ME 24-688 – Week 13

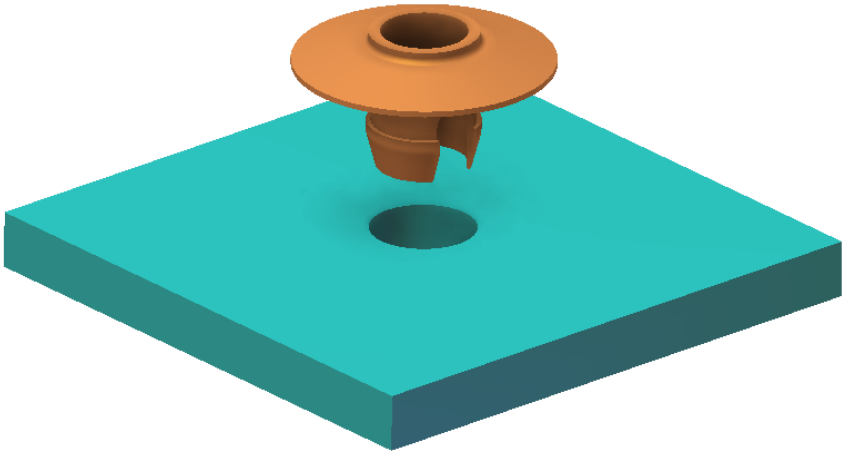
Design Modifications

Design Modifications and Iterations

Validating designs to ensure the performance is as required is a common use for analysis applications. Another use is to use the analysis results to identify early improvement areas and benchmark different design options against each other to improve the overall design. The goal is to create a more optional design for the design objectives.

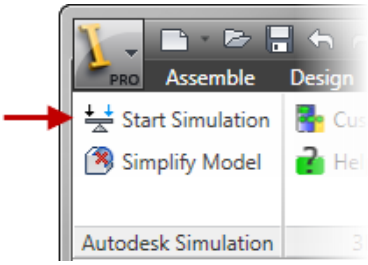
1.1 Project 2 – Snap Fit Plastic Bushing

During this project we will explorer the different methods for making design changes to the native CAD model file and updating the analysis model. The initial design will be analysis to identify design improvements and compare two material options before complete the design of a plastic push bushing.

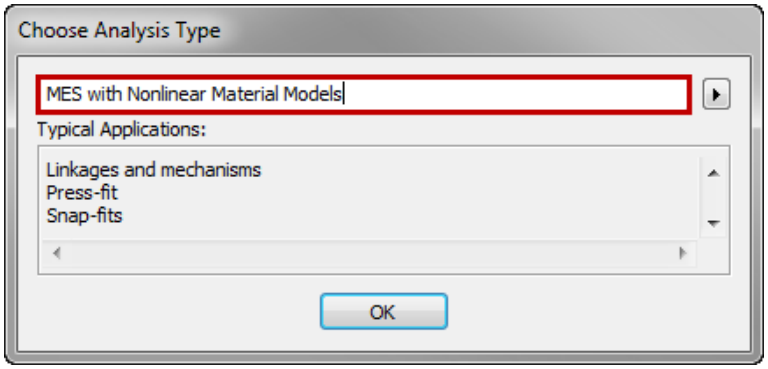


1. **Open** the "*Plastic Bushing Assembly.iam*" file in Autodesk Inventor Professional.

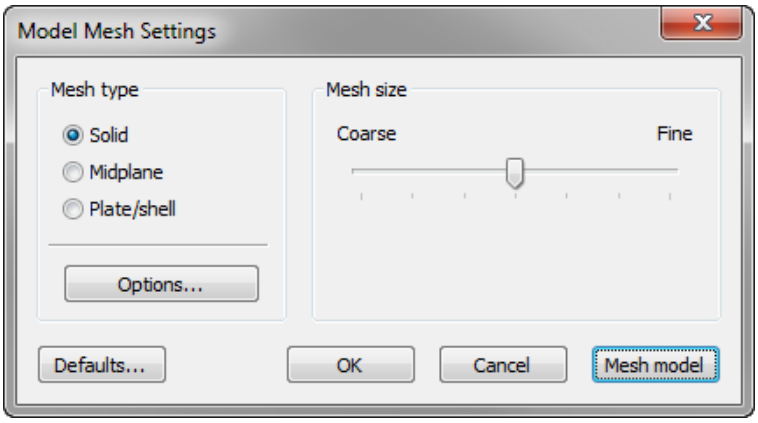
2. To start an Autodesk Simulation analysis from within Autodesk Inventor click **Add-Ins tab | Autodesk Simulation panel | Start Simulation**.



