Press Fit Nonlinear Static Analysis in WaveFEA

1. Creating Geometry

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

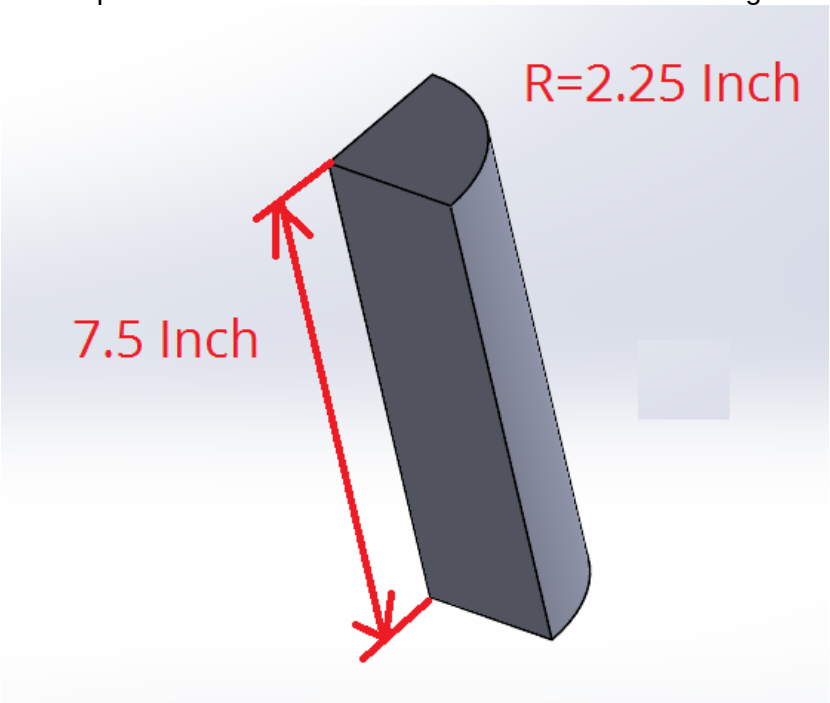

Figure 1. Quarter Cylinder Geometry

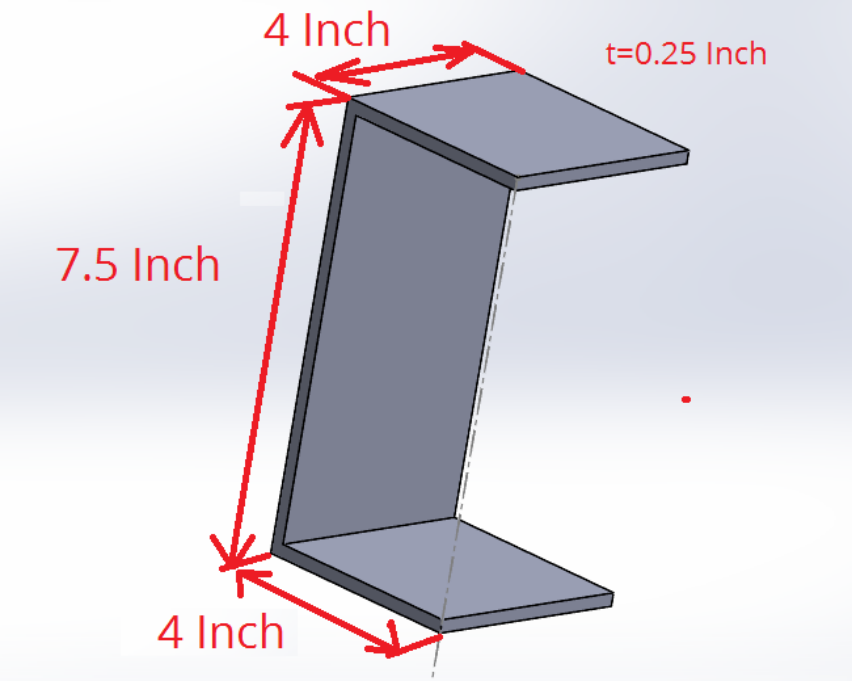

Figure 1. Clip Geometry

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.

