ME 24-688 – Week 12

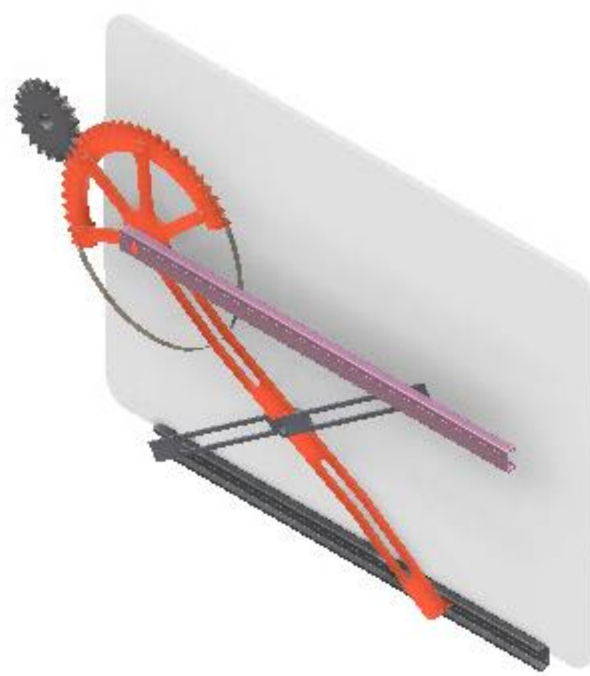
Linear Analysis from Dynamic Simulation

Linear Analysis from Dynamic Simulation

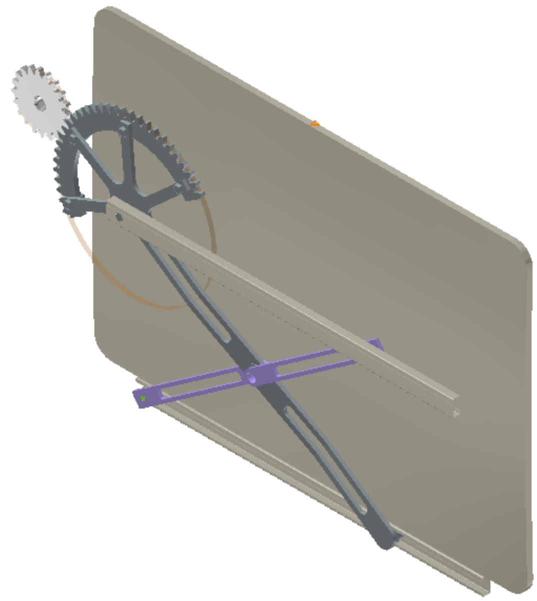
Determining the loads that are applied to a part within a mechanism is critical to performing an accurate finite element analysis (FEA) on the part. One method to do this from your digital prototype 3D model is to transfer the result information from a Dynamic Simulation as loads for the FEA. During this project you will use Autodesk Inventor Professional to complete the Dynamic Simulation and then complete a linear analysis on a part from the calculated information.

1.1 Project 1 – Window Glass Mechanism Linear FEA

In this project, you run a Dynamic Simulation on the window glass mechanism. Then you export the maximum loads on a part to the Stress Analysis environment to validate the robustness of the part with a linear analysis. The initial results indicate that the original design achieves a safety factor above 1.0 but could be make slightly stronger to ensure good performance over the lifespan of the part.

