ME 24-688 – Week 9

Design Optimization & Convergence

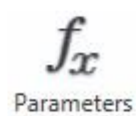
Design Optimization and Convergence

1.1 Project 4A – Design Optimization Study

In this project, you analyze several configurations of a link arm to determine the effect of varying the thickness and adding a rib. Using a parametric study to perform the analysis, you study six different combinations of parameters in just one simulation run.

1. Open *Parametric Analysis.ipt* from the location of your project files.

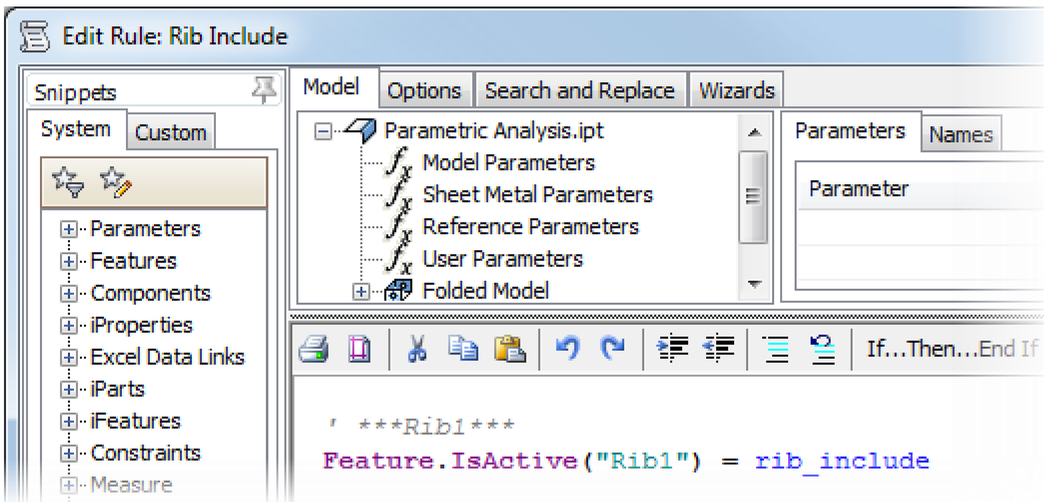
2. Click **Manage tab | Parameters panel | Parameters** from the Ribbon to start the **Parameters** dialog.



Under **User Parameters**, there is a single parameter named *rib_include* that is equal to 1.

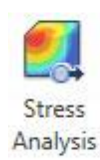
3. Close the **Parameters** dialog.

4. Click **Manage** tab **|** **iLogic** panel **|** **iLogic Browser**, in the iLogic Browser right-click *Rib Include* rule and select **Edit Rule** from the menu. Notice that the rib feature is suppressed if the parameter *rib_include* is equal to `0`.



Close the **Edit Rule** dialog.

5. Enter the **Stress Analysis** environment by picking **Environments tab | Begin panel | Stress Analysis** from the Ribbon.



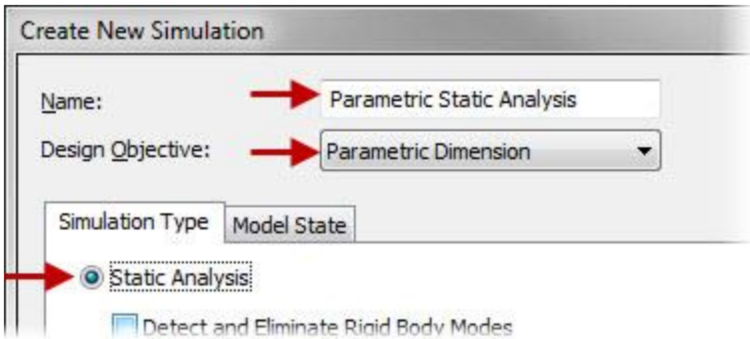
6. On the **Manage** panel, click **Create Simulation**.

7. In the **Create New Simulation** dialog, do the following:

   ■ Enter `Parametric Static Analysis` for the **Name**.

