

# Press Fit Nonlinear Static Analysis in WaveFEA

## 1. Creating Geometry

Quarter cylinder and C clips Geometries are drawn as shown in below image

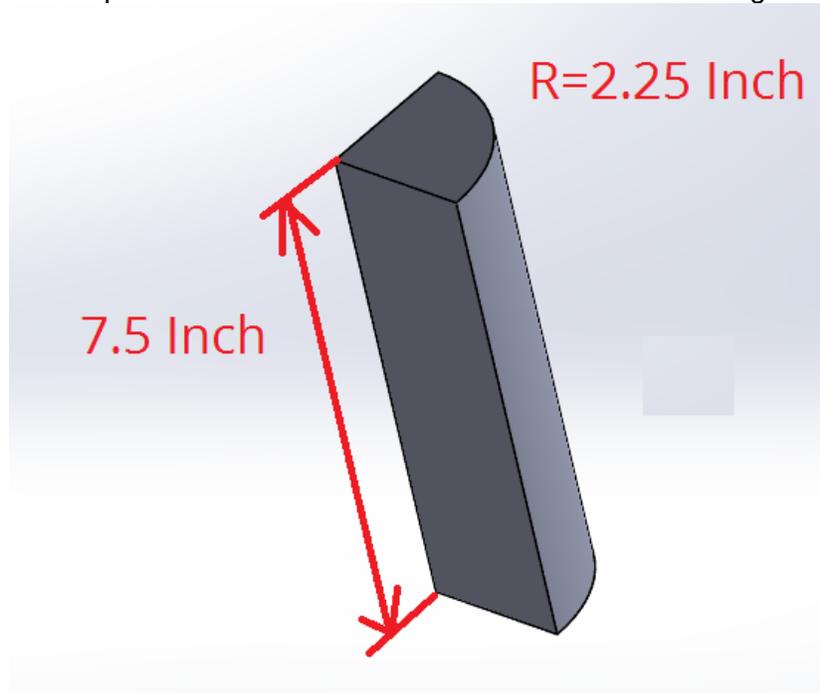


Figure 1. Quarter Cylinder Geometry

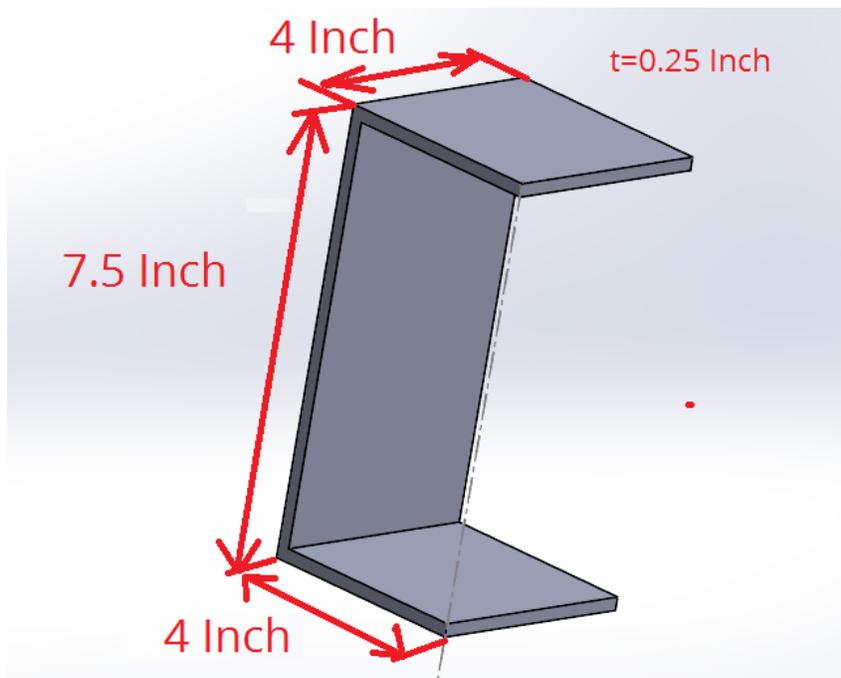


Figure 1. Clip Geometry

## 2. Creating Assembly

Go to the Assembly and import above parts four times to create the assembly model as shown below. You should now see the geometry of a cylinder housed within a thin walled box, as shown below.