3. Once the Autodesk Simulation application starts select **MES with Nonlinear Material Models** as the **Analysis Type** and click **OK**.

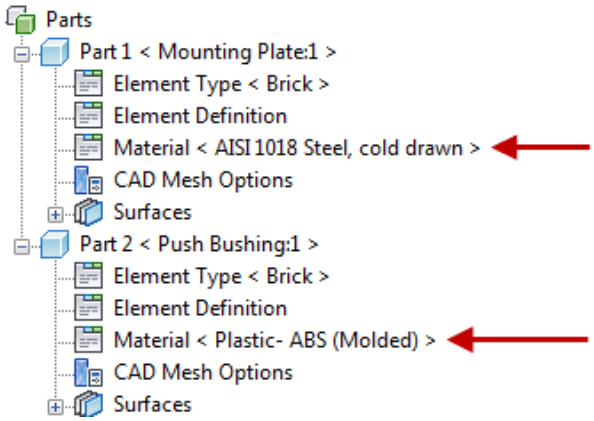


4. Click **Mesh tab | Mesh panel | 3D Mesh Settings** and keep the default 100% **Mesh Size**. Click **Mesh Model** button to process the surface meshes of the model. When prompted to view mesh results click **No**.



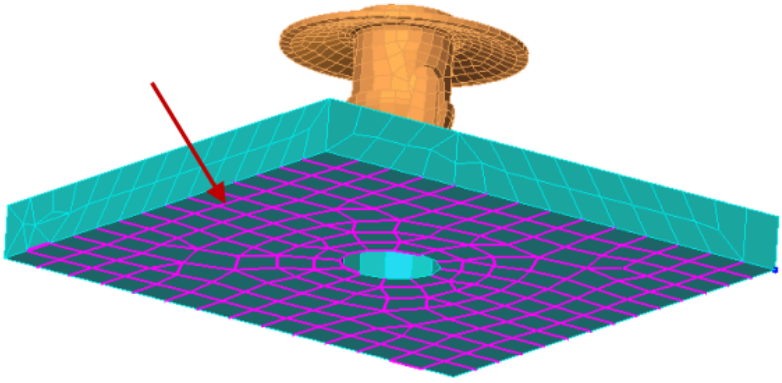
5. Set the materials for each of the parts as shown below:

   - Part 1 <Mounting Plate> = AISI 1018 Steel, Cold Drawn
   - Part 2 <Push Bushing> = Plastic – ABS (Molded)



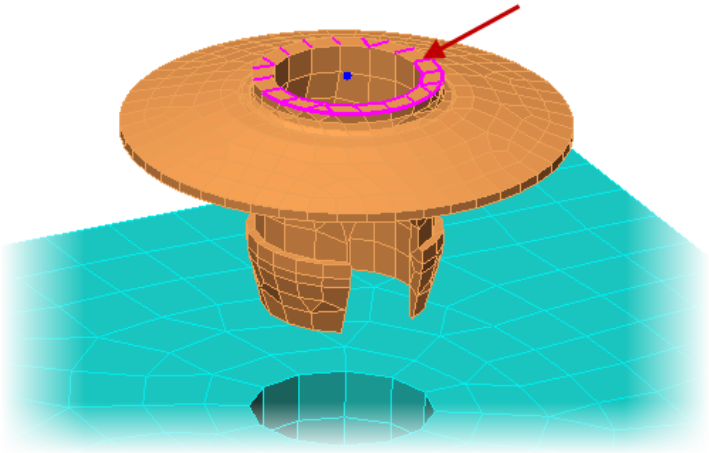
6. Set the section settings to **Point** for **Shape** and **Surfaces** for **Select**.

7. Select the bottom surface of the *Mounting Plate* part as shown below.



8. Right-Click in the graphics window and select **Add | Surface Boundary Condition**. Select **Fixed** to lock all degrees of freedom and click **OK** to complete.