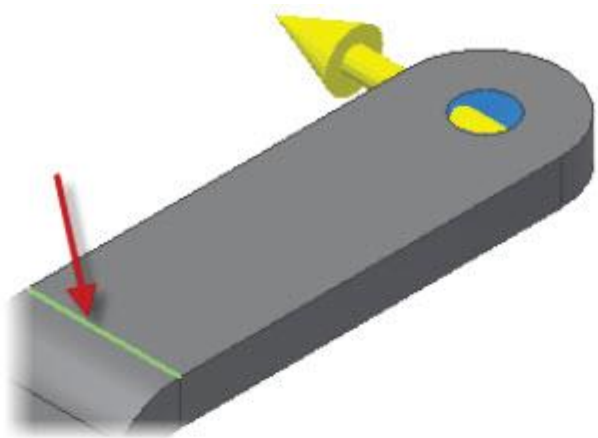
   ■ Confirm that **Static Analysis** is selected on the **Simulation Type** tab.

   ■ Choose **Parametric Dimension** from the **Design Objective** list.



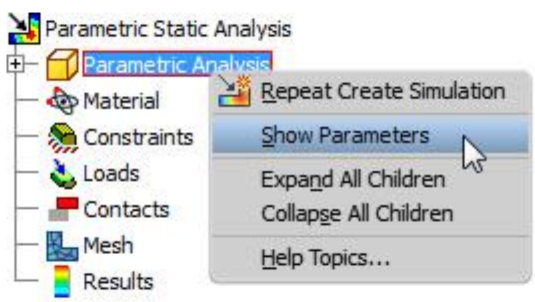
   ■ Click **OK** to dismiss the dialog.

8. Add a `2000N` **Bearing** load to the hole as shown. Align the force using the edge of the face as shown.



9. Add **Fixed** constraints to four flat inside faces of the square hole.

10. In the Browser, under the **Parametric Static Analysis** node, right-click **Parametric Analysis** and choose **Show Parameters** from the Browser menu.



11. In the **Select Parameters** dialog:

■ Under **Model Parameters**, click the check box next to *Thickness*.

■ Under **User Parameters**, click the check box next to *rib_include*.

■ Click **OK**

12. Choose **Environments tab | Manage panel | Parametric Table** from the Ribbon.

13. In the **Parametric Table** dialog, under **Parameters**, do the following:

    ■ In the **Values** column, for *Thickness*, enter `12,16,20`.

    ■ In the **Values** column, for *rib_include*, enter `0,1`.

    ■ Right-click one of the column headers and choose **Best Fit** (all columns).

Parameter Name	Values		Curr
Thickness	12,16,20		20
rib_include	0,1		1

14. Under **Parameters**, right-click anywhere within the grid. Choose **Generate All Configurations** from the menu.

15. Select different parameter values using the value sliders and notice how the model updates in the graphics window.

16. Right-click any cell and choose **Show Base Configuration** from the menu.

17. In the **Parametric Table** dialog, under **Design Constraints**, do the following:

    ■ Right-click in the first empty row. Click **Add Design Constraint**.

    ■ Select **Von Mises Stress**. Click **OK**.

    ■ Repeat to add **Mass** and **X Displacement**.

    ■ Click the **Constraint Type** column for Max Von Mises Stress.

    ■ Select **Upper Limit** from the list. In the **Limit** column, enter `250`.

    ■ In the **Safety Factor** column, enter `2`.

Parametric Table

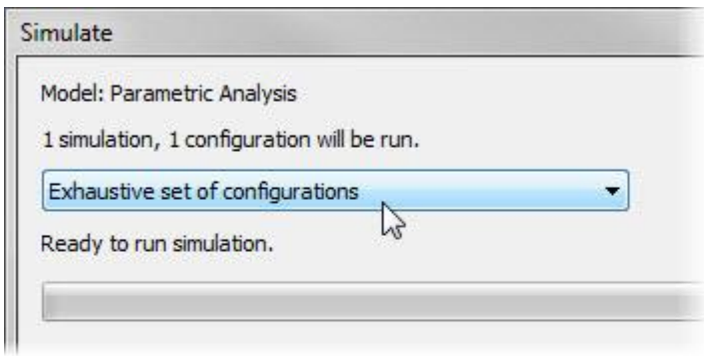
Design Constraints

Constraint Name	Constraint Type	Limit	Safety Factor
Max Von Mises Stress	Upper limit	250	2
Mass	View the value		
Max X Displacement	View the value		

- ■ Close the **Parametric Table** interface.

