ME 24-688 – Week 9

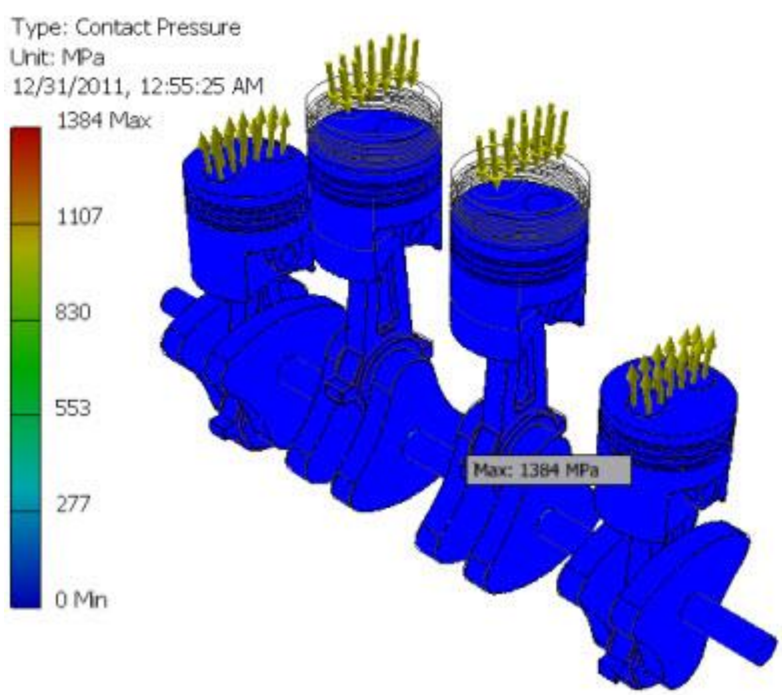
Assembly Simulation Analysis

Determine Stress in an Assembly

1.1 Project 3 – Determine Stress in an Assembly

In this project, you determine the stress areas within an assembly. You apply contacts, edit contacts, apply loads and constraints, run the analysis, and view the results.

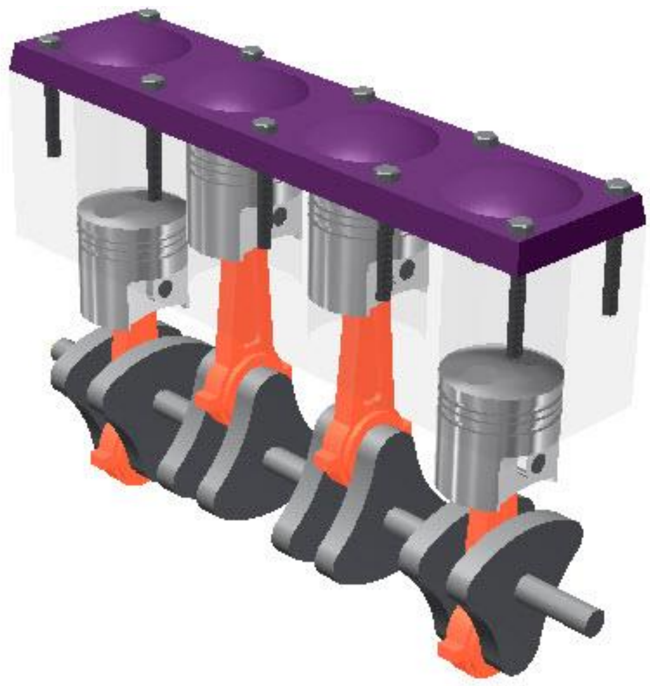
Please note that this analysis is performed at one position of the engine and does not include inertial forces. There may be other positions of the assembly that result in higher forces in the bearings. Dynamic Simulation can be used to solve for the inertial forces and to help you locate the position that results in the highest forces in the components.



