ME 24-688 – Week 9

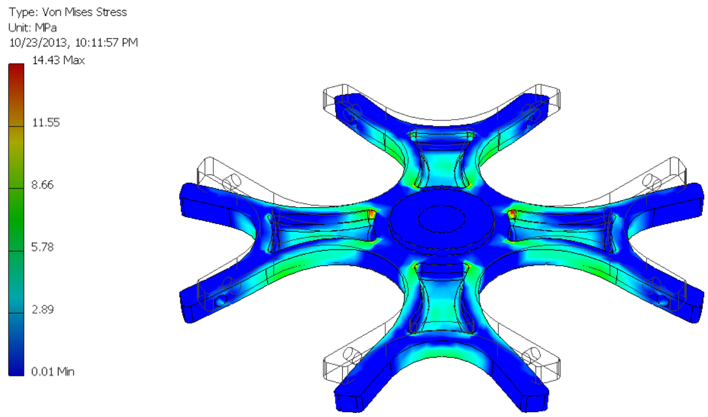
Adding Loads and Constraints

Adding Loads and Constraints

During these projects you will create different loads and constraint and explorer the general interface of the linear FEA capabilities available within Autodesk Inventor Professional 2014.

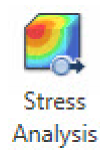
1.1 Project 1A – Create Loads and Constraints

In this project, you create the required loads and constraints to determine the stress and deformation of a rotor for a swing-bucket centrifuge rotating at 4,500 rpm.



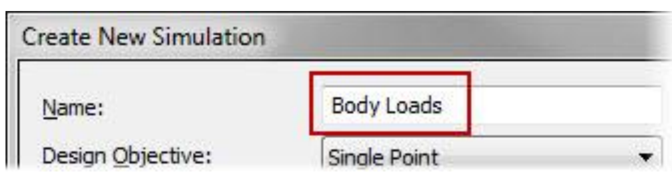
1. Open *Preparing Simulations.ipt* from the location of your project files.

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

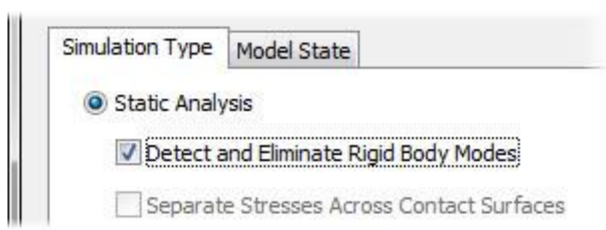


3. On the **Manage** panel, click **Create Simulation**.

4. In the **Create New Simulation** dialog, enter **Body Loads** for the **Name**.



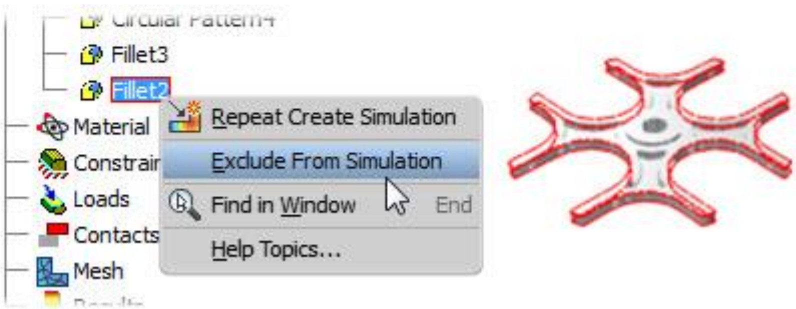
5. In the **Create New Simulation** dialog, select the **Detect and Eliminate Rigid Body Modes** check box to turn it on. This setting will use an algorithm to stop movement in the part if there are not enough constraints to fix it based off the center of mass of the component. This is needed because of the pin constraint added later is the only constraint leaving one degree of freedom.



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

7. In the Browser, expand *Preparing Simulations* part so that you can see all of the individual features.

8. Right-click on *Fillet2* and pick **Exclude From Simulation** on the Browser menu



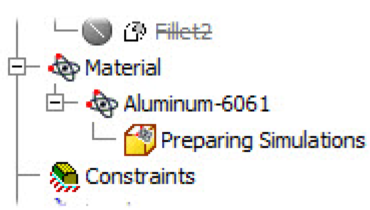
- The Browser will update and the glyph will change to indicate that the feature is no longer part of the simulation.