9. Select the top surface of the *Push Bushing* part as shown below.



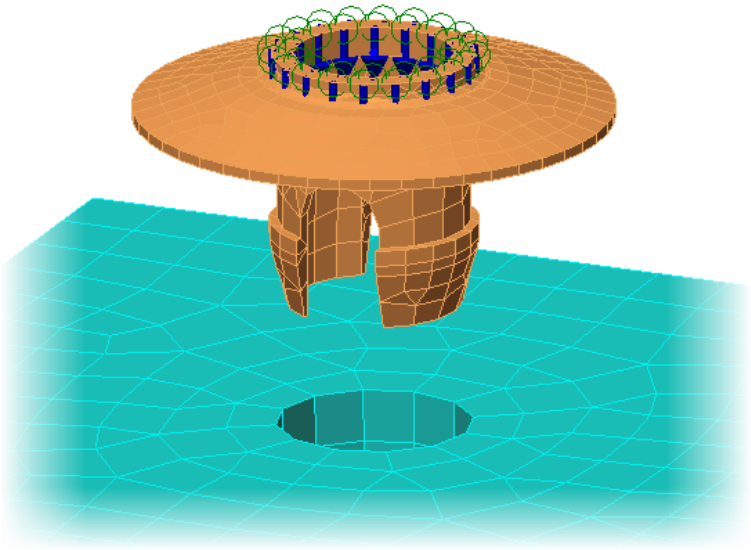
10. Right-Click in the graphics window and select **Add | Surface Boundary Condition**. Check the **Tx** and **Ty** checkboxes to lock the X and Y transitional degrees of freedom. Click **OK** to complete. This will help hold the position of the part as it is pressed into the plate.

11. Select the same top surface of the Push Bushing part and right-click in the graphics window and pick **Select Subentities | Vertices**. This will select all of the nodes on this surface.

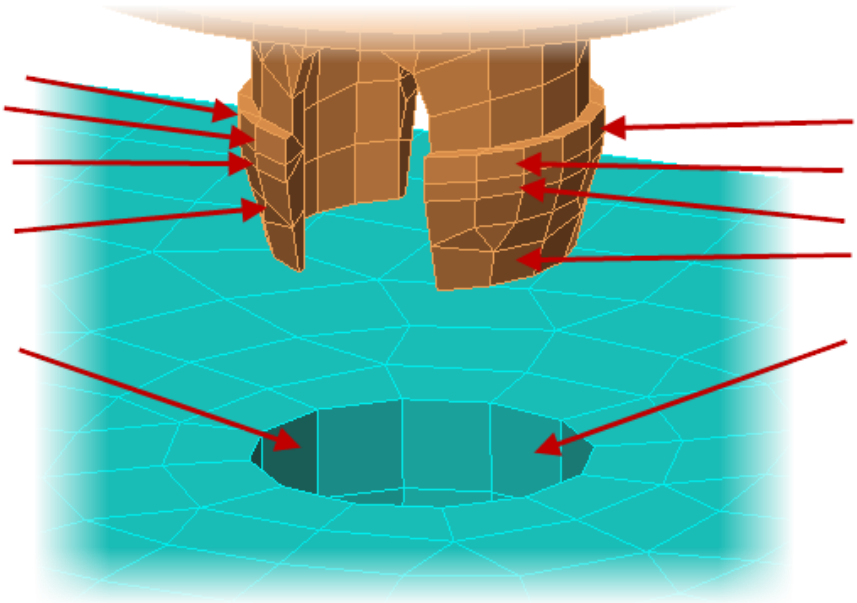
12. Right-Click the graphics window and select **Add | Nodal Prescribed Displacements**. Enter -30 for the **Magnitude** and select **Scalar Z** as the **Direction**. Click **OK** to complete.

13. Set the section settings to **Point** for **Shape** and **Surfaces** for **Select**.

14. **Orbit** and **Zoom** the model into a position as shown below.



15. Setup a **Surface Contact** pair between the surfaces marked below. There should be a total of 16 contact pairs created. Each of the 8 surfaces on the Push Bushing part should be paired with both of the hole surfaces.