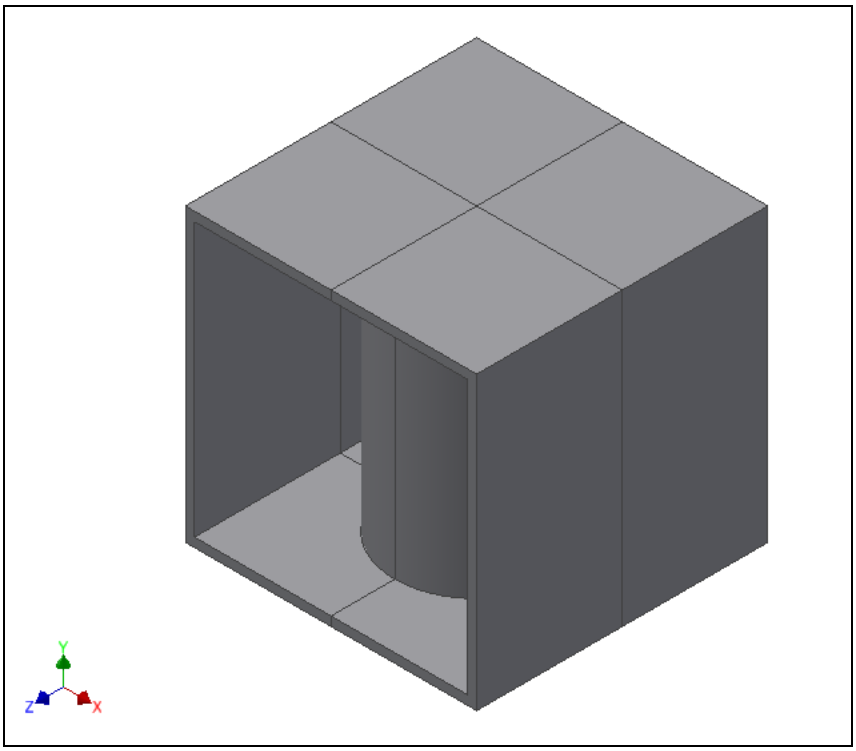

Figure 1. Housed Cylinder Geometry

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.**

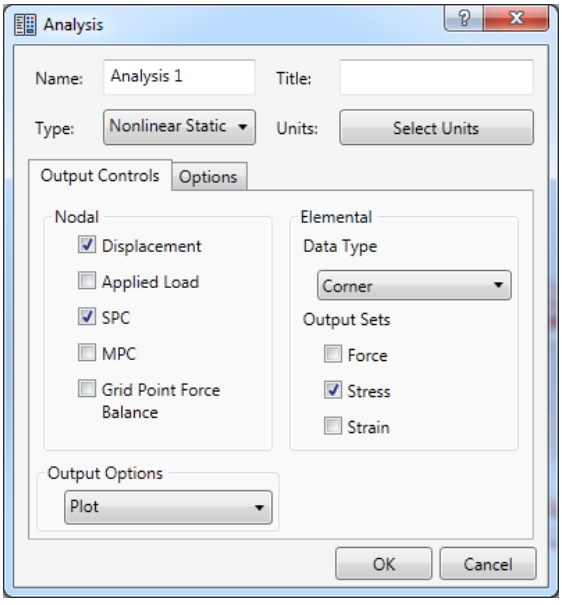

Figure 2. Analysis Form

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.

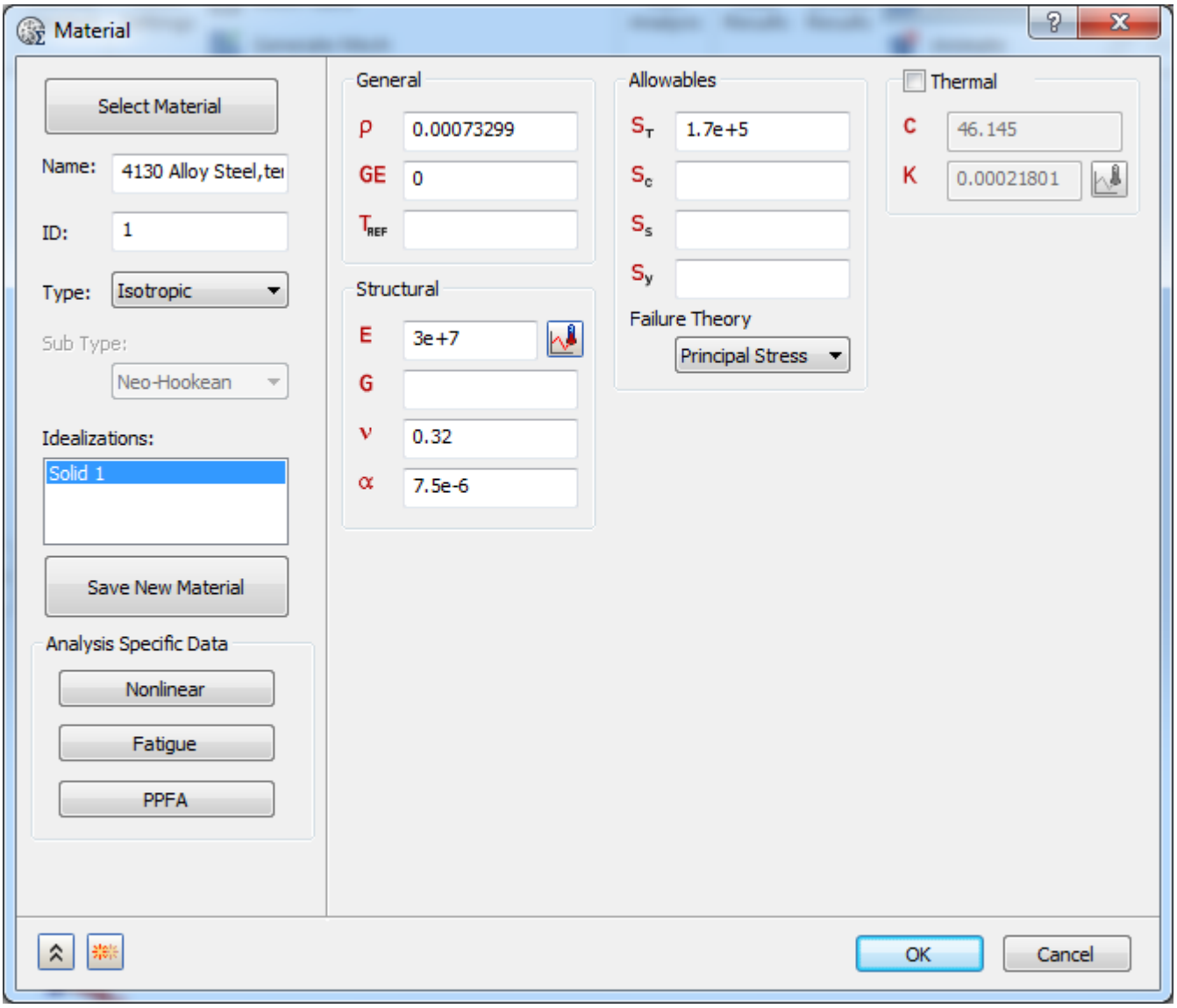

Figure 3. Material Form

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.