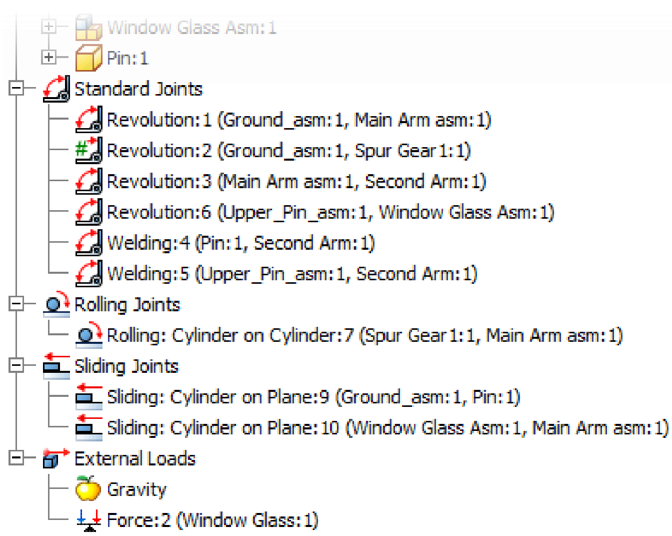


1. **Open** the provided *WindowGlassLeverFEA.iam* Autodesk Inventor file in Autodesk Inventor Professional.



2. Active the Dynamic Simulation environment by clicking **Environments tab | Begin panel | Dynamic Simulation**. If prompted to view the tutorial click **No** to close the window.

3. Review the **Dynamic Simulation Browser** and notice the already created joints and loads.

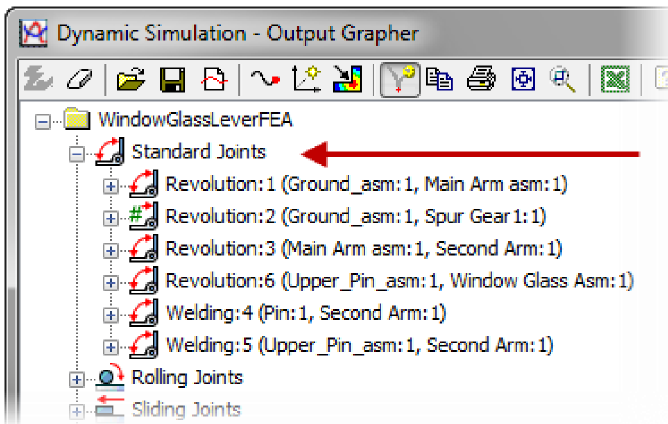


ME 24-688 – Week 12

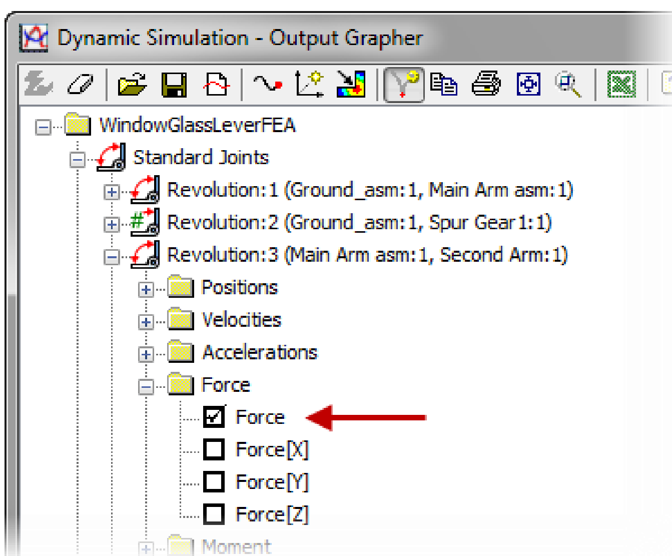
Linear Analysis from Dynamic Simulation

4. Within the **Simulation Player** dialog click **Run or Reply the Simulation** to review the motion of the mechanism.

5. One the **Results** panel click **Output Grapher**. In the **Output Grapher** Browser three expand the **Standard Joints** node.



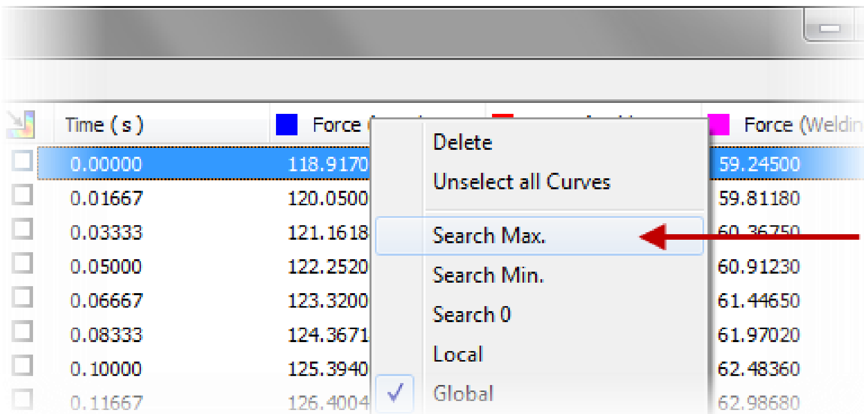
6. Under *Revolution:3 (Main Arm asm asm:1, Scond Arm:1)*, expand **Force** and select the **Force** checkbox.