16. Click **Setup tab | Model Setup panel | Parameters** to open the **Analysis Parameters**.

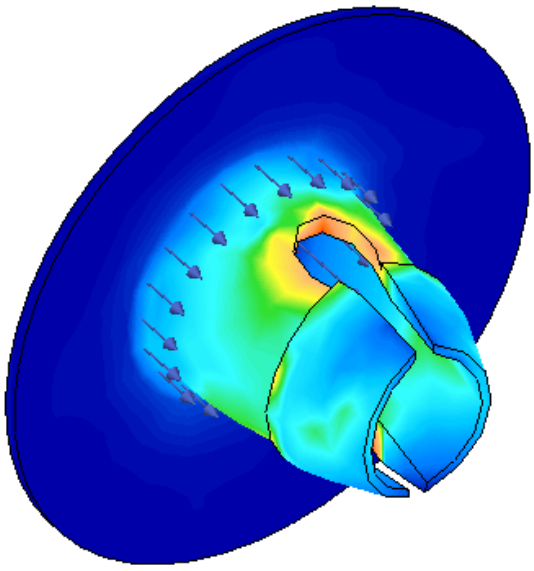
17. Enter 40 as the **Capture Rate** and click **OK** to complete.

18. You can now run the simulation to establish a performance baseline for the *Push Bushing* part. Click **Analysis tab | Analysis panel | Run Simulation**. The simulation will take approximately three (3) minutes to solve depending on computer resources.

19. **Save** the Analysis.

20. To focus on the *Push Bushing* part right-click on the **Mounting Plate** part in the **Results Browser** and turn the **Visibility** off.

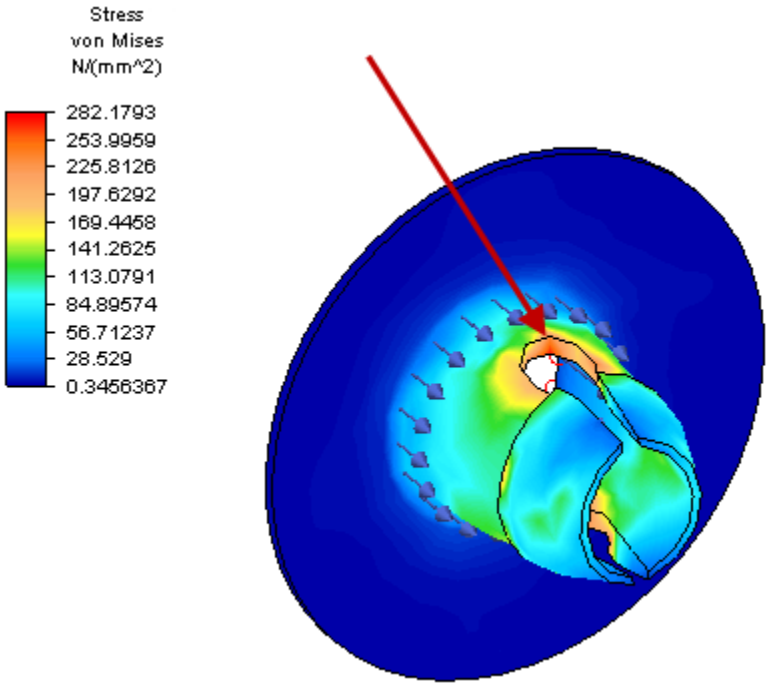
21. **Start** the results under the **Captures** panel to review the movement and displacement of the *Push Bushing* part. You will notice that there is high stress at the top of the notch of the part.



ME 24-688 – Week 13

Design Modifications

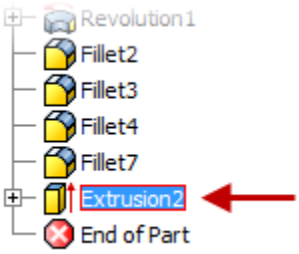
22. During the time steps 35-39 you will notice the stress at the top of the notice is over 250 N/(mm^2) with this current design. Also notice that the end of the bushing almost touches when it is compressed into the hole.



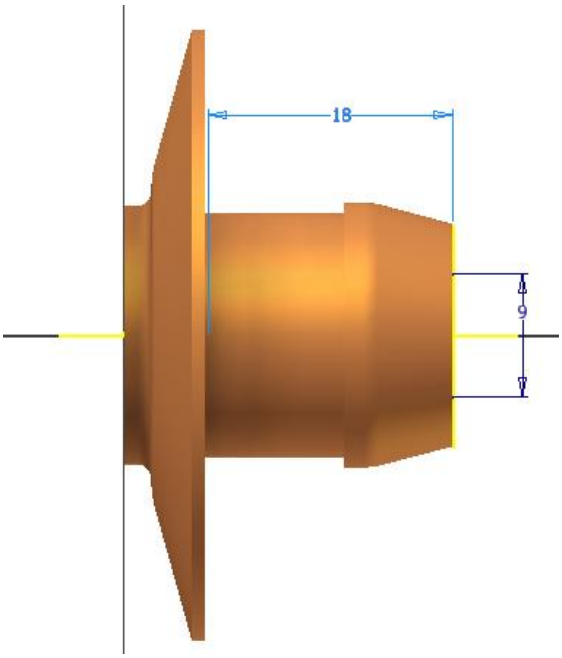
23. Click the **FEA Editor** tab of the **Browser**.