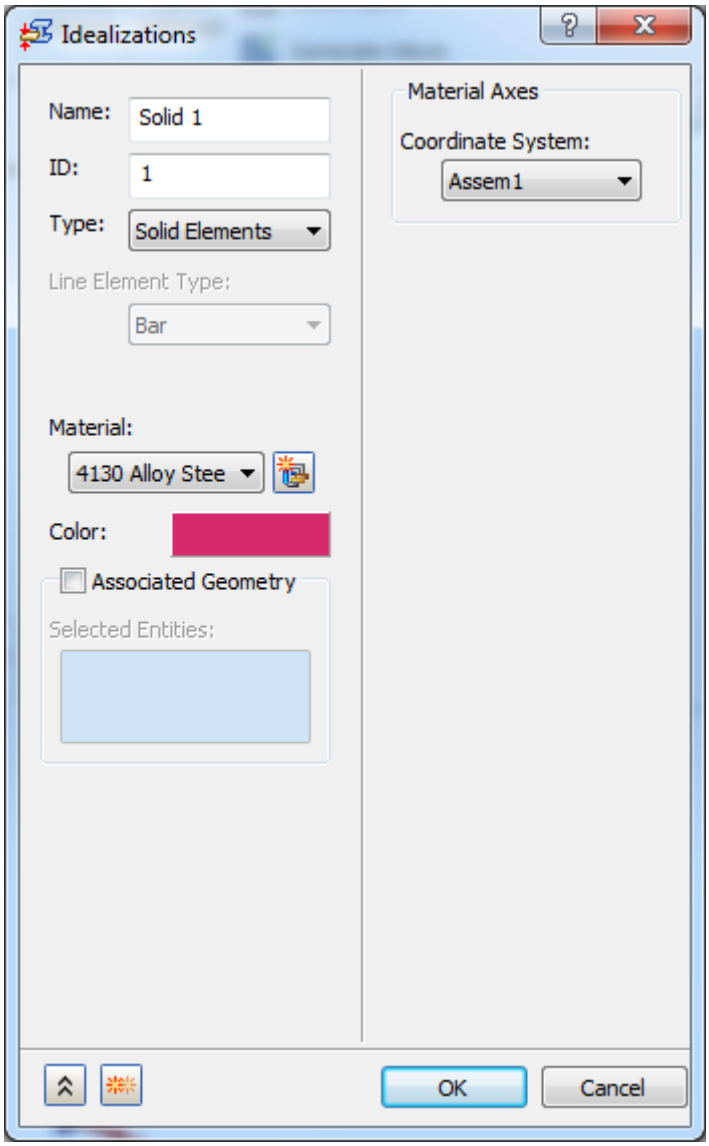

Figure 4. Physical Property Form

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**.

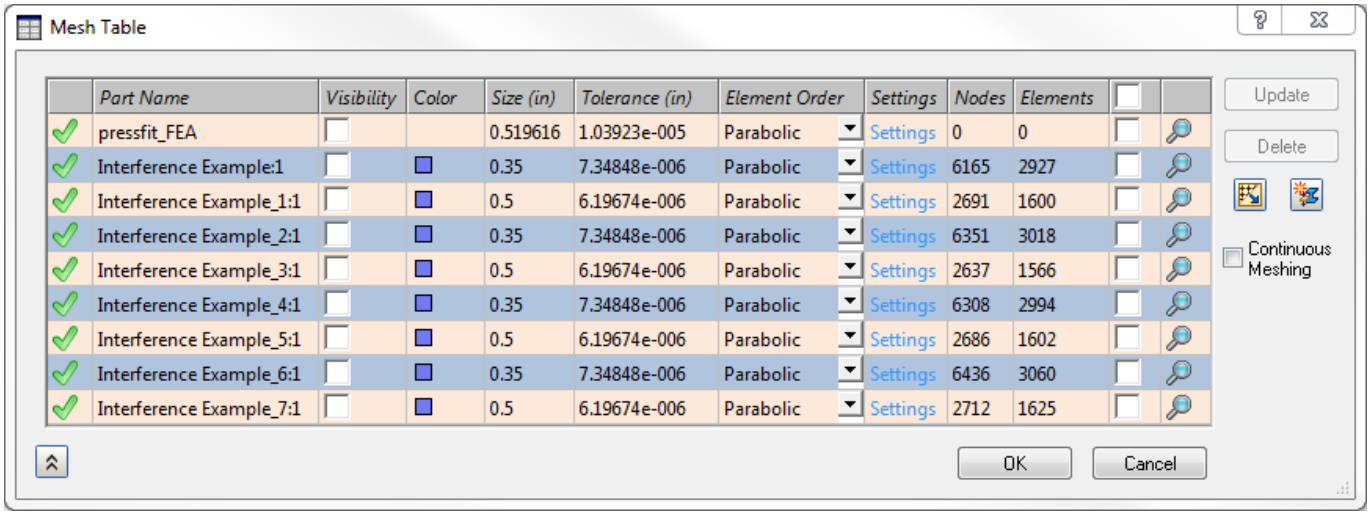

Figure 5. Mesh Table Form

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.

