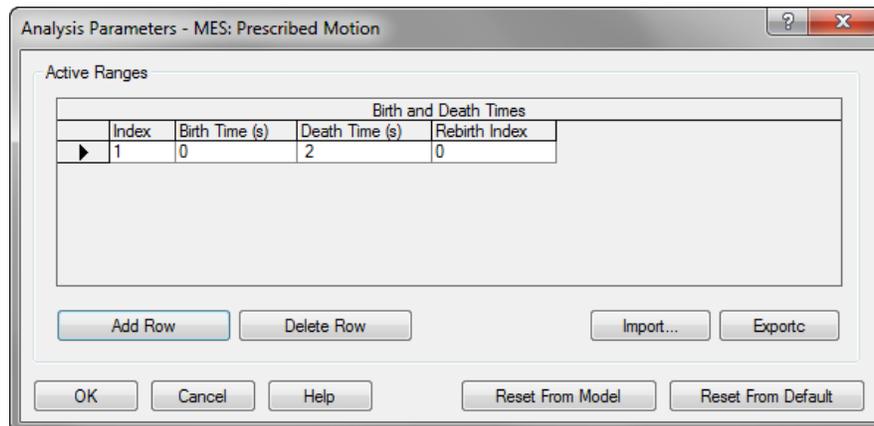
32. With the nodes selected we will assign a displacement movement. Click **Setup tab | Constraints panel | Prescribed Displacement** to open the **Prescribed Displacement** dialog. We will use a translation displacement and enter **-0.6 Magnitude** value. Then select **Scalar Y** as the **Direction** 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.



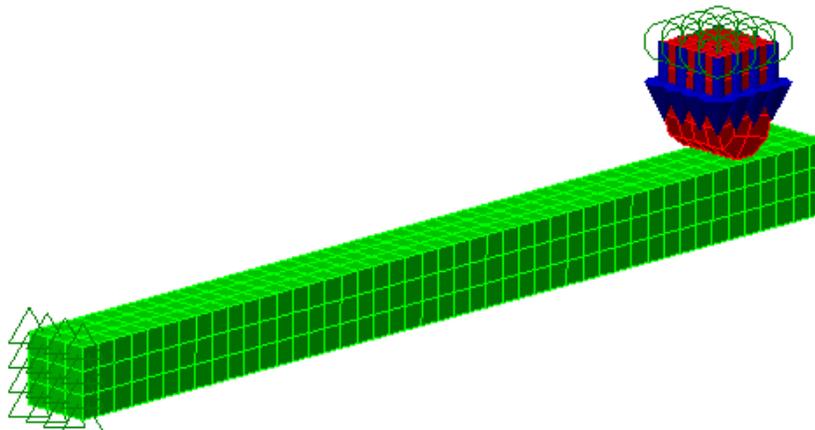
ME 24-688 – Week 10

Nonlinear Static Stress

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.



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

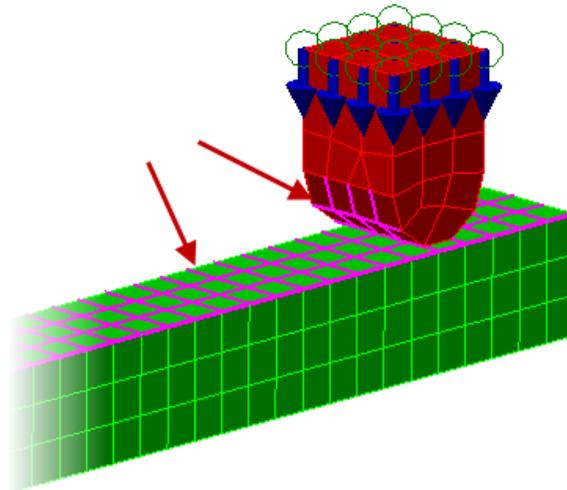


ME 24-688 – Week 10

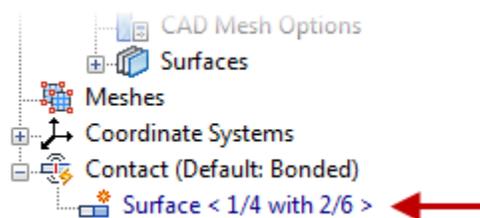
Nonlinear Static Stress

39. Now we need to start the process of assigning the contact 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 contact relationship between the two surfaces. Hit the **Enter** key to accept the default contact pair name in the browser to complete the command.