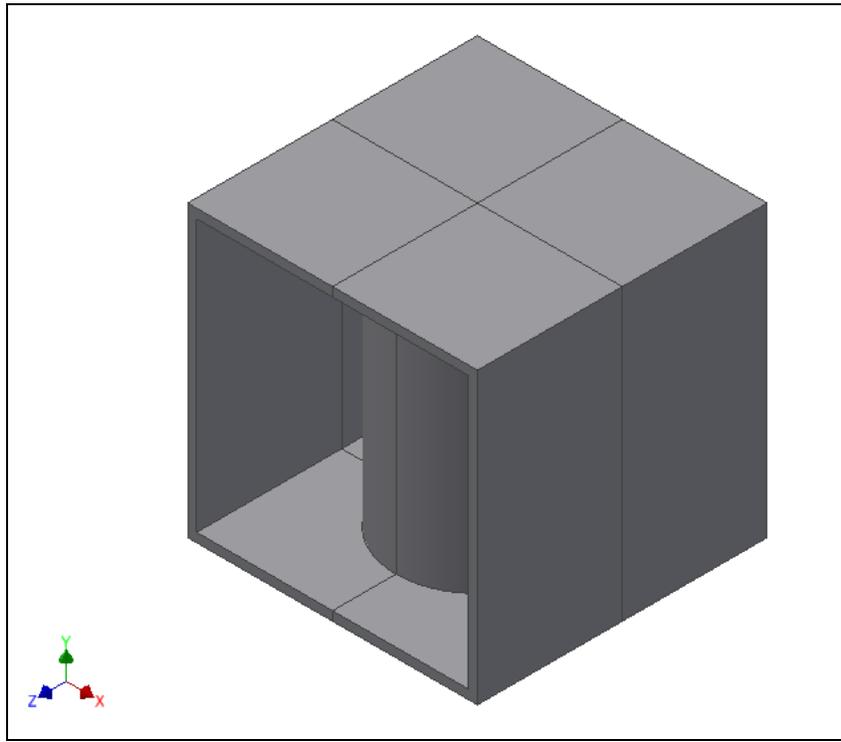


Figure 1. Housed Cylinder Geometry

## 3. Selecting the Analysis Type

Now right-click on **Analysis1** and select **Edit**. You will see the **Analysis** window as shown in Figure 2. Then change the analysis **Type** to **Nonlinear Static** and click **OK**.

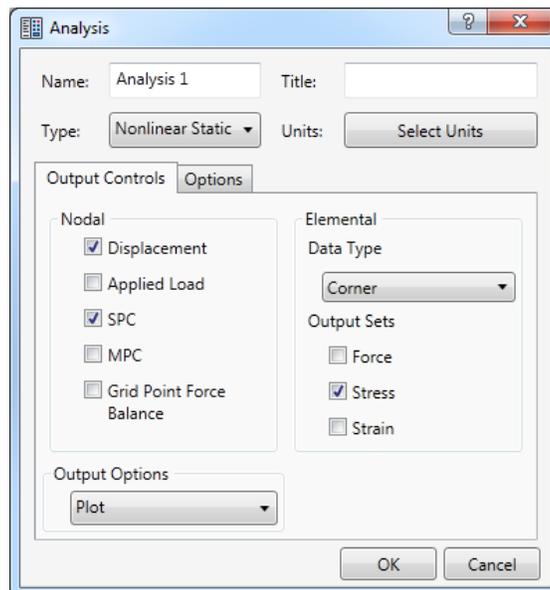


Figure 2. Analysis Form

## 4. Creating the Material

From the **WaveFEA Model Tree**, right-click on **Materials** and select **New**. Now you will see the **Material** window as shown below.

The Material Form dialog box is shown with the following data:

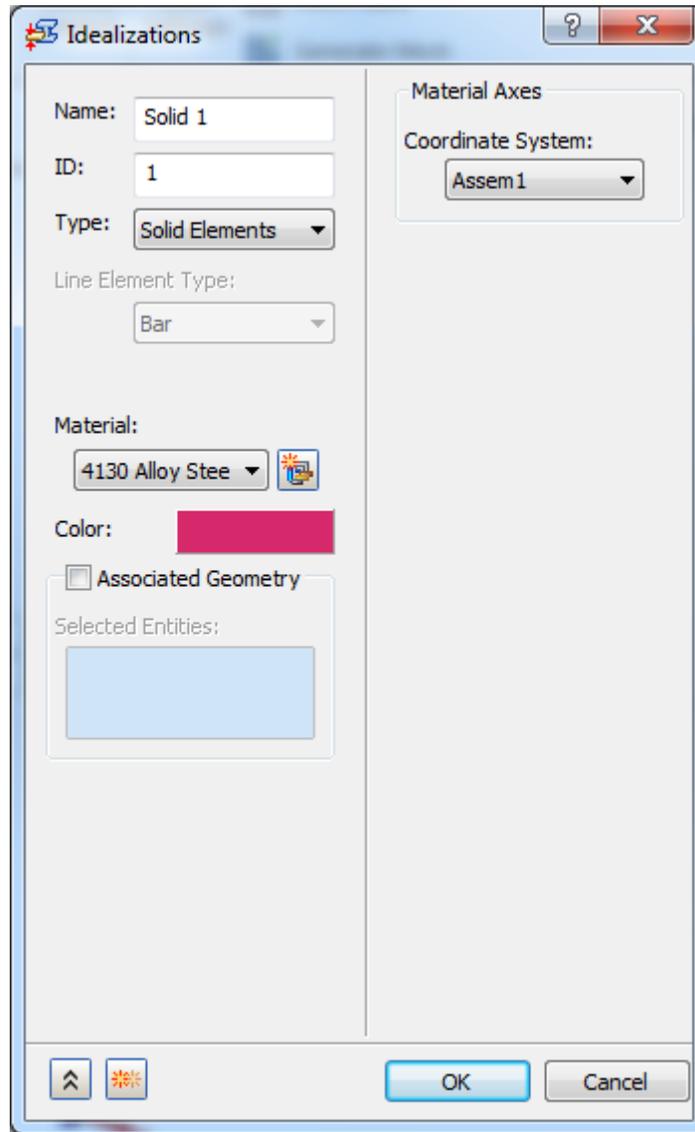
| Section    | Property  | Value      |
|------------|-----------|------------|
| General    | $\nu$     | 0.00073299 |
|            | GE        | 0          |
|            | $T_{REF}$ |            |
| Structural | E         | 3e+7       |
|            | G         |            |
|            | $\nu$     | 0.32       |
|            | $\alpha$  | 7.5e-6     |
| Allowables | $S_T$     | 1.7e+5     |
|            | $S_c$     |            |
|            | $S_s$     |            |
|            | $S_y$     |            |
| Thermal    | C         | 46.145     |
|            | K         | 0.00021801 |