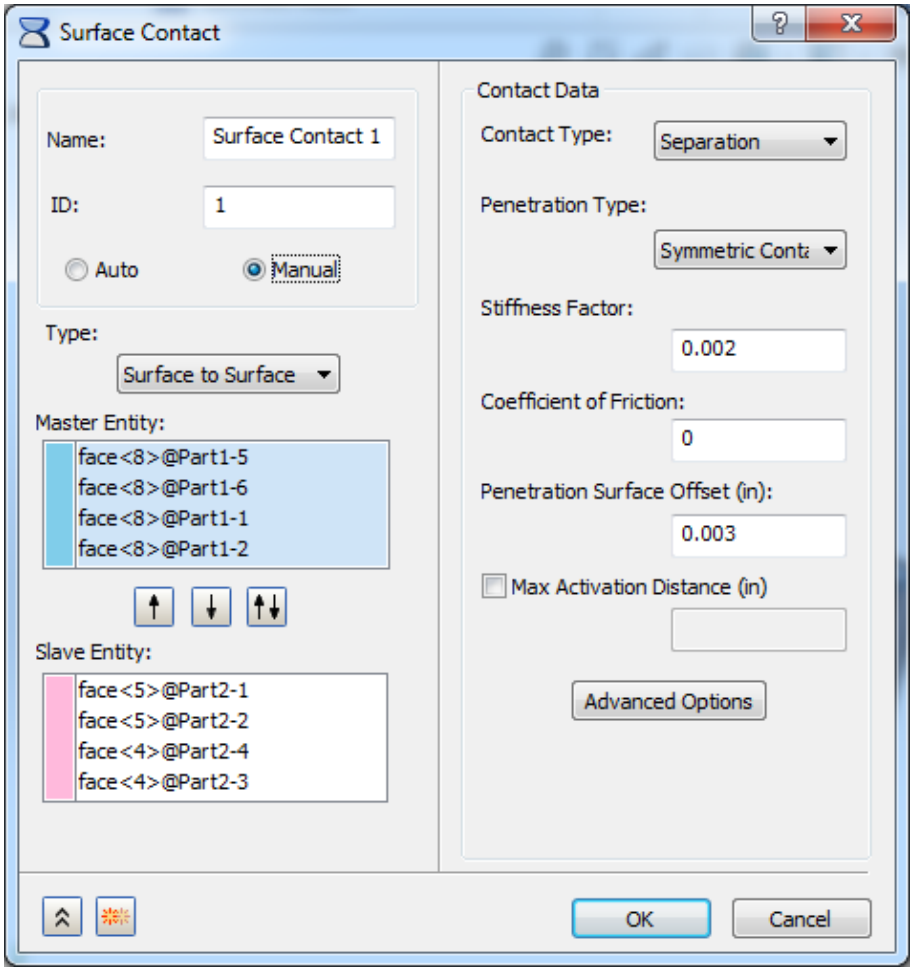

Figure 6. Surface Contact Form

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.

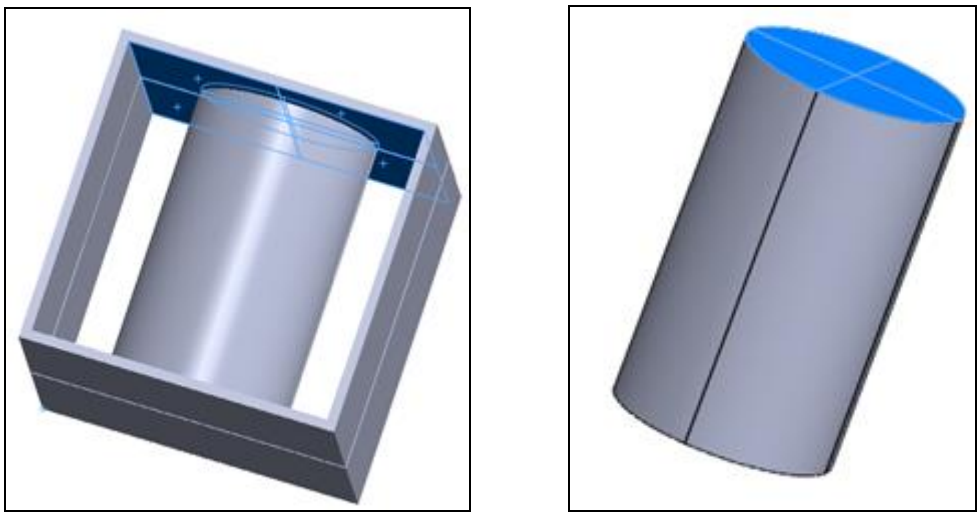

Figure 7. Top Box and Cylinder Contacts

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.