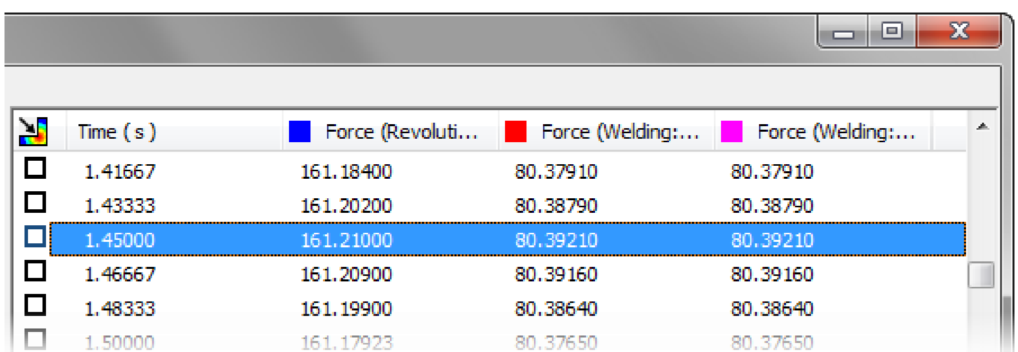
7. Under *Welding:4 (Pin:1, Second Arm:1)*, expand **Force** and select the **Force** checkbox.

8. Under *Welding:5 (Upper_Pin_asm:1, Second Arm:1)*, expand **Force** and select the **Force** checkbox.

9. Right-Click in the column for *Force (Revolution:3) (N)* and select **Search Max**.

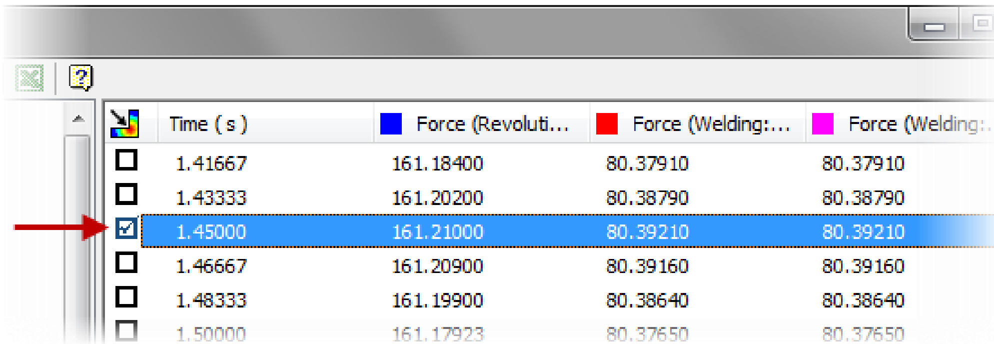


10. Review the time-steps pane and the graphics window. The maximum load occurs at 1.45 seconds and the load is 161.21 N.