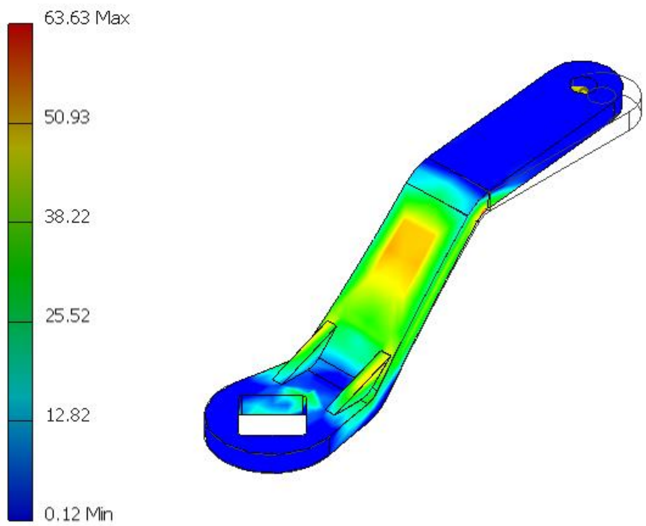
18. Start the **Simulate** dialog by choosing **Environments tab | Solve panel | Simulate** from the Ribbon or **Simulate** from the Marking Menu.

In the **Simulate** dialog, select **Exhaustive Set of Configurations**.



Pick **Run** to continue.

19. The *Von Mises Stress* is displayed.



20. Choose **Environments tab | Manage panel | Parametric Table** from the Ribbon.

21. In the **Parametric Table** dialog box, observe the values of the stress, mass, and displacement as you select different parameter values using the value sliders.

22. Select different parameter values using the value sliders and observe the *Von Mises Stress* in the graphics window. Notice that the scale for the color bar stays the same for all configurations.

23. Pick **Environments tab | Display panel | Same Scale** from the Ribbon to turn it off.

24. Select different parameter values again and notice how the scale for the color bar changes to match each configuration's stress results.

25. Pick **Environments tab | Display panel | Same Scale** from the Ribbon to turn it back on.

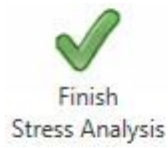
26. Under **Design Constraints**, in the **Constraint Type** column for **Mass**, select **Minimize**. The combination of parameters is found that results in the minimum mass while meeting the stress criteria.

Constraint Type	Limit	Safet	Result Value	Unit
Upper limit	250	2	122.259	MPa
Minimize			5.07448	kg
View the value			1.36832	mm

27. Under **Parameters**, right-click in the table area. Click **Promote Configuration to Model** and choose **Yes** when prompted.

28. Close the **Parametric Table** dialog.

29. Exit the **Stress Analysis** environment by picking **Exit panel | Finish Stress Analysis** from the Ribbon.



30. In the part environment, review the updated model.

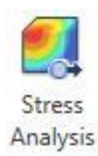
31. Close all files without saving.

1.2 Project 4B – Result Convergence

In this project, you perform a convergence study to determine if results are converged. You see how common causes of stress singularities prevent stress results from converging. You then correct some of the singularities and perform a convergence study.

1. Open *Stress Convergence.ipt* from the location of your project files.

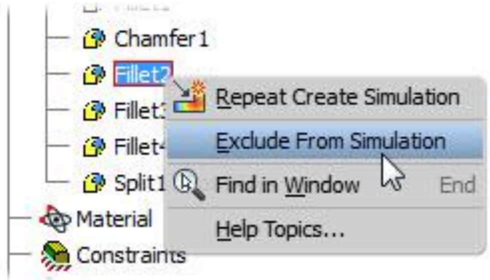
2. Enter the **Stress Analysis** environment by picking **Environments tab | Begin panel | Stress Analysis** from the Ribbon.