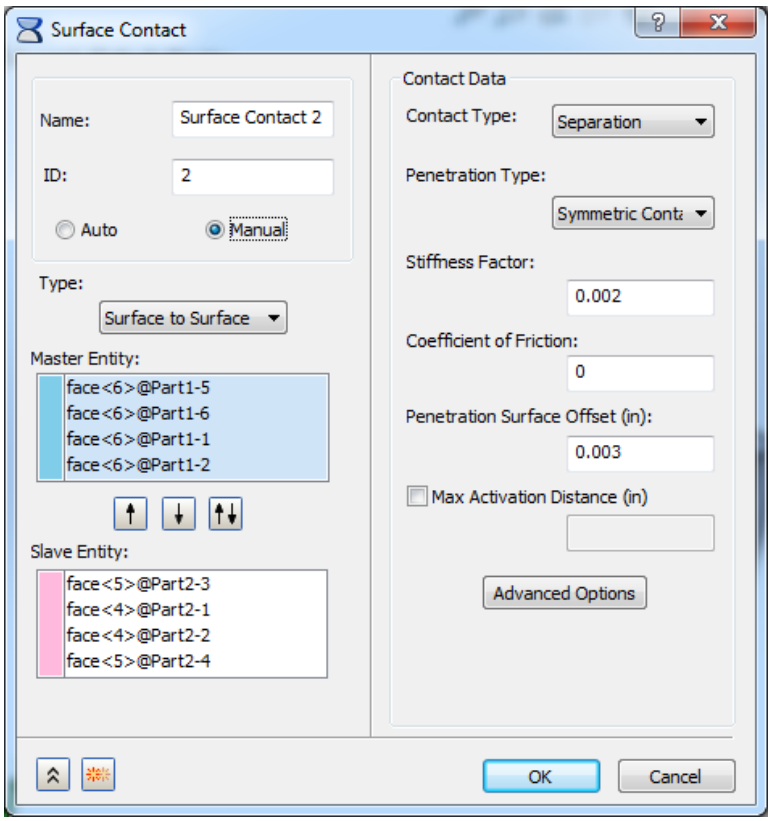

Figure 8. Surface Contact Form

Your selected regions should resemble the following pictures:

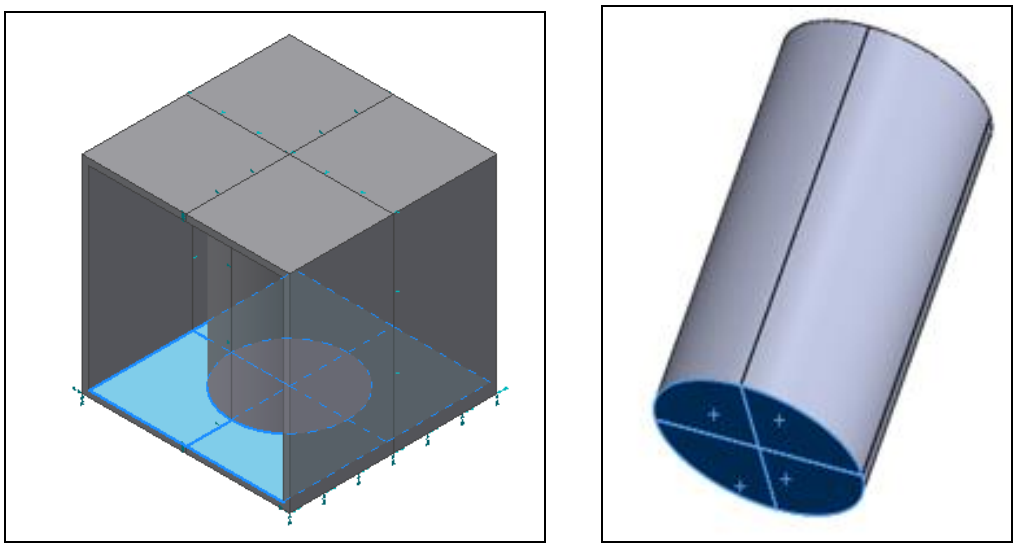

Figure 9. Bottom Box and Cylinder Contacts

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 **Fy** box with the value **-386.4**. Your window should look like the one shown below.

