# Setting up a Concrete Model in WaveFEA

## 1. Introduction

The model shown in Figure 1 is a 20 in. wide by 100 in. long by 8 in. tall slab of concrete that is simply supported on one end and simply supported with a roller on the other. The concrete has bar elements modeled along the upper portion of the bottom layer of the thickness of the slab along the lengths to model re-bar. There is a uniform pressure load applied to the surface of the slab. The concrete material is modeled using a nonlinear elastic material with a compressive Young's modulus 100 times greater than the tensile Young's modulus.

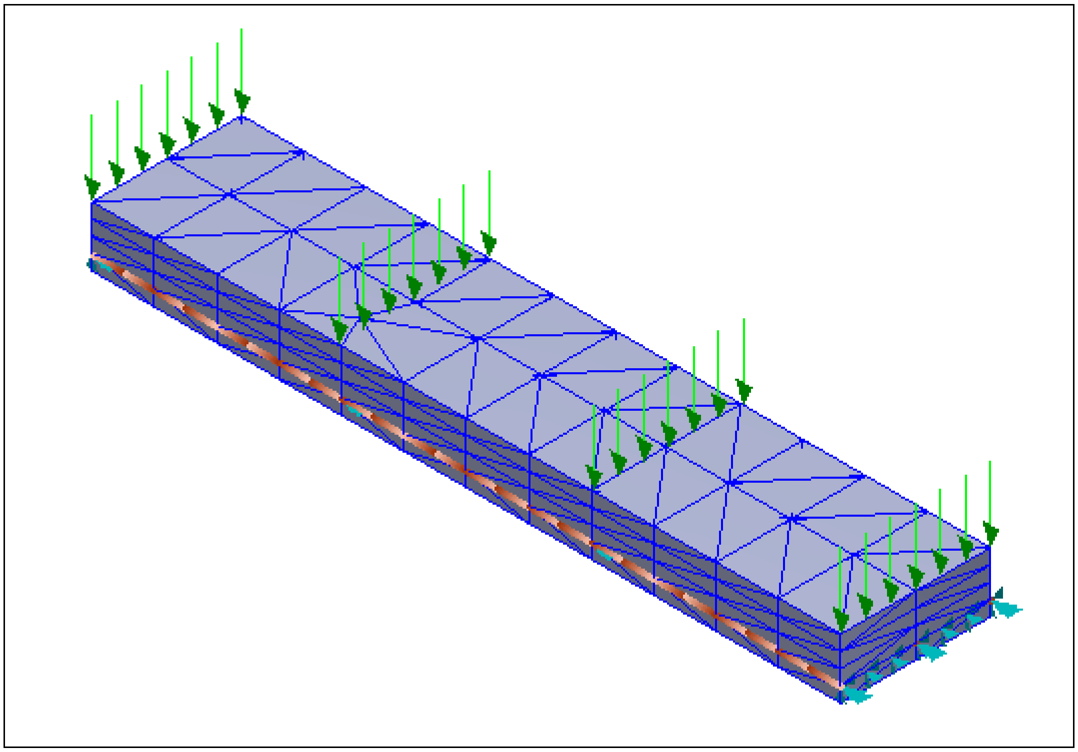

**Figure 1. Concrete slab model**

## 2. Create the Rod Model

Here first need to create a part model, then insert it into an assembly for analysis.

### 2.1 Create a part

Create a 20 in. by 100 in. rectangle. Go to **sketch** select **Front plane** to create a **Rectangle** shown in Figure 2.

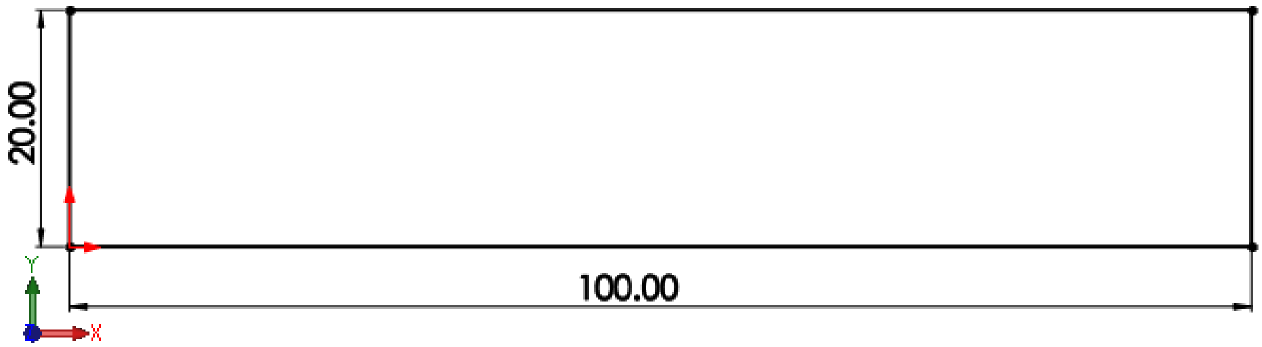


**Figure 2. Creating a Rectangle Location**

Finish the sketch.

Select **Extruded Boss/Base** command from **Features,** Select the created **Sketch** and **Extrude** up to **2 inch.**

Select **OK** to finish.

The extruded model should look as shown in Figure 3

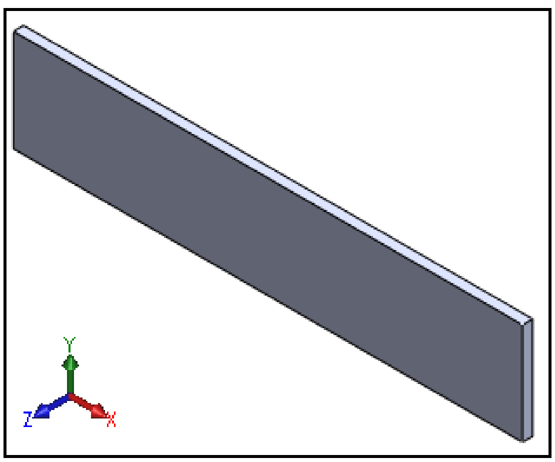


**Figure 3. 2x10x100 slab**

Create a **Sketch** on one of the surface of the **Extruded** face, as shown in the below figure 4

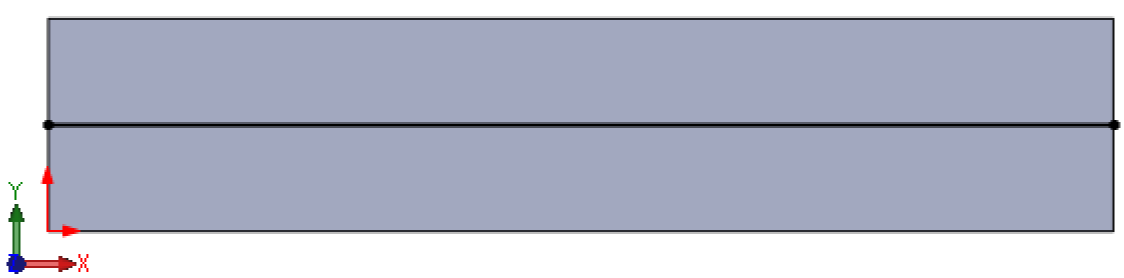


**Figure 4. Sketch on the slab**

Finish the sketch, Go to **Tool/Ribbon bar** and select **Split Line.**

Select the Created sketch in **Sketch to Project** and the two opposite faces to be split in the **Faces to Split.**

Click on **OK** to finish.  After splitting the two surfaces the model should look as shown in Figure 4

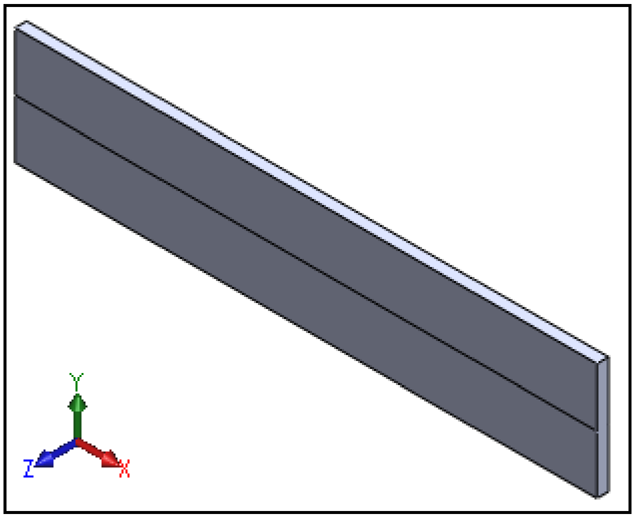


**Figure 4.  Split slab**

Save the model as **"20_100_2_tall_slab (.sldprt)".**

## 2.2. Creating an assembly model

Open a new assembly document and insert the part created **20_100_2_tall_slab (.sldprt)**

The same part should be in four times, assemble all the four parts one on another, the entire model of the assembly as shown in Figure 5.

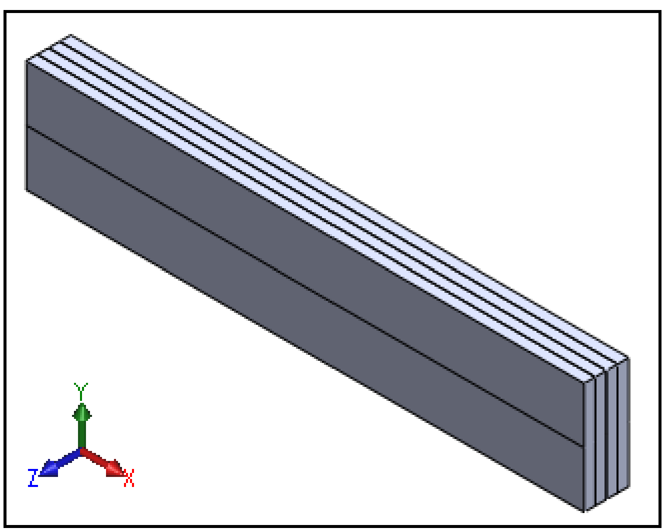


**Figure 5.  Assembled 8x10x100 slab**

Save the model as  **"20_100_8_tall_slab (.sldasm)".**

# 3. Define the Material Property

We now need to define the material properties for the concrete. Go to **Model** right click on **Materials** and choose **New,** Rename the title the material as **Concrete** and enter a Young's **Modulus E** of **3.6E6** and a **Poisson's ratio** of **0.15** as shown in Figure 6.

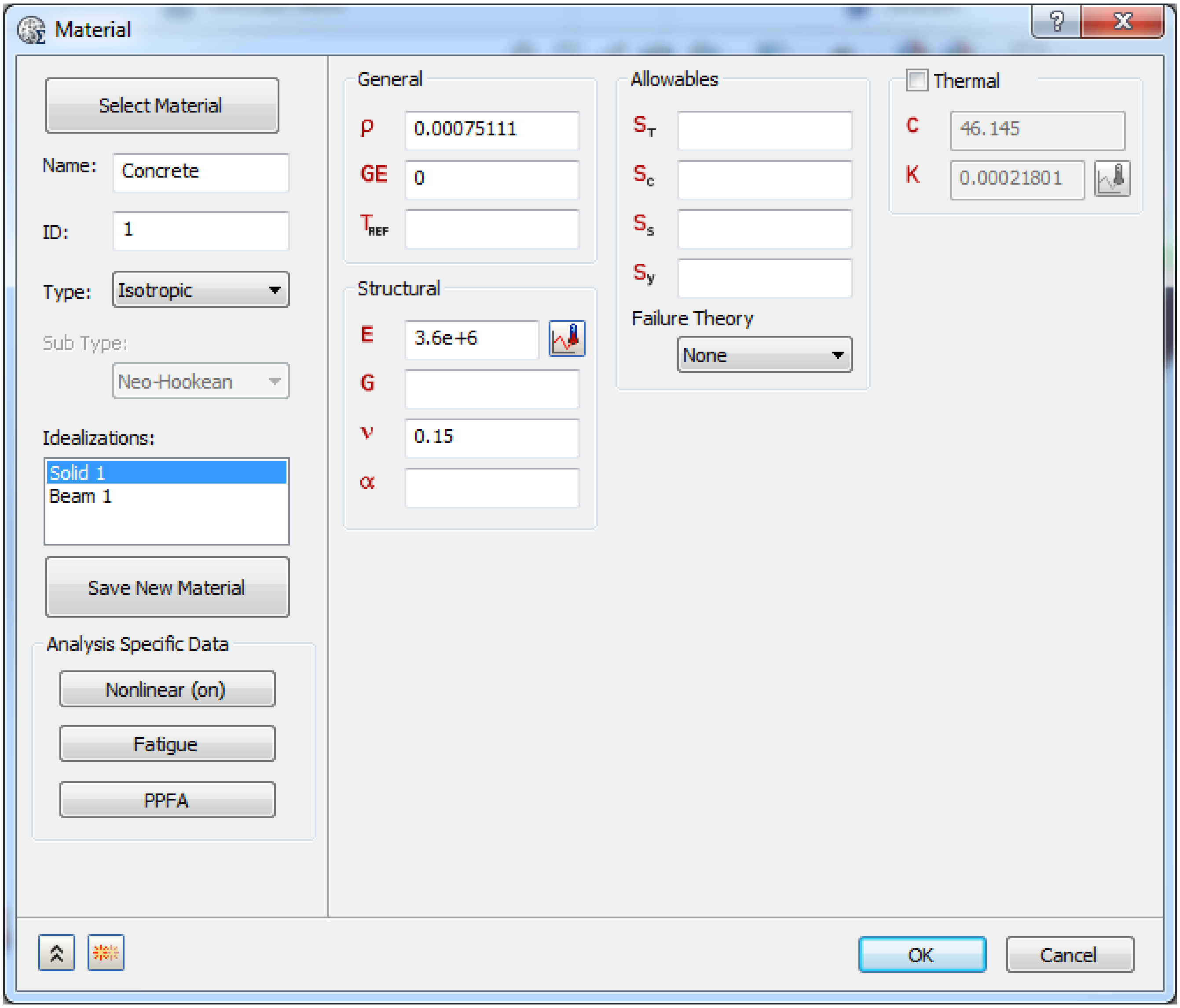


**Figure 6. Isotropic Material definition**

After defining material data, click on **Nonlinear** button from the **Analysis Specific Data,** a **Nonlinear Material Data** window will appear, select **Nonlinear Elastic** radio button, enter the values of **Strain and Stress** into their respective fields as shown in the Figure 7 below.

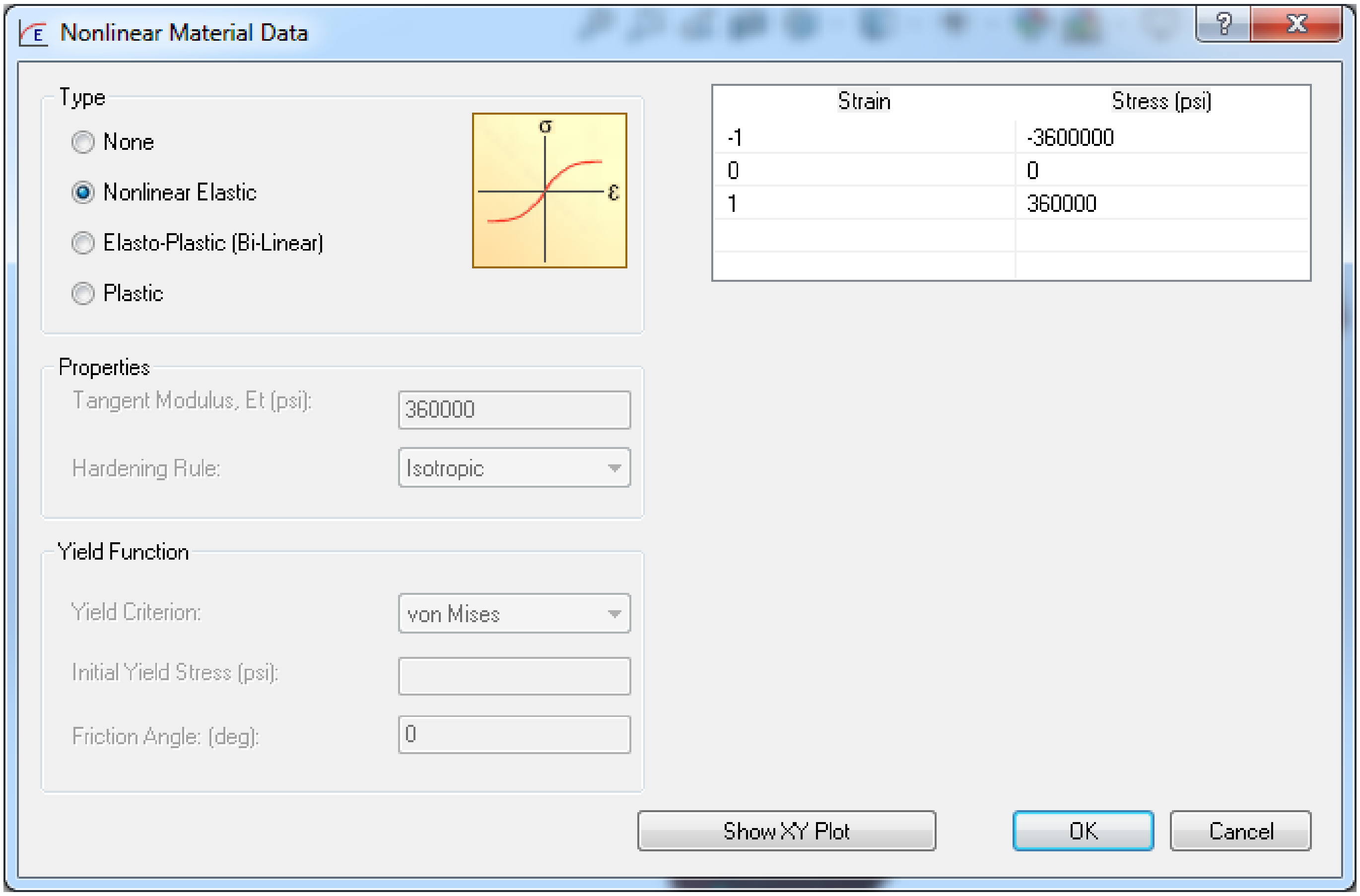


**Figure 7. Nonlinear material settings for the concrete**

The Concrete Function that was just created can be seen in graphical form in Figure 8.

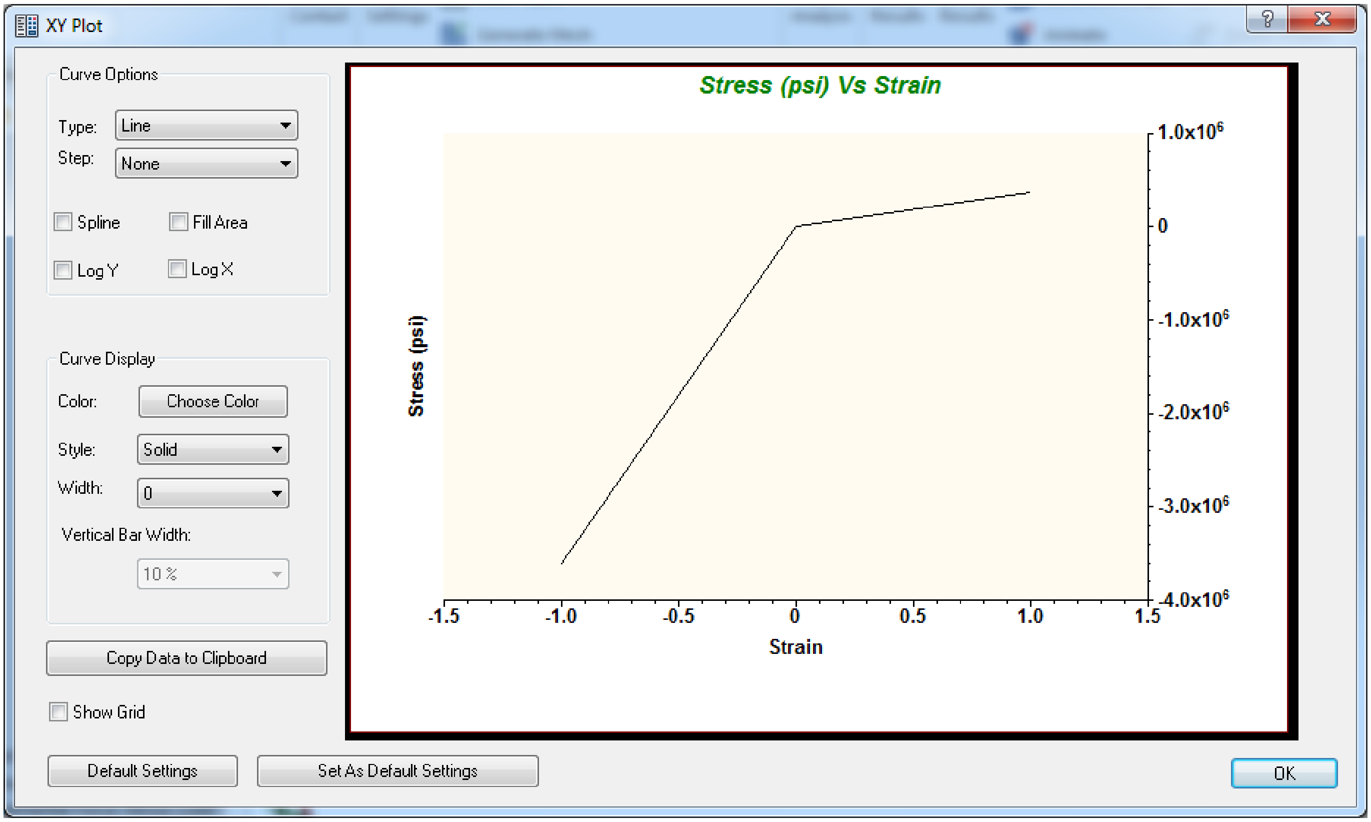


**Figure 8. Stress vs. Strain Concrete Function**

Click **OK** thrice to create **Concrete** Material.

Create new material for the bar elements, go to **WaveFEA Model Tree** right click on **Materials** and choose **New,** Rename the title the material as **AISI 4340 Steel.** The window should now look like Figure 9.

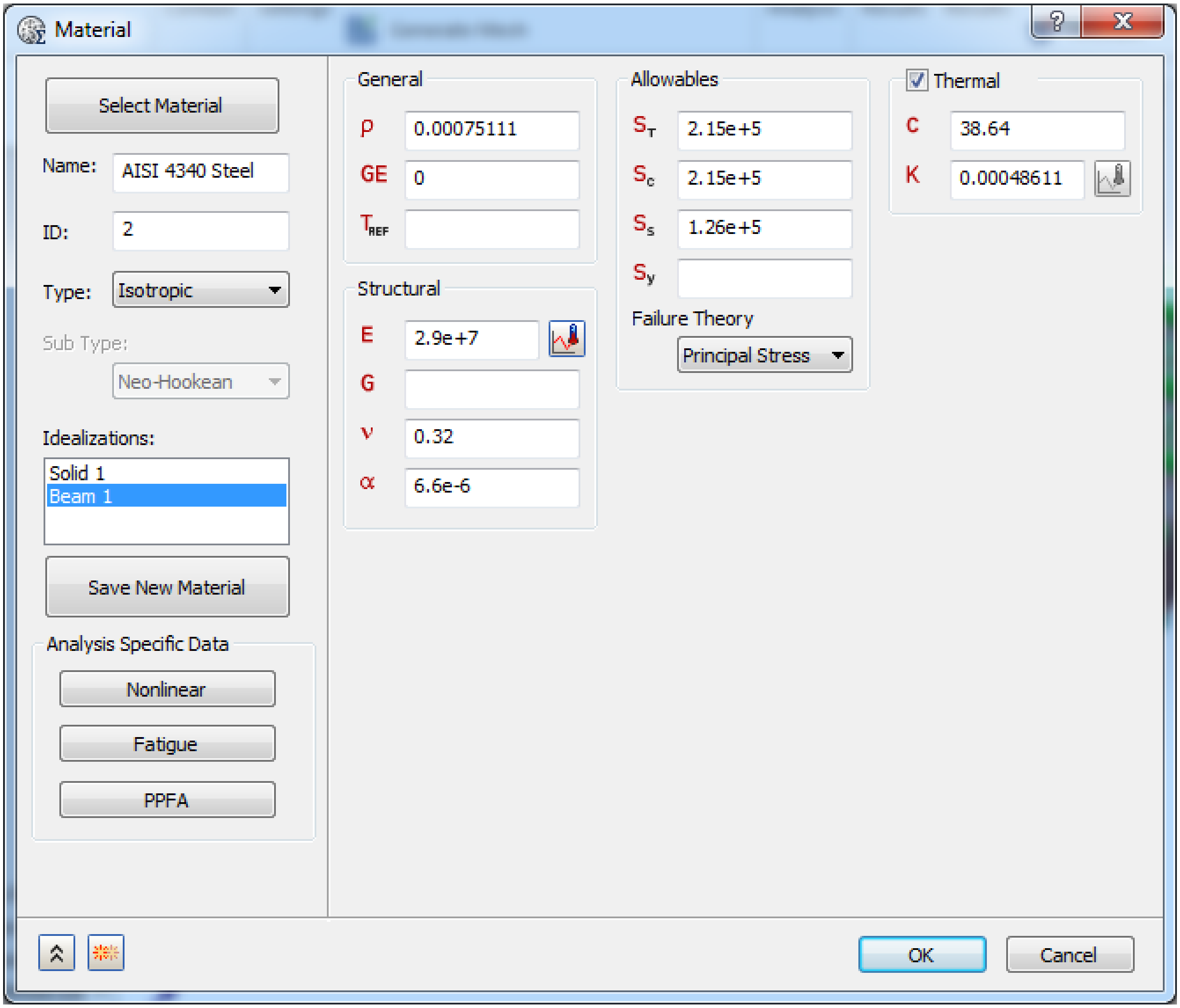


**Figure 9. Isotropic Material Definition**

Click **OK.**

## 4. Define the Physical property

Here two physical properties should be created, one for bar elements another for solid elements.

Create a solid physical property by going to **WaveFEA Model Tree** right click on **Physical Properties** and choose **New,** rename the title the material as **Concrete Property**, select **Solid Elements** from the **Type** drop down menu, select **Concrete** from the **Material** drop down menu, click **OK** to define **Physical Property** for **Concrete.**

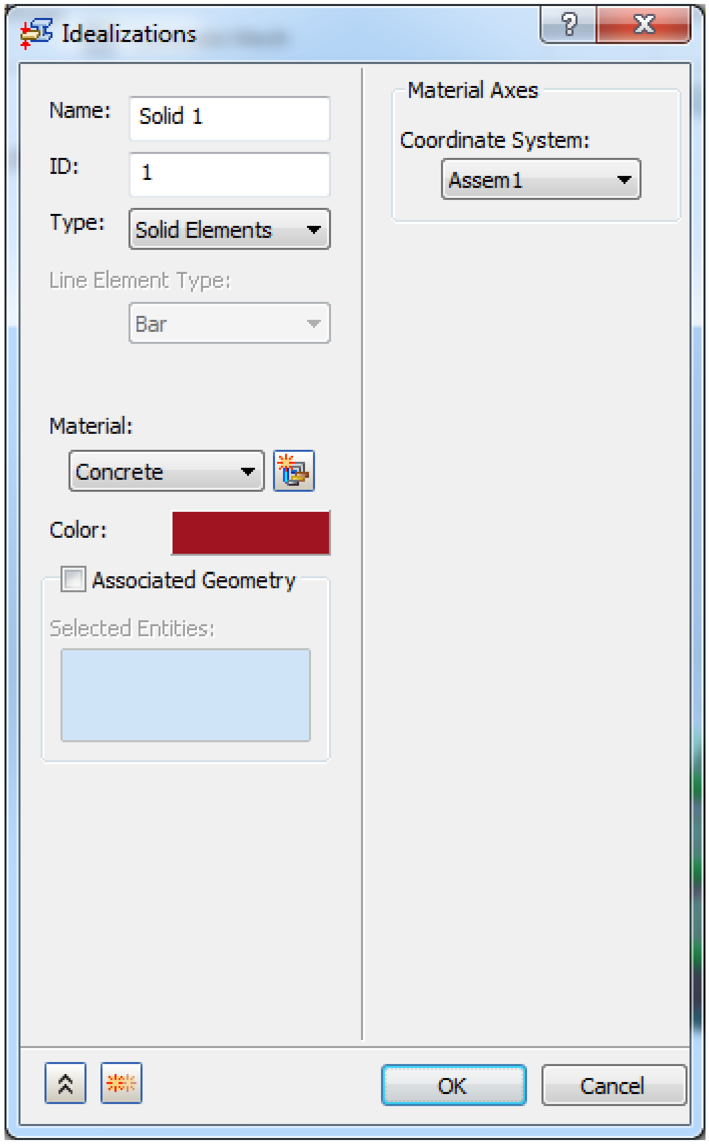


**Figure 10. Physical Property Definition**

The second property is for bar element. Before selecting the **Associated Geometry,** Need to hide the first three parts by switching to **Model Tree**.

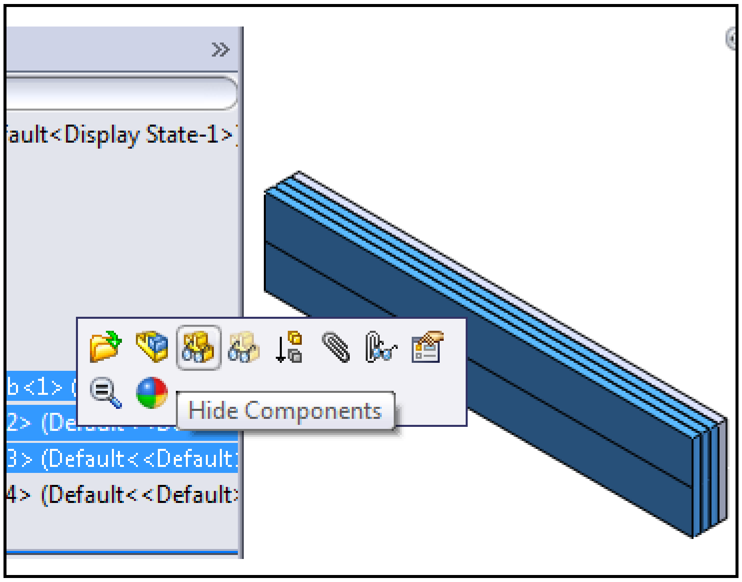


**Figure 11. Hide parts in Model tree**

After hiding the three parts switch to **WaveFEA Model Tree** view

Right-click on **Beams** and choose **New,** rename the title as **Rod Property**, select **Line Elements** from the **Type** drop down menu, select **AISI 4340 Steel** from the **Material** drop down Menu, Select **Cross Section** radio button and click on **Define** button, select **Rod** from the **Shape** drop down and enter **0.707** in **DIM1 (in)** Box. Click **OK**, check **Associated Geometry** select the three edges as shown in the figure.

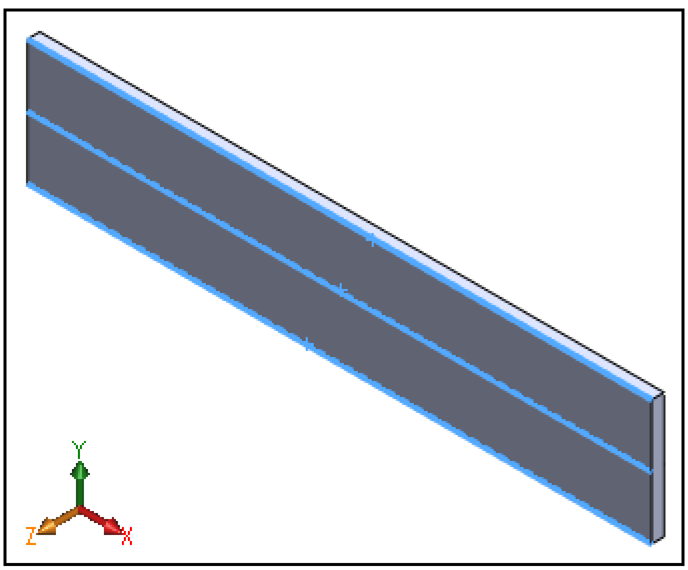


**Figure 12. Beam property selection**

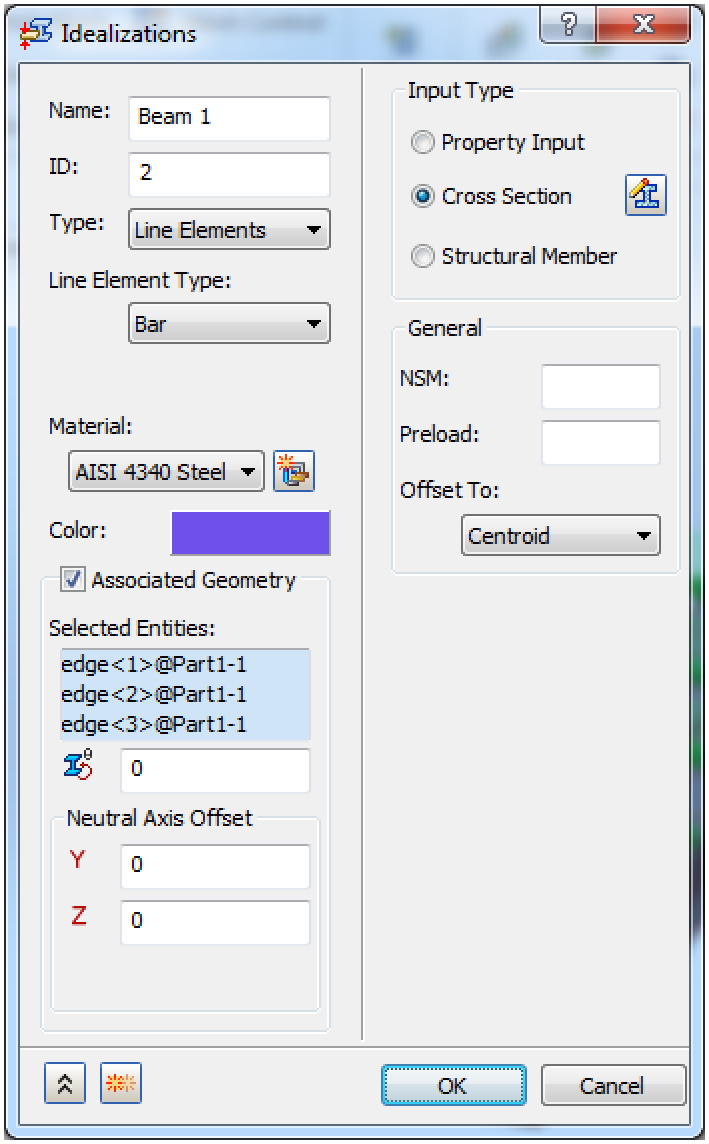


**Figure 13. Beam property definition**

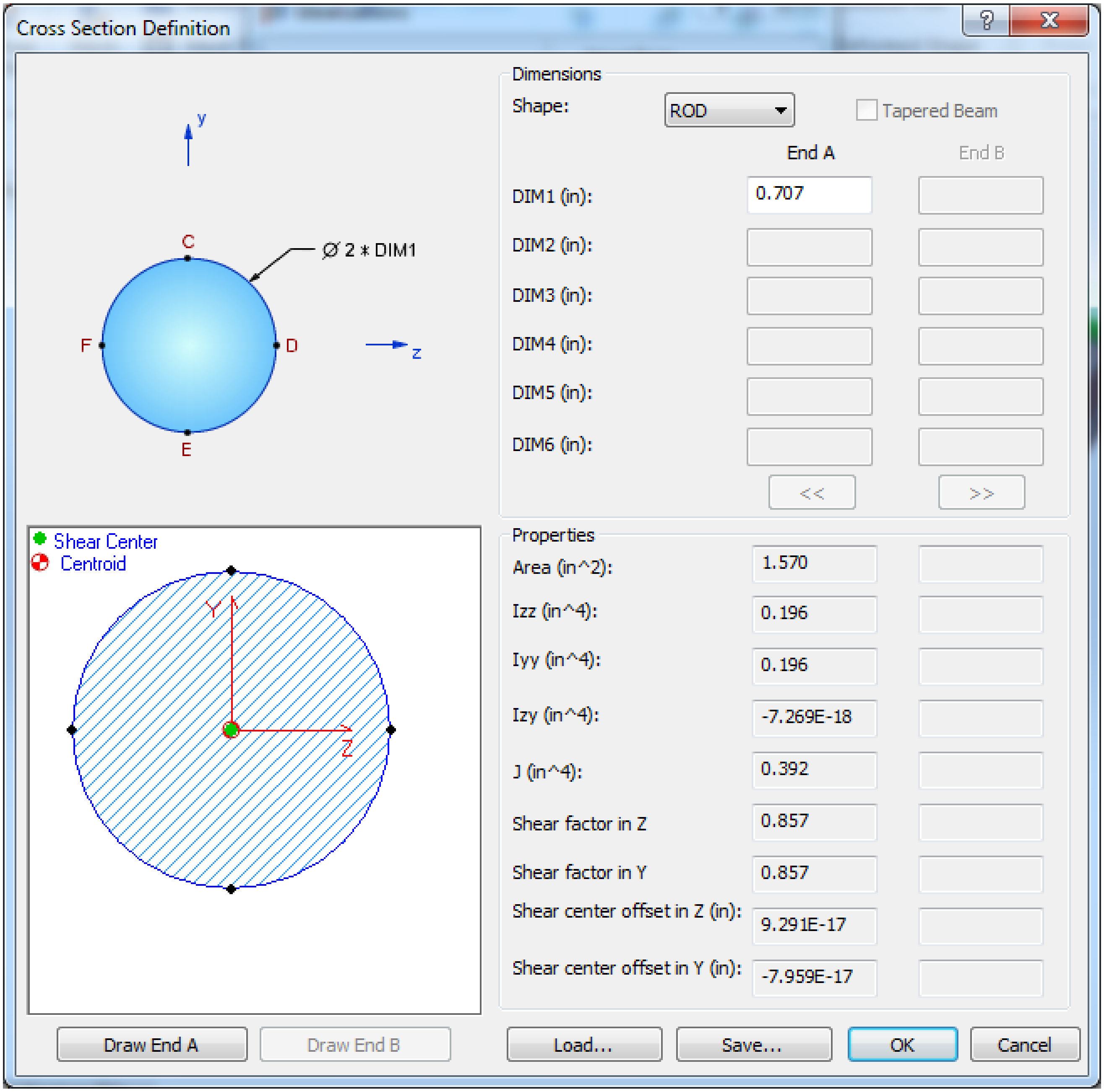


**Figure 14. Rod cross section definition**

Click **OK** twice to define **Rod Physical Property.**

## 5. Mesh the Model

Unhide the parts for meshing, as hidden parts will take default size for meshing.

Right-click on **Mesh Model** and choose **Edit**. Set the **Element Size** to **10 with linear type**, click **OK**. The meshed model should look as shown in Figure 15 below.

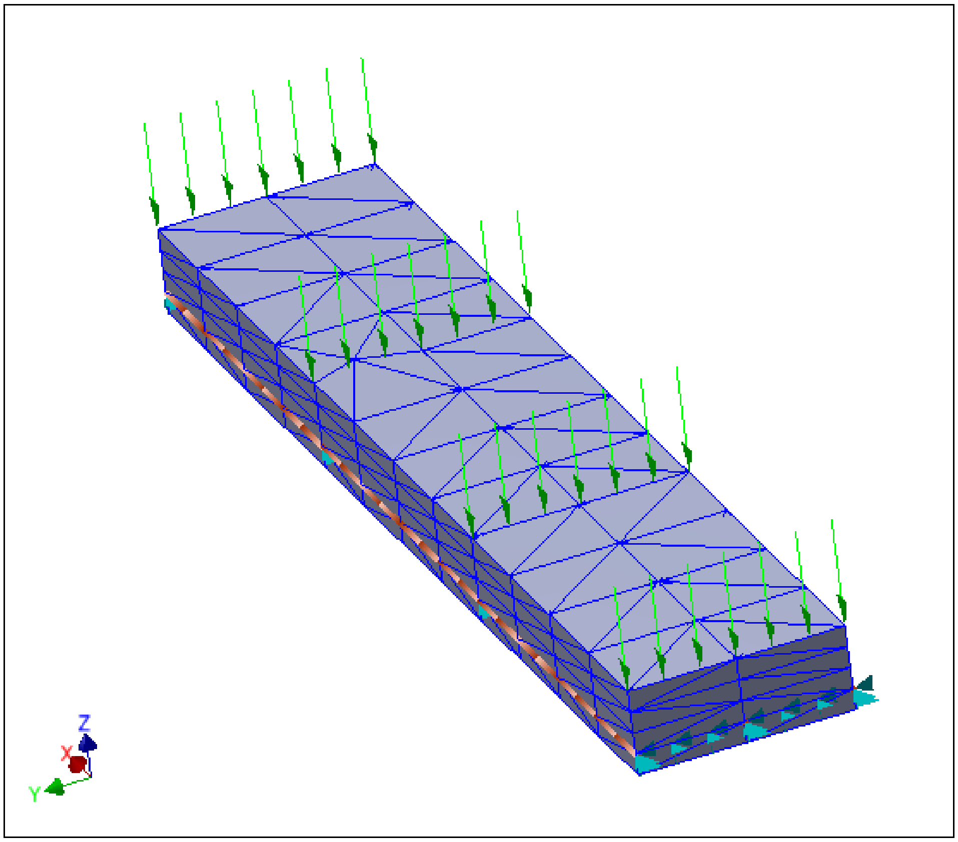


**Figure 15. Auto meshed assembly part**

## 6. Creating Constraints

Next, the constraints need to be defined.

From the **WaveFEA Model Tree** right-click on **Mesh Model** and choose **Hide all Meshes.**

Right-click on **Constraints** and choose **New,** rename the title as **TxTyTz**, Select **Pinned** button, In the **Selected Entities** Select the four short edges where Bar elements were defined, as shown in Figure 16.

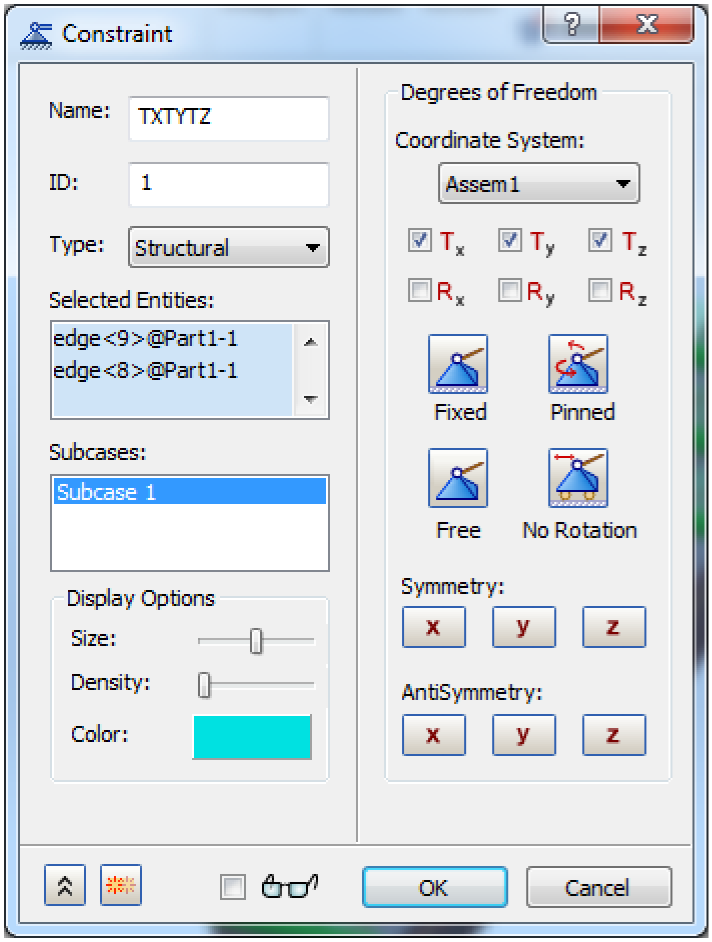


**Figure 16. Fix Translations constraint 1 definition**

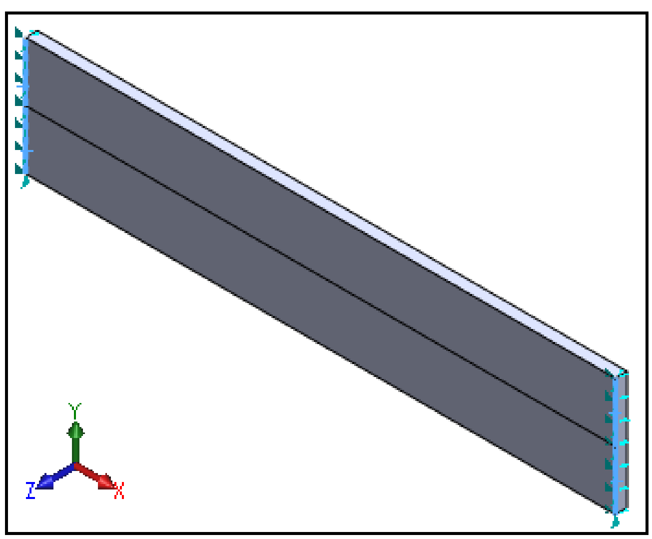


**Figure 17. Edges selected for constraint 1 definition**

Create another constraint by going to **WaveFEA Model Tree** right-click on **Constraints** and choose **New,** rename the title as **Rx**, Check only **Rx**, In the **Selected Entities** select the long short edges where bar elements were defined, . The model should now have constraints as shown in Figure 18

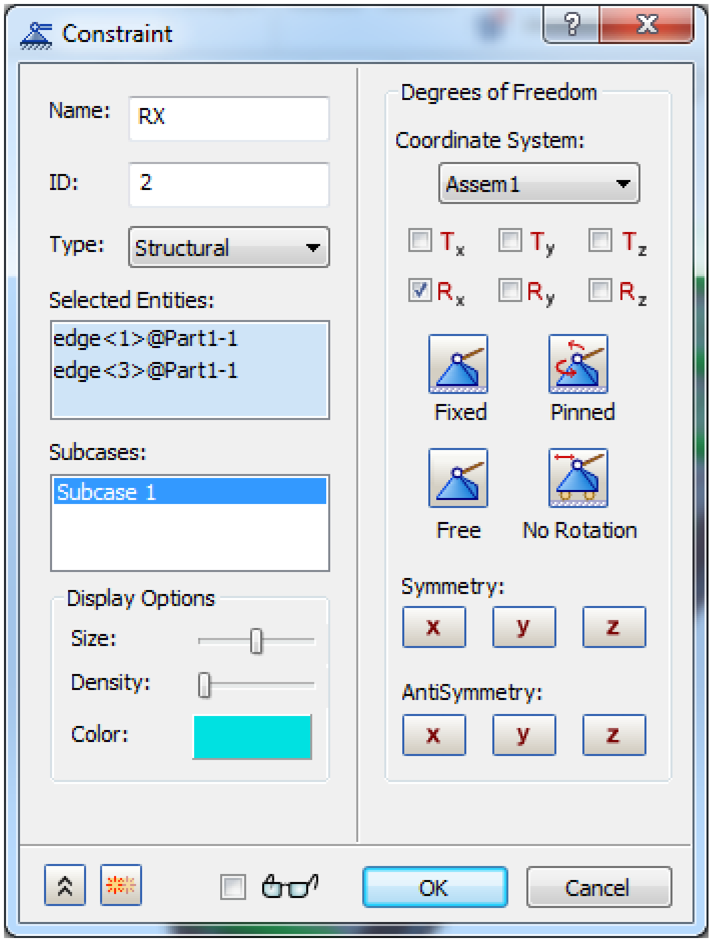


Figure 18. Rx Constraint 2 definition

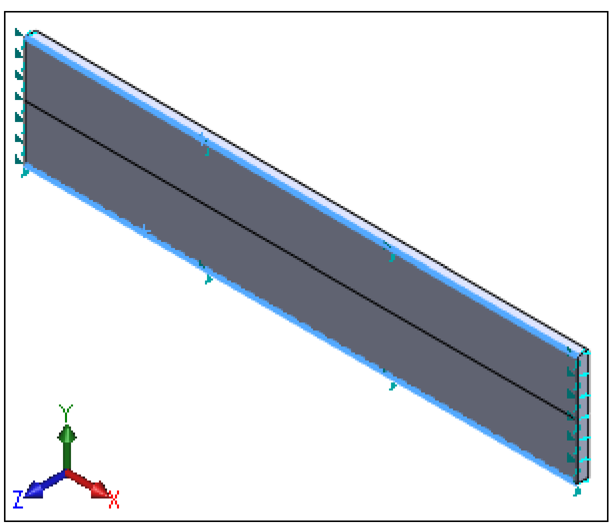


Figure 19. Edges selected for constraint 2 definition

## 7. Creating the Pressure Load

In **WaveFEA Model Tree** right-click on **Assem1** and choose **Display all Bodies**

Right-click on **Loads** and choose **New,** rename the title as **Pressure Load**, select **Pressure** from the **Type** drop down menu, enter **10** in the **Magnitude (psi)**, In the **Selected Entities** select the two top surface as shown in Figure 20.

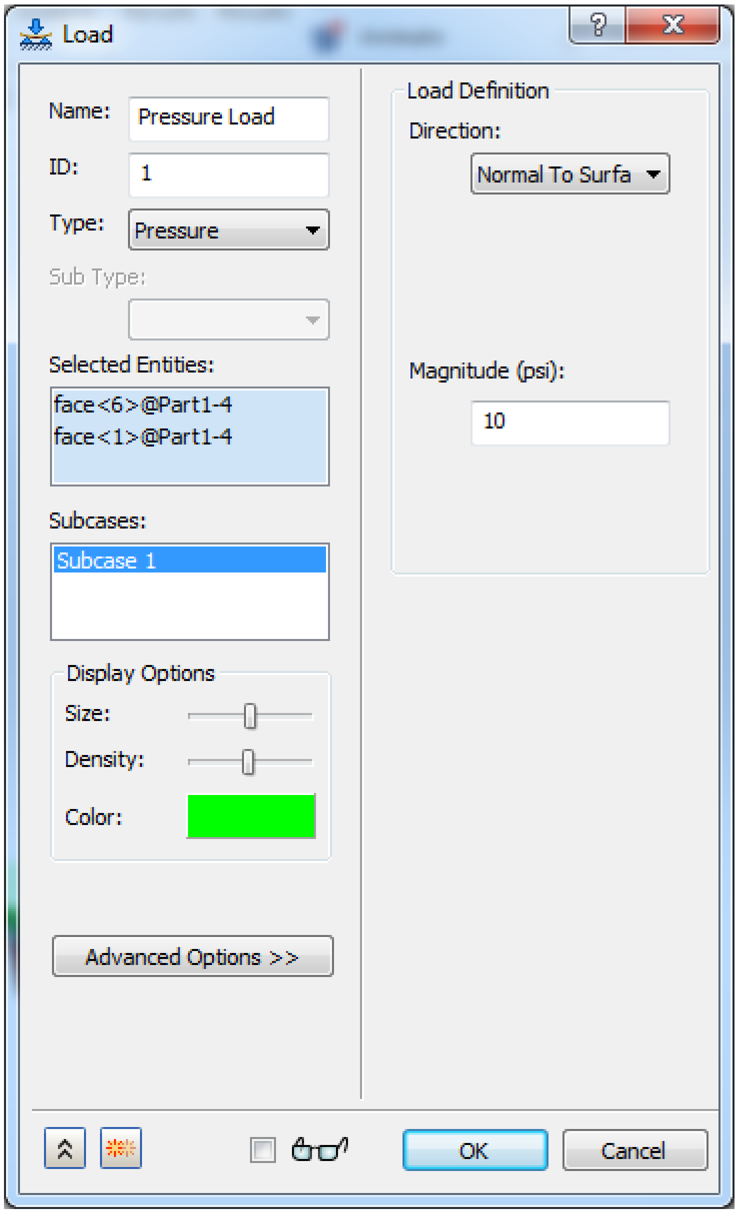


**Figure 20. Pressure load definition**

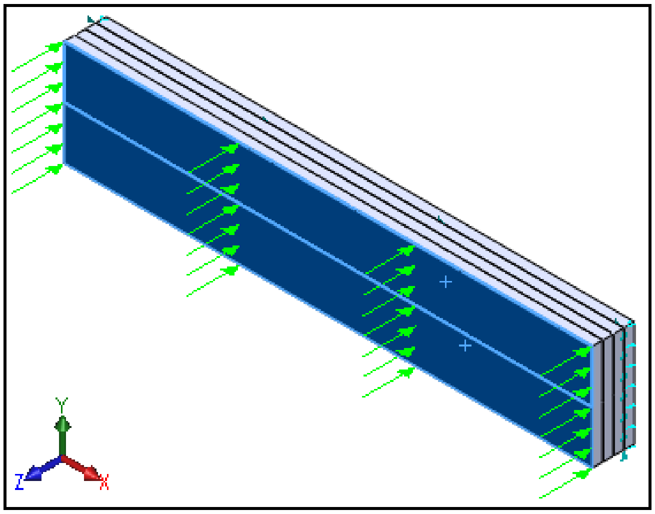


**Figure 21. Face selection for pressure load definition**

Click **OK** to create Pressure Load.

## 8. Set up a Nonlinear Analysis

Next, the **Nonlinear analysis** options need to be setup. Right click on **Analysis 1** and select **Edit,** Select **Nonlinear Static** from the **Type** drop down menu, switch **Off** the **Large Displacements** as shown in Figure 22. Click **OK.**

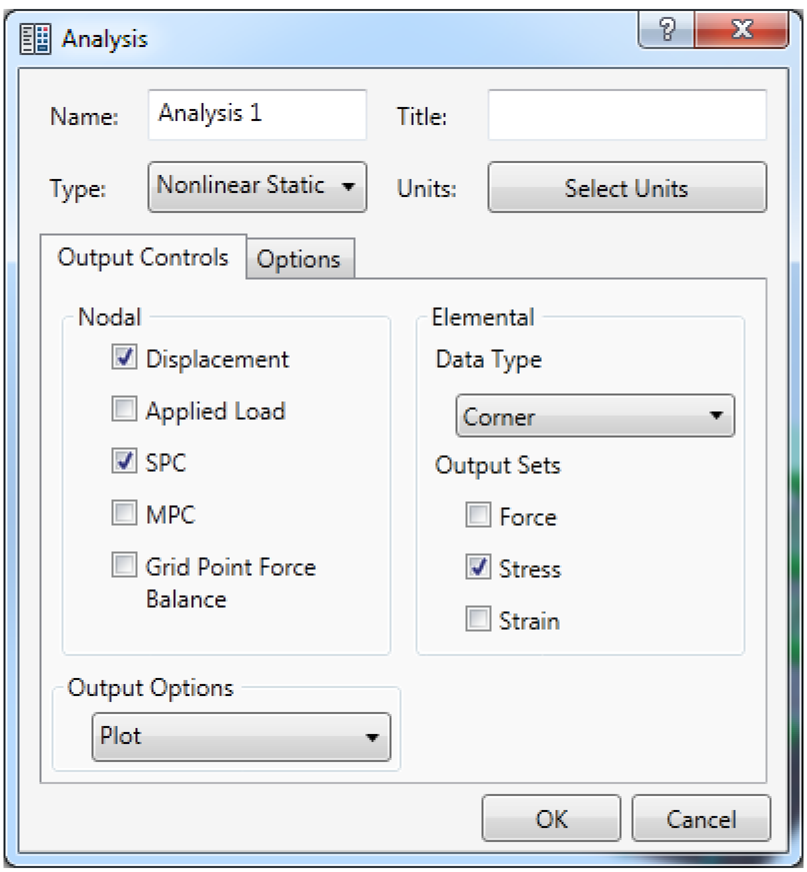


**Figure 22. Analysis dialog**

Now, from the **Subcase 1**, right-click on **Nonlinear setup 1** and select **Edit.** In the **Number of Increments** field, enter **20** Keep **Intermediate Output** as **ON** as shown in Figure 23

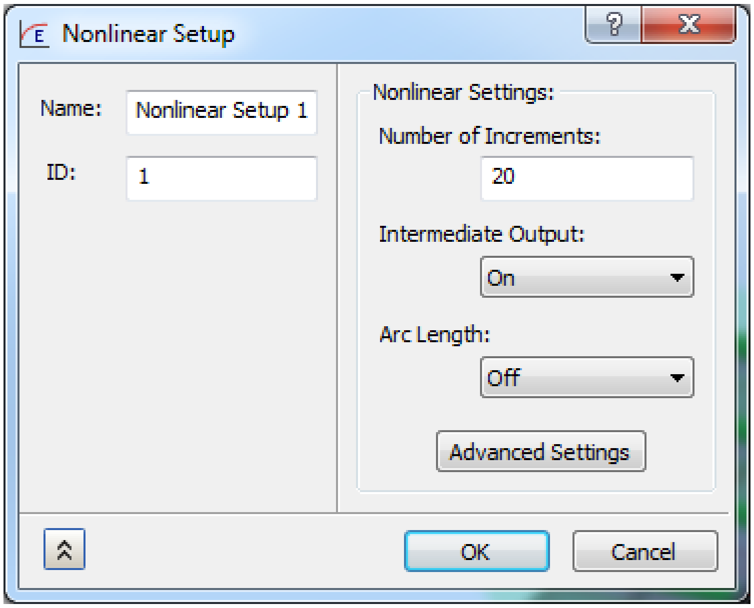


**Figure 23. Load Set Options for Nonlinear Analysis**

## 8. Analyze the model

Save the model.

Right-click on **Analysis1** and select **Solve in Nastran.**

Your model should now and it takes nearly a minute.
When the **Autodesk Nastran** solution completes**,** click **OK** to close that message window.

## 9. Results review

For this case, you will animate your results to view the VonMises stress for the concrete slab, right-click on **Results** and select **Edit,** from the **Subcases** drop box select **'Increment 20, Load=1.0',** select **SOLID VON MISES** under **Contour options,** check **Deformed options** and select **Displacement** under **Deformed options. Click Display**. You should get results plot like the ones shown in Figure 24.

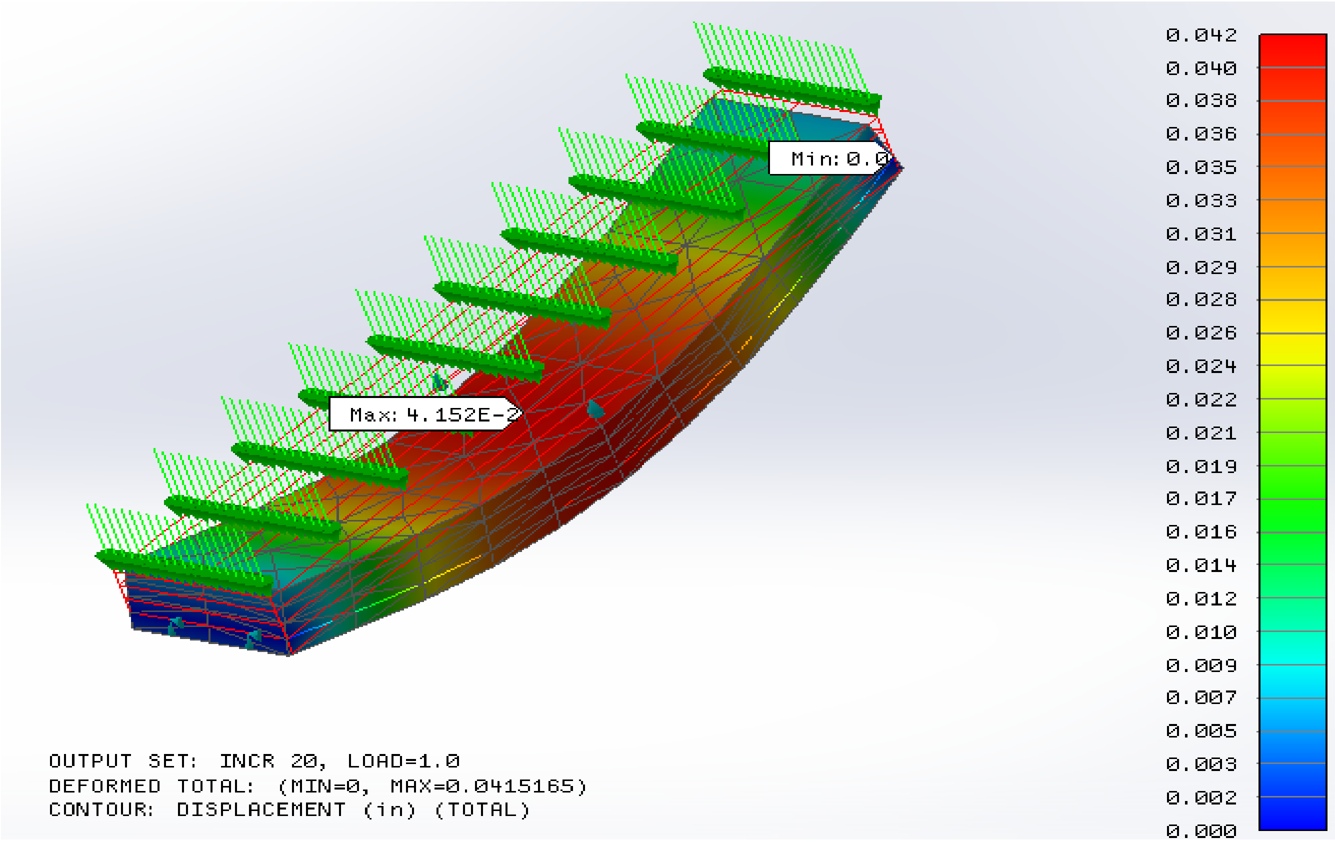


**Figure 24. Displacement and Total Translation results**

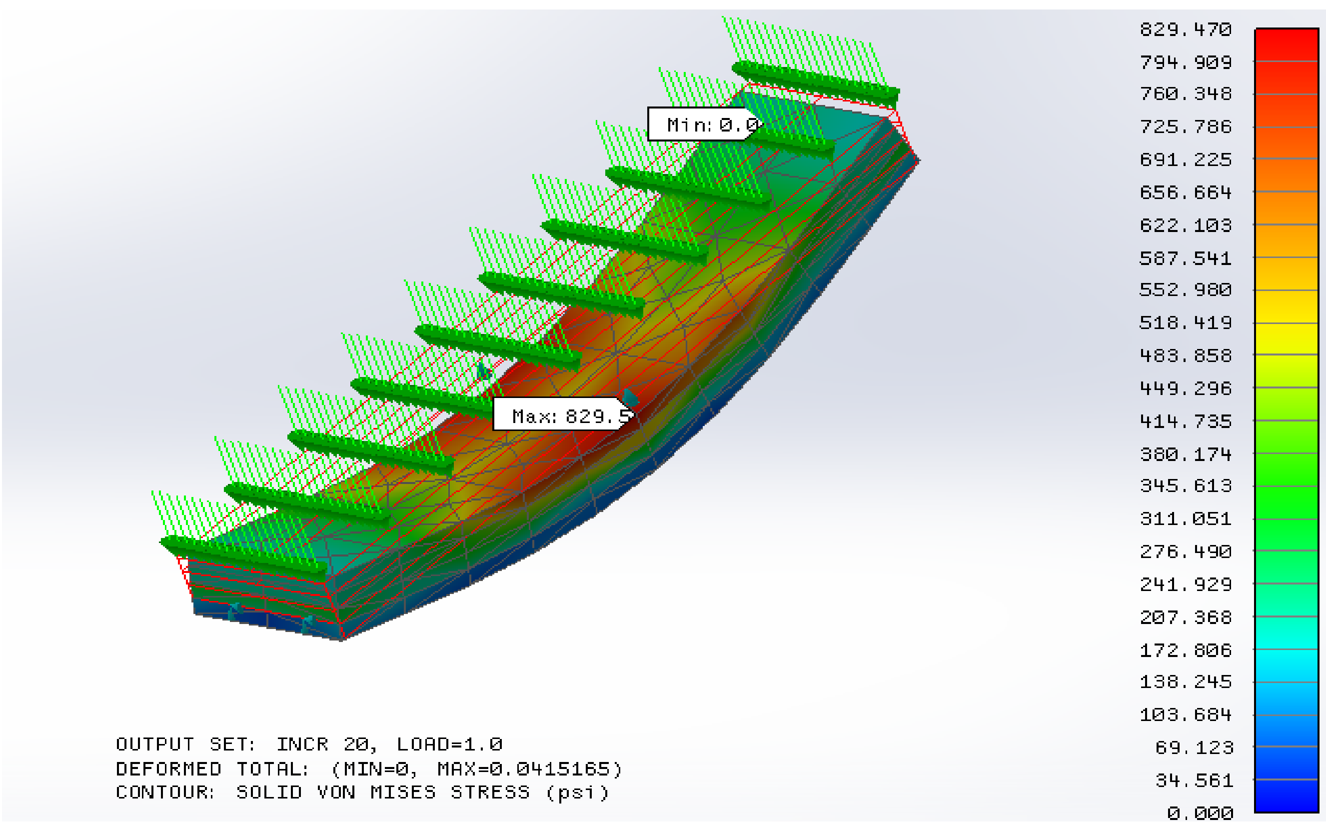


**Figure 25. Solid von Mises Stress and Total Translation results**

## 10. Conclusion

Concrete model setup, concrete has bar elements modeled along the upper portion of the bottom layer of the thickness of the slab. This case shows WaveFEA versatility for handling such a problem which has mixed mesh of beams and solid. It also shows the capability of WaveFEA to use merging nodes as welded in a non-linear analysis to connect dissimilar meshes like beams and solid.