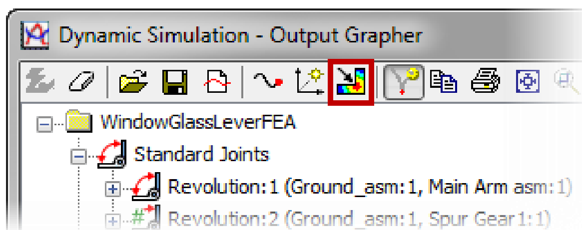


Linear Analysis from Dynamic Simulation

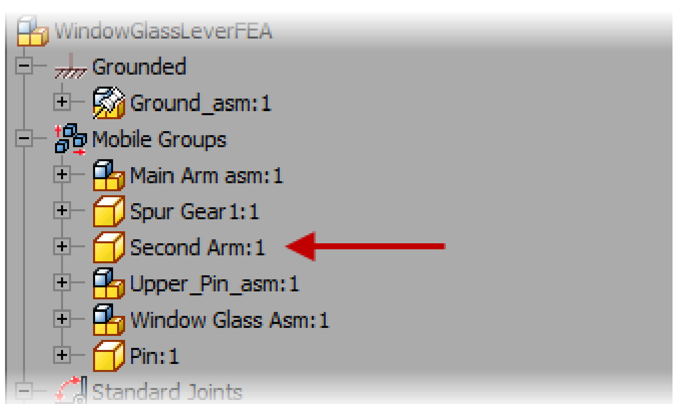
11. Now that we have identified the condition that applies the maximum about of force we will transfer the loads to the FEA environment within Autodesk Inventor Professional. Check the **Export to FEA** checkbox next to that time-step as shown below.



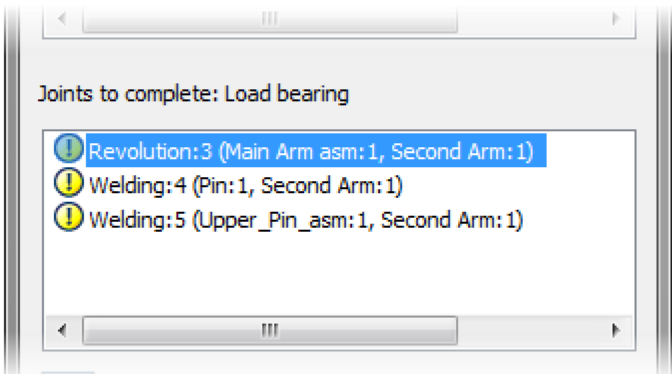
12. On the **Output Grapher** toolbar click **Export to FEA**.



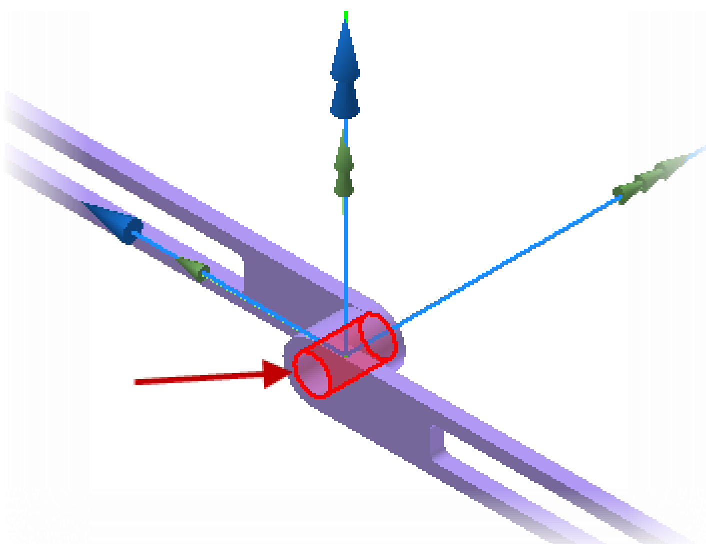
13. When prompted to pick a part select the *Second Arm* part in the browser. Click **OK** once selected.