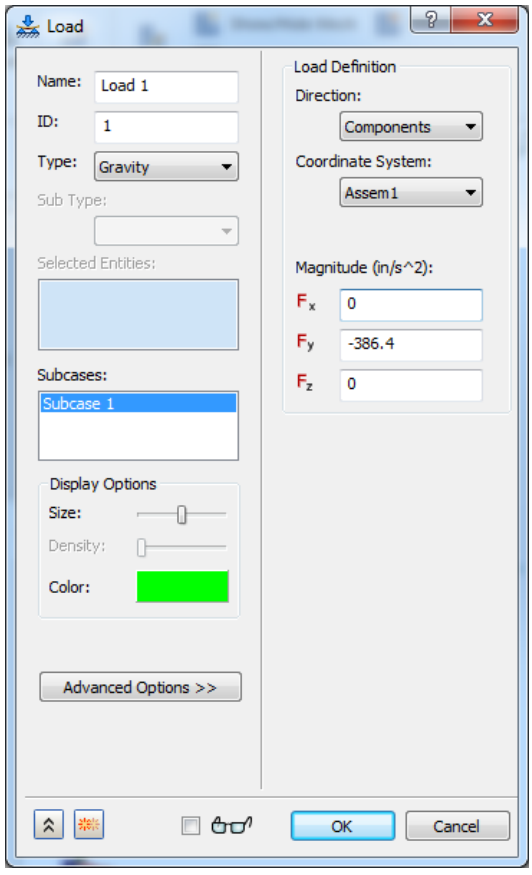

Figure 10. Gravity Loads

9. Creating the Constraints

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

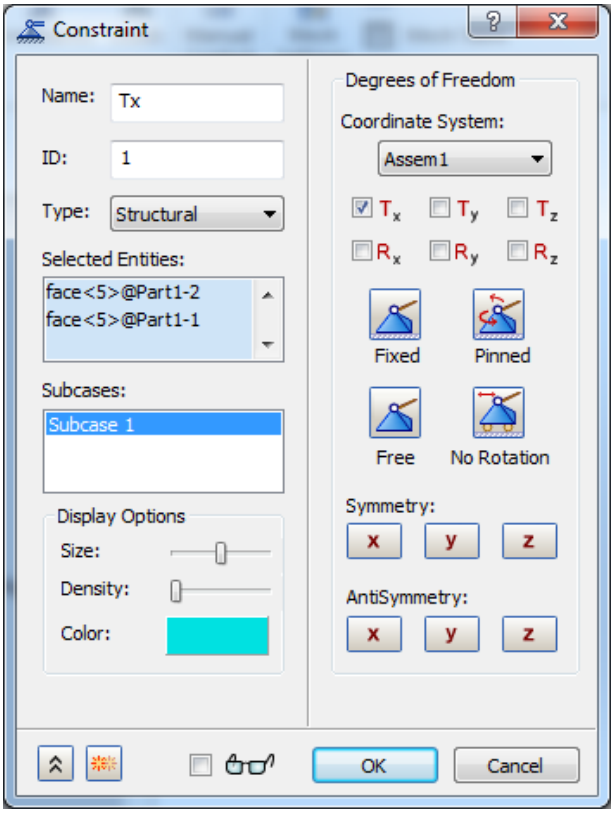

Figure 11. Constraint1 Face Selection

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**.

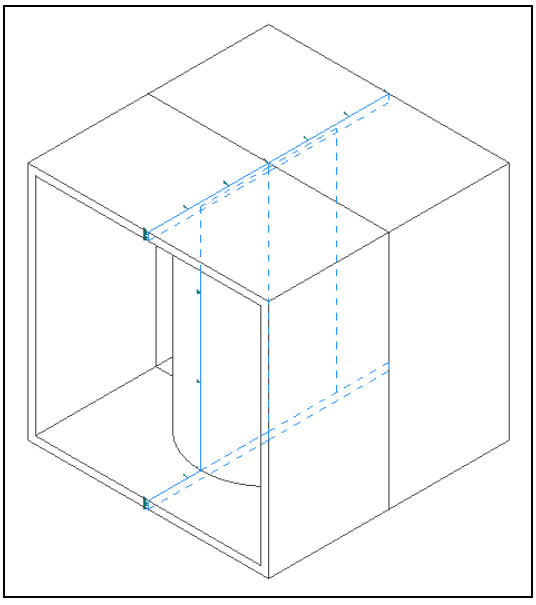

Figure 12. Y-Z Plane Parallel Face Selection

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**.