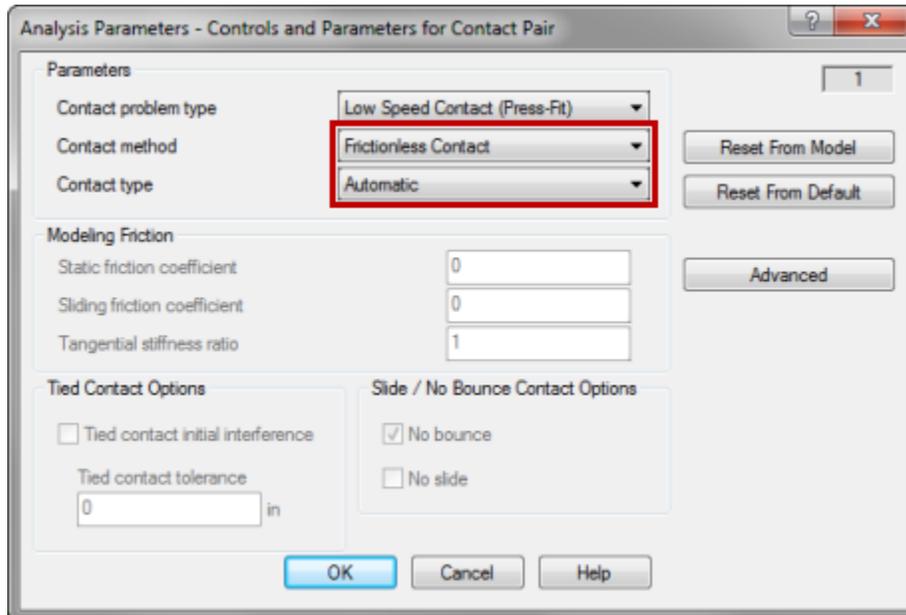


42. Select the *Surface <1/4 with 2/6>* contact in the browser then right-click and select **Settings** to open the **Controls and Parameters for Contact Pair** dialog.

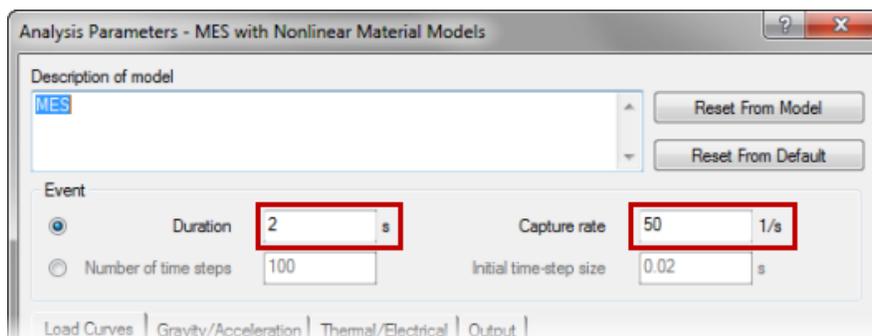
ME 24-688 – Week 10

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.



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.



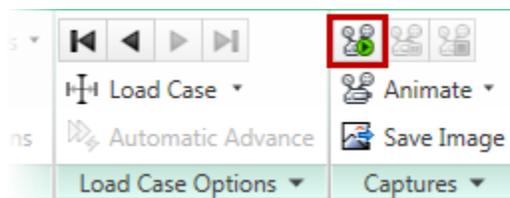
ME 24-688 – Week 10

Nonlinear Static Stress

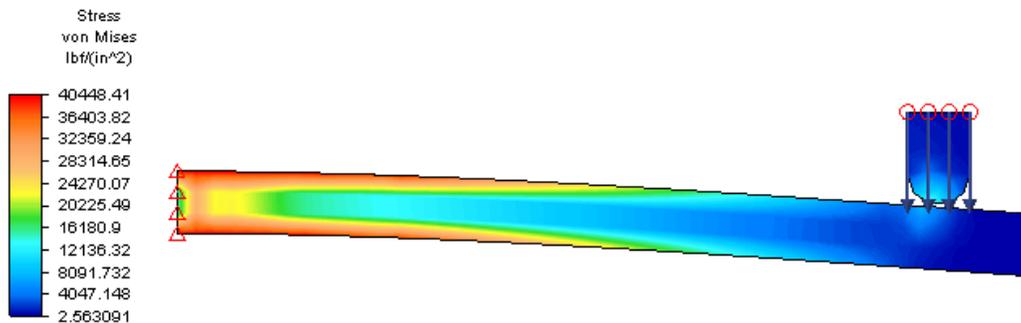
45. To run the simulation click **Analysis tab | Analysis panel | Run Simulation**. The complete mechanical event simulation will be solved now.

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 it is cycling through each of the 100 time steps and the Von Mises Stress Legend values are updating at each step.

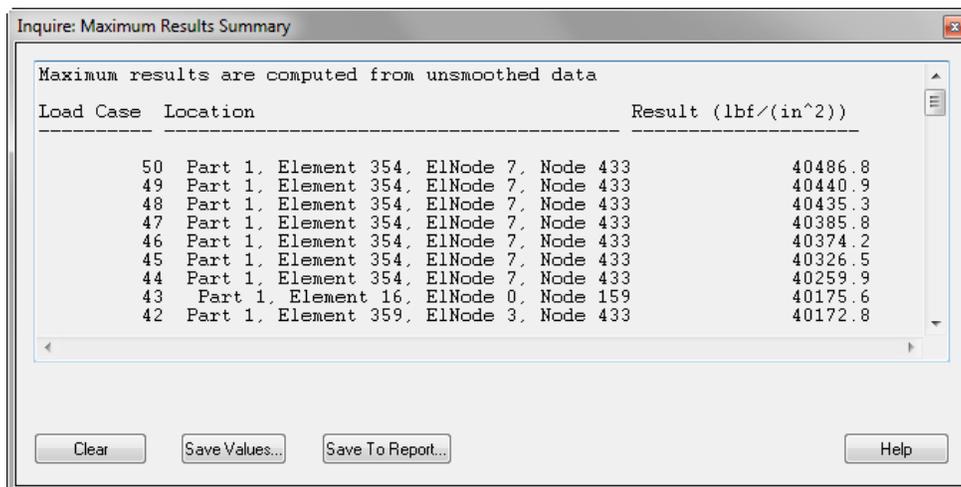


49. Click **Stop** under the **Captures** panel when done.

ME 24-688 – Week 10

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*



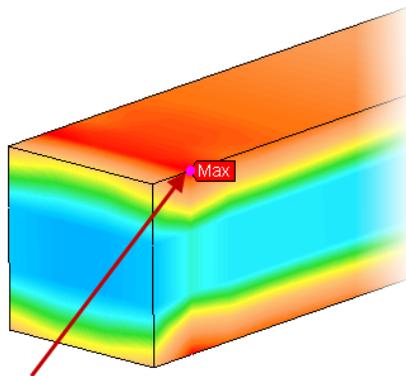
ME 24-688 – Week 10

Nonlinear Static Stress

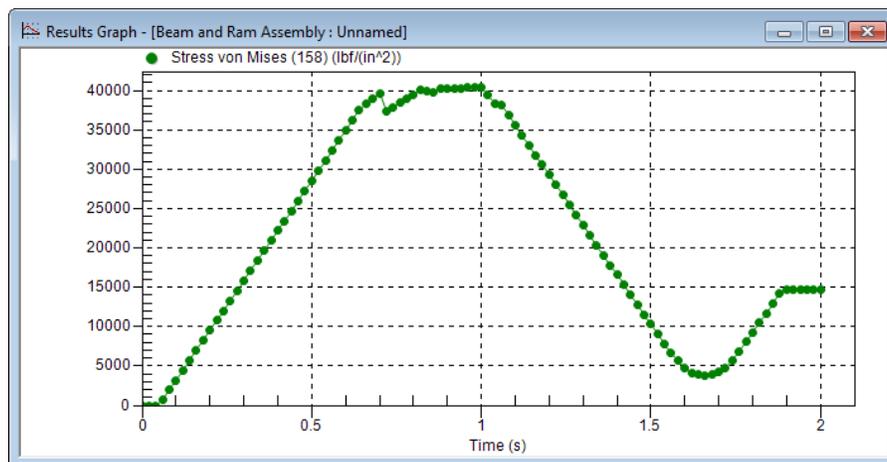
55. Click back to the **Results Contours** tab and ensure **Von Mises** is selected in the **Stress** panel.

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 Hardening 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

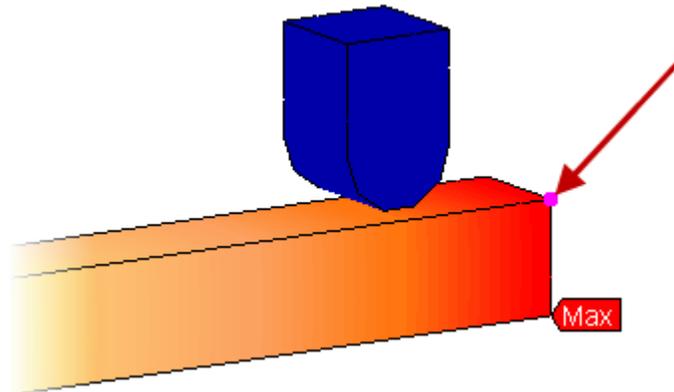
59. **Close** the graph display window.

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.

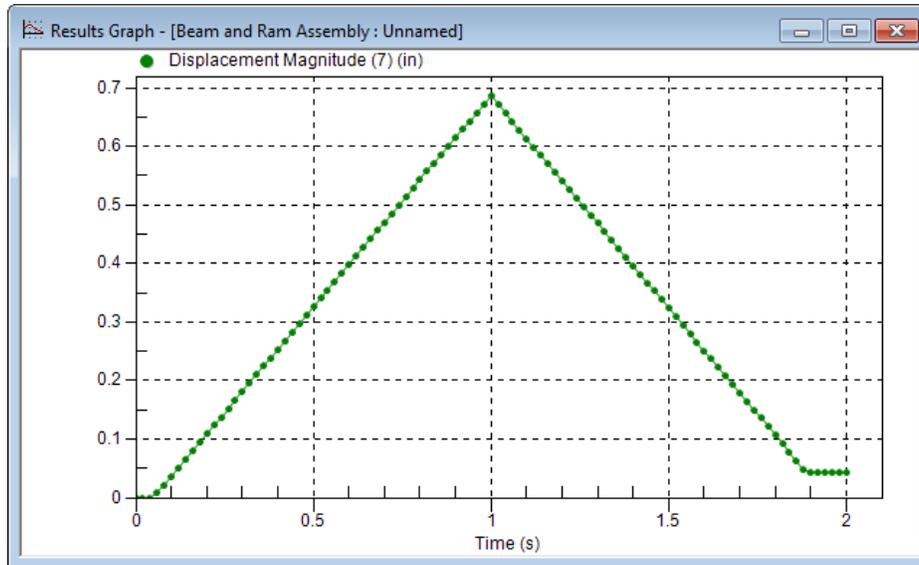


64. Right-Click in the graphics window and select **Graph Value(s)**. This will open a new window with a graph of the selected node showing the displacement at each load case.

ME 24-688 – Week 10

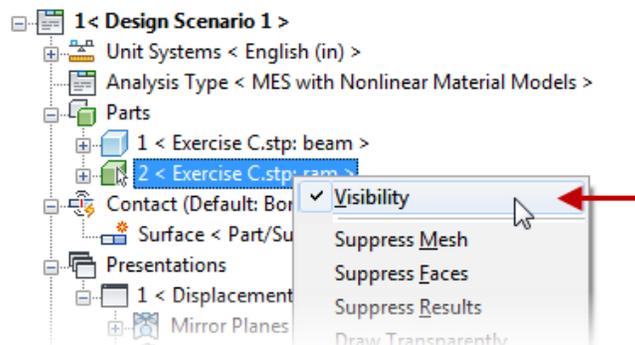
Nonlinear Static Stress

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.

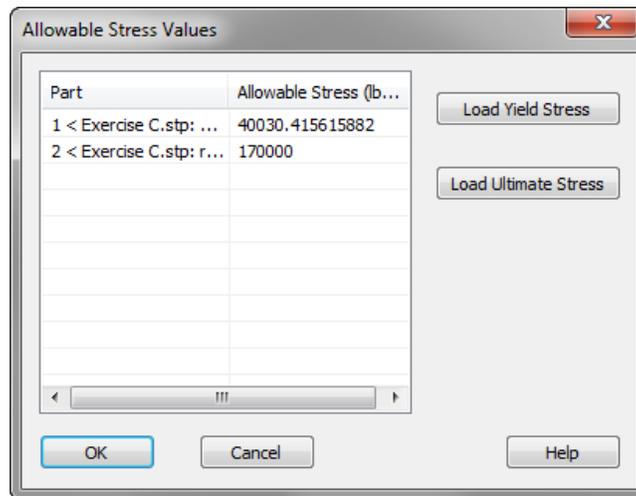
67. Now that we know the beam part is our main concern we can turn off the ram part from the results. Select the *Part 2* in the **Results Browser** and right-click to turn the **Visibility** off.



ME 24-688 – Week 10

Nonlinear Static Stress

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.



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.