Figure 3. Material Form

Click on the **Load Material...** button on the left-hand side and select **4130 Alloy Steel,tempered,Ultra High Strength** or any other compatible material from the material database. Click **Close** and then click **OK**.

## 5. Creating the Physical Property

Create a physical property by right-clicking on **Physical Properties** and selecting **New**. Now you will see the **Physical Property** window as shown in Figure 4 below.



The screenshot shows the 'Idealizations' dialog box with the following settings:

- Name: Solid 1
- ID: 1
- Type: Solid Elements
- Line Element Type: Bar
- Material: 4130 Alloy Steel
- Color: A red color swatch
- Associated Geometry:  (unchecked)
- Selected Entities: An empty list box
- Material Axes: Coordinate System: Assem1

Buttons at the bottom include an up arrow, a refresh icon, and OK/Cancel buttons.

Figure 4. Physical Property Form

Select the type as **Solid Elements**, then in the pull-down menu for **Material** select **4130 Alloy Steel,tempered,Ultra High Strength**. Check the **Associate Geometry** checkbox and select the four box parts from the model, then click **OK**. The material and physical property have now been created.

Repeat the same steps to define the physical property for the four cylinder parts of the model.

## 6. Meshing the Model

Right-click on **Mesh Model** from the **WaveFEA Model Tree** and select **Mesh Table** from expand menu and **Edit** the Mesh Model. You will see the **Mesh Table** window. Define the mesh size as shown in Figure 5. Make sure **Continuous Meshing** checkbox is unchecked and click **OK**.

|   | Part Name                | Visibility               | Color                    | Size (in) | Tolerance (in) | Element Order | Settings | Nodes | Elements |                          |  |
|---|--------------------------|--------------------------|--------------------------|-----------|----------------|---------------|----------|-------|----------|--------------------------|--|
| ✓ | pressfit_FEA             | <input type="checkbox"/> |                          | 0.519616  | 1.03923e-005   | Parabolic     | Settings | 0     | 0        | <input type="checkbox"/> |  |
| ✓ | Interference Example:1   | <input type="checkbox"/> | <input type="checkbox"/> | 0.35      | 7.34848e-006   | Parabolic     | Settings | 6165  | 2927     | <input type="checkbox"/> |  |
| ✓ | Interference Example_1:1 | <input type="checkbox"/> | <input type="checkbox"/> | 0.5       | 6.19674e-006   | Parabolic     | Settings | 2691  | 1600     | <input type="checkbox"/> |  |
| ✓ | Interference Example_2:1 | <input type="checkbox"/> | <input type="checkbox"/> | 0.35      | 7.34848e-006   | Parabolic     | Settings | 6351  | 3018     | <input type="checkbox"/> |  |
| ✓ | Interference Example_3:1 | <input type="checkbox"/> | <input type="checkbox"/> | 0.5       | 6.19674e-006   | Parabolic     | Settings | 2637  | 1566     | <input type="checkbox"/> |  |
| ✓ | Interference Example_4:1 | <input type="checkbox"/> | <input type="checkbox"/> | 0.35      | 7.34848e-006   | Parabolic     | Settings | 6308  | 2994     | <input type="checkbox"/> |  |
| ✓ | Interference Example_5:1 | <input type="checkbox"/> | <input type="checkbox"/> | 0.5       | 6.19674e-006   | Parabolic     | Settings | 2686  | 1602     | <input type="checkbox"/> |  |
| ✓ | Interference Example_6:1 | <input type="checkbox"/> | <input type="checkbox"/> | 0.35      | 7.34848e-006   | Parabolic     | Settings | 6436  | 3060     | <input type="checkbox"/> |  |
| ✓ | Interference Example_7:1 | <input type="checkbox"/> | <input type="checkbox"/> | 0.5       | 6.19674e-006   | Parabolic     | Settings | 2712  | 1625     | <input type="checkbox"/> |  |