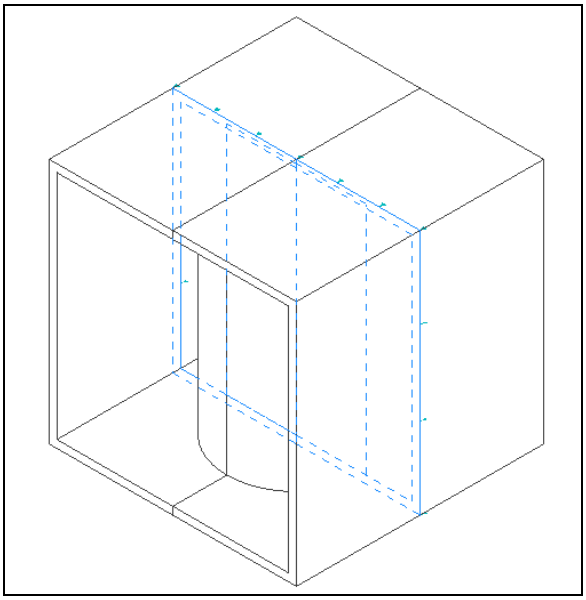

Figure 13. X-Y Plane Parallel Face Selection

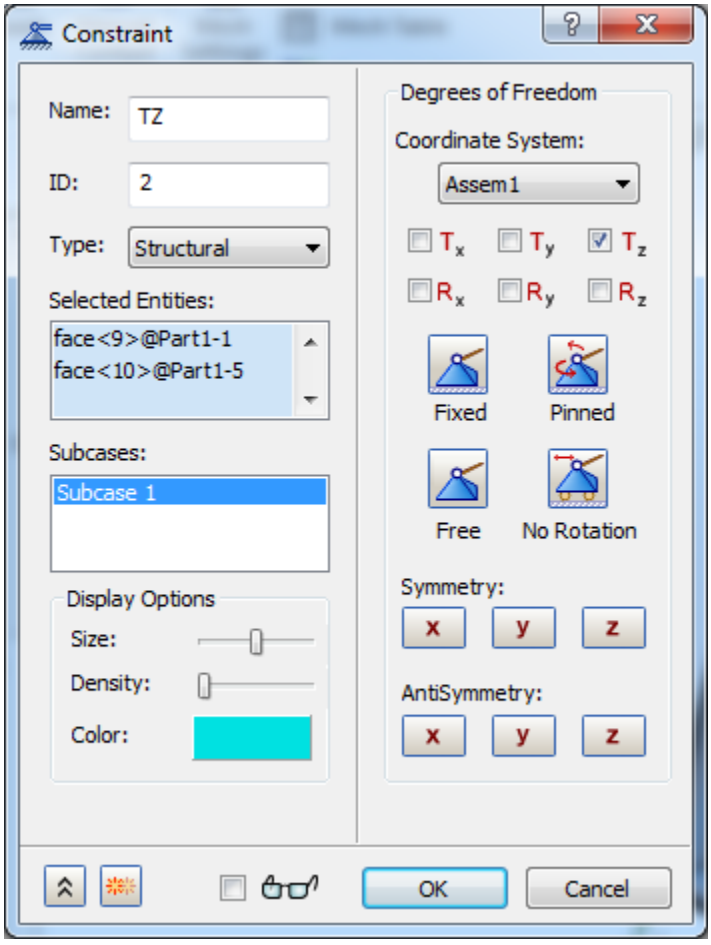

Figure 14. Constraint 2 Face Selection

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.

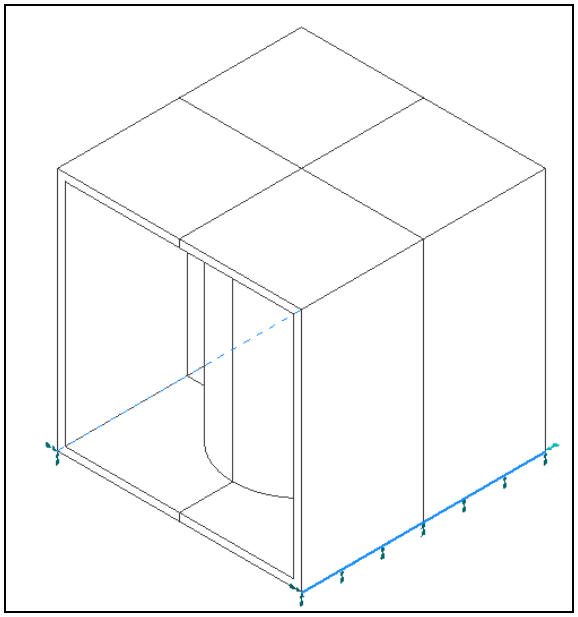

Figure 15. Fixed in Bottom Face Parallel Edge Selection

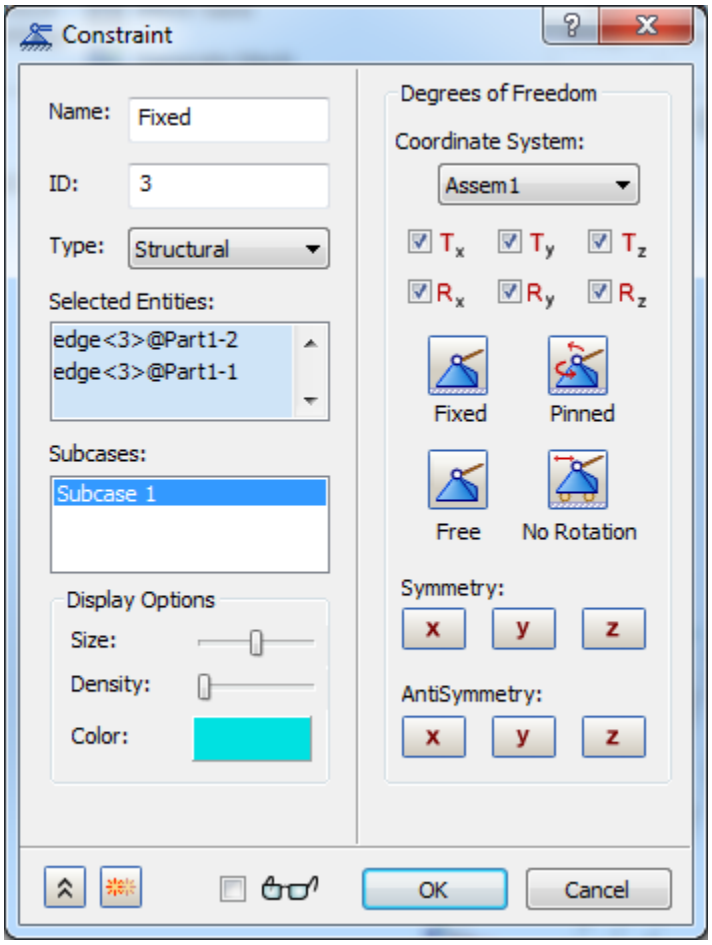

Figure 16. Constraint 3 Face Selection

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**.