24. Based on this initial baseline data we will make several design changes. Switch back to Autodesk Inventor which should still be open on the computer.

25. Within Autodesk Inventor double-click the *Push Bushing* part in the **Browser** to edit. Then right-click on the *Extrusion 2* feature and select **Edit Sketch**.



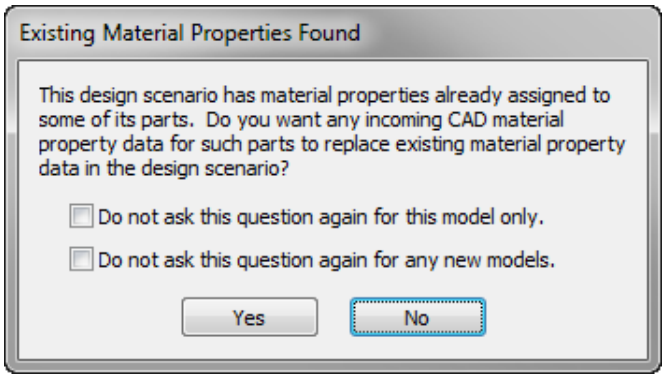
26. Edit the dimensions of the sketch to be 18 long and 9 tall as shown below.



27. Click **Finish Sketch** on the **Exit** panel. Then click **Return** on the **Return** panel.

28. **Save** the new design in Autodesk Inventor.

29. To update the model in Autodesk Simulation we start the simulation process again. Click **Add-Ins tab | Autodesk Simulation panel | Start Simulation**. The open will be opened again in Autodesk Simulation and the prompt shown below will be displayed.



30. Click **No** to keep all of the material assignments that were already made in the previous Autodesk Simulation file.

    Note: You can also update your Autodesk Simulation model by reopening the native CAD file from within Autodesk Simulation. If there is an Autodesk Simulation file already existing it will just updated that file with the latest model geometry.

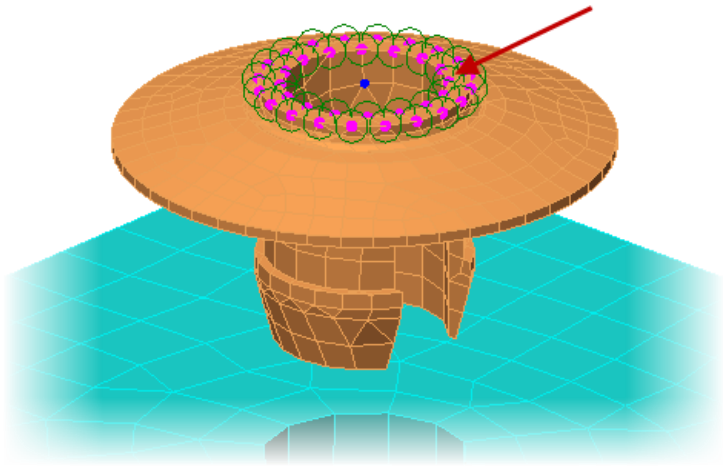
31. Now that the model has been updated the model will need to be re-meshed in Autodesk Simulation. Click **Mesh tab | Mesh panel | Generate 3D Mesh**.

32. Notice that the prescribed displacement constraint has been removed. This was removed because the model was re-meshed and the node numbers are different. Also notice the contact pairs were removed because the surface ID's changed during the model update.

ME 24-688 – Week 13

Design Modifications

33. Select the nodes of the top surface as shown and add back the -30 mm prescribed displacement in the **Z Axis**.



34. Now go back and create the 16 different Surface Contact pairs.

35. Run the simulation again to recalculate the results using the new model geometry.

36. Review the new results like before and you will notice the stress is reduced in the back of the notch per the new design. This will help improve the design of the part and still achieve the design objectives of the plastic bushing.