9. Pick the **Assign** button from the **Material** panel of the Ribbon.

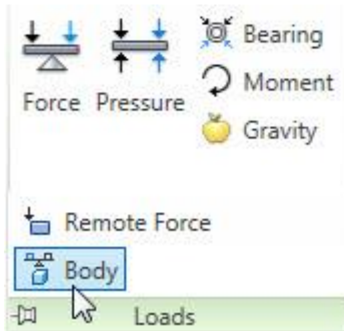
10. In the **Assign Materials** dialog, Select *Aluminum 6061* in the **Override Material** column.

- Next, pick **OK** to dismiss the **Assign Materials** dialog.

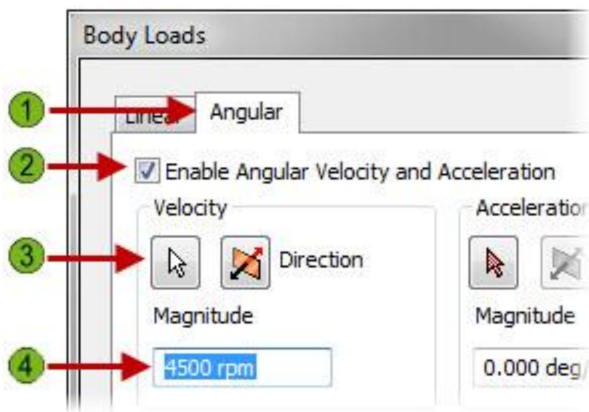
11. Expand the **Material** node in the Browser. Then expand the *Aluminum-6061* node. Notice the Browser displays the material override.



12. Pick **Body Load** from the **Loads** panel of the Ribbon.



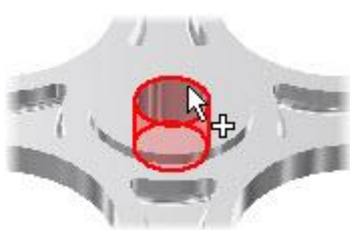
13. In the **Body Loads** dialog, perform the following actions:



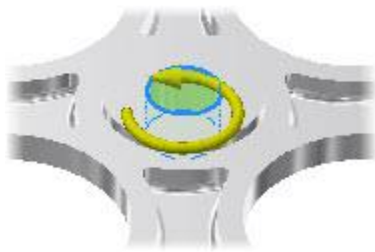
1 Click the **Angular** tab.

2 Select the **Enable Angular Velocity and Acceleration** check box to turn it on.

3 Under **Velocity**, pick the selection arrow. Then select the inner surface of the hole in the center of the part.



4 Enter **4,500 rpm** for the **Magnitude**. Your model should now resemble the figure below.



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

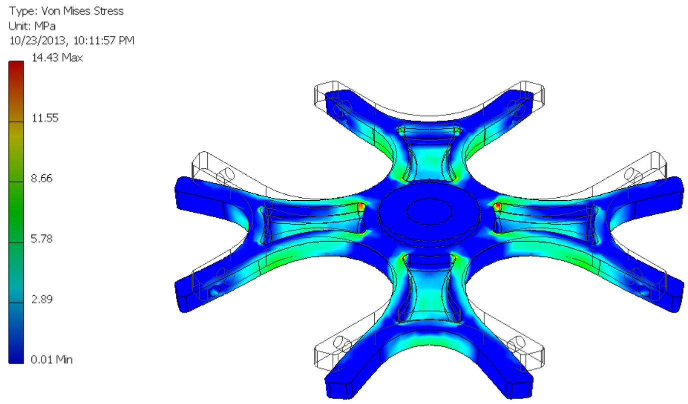
14. On the **Constraints** panel, click **Pin**.

- Select the inner cylindrical surface of the hole in the center of the hub.

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

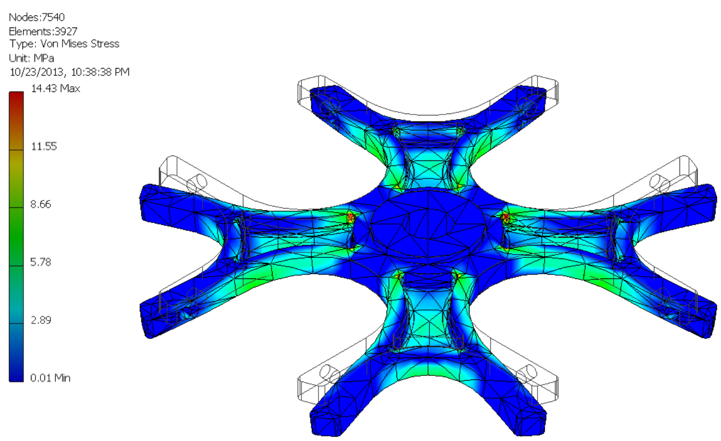
15. Start the **Simulate** dialog by choosing **Solve** from the **Simulate** Ribbon or **Simulate** from the Marking Menu. Pick **Run** to continue.