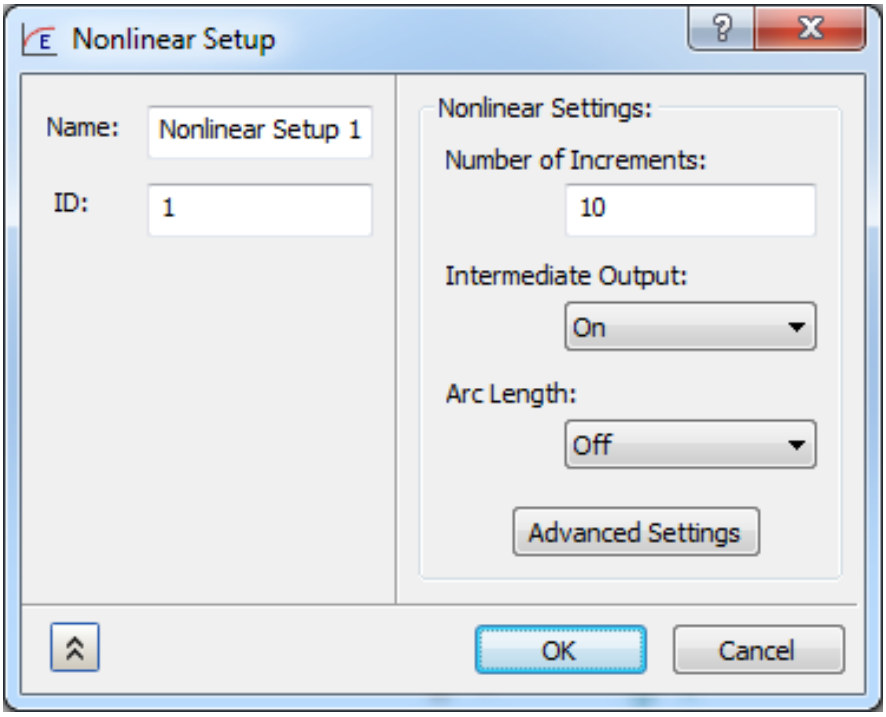

Figure 17. Nonlinear Setup Form

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.

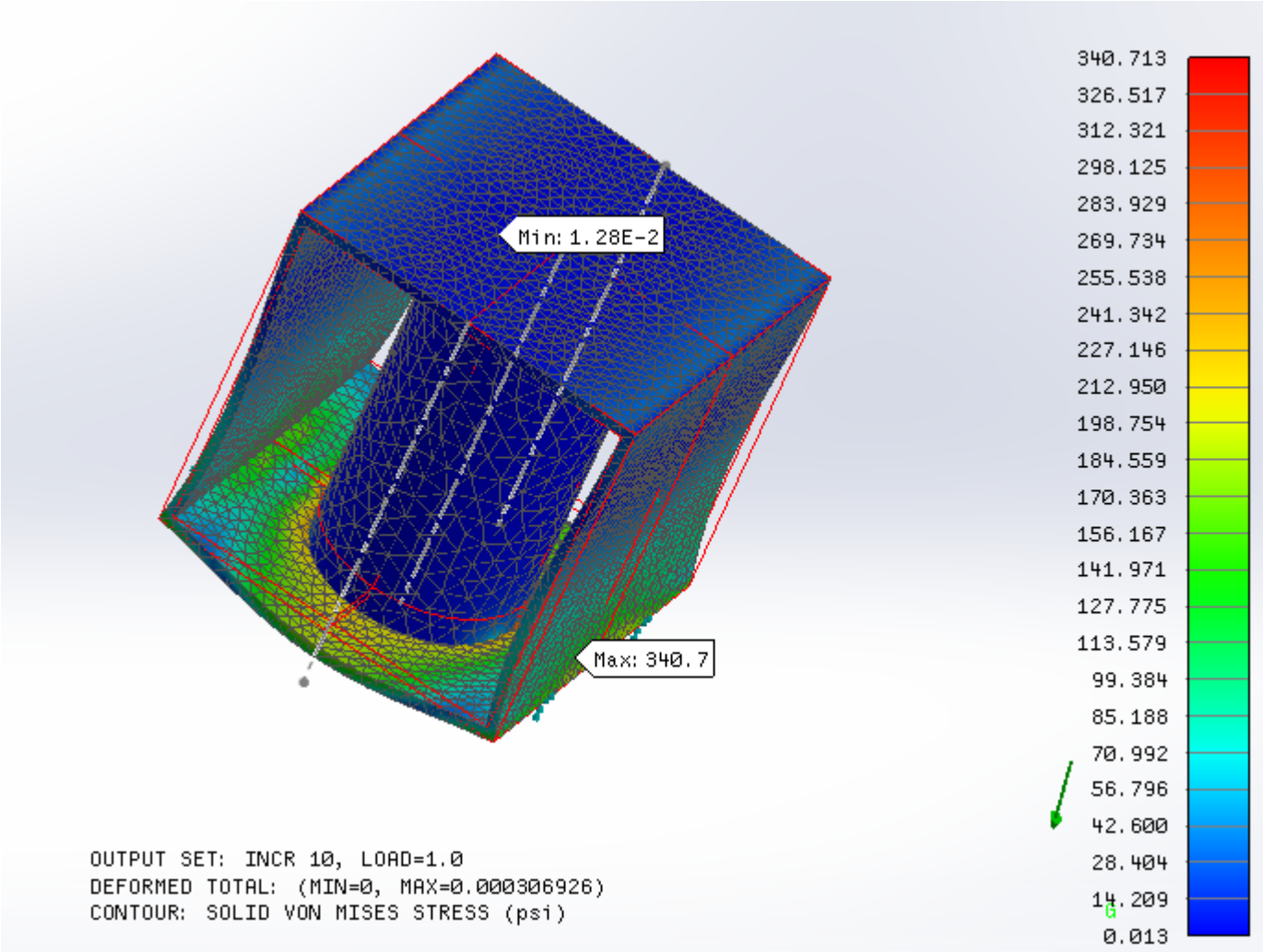

Figure 18. Press Fit Nonlinear Static Analysis Results

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.