Figure 5. Mesh Table Form

## 7. Generating the Contacts

Right-click on **Surface Contacts** and select **New**. You will see the **Surface Contact** window as shown in Figure 6 below.

| Surface Contact  |  |
|--|--|
| Name:  | Surface Contact 1  |
| ID:  | 1  |
| <input type="radio"/> Auto <input checked="" type="radio"/> Manual   |  |
| Type:  | Surface to Surface   |
| Master Entity:   | face<8>@Part1-5<br>face<8>@Part1-6<br>face<8>@Part1-1<br>face<8>@Part1-2 |
| Slave Entity:  | face<5>@Part2-1<br>face<5>@Part2-2<br>face<4>@Part2-4<br>face<4>@Part2-3 |
| <b>Contact Data</b><br>Contact Type: Separation<br>Penetration Type: Symmetric Cont.<br>Stiffness Factor: 0.002<br>Coefficient of Friction: 0<br>Penetration Surface Offset (in): 0.003<br><input type="checkbox"/> Max Activation Distance (in) |  |
| <input type="button" value="Advanced Options"/>  |  |
| <input type="button" value="OK"/> <input type="button" value="Cancel"/>  |  |

Figure 6. Surface Contact Form

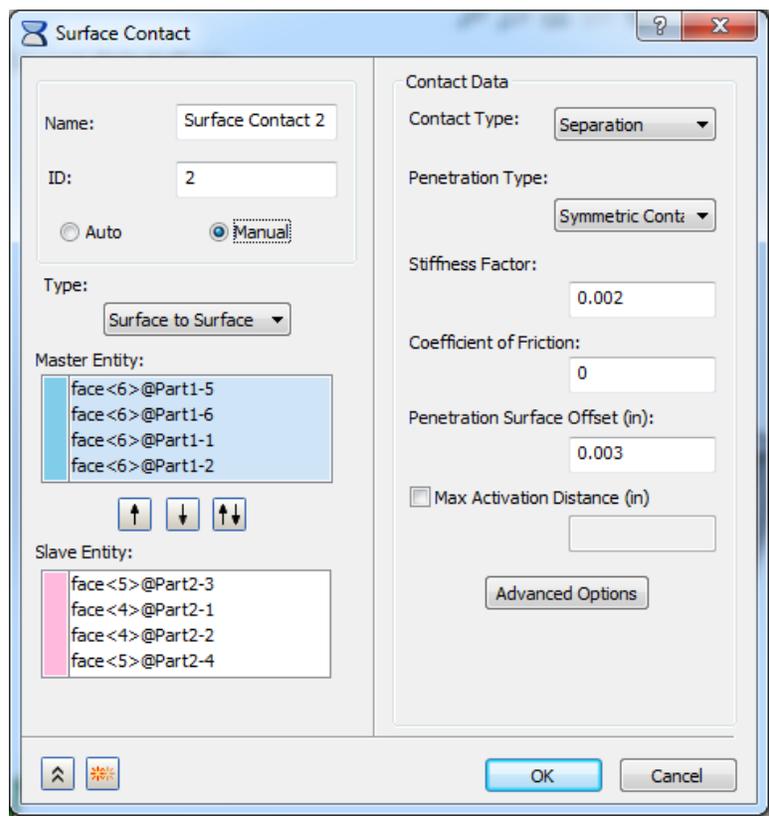
In the **Master Entities** select the inside upper surface of the four box faces. See Figure 7 below. Use the axes to help orient the model, be careful not to accidentally select the outer upper surface of the box.

In the **Slave Entities** select the inside upper surface of the four cylindrical faces. See Figure 7 below.



**Figure 7. Top Box and Cylinder Contacts**

You will need to repeat the process of creating a **New Connection Region** for the inner bottom surface of the box and the bottom faces of the cylinder making sure to title them to easily distinguish between the two (example: *bottomBox*, *bottomCylinder*). See Figure 8.