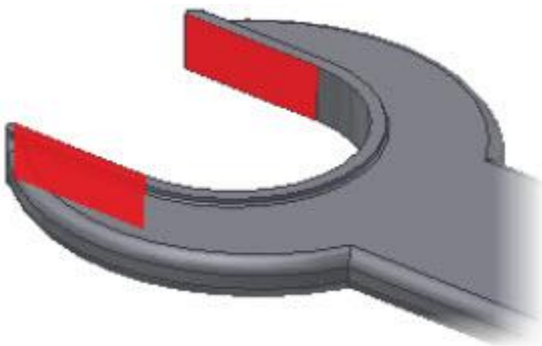
3. Under the **Environments** tab, on the **Manage** panel, click **Create Simulation**.

   Accept all of the default settings by clicking **OK** to continue.

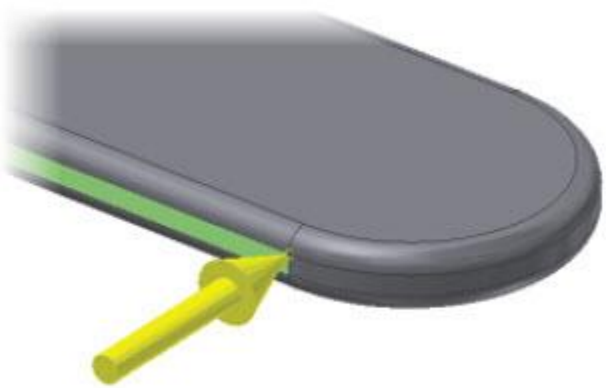
4. Exclude *Fillet2* from the simulation.



5. Add a **Fixed** constraint to the two faces as shown.



6. Add a `700N` force to the **Vertex** on the handle as shown. Use the adjacent flat face to specify the load direction.



7. On the **Mesh** panel, click **Mesh Settings** . Do the following:

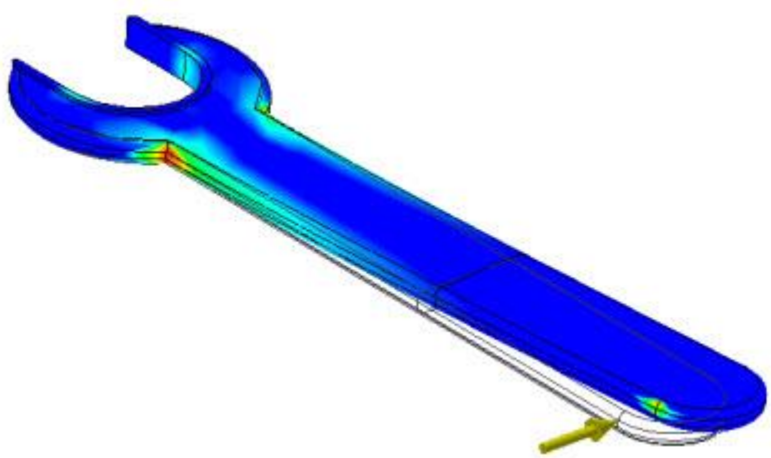
   ■ Confirm that the **Create Curved Mesh Elements** check box is checked.

   ■ Click **Cancel**.

8. Start the **Simulate** dialog by choosing **Environments tab | Solve panel | Simulate** from the Ribbon or **Simulate** from the Marking Menu. Pick **Run** to continue.

9. On the **Display** panel, click **Show Maximum Value**.

10. Review the stress results. The area under the load has high stress; however, the stress is not realistic because the load was applied to a vertex. This is known as a stress singularity.



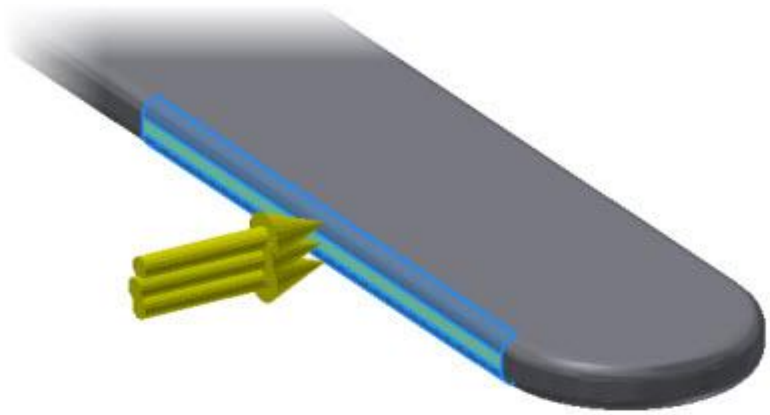
11. In the browser, expand **Loads**. Double-click the **force** load.

12. To remove the load at the vertex and move it to a face, do the following:

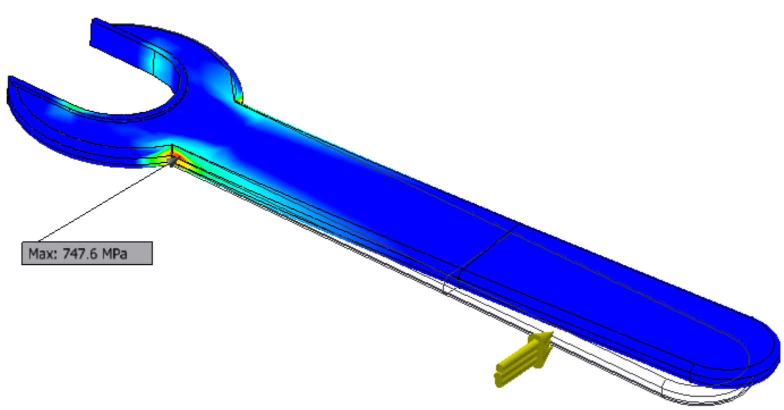
    ■ Press and hold down the CTRL key while selecting the vertex where the load is applied. Release the CTRL key.

    ■ Select the three adjacent faces as shown.

    ■ Click **OK**.



13. Run the simulation. The maximum stress is now at the inside corner.



14. To review if the results have converged click on **Results panel | Convergence Plot** to open the **Convergence Plot** dialog. You will see the results are still over 30% so they should not be trusted at this time.

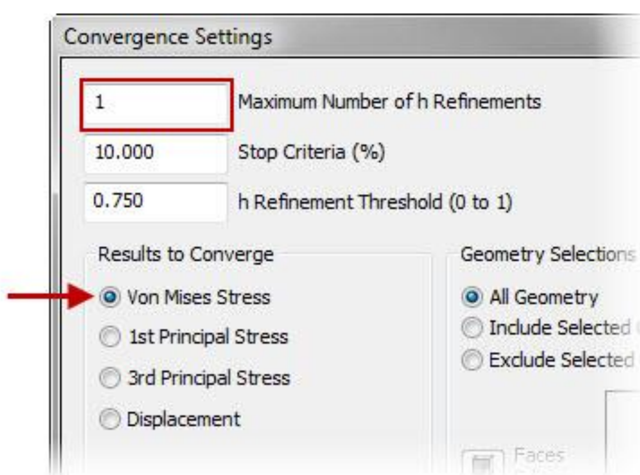
15. Now that you have removed the obvious stress singularity at the load, you perform a convergence study to see if the results are converged.

    On the **Mesh** panel, click **Convergence Settings**. Do the following:

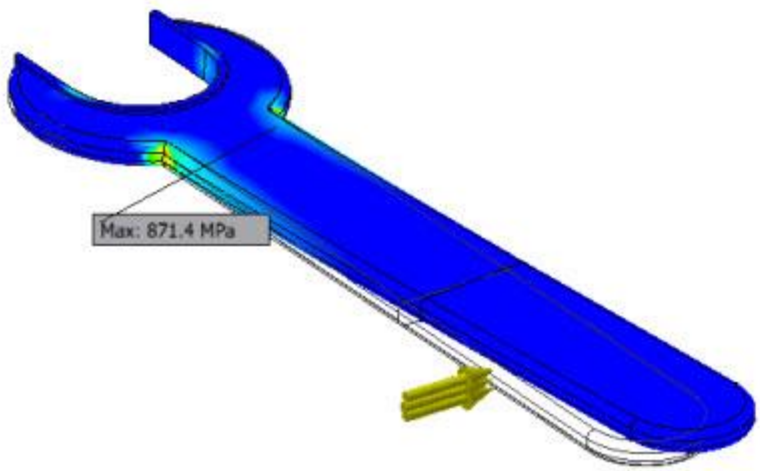
    ■ For **Maximum Number of H Refinements**, enter `1`.

    ■ Under Results to Converge, confirm that **Von Mises Stress** is selected.

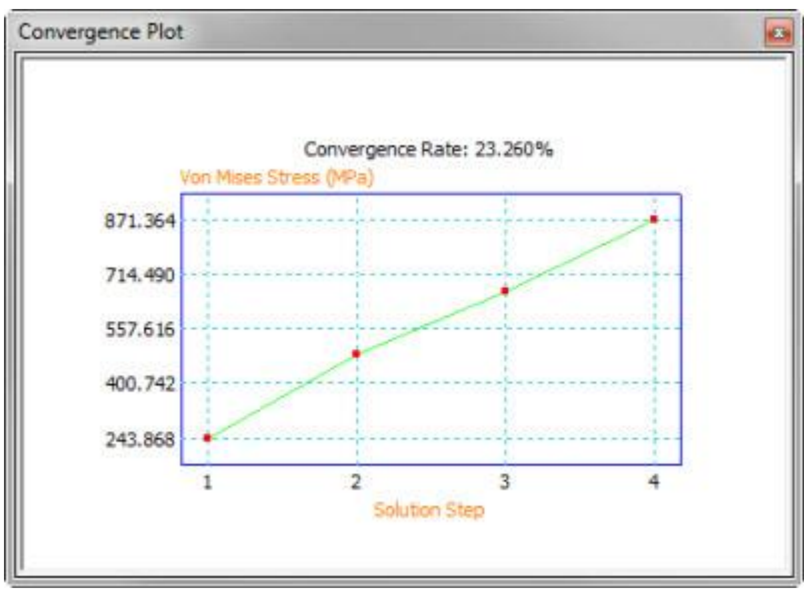
    ■ Click **OK**.



16. Run the simulation. The stress at the sharp corner has increased to over 800 MPa.



17. On the **Result** panel, click **Convergence Plot**. The sharp corner prevents the results from converging so more work is required to ensure good results.



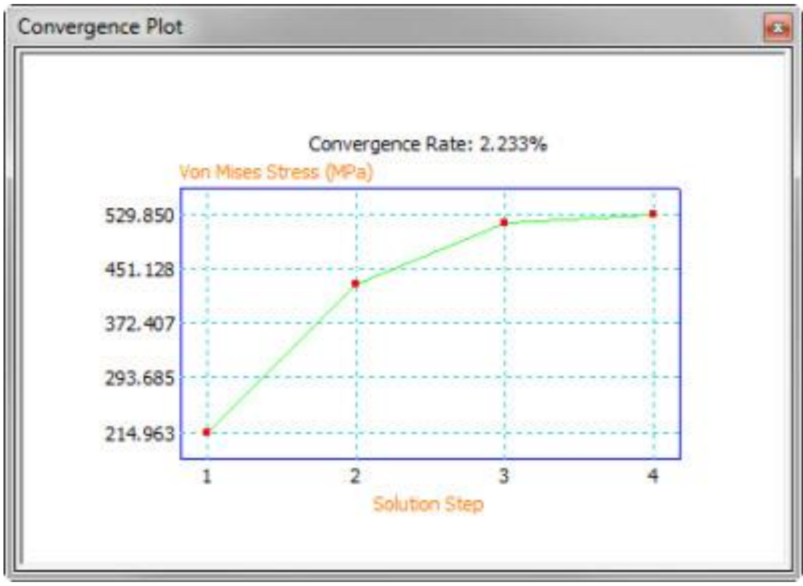
18. Exit the **Stress Analysis** environment.

19. Unsuppress *Fillet1*

20. Enter the **Stress Analysis** environment.

21. Run the simulation.

22. Display the Convergence plot. As you can see the results have converged with the one additional H refinement. For part files three P method refinements are always used first.

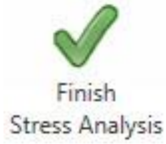


23. In the browser, right-click the **Results** node. Click **Simulation Log**.

24. Scroll to the end of the simulation log. Review the results from the last two runs. Notice the number of refinements, and the number of elements and nodes for each refinement.

    Note: Three convergence cycles using the P method take place before the first h refinement. Although they are not detailed in the simulation log, they are shown as the first three points in a convergence plot.

25. Exit the **Stress Analysis** environment by picking **Exit panel | Finish Stress Analysis** from the Ribbon.



26. Close all files without saving.