

Concrete Slab in WaveFEA

1. Introduction

The model shown in Figure 1 is a 100"x100"x8" slab of concrete that is fixed around the perimeter. The concrete has 10 bars modeled along the tension side of the slab to represent the 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 that is 100 times greater than the Young's modulus in tension.

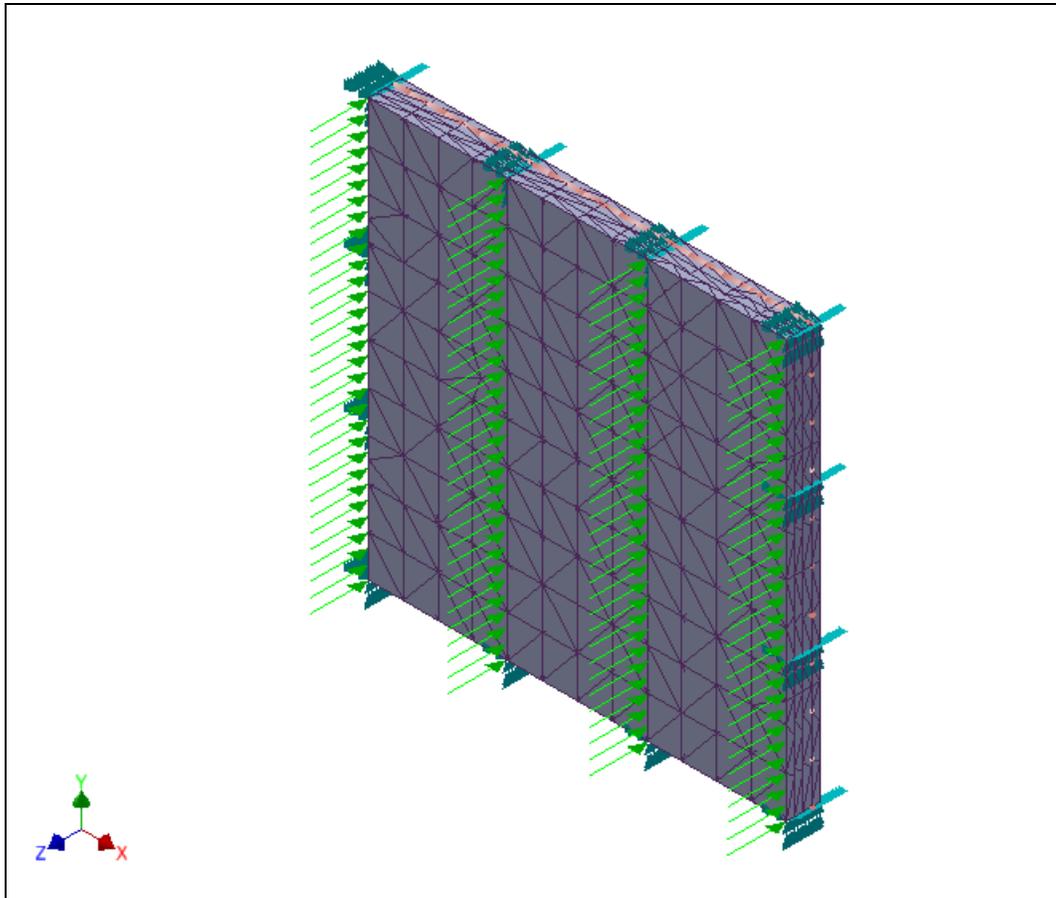


Figure 1. Concrete Slab Model

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

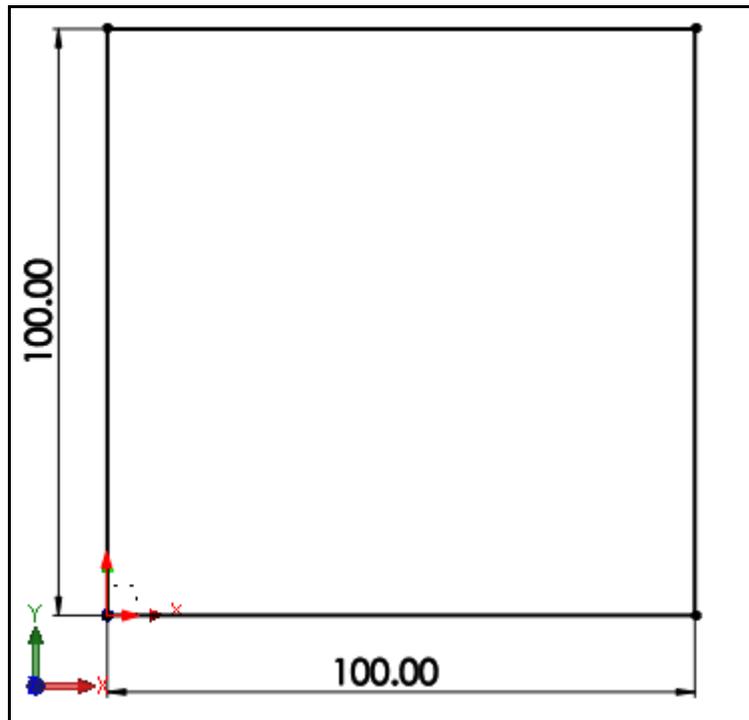