**Figure 8. Surface Contact Form**

Your selected regions should resemble the following pictures:

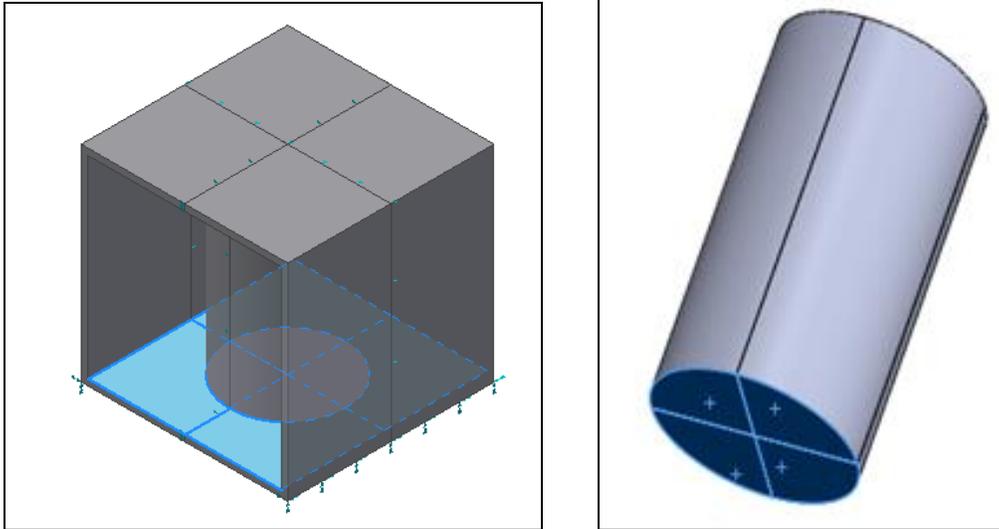


Figure 9. Bottom Box and Cylinder Contacts

## 8. Creating the Loads

To create a body load, right-click on **Loads** and select **New**. You will now see the **Load** window. Select the type as **Gravity** and fill in **F<sub>y</sub>** box with the value **-386.4**. Your window should look like the one shown below.

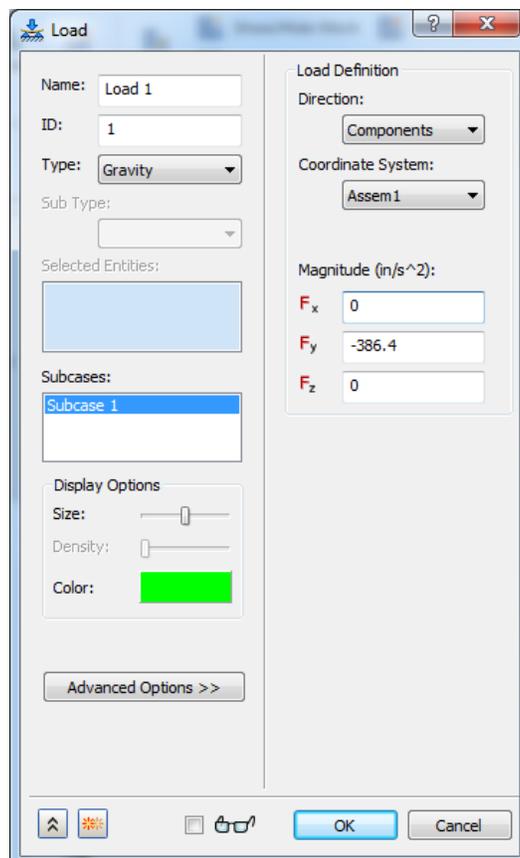
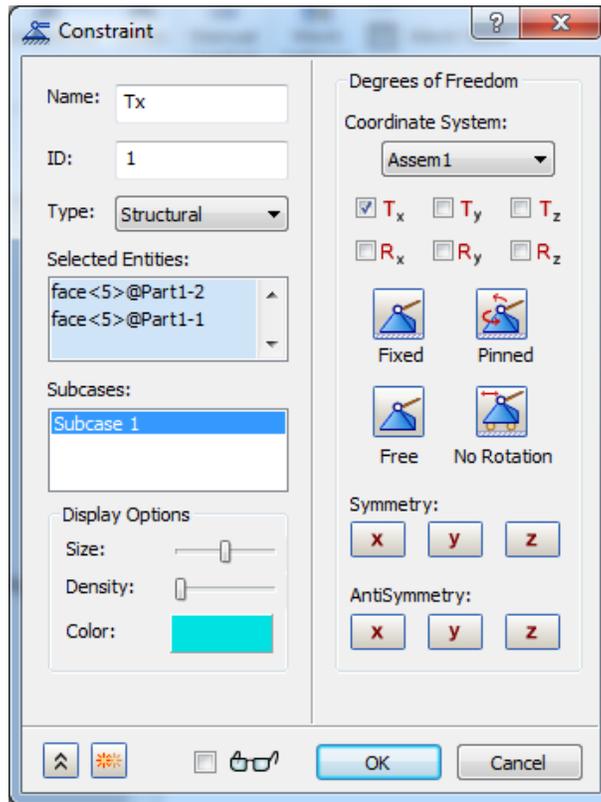