16. When the analysis is complete, the *Von Mises Stress* is displayed on the displaced model. The stress is far below the yield strength of the material.



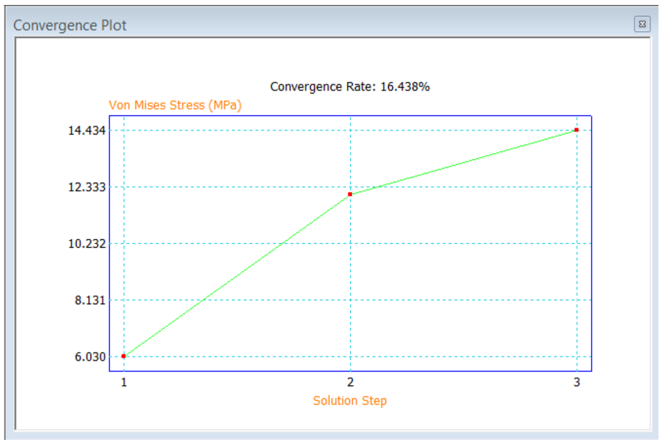
17. Pick **Prepare | Mesh View** from the Ribbon or **Mesh View** from the Marking Menu.

- The mesh is displayed.



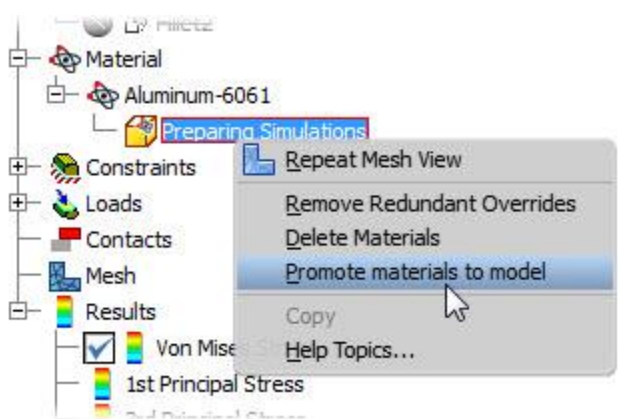
- Click **Mesh View** again to turn off the mesh.

18. To ensure the results are valid and there are no stress singularities display the convergence plot by clicking **Result panel | Convergence**. This will show how the solution was solved three times and they converged within 16 %. Autodesk Inventor Professional automatically runs three refinements for parts and two for assemblies known as "P" refinement.

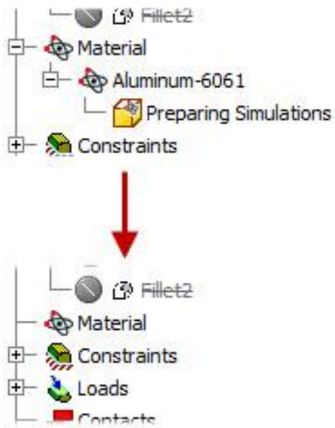


19. You now promote the material to the part so that the material is set in the part environment.

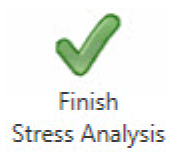
- Right-click **Preparing Simulations** under the **Material |** *Aluminum-6061* node in the Browser and choose pick **Promote Materials to Model** from the Browser Menu.



- Notice the change in the Browser, in which the folders below the **Material** folder are removed because there are no longer material overrides in the **Stress Analysis** environment.