Figure 2. Creating a Rectangle Location

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

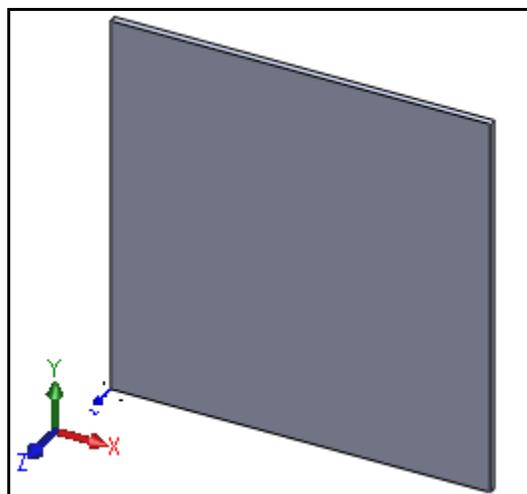


Figure 3. 2_100_100_slab

Create a **Sketch** on one of the surface of the **Extruded** block at 10mm distance, as shown in the below figure 4

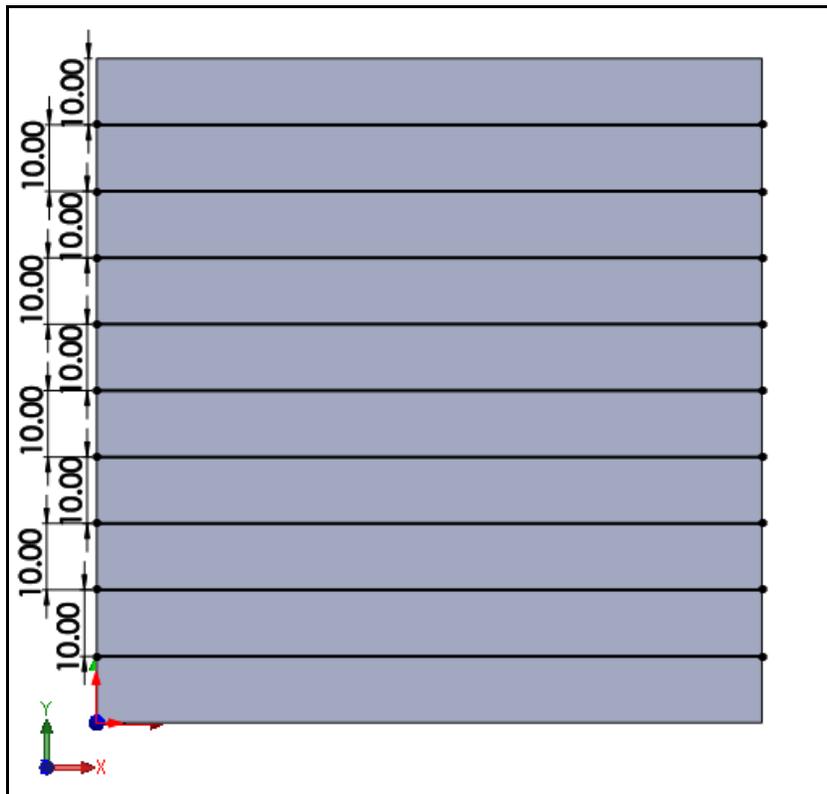


Figure 4. Sketch on the slab

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

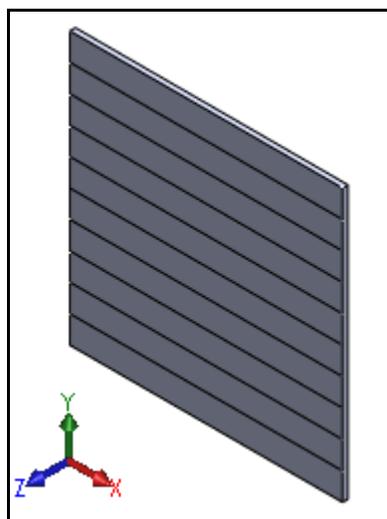


Figure 4. Split slab

Save the model as "**100_100_2_tall_slab (.sldprt)**".