Figure 10. Gravity Loads

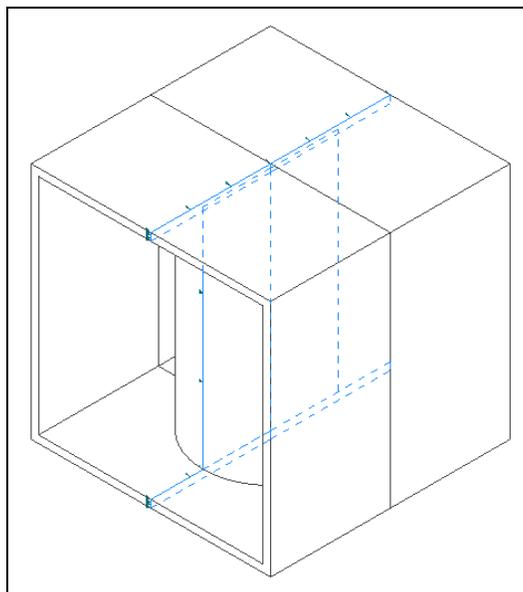
## 9. Creating the Constraints

Right-click on **Constraints** and select **New**. You will see the **Constraint** window as shown in Figure 11.



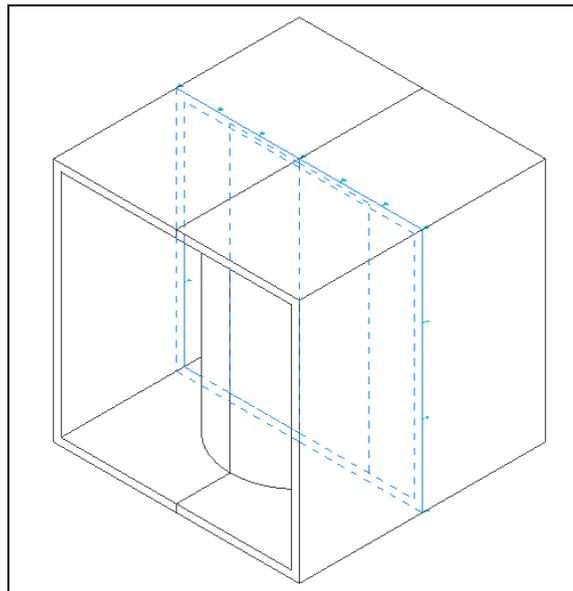
**Figure 11. Constraint1 Face Selection**

Give it a title. Select all faces that are parallel in the Y-Z plane (there are four cylinder faces and eight box faces that are in the Y-Z plane as shown in Figure 13). After selecting all twelve faces, check only **TX** and then click **OK**.

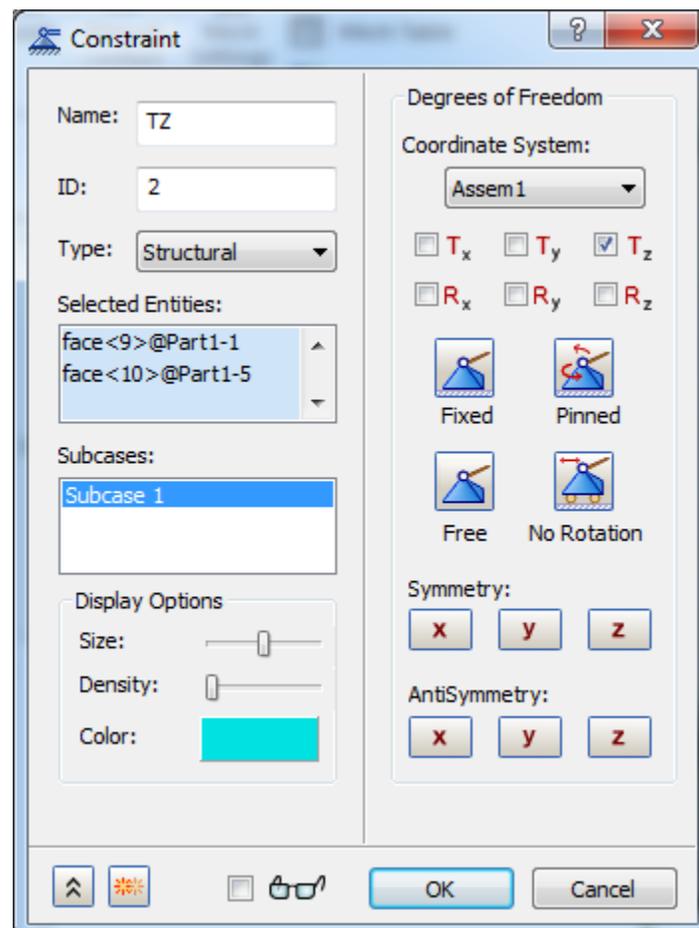


**Figure 12. Y-Z Plane Parallel Face Selection**

Repeat the process of creating a new **Constraint** to set up constraints that are parallel to the X-Y plane. There are four cylinder faces and four box faces that are parallel to the X-Y plane. After selecting all eight faces, check only **TZ** and then click **OK**.