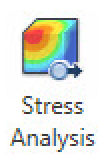
ME 24-688 – Week 9

Assembly Simulation Analysis

1. Open *Analyze Assemblies.iam* from the location of your project files.



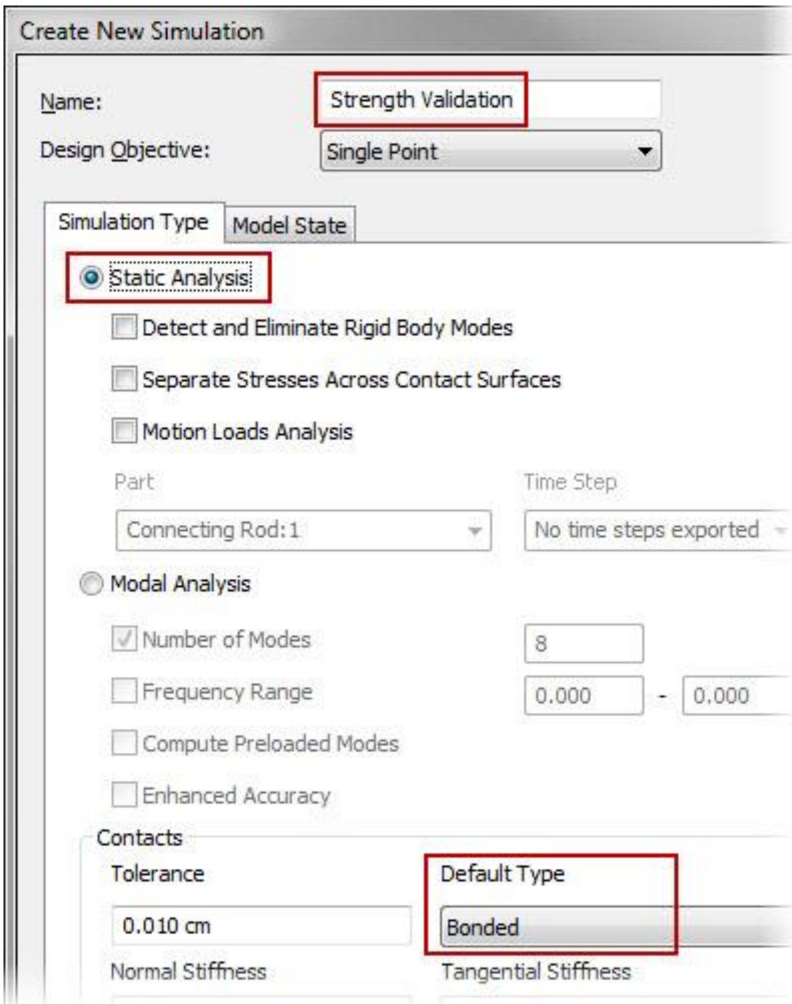
2. Enter the **Stress Analysis** environment by picking **Environments | Begin | Stress Analysis** from the Ribbon.



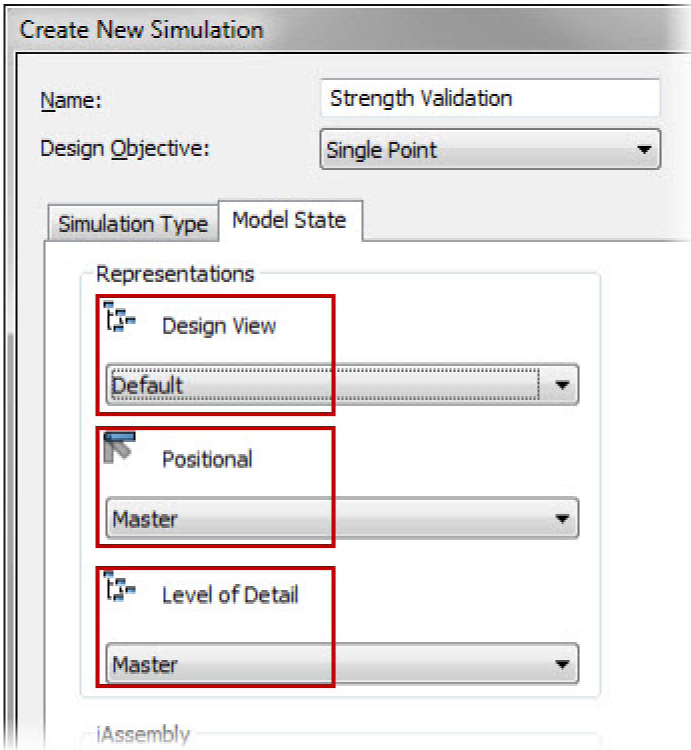
3. On the **Manage** panel, click **Create Simulation**.