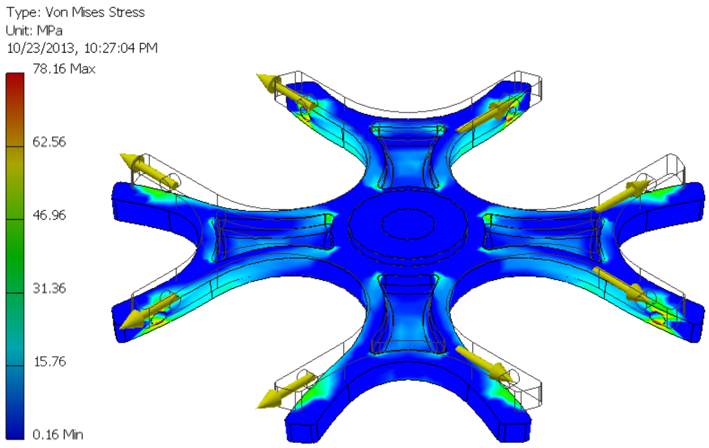
20. Leave the **Stress Analysis** environment by picking **Exit | Finish Stress Analysis** from the Ribbon.



21. Close all files without saving.

1.2 Project 1B – Simulate Loads Project

In this project you add forces to a part that simulate the loads and constraints imposed by other components in an assembly. The magnitude of the forces was calculated based on the rotational velocity, mass of the other components, and swing radius.

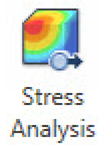


1. Open *Preparing Simulations - Swing Buckets.ipt* from the location of your project files.

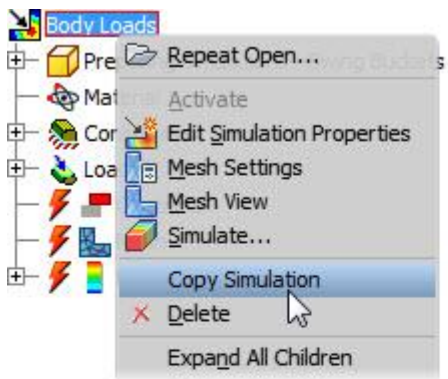
ME 24-688 – Week 9

Adding Loads and Constraints

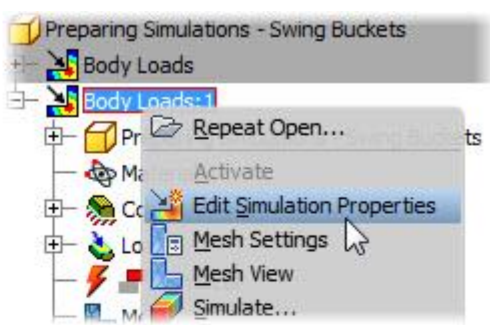
2. Enter the **Stress Analysis** environment by picking **Environments | Begin | Stress Analysis** from the Ribbon.



3. Right-click over the **Body Loads** node and choose **Copy Simulation** from the Browser menu.



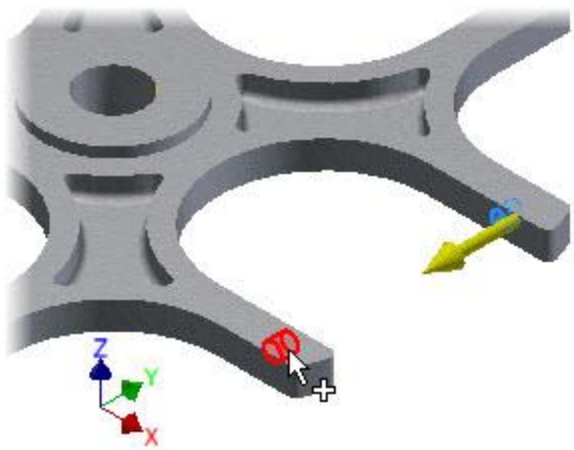
4. Right-click over the new simulation, **Body Loads:1**, node and pick **Edit Simulation Properties** from the Browser menu.



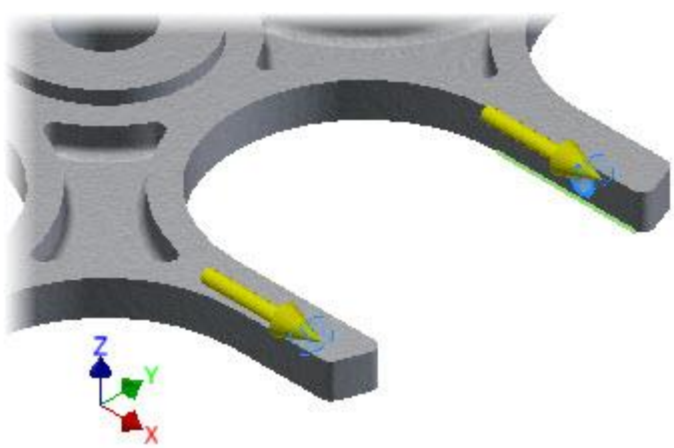
5. In the **Simulation Properties** dialog, enter **Body and Force Loads** in the **Name** field. Click **OK** to dismiss the dialog.