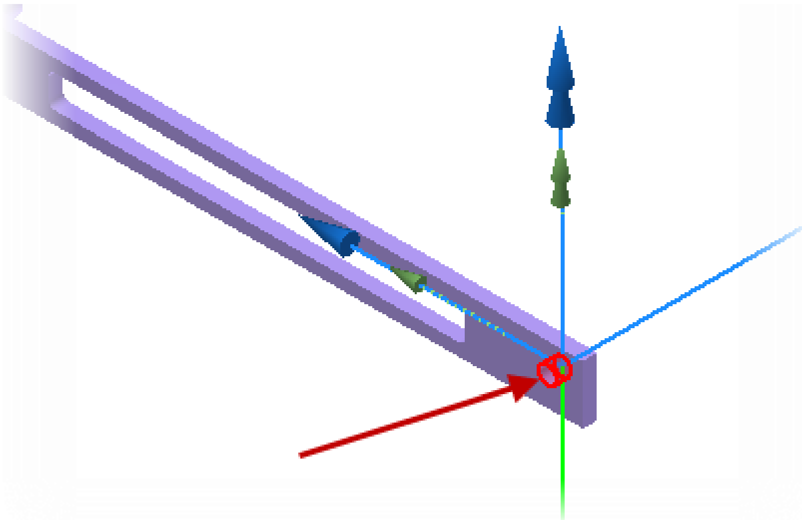
14. In the **FEA Load-Bearing Faces Selection** dialog, under **Joints to Complete: Load Bearing**, make sure that **Revolution:3** is selected. This is a listing of selections that need to be made to complete the transfer from the assembly to the part.



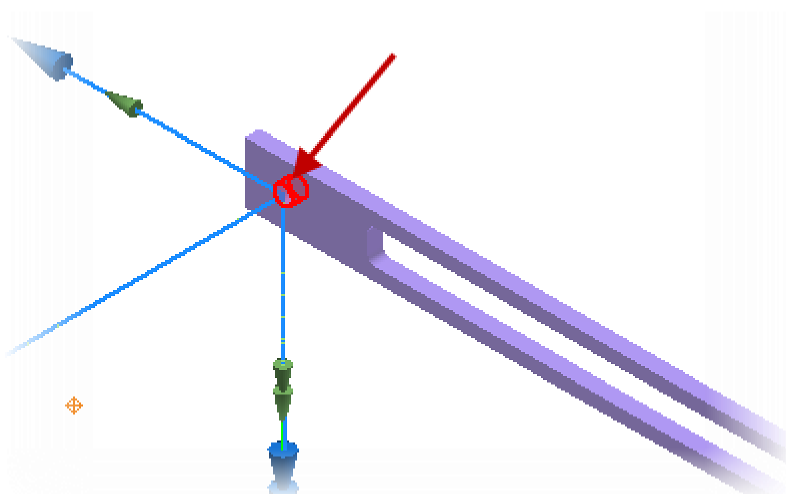
15. Select the load bearing face as shown below.



16. Under **Joints to Complete: Load Bearing**, select **Welding:4** and select the load bearing face as shown below.



17. Under **Joints to Complete: Load Bearing**, select **Welding:5** and select the load bearing face as shown below.

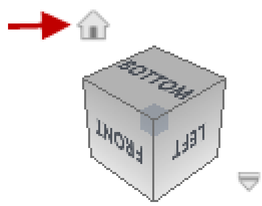


18. Click **OK** to complete the transfer and setup.

19. On the **Simulation Player** click **Rewind to the Beginning of the Simulation**.

20. Click the **Construction Mode** icon to exit the results view.

21. On the **View Cube** click the **Home** view icon.



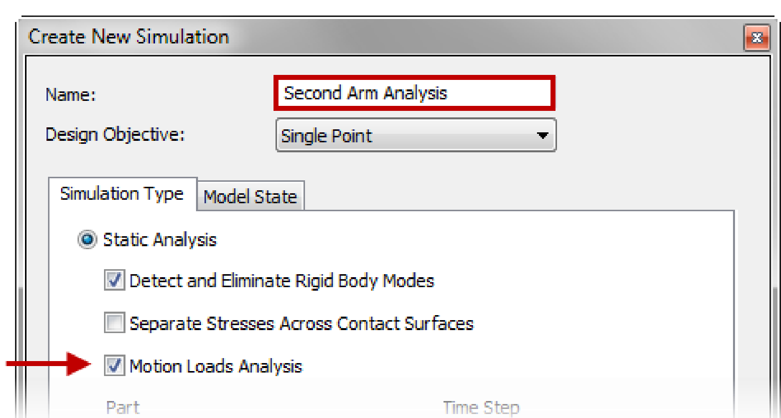
22. Exit the **Dynamic Simulation** environment by click the **Finish Dynamic Simulation** on the **Exit** panel.