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

   - Enter **Strength Validation** for the **Name**.
   - Confirm that **Static Analysis** is selected on the **Simulation Type** tab.
   - Confirm that **Bonded** is the **Default Type** of contact. This will assign bonded contacts between parts by default. They can be edited later if required.



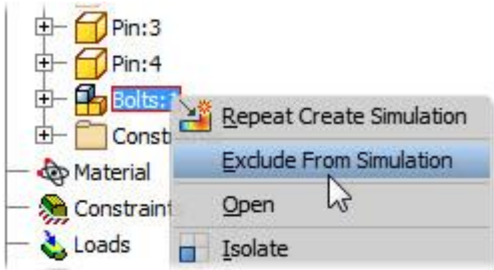
5. In the **Create New Simulation** dialog, click the **Model State** tab.
   - Confirm the **Design View** is set to *Default*.
   - Confirm that **Positional** and **Level of Detail** are both set to *Master*.



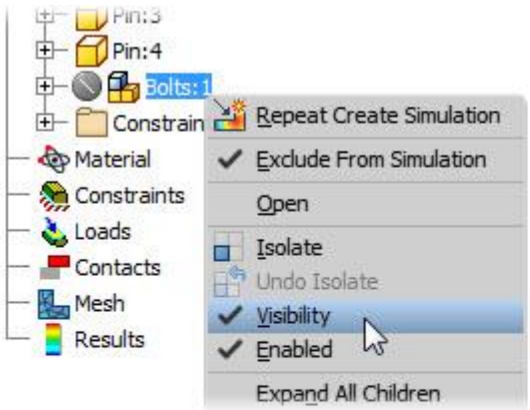
6. Click **OK** to dismiss the **Create New Simulation** dialog and create the new simulation.

7. Expand the *Analyze Assemblies* assembly node in the Browser.

8. Right-click *Bolts:1* in the Browser and choose **Exclude from Simulation** from the Browser menu.



- Right-click ***Bolts:1*** again and choose **Visibility** from the Browser menu.



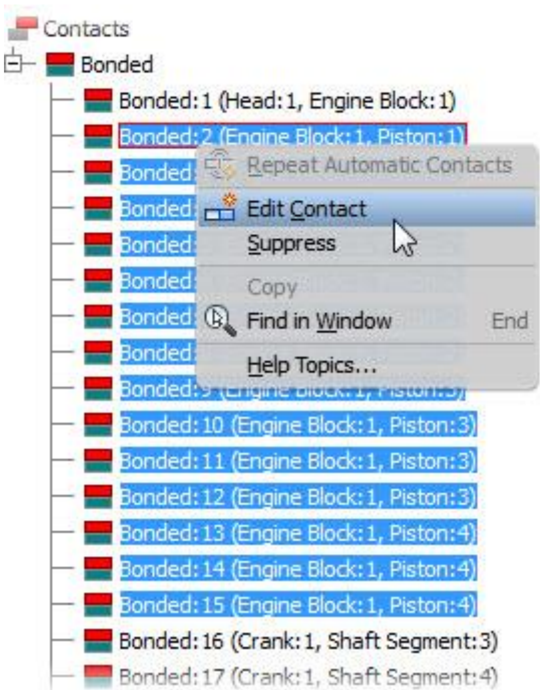
9. You now add contacts to the model by first adding automatic contacts and then modifying some of the resulting contacts.

- Click **Contacts | Automatic** from the Ribbon to apply automatic contacts.