6. Start the **Bearing Load** dialog by choosing **Loads | Bearing** from the Ribbon.

7. Starting with the cutout that is aligned with the **positive X** axis, select the two holes on either side of the cutout.

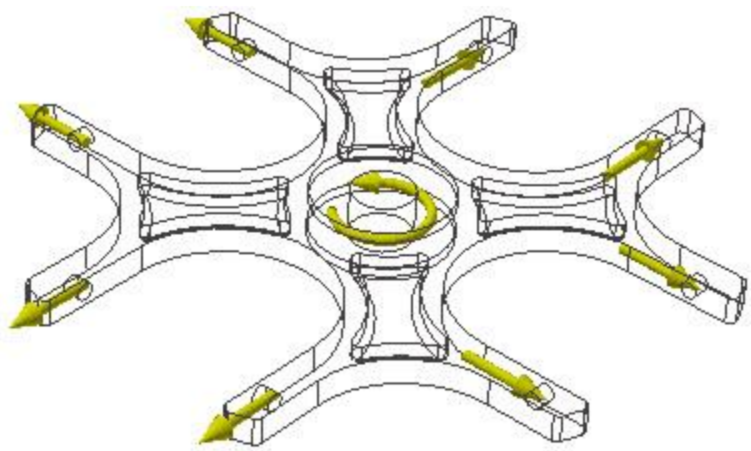


8. Pick the **Direction** button from the **Bearing Load** dialog. Select an **edge** that is aligned with the **positive X** axis so that the forces point in the positive X direction, outward from the center of the model. Use the **Flip Direction** button if necessary.



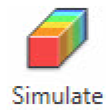
9. Enter **1600N** for the **Magnitude** and click **Apply**, but do not dismiss the **Bearing Load** dialog.

10. Repeat the Bearing Load application steps for the other three cutouts, ensuring that the loads point out from the center of the part, as shown below.

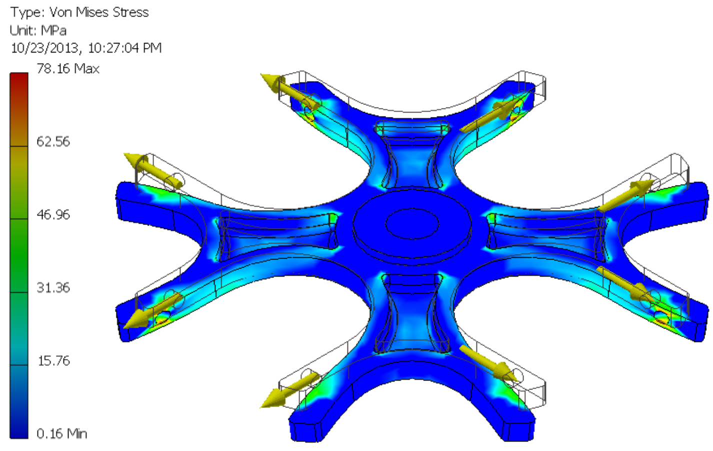


- Dismiss the **Bearing Load** dialog when all four pairs of loads have been applied.

11. 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.



12. We will now change the magnitude of the loads. If you have multiple load values to change, it is often easier to modify the values in the Parameters dialog box rather than editing each load from the Stress Analysis browser.

    - Click **Manage | Parameters | Parameters** from the Ribbon.

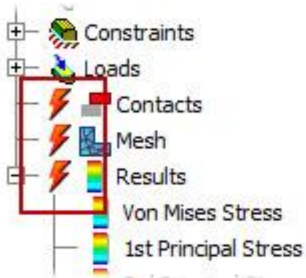
13. When the **Parameters** dialog appears scroll down until the **Body and Force Loads** section is displayed.

14. Change the four **1600N** entries to **2000N**.

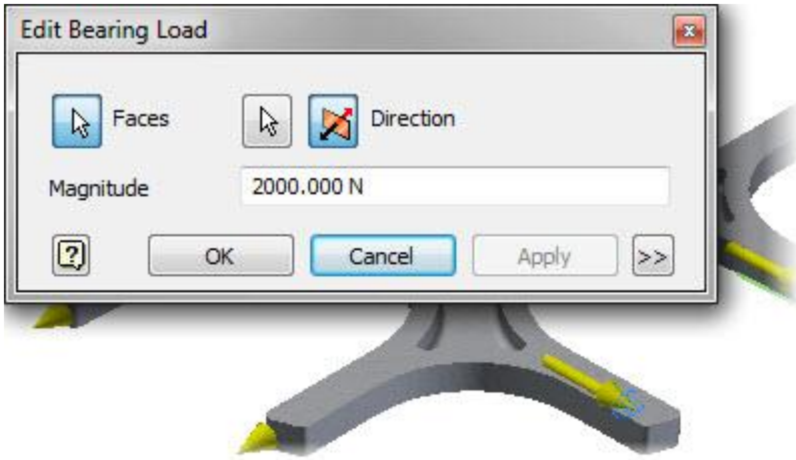
Body and Force Loads:1		
sa_2	deg/s	4500 rpm
sa_3	deg/s...	0.000 deg/s^2
sa_4	N	2000 N
sa_5	N	2000 N
sa_6	N	2000 N
sa_7	N	2000 N

    - Click **Done** to confirm the changes and dismiss the Parameters dialog.

15. Click the **Stress Analysis** tab from the Ribbon. Notice that the Browser now indicates that the results are out of date with red lightning bolt glyphs.

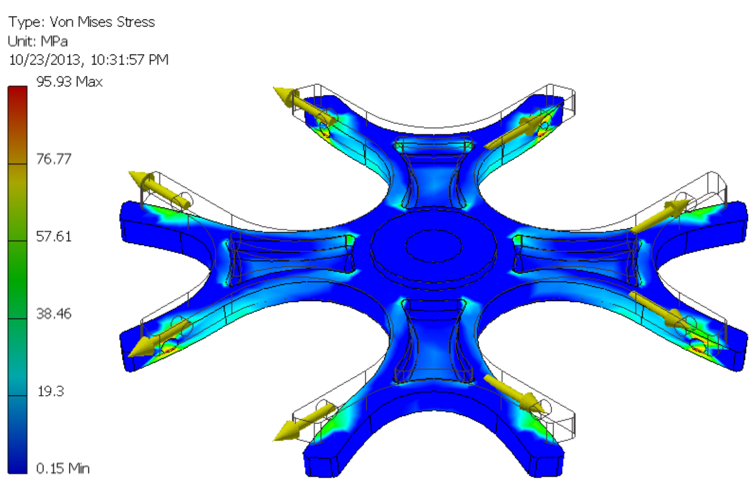


16. Expand the **Loads** node of the Browser and double-click any one of the bearing loads. Confirm that the **Magnitude** is now *2000N*, and then click **OK** to dismiss the dialog.

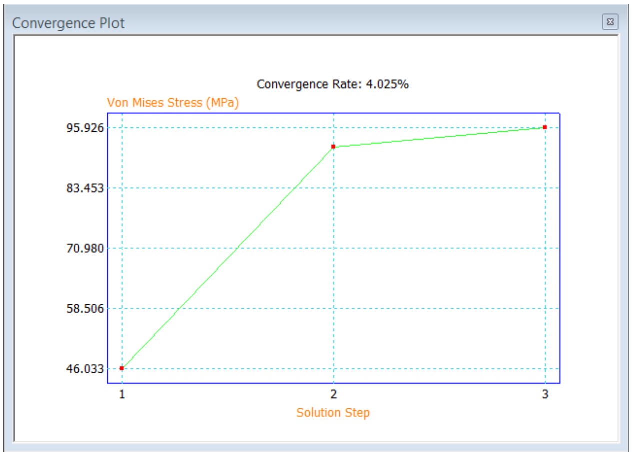


17. Again start the **Simulate** dialog by choosing **Solve | Simulate** from the Ribbon or **Simulate** from the Marking Menu. Pick **Run** to continue.

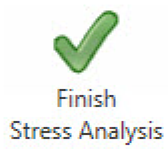
- The updated *Von Mises Stress* is displayed for the *2000N* loads.



18. Review the convergence plot by clicking **Result panel | Convergence** to ensure the results are valid. You will see the convergence rate is 4% so they are within the 10% default limit.



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



20. Close all files without saving.