**Figure 13. X-Y Plane Parallel Face Selection**



**Figure 14. Constraint 2 Face Selection**

To constrain the box, you need to create a new constraint by right-clicking on **Constraint**. Select the bottom edges of the box that are on the X-Z plane and run parallel to the z-axis. Since the box's geometry is broken up, there will be a total of four selections. After selecting the edges, select the **Fixed** option and click **OK**. The geometry is now properly constrained.

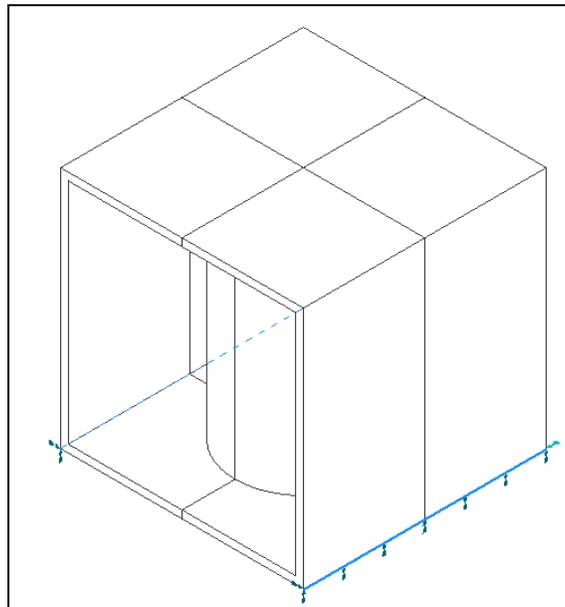


Figure 15. Fixed in Bottom Face Parallel Edge Selection

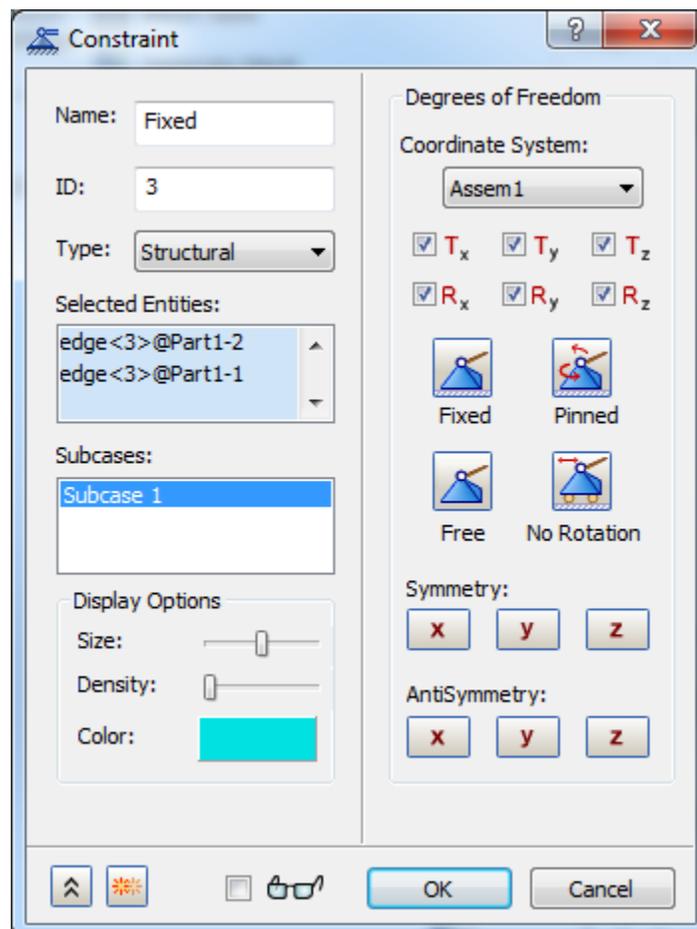


Figure 16. Constraint 3 Face Selection

## 10. Nonlinear Analysis Setup

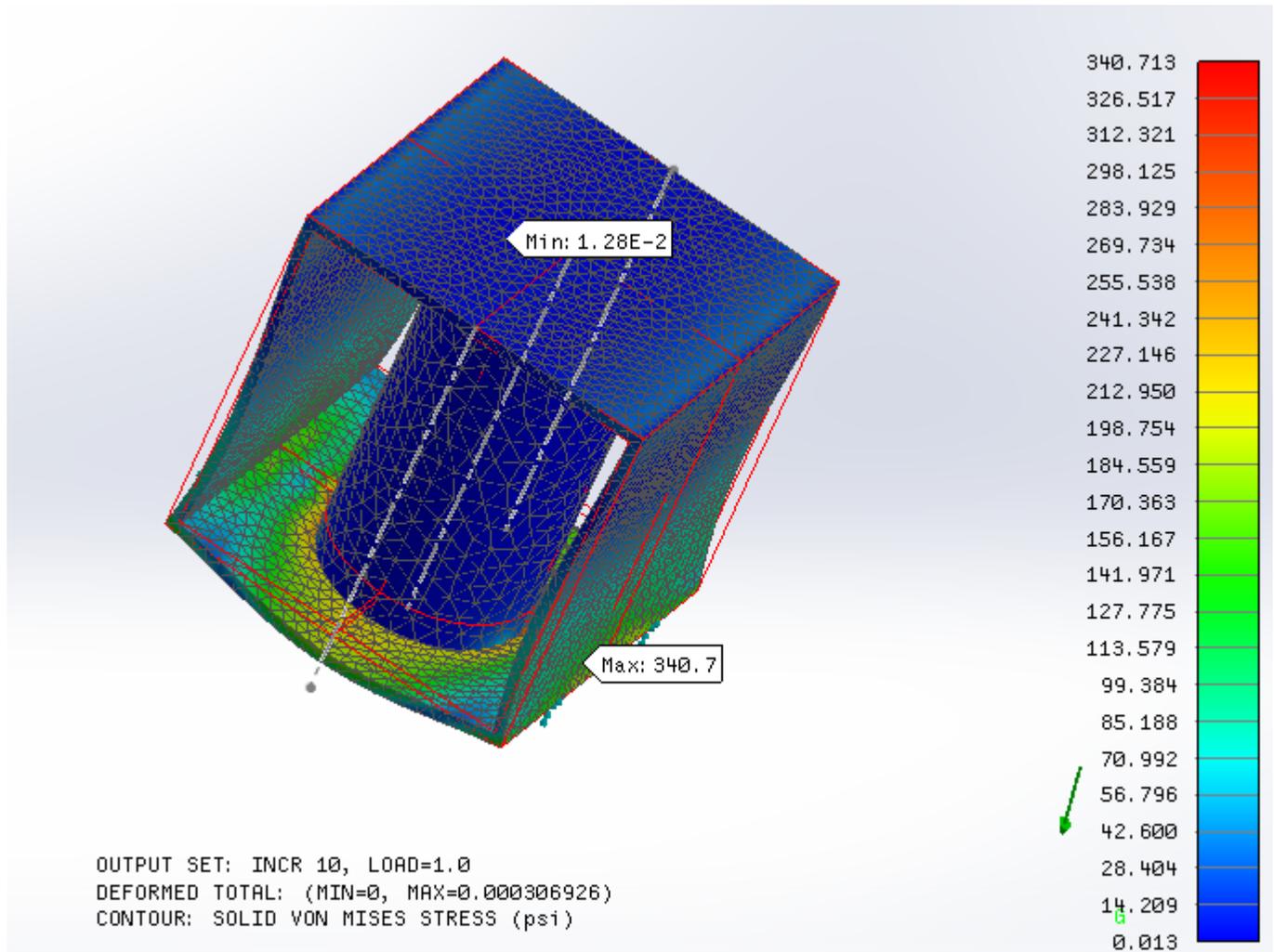
Right-click on **Nonlinear Setup 1**. You will now see the Load Set Option for Nonlinear Analysis window. Type in **10** for **Number of Increments**, and for **Intermediate Output**, select **On** from the pull-down menu. Click **OK**.

The image shows a software dialog box titled "Nonlinear Setup". The dialog is split into two panes. The left pane contains the following fields:  
Name: Nonlinear Setup 1  
ID: 1  
The right pane, titled "Nonlinear Settings:", contains the following fields:  
Number of Increments: 10  
Intermediate Output: On (selected from a dropdown menu)  
Arc Length: Off (selected from a dropdown menu)  
Below these settings is a button labeled "Advanced Settings". At the bottom of the dialog are "OK" and "Cancel" buttons. A small icon is visible in the bottom-left corner of the dialog.

Figure 17. Nonlinear Setup Form

## 11. Running the Analysis

At this point you should save your work, then right-click on **Analysis1** and select **Solve in Nastran**. After running the analysis, right-click on **Results**, then **Edit** to view the results. You can reset the deformation scale to 1% for a less exaggerated view. The final results should resemble the picture below.



**Figure 18. Press Fit Nonlinear Static Analysis Results**

## 7. Conclusion

**Press fit** analysis can be very useful when dealing with interference fit cases. This case shows WaveFEA versatility for handling such a problem. It also shows the capability of WaveFEA to use different contacts and split faces constraining in a **nonlinear** analysis to setup the interference fit.