23. Now we will enter the **Stress Analysis** environment to complete the analysis of the *Second Arm* part using the provided loads. Click **Environment tab | Begin panel | Stress Analysis**.

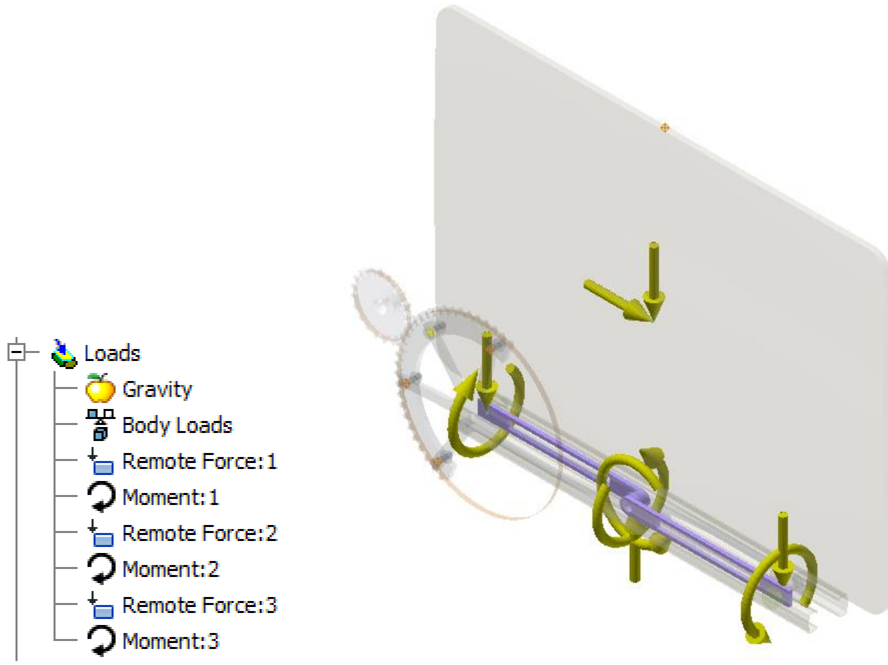
24. On the **Manage** panel click **Create Simulation**.

25. In the Create New Simulation dialog complete the following:

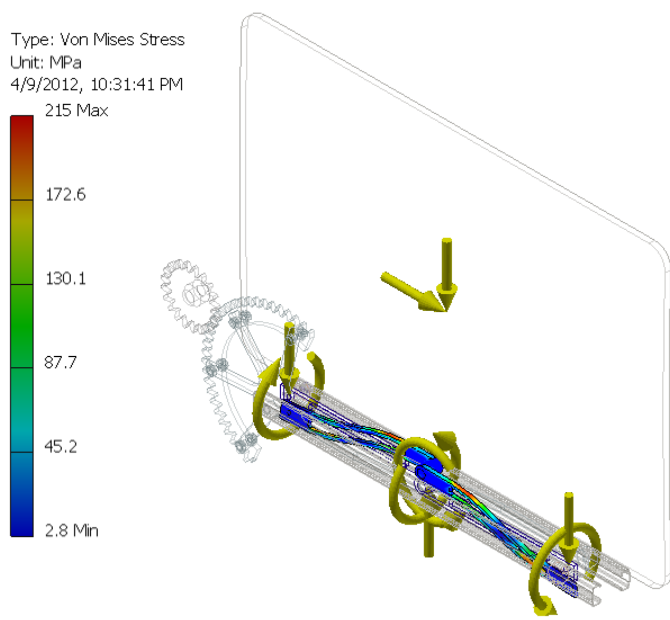
    - For **Name** enter **Second Arm Analysis**
    - On the **Simulation Type** tab select the **Motion Loads Analysis** checkbox
    - Click **OK** to create the new simulation.