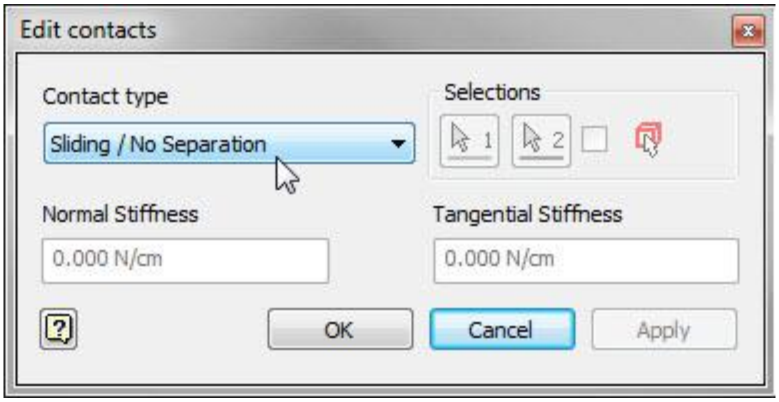
- **Bonded** contacts are applied to all locations per the default settings.

10. In the Browser, expand the **Contacts** folder. Click to expand **Bonded**.

11. Select all of the contacts between the **Engine Block** and **Piston** (14 in total). Right-click and choose **Edit Contact** from the Browser menu.



12. When the **Edit Contacts** dialog appears, change the **Contact Type** to **Sliding / No Separation**. This will allow the parts to move but not separate from each other as they move up and down the cylinders.

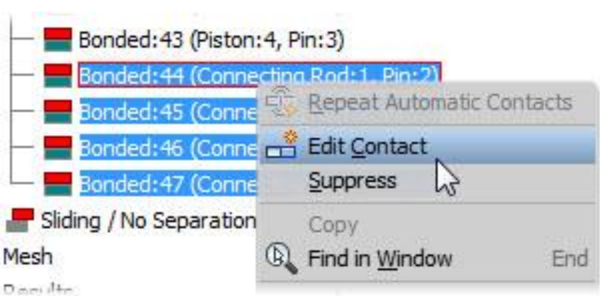


- Pick **OK** to continue.

13. Review the changes in the Browser. Notice the edited contacts now appear under the **Sliding/No Separation** node in the Browser.

14. To allow the connecting rods to rotate freely on the crankshaft, change the 12 contacts between the **Crank** and **Connection Rod** to **Sliding / No Separation**.

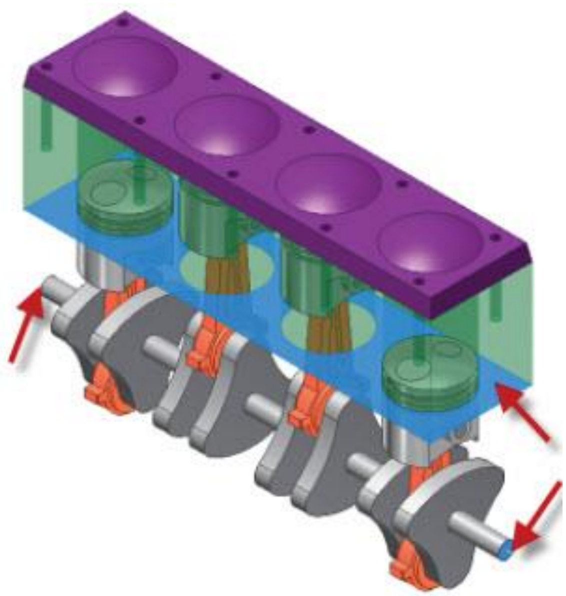
15. To allow the wrist pin to rotate freely in the connecting rod, change the four contacts between the **Connecting Rod** and **Pin** to **Sliding / No Separation**.



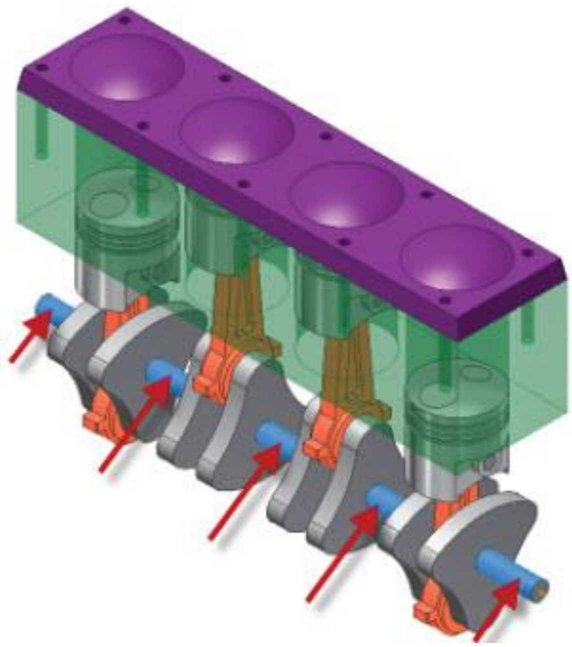
16. You now add loads and constraints. The cylinder is loaded with 10MPa pressure to simulate combustion pressures. The pistons in the down position are loaded with -0.5 MPa to simulate suction. The pressure loads are applied to just the top of the pistons because the stress in the head and block is not the objective of the analysis.

    - Pick **Constraints | Fixed Constraint** from the Ribbon.

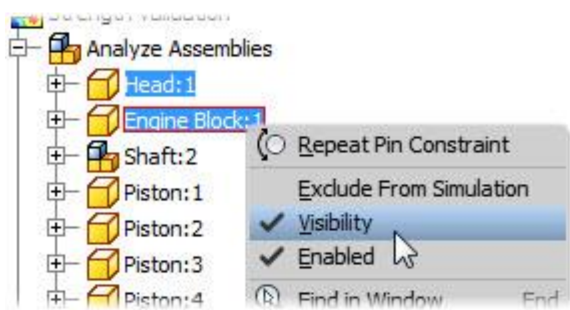
17. Add a **Fixed** constraint to the bottom face of the block and the two end shaft faces of the shaft as shown in the image.



18. Add **Pin** constraints to each of the middle shaft segments as shown.

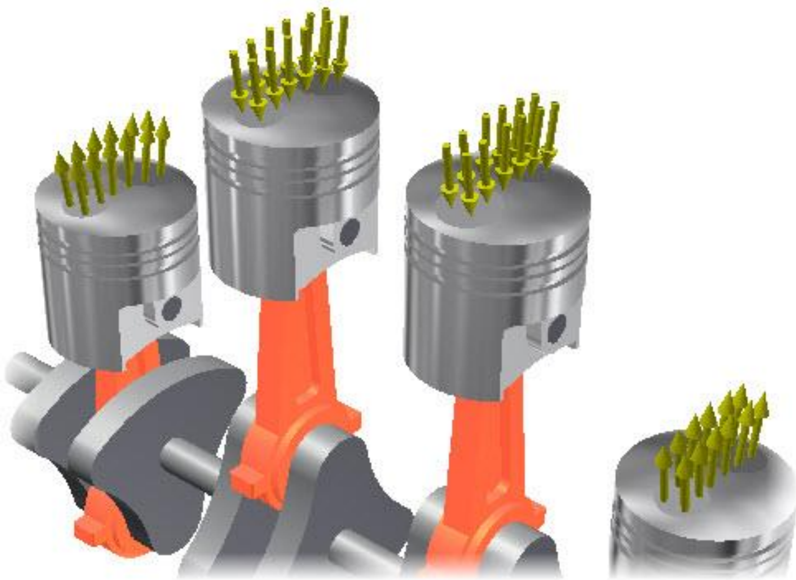


19. In the Browser select both the *Engine Block* and *Head* part components. Right-click and choose **Visibility** from the Browser menu to turn off visibility for the two components.



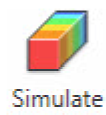
20. Pick **Loads | Pressure** from the Ribbon. Apply a pressure of **10 MPa** to the top faces of the two pistons that are in the top position. Be sure that you select all three faces for each piston for a total of six.

21. Apply a **Pressure** load of **-0.05** `MPa` to the top faces of the other two pistons.

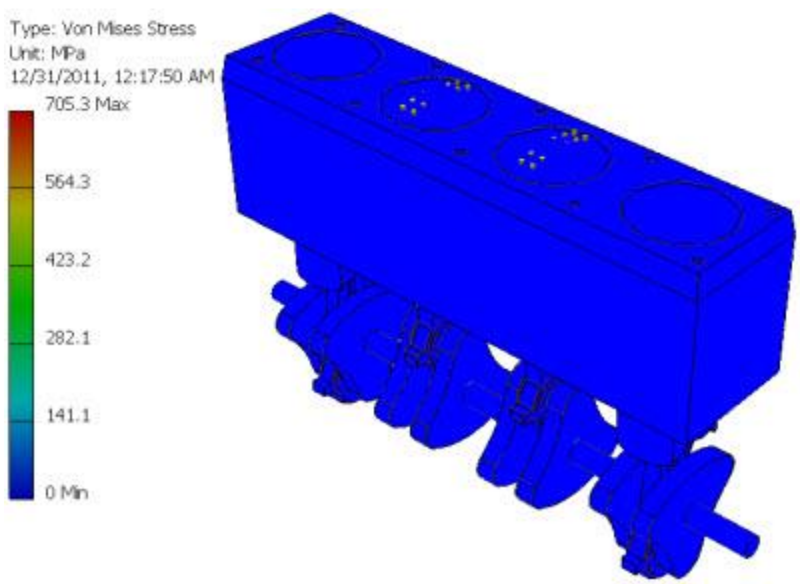


22. Turn the visibility of the *Engine Block* and *Head* components back to **On**.

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

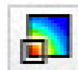


24. The *Von Mises Stress* is displayed.

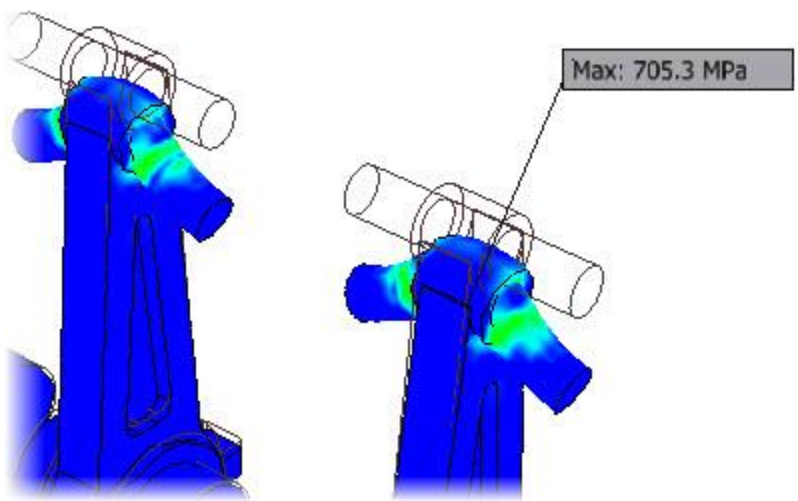


The maximum reported *Von Mises Stress* is over 700 MPa, higher than the yield strength of steel. The stresses on the outside of the engine are low. Therefore, the area of maximum stress must be hidden by other components.

25. Pick **Display | Show Maximum Value** from the Ribbon.

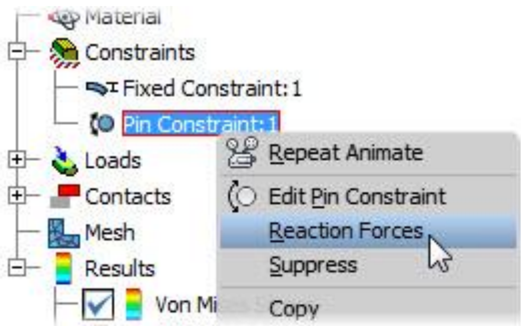


26. Turn off the visibility of the head, engine block, and the four pistons. The maximum stress is on the pin.

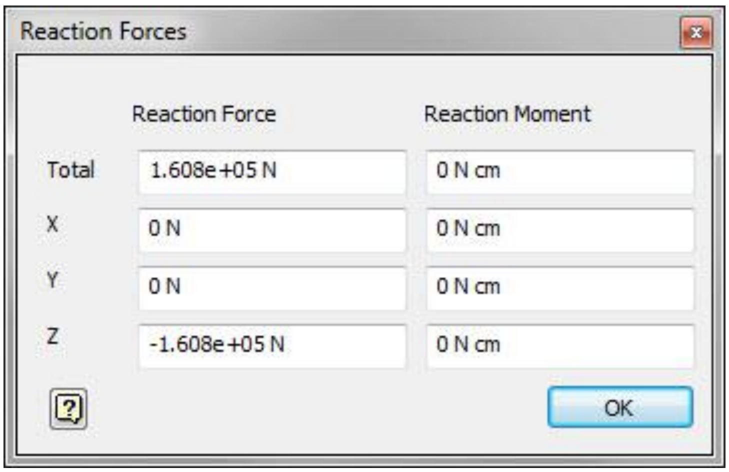


27. Animate the results to view the deformation by picking **Result | Animate** from the Ribbon.

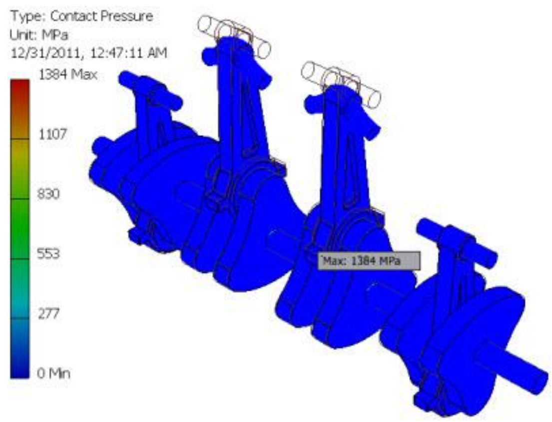
28. Expand the **Constraints** node of the Browser. Right-click over the **Pin Constraint** and choose **Reaction Forces** from the Browser menu.



- Review the results and then click **OK** to dismiss the dialog.



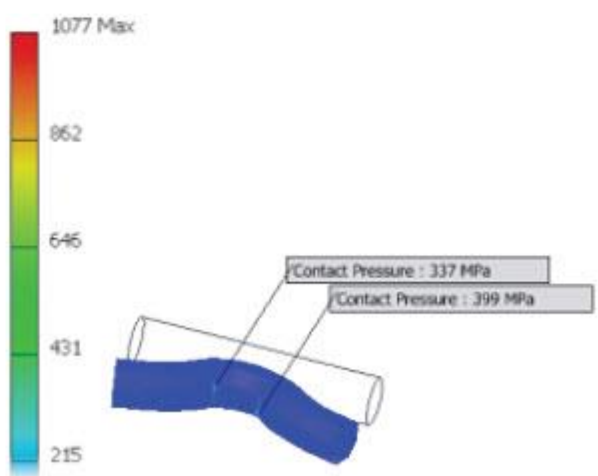
29. Expand the **Results** node of the Browser, followed by **Contact Pressure**. Double-click **Contact Pressure**.



30. Turn off the visibility of *Connecting Rod:4*.

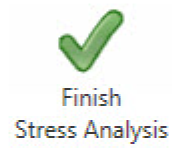
31. Rotate your view to view the pin from below.

32. Probe the results to view the magnitude of the contact pressure.



33. Turn on the visibility of all components.

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



35. Close all files without saving.