2.2. Creating an assembly model

Open a new assembly document and insert the part created **100_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.

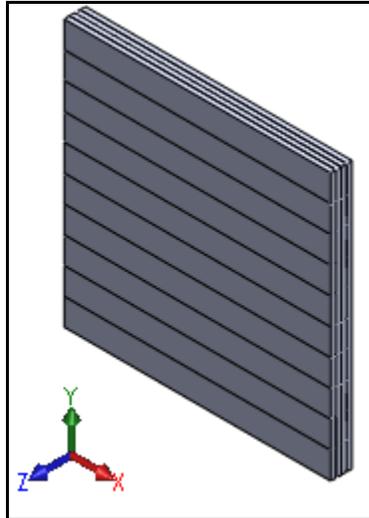


Figure 5. Assembled 8x100x100 Slab

Save the model as "**100_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.

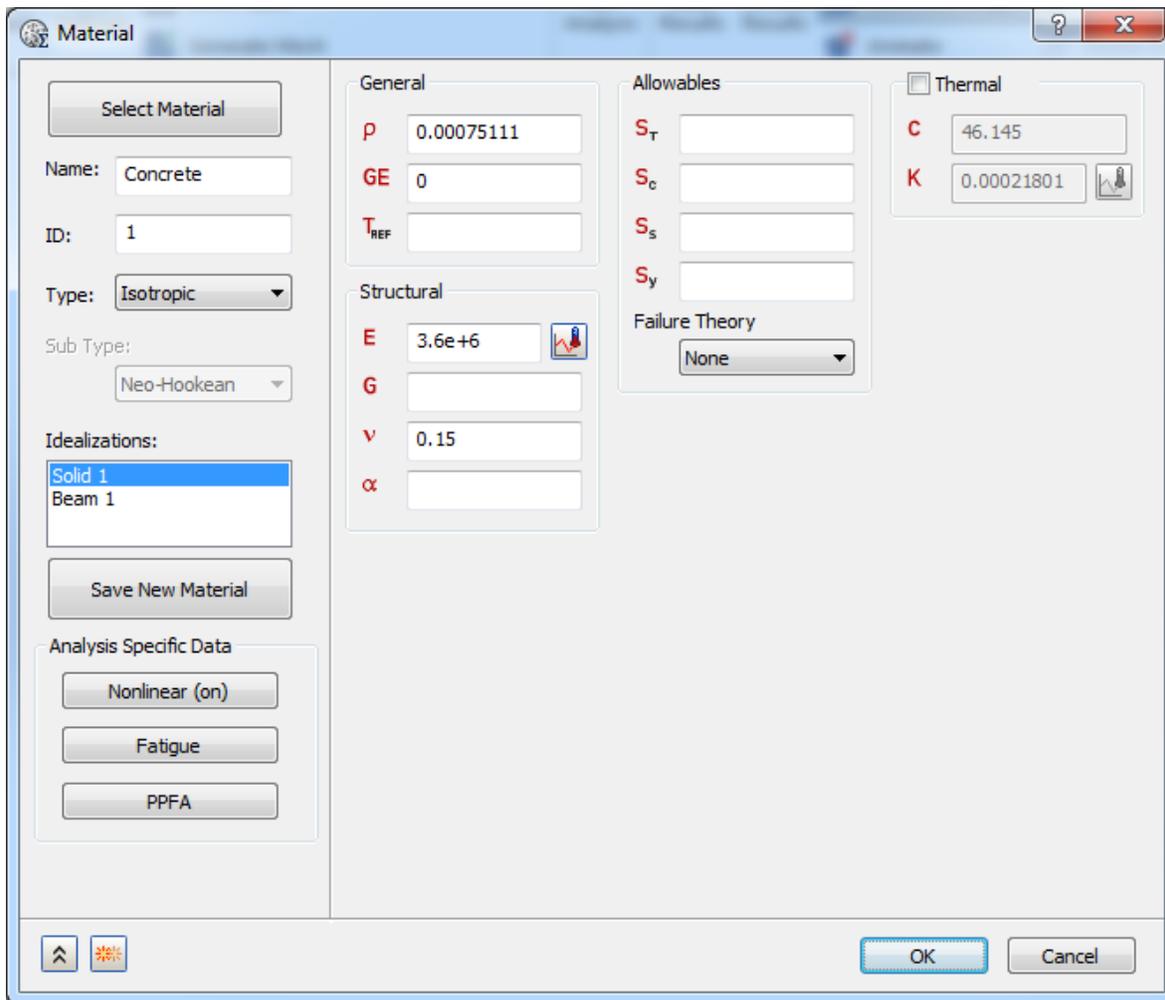


Figure 6. Isotropic Material Definition

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.

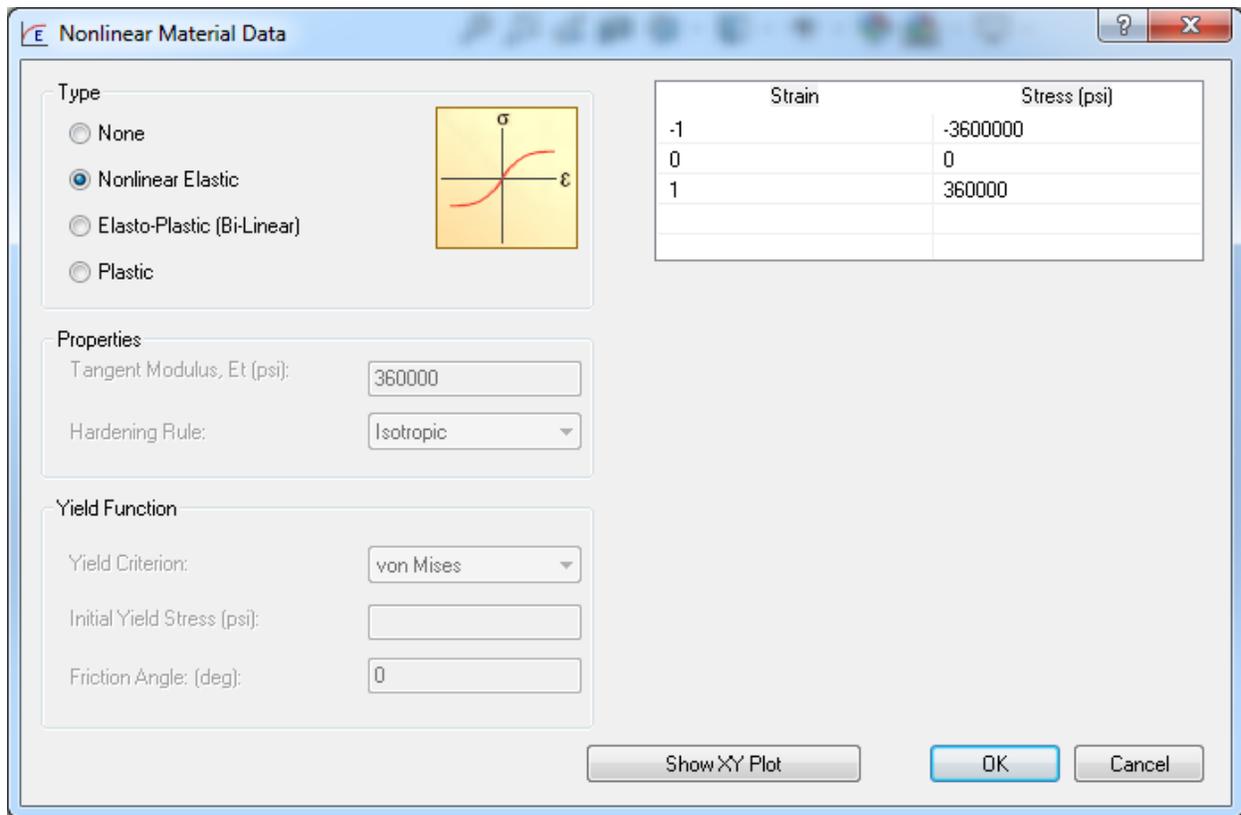


Figure 7. Nonlinear Material Settings for the Concrete

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

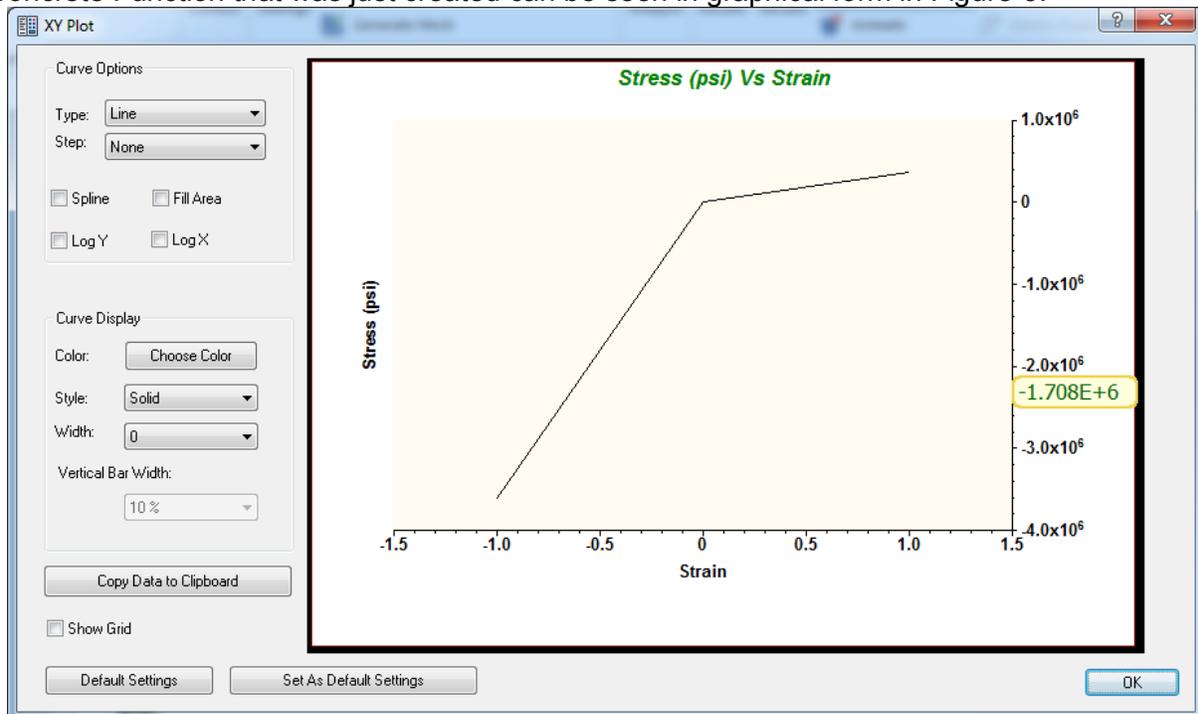


Figure 8. Stress vs. Strain Concrete Function

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.

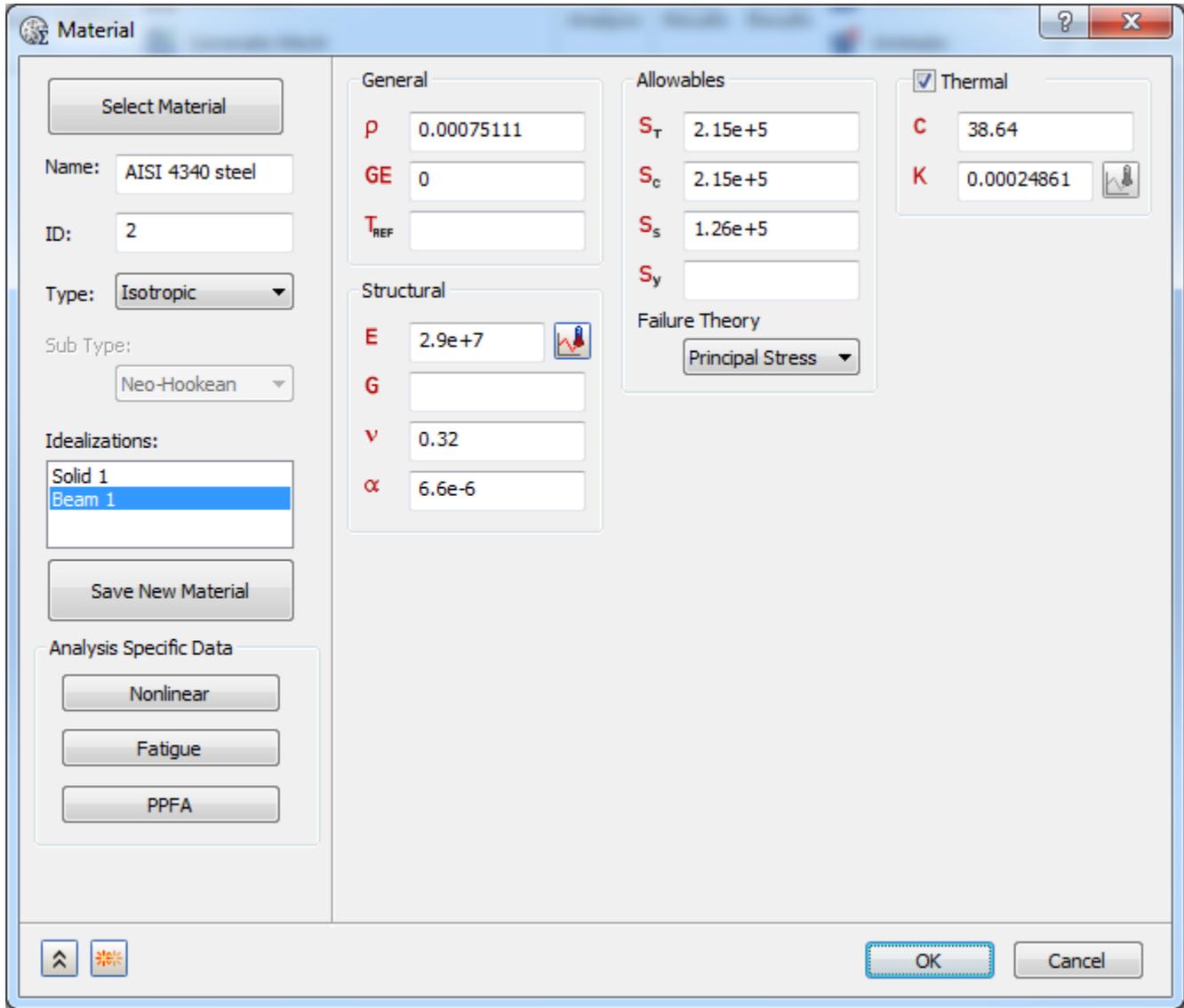


Figure 9. Isotropic Material Definition

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 **Solid** 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, check **Associated Geometry** and select all four imported parts, click **OK** to define **Physical Property** for **Concrete**.

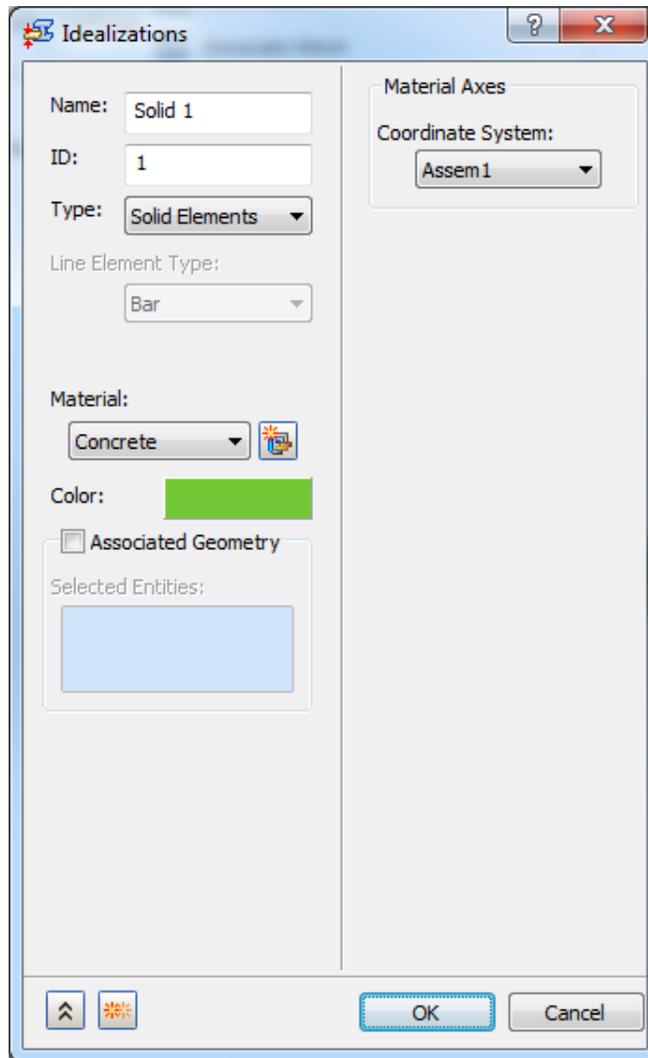


Figure 10. Physical Property Definition

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

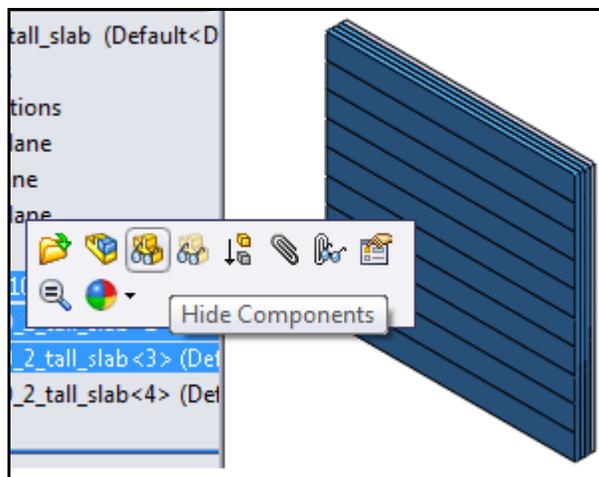


Figure 11. Hide Parts in Model Tree

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 **Property input** radio button and enter the particulars as in the Figure 13. Click **OK**, check **Associated Geometry** select the 11 edges as shown in the figure.

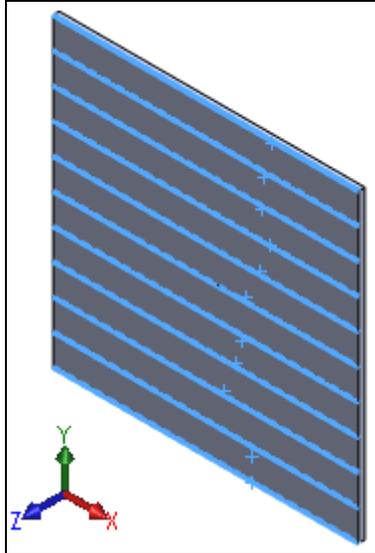


Figure 12. Beam Property Selection

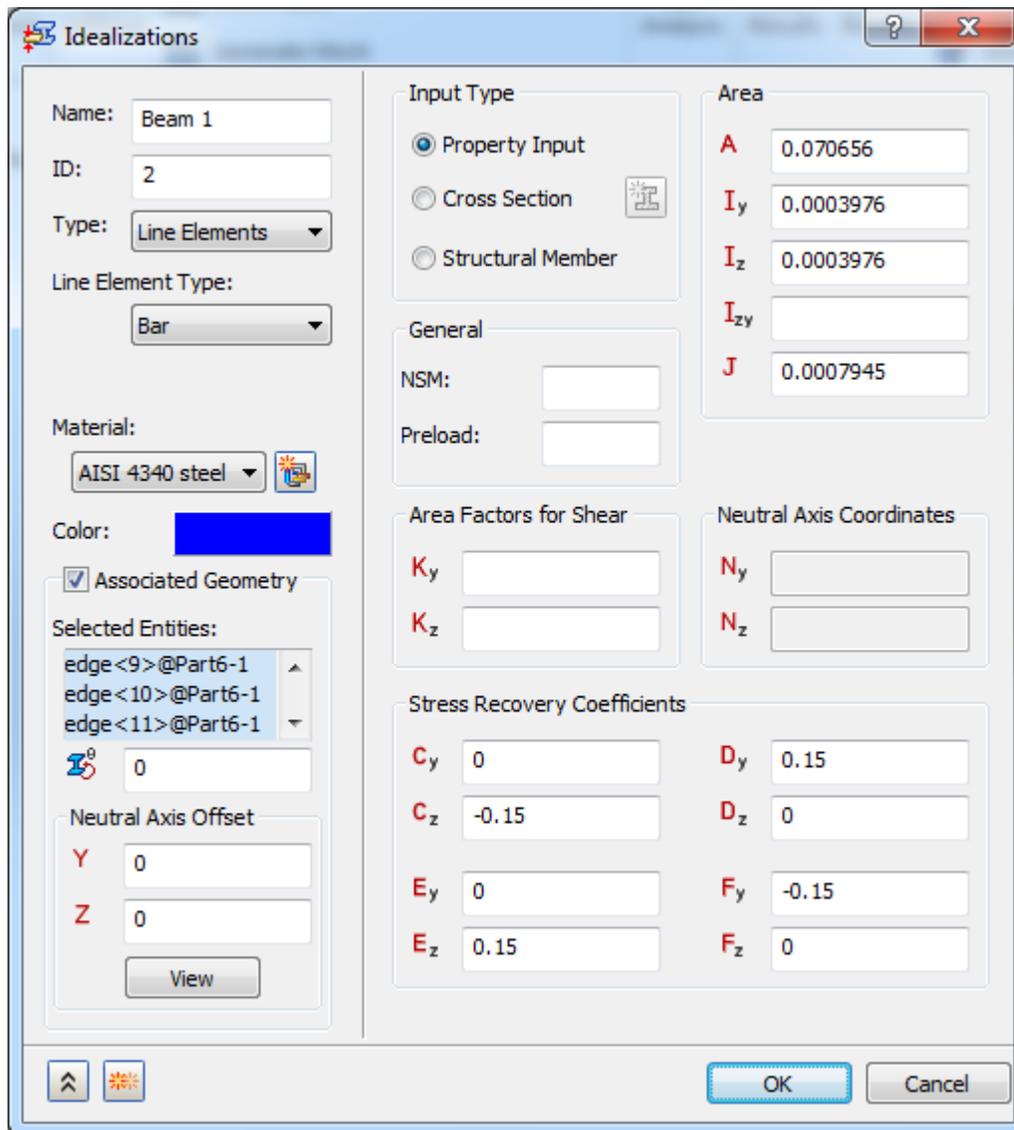


Figure 13. Beam property definition

Click **OK** 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**, click **OK**. The meshed model should look as shown in Figure 15 below.

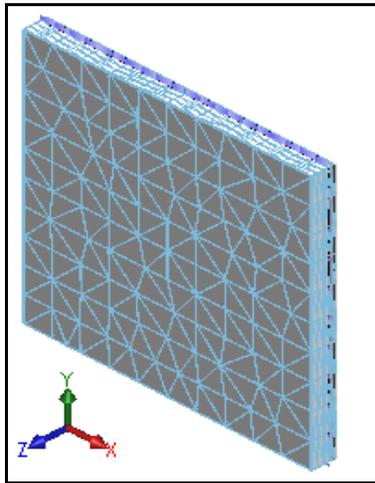


Figure 15. Auto Meshed Assembly

6. Creating Constraints

Next, the constraints need to be defined.

From the **WaveFEA Model Tree** right click on **Assem1** and choose **Display all Meshes**. Right-click on **Mesh Model** and choose **Hide all Meshes**.

Right-click on **Constraints** and choose **New**, rename the title as **Fixed**. In the **Selected Entities** select all the surrounding faces of the slab, as shown in Figure 16 and 17. Click **OK**.

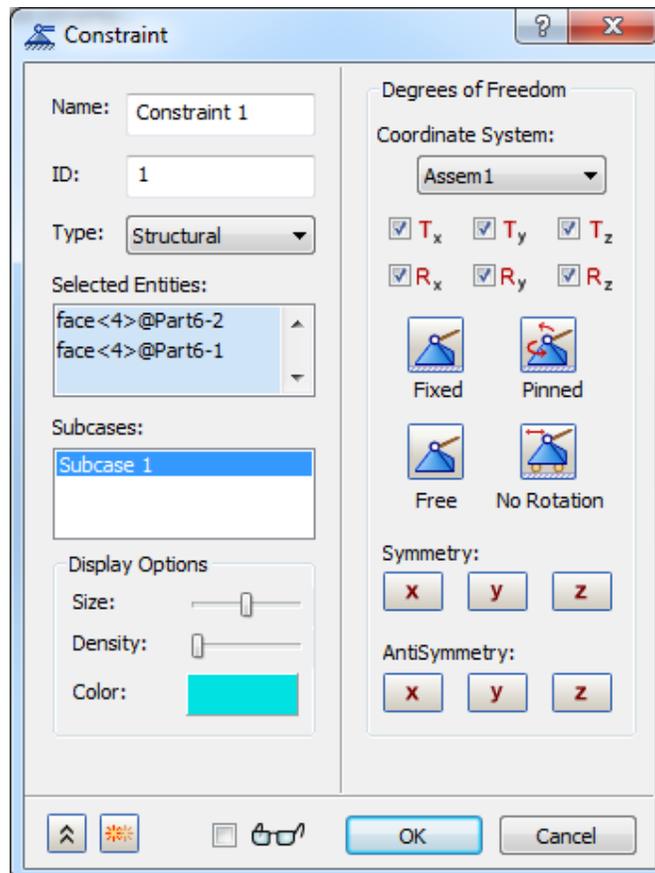


Figure 16. Fix Translations Constraint Definition

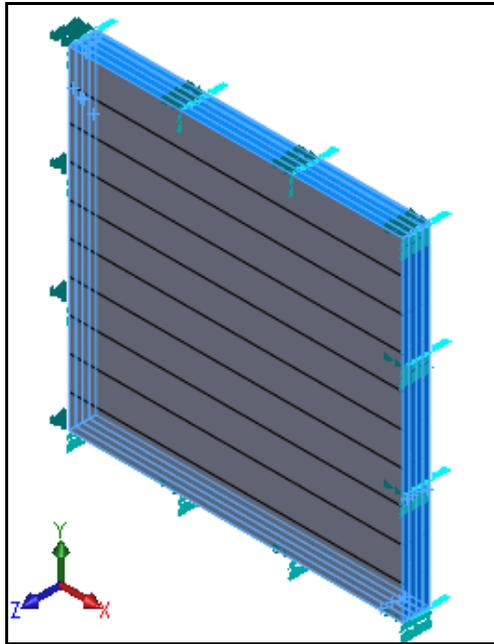


Figure 17. Edges Selected for Constraint Definition

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 top 10 surfaces as shown in Figure 20.