26. Now you will see the different loads that have been created from the Dynamic Simulation.

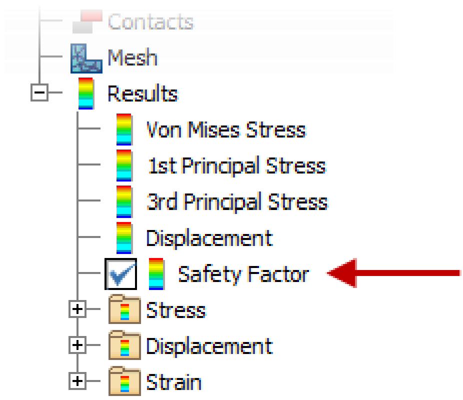


27. To solve the linear analysis with all of the default settings click **Simulate** on the **Solve** panel then click **Run**.

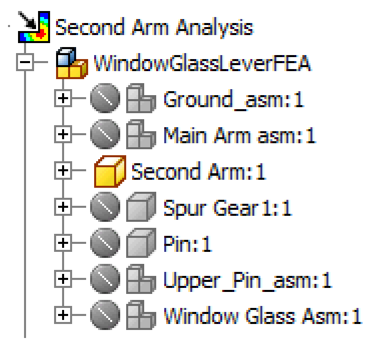


28. Use the view tools to review the stress contours on the part.

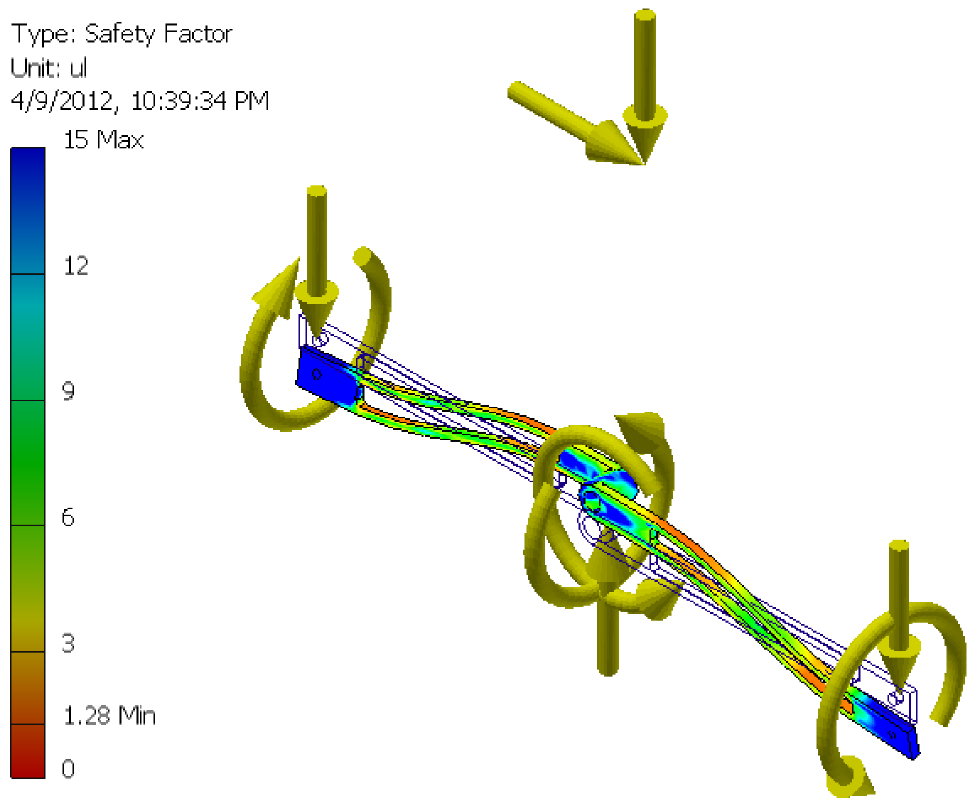
29. In the **Stress Analysis Browser** under **Results** double-click **Safety Factor**.



30. In the Browser expand the assembly model and select all of the components that are not included in the analysis and turn their visibility off.



31. Review the part and notice the Safety Factor is around 1.28 which may indicate that this part needs to be made stronger to ensure it withstands the loads. Having the part with a Safety Factor of 2 or more would be acceptable in this case.



32. **Save** the file so you can review later as required.