ME 24-688 – Week 9

Viewing Analysis Results

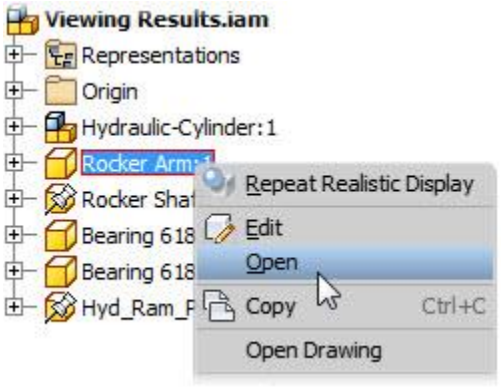
Viewing Analysis Results

1.1 Project 2 – Viewing Analysis Results

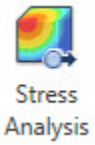
In this project you learn additional stress analysis options and how to thoroughly review the simulation results.

1. Open *Viewing Results.iam* from the location of your project files.

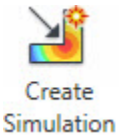
2. Right-click over *Rocker Arm:1* in the Browser and choose **Open** from the Browser menu.



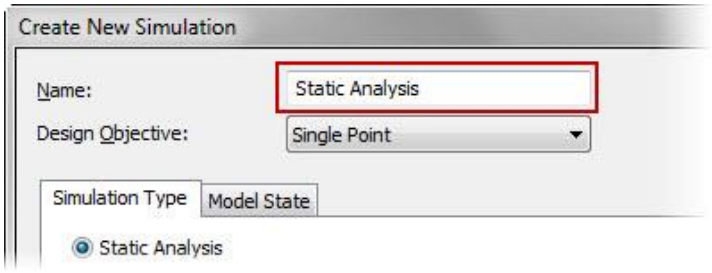
3. Enter the **Stress Analysis** environment by picking **Environments | Begin | Stress Analysis** from the Ribbon.



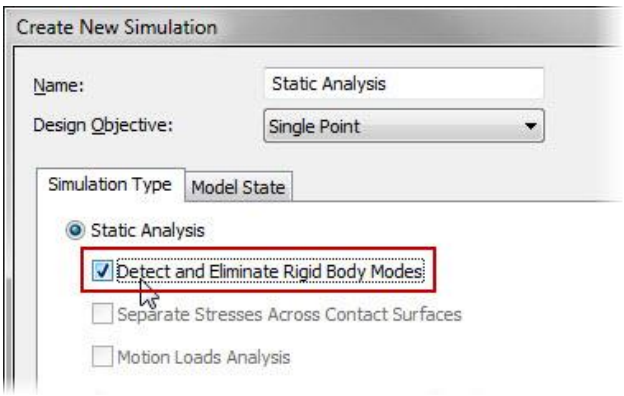
4. On the **Manage** panel, click **Create Simulation**.



5. In the **Create New Simulation** dialog, enter **Static Analysis** for the **Name**.



6. In the **Create New Simulation** dialog, select the **Detect and Eliminate Rigid Body Modes** check box to turn it on.

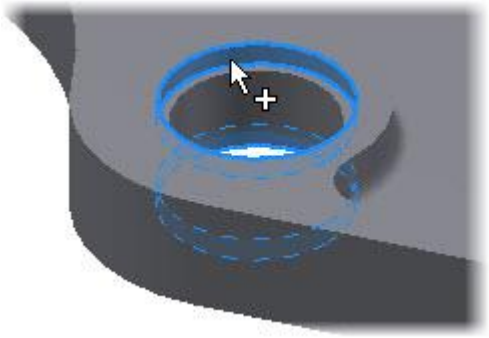


7. Click **OK** to dismiss the **Create New Simulation** dialog.

8. You will now simplify the model by suppressing features of the model to reduce elements in the mesh. This process is used to improve speed of the simulation and increase focus on the important areas of concern. Expand the *Rocker Arm* part node in the Browser and right-click over *ExteriorFillets*. Choose **Exclude from Simulation** on the Browser menu.

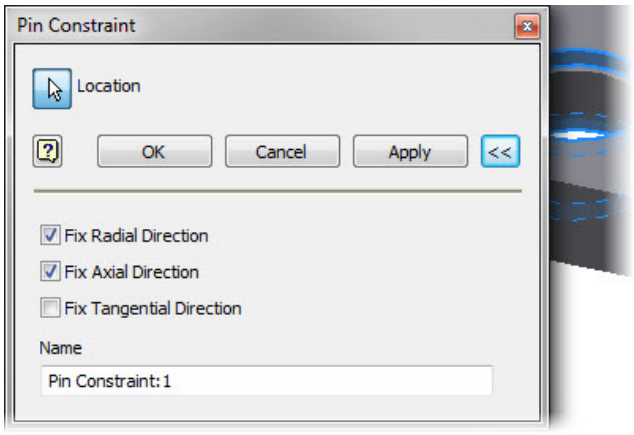
9. On the **Constraints** panel, click **Pin**.

   - Select the two cylindrical faces that contact the bearings.



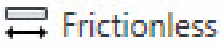
10. Remaining in the **Pin Constraint** dialog:

    - Click **>>** to expand the dialog.
    - Ensure the **Fixed Radial Direction** option is checked.
    - Ensure the **Fix Axial Direction** option is checked.
    - Ensure the **Fix Tangential Direction** option is unchecked.

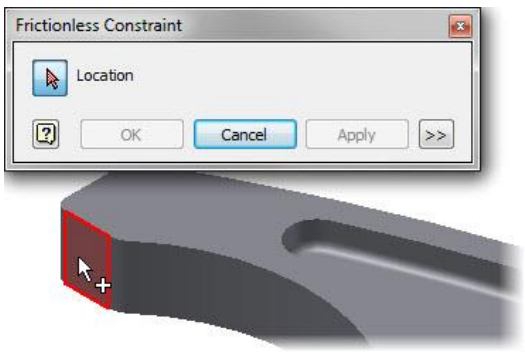


    - When you have confirmed these settings, click **OK** to dismiss the dialog.

11. To allow the part clamping surface to move and slide but not pull away we will add a frictionless constraint. Do this by clicking **Constraints panel | Frictionless**.



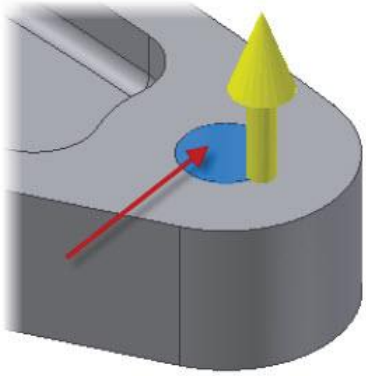
    - Select the face as shown in the figure below.



    - Click **OK** to dismiss the dialog.

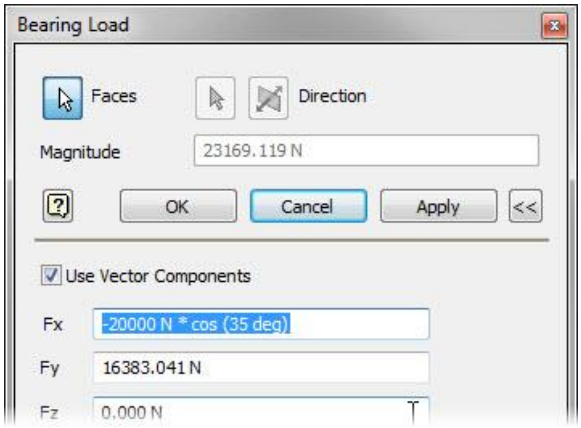
12. Start the **Bearing Load** feature by choosing **Loads | Bearing** from the Ribbon.

- Select the circular face on the inside of the hole.



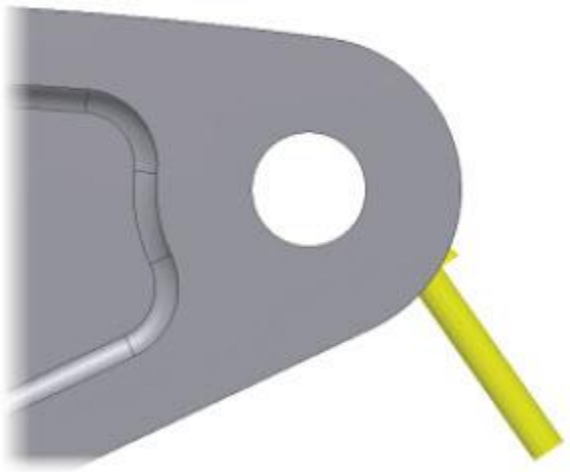
13. In the **Bearing Loads** dialog enter the following directional information that was direction from an assembly motion analysis:

- Pick **>>** to expand the dialog.
- Select the **Use Vector Components** check box to specify the force magnitude and direction.
- For **Fx**, enter **-20000 N * sin(35 deg)**. Note the negative sign. Also note that the expression is evaluated and replaced with the value when the next field is picked.
- For **Fy**, enter **20000 N * cos(35 deg)**.



- Click **OK** to dismiss the dialog.

14. View the model from the front and confirm that the load direction is as shown. If the load is not oriented correctly, verify and edit the vector component values.



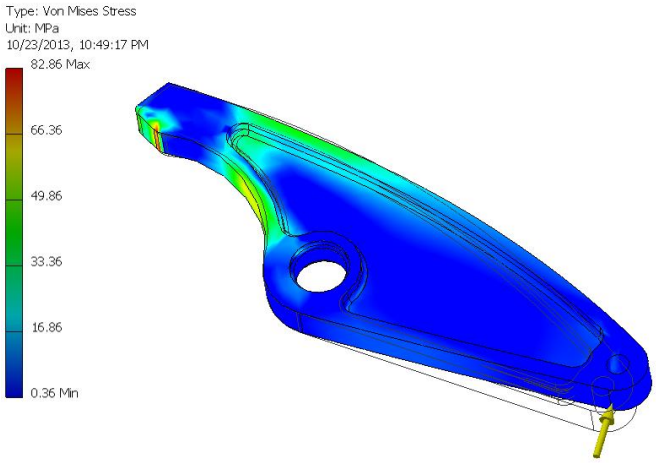
15. To ensure the proper material is assigned to the part click **Material panel | Assign** to verify *Steel, Mild* is the original material. The material property was assigned at the part level.

ME 24-688 – Week 9

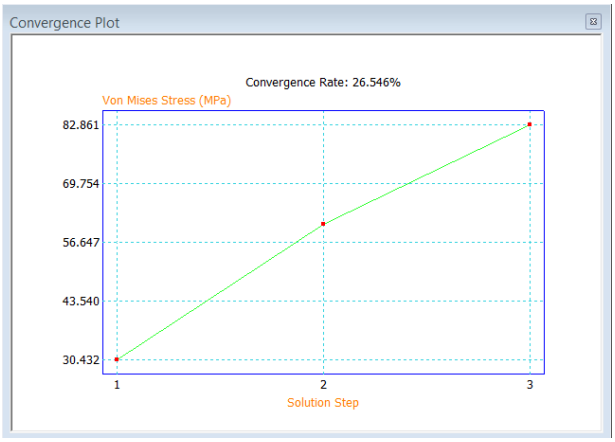
Viewing Analysis Results

16. Start the **Simulate** dialog by choosing **Solve | Simulate** from the Ribbon or **Simulate** from the Marking Menu. Pick **Run** to continue.

- The *Von Mises Stress* is displayed, as shown below.



17. After reviewing the results you will notice a high stress area near the clamping surface. Review the convergence by clicking **Result panel | Convergence** to open the **Convergence Plot** dialog. You will see the simulation as not converged yet with a rate of 26 ~~25~~%. We will cover this in greater detail later.

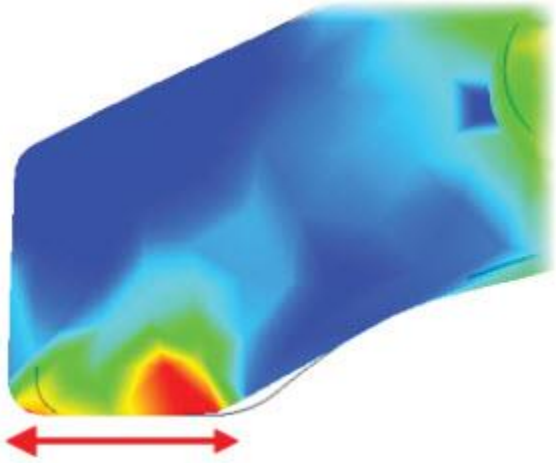


18. Change your view to the **Front** of the model.

19. Pick **Result | Animate** from the Ribbon to bring up the **Animate Results** dialog.

- When the **Animate Results** dialog appears, choose **Fastest** from the **Speed** list and click **Play**.

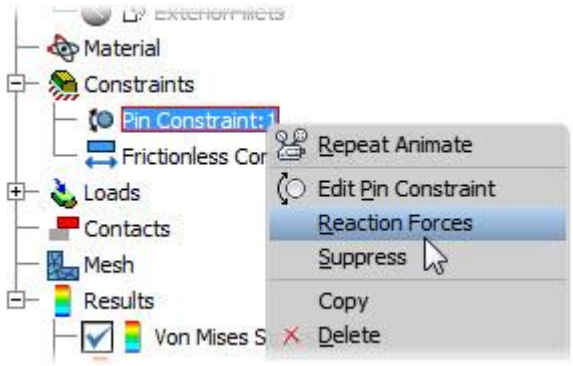
20. Zoom in to the left end of the *Rocker Arm*. Confirm that the model moves sideways along the frictionless surface, not up and down.



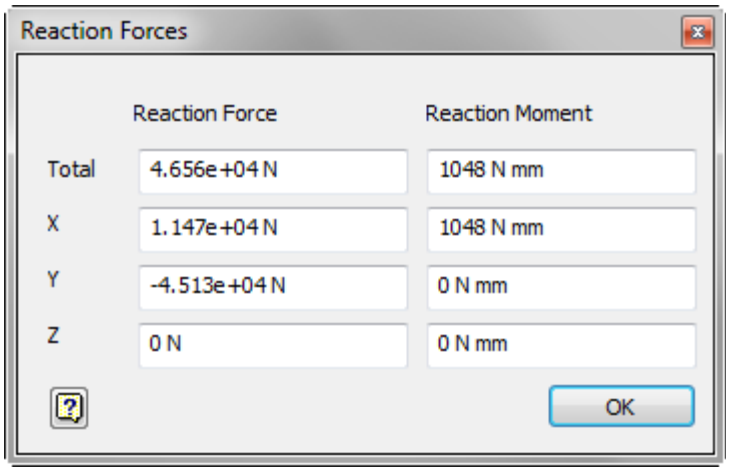
21. Click **OK** to dismiss the **Animate Results** dialog.

- Return to the **Home** view.

22. Expand the **Constraints** node in the Browser. Right-click the *Pin Constraint* and choose **Reaction Forces** from the Browser menu.



- Review the forces and moments. As expected, the Y moment is zero.



- Click **OK** to dismiss the dialog.

ME 24-688 – Week 9

Viewing Analysis Results

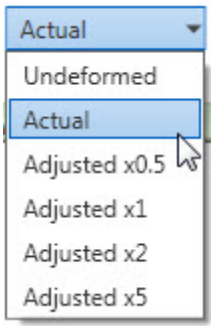
23. On the **Display** panel, toggle each of the following and make note of the effect:

    - **Maximum Value**

    - **Minimum Value**

    - **Boundary Conditions**

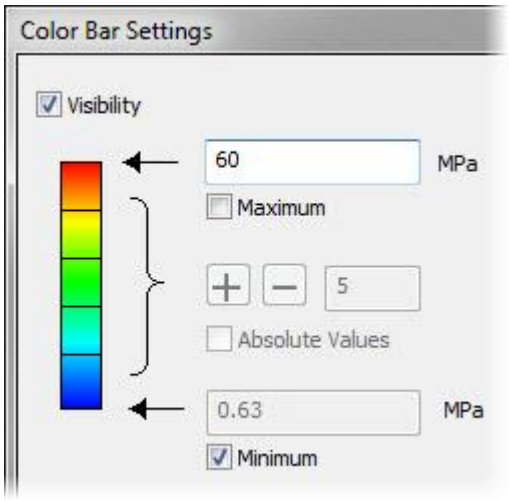
24. On the **Display** panel, from the **Adjust Displacement Scale** list, do the following:



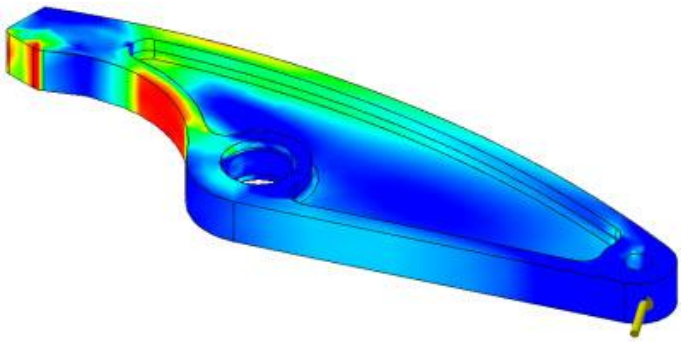
    - Select **Actual**. The actual deformation is small.
    - Select **Adjusted X5**. The deformation is greatly exaggerated.
    - Select **Adjusted X1** to return the displacement scale to the default value.

25. On the **Display** panel, click **Color Bar**.

- In the **Color Bar Settings** dialog box, do the following:



- Clear the **Maximum** check box
- For the **Maximum** value, enter **60**
- Click **Apply**

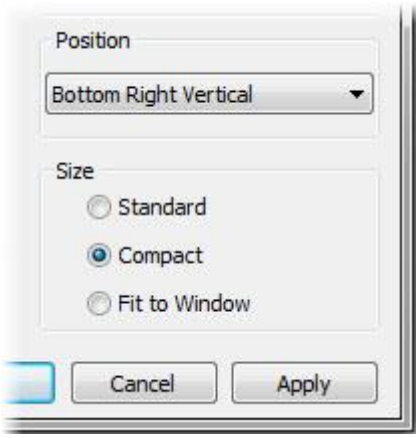


- The results update on the model.

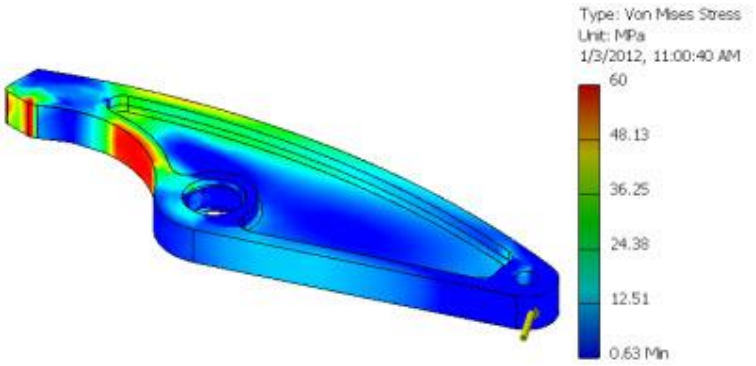
ME 24-688 – Week 9

Viewing Analysis Results

26. In the **Color Bar Settings** dialog box, do the following:



- Under **Position**, select **Bottom Right Vertical**
- Under **Size**, click **Compact**
- Click **OK**



27. In the Browser, under the **Results** folder, expand all of the subfolders. Do the following:

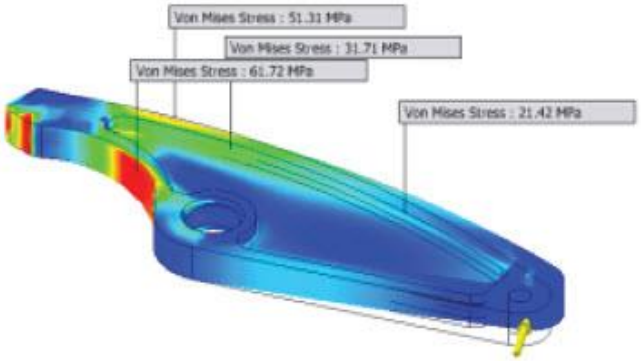
- Under **Strain**, double-click **Equivalent Strain**. Review the results.
- Under **Results**, double-click **Displacement**. Review the displacement.
- Review the **Safety Factor**.
- Return to the *Von Mises Stress* view.

ME 24-688 – Week 9

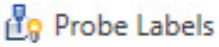
Viewing Analysis Results

28. On the **Result** panel, pick **Probe**.

- Select a few locations on the model to display the results.



29. On the **Display** Panel, click **Probe Labels** to turn off the labels.



30. Begin creating a report by picking **Report | Report** from the Ribbon.

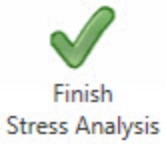
31. In the **Report** dialog box, do the following:

- For **Report Title**, enter **Rocker Arm Stress Analysis**.
- For **Report Author**, enter **your name**.
- For **Summary**, enter **Stress analysis results for rocker arm**.
- Under **Report location**, hover the cursor over the **Path** entry. Review the location. If necessary, change the location to one that you can easily locate.
- Review the **Properties** tab.
- Click the **Simulations** tab.
- Under **Results**, clear the **Stress** and **Strain** check boxes.
- Click **OK**.

32. Review the report in your Internet browser.

33. Close the report in your Internet browser.

34. Leave the **Stress Analysis** environment by picking **Exit | Finish Stress Analysis** from the Ribbon.



35. Close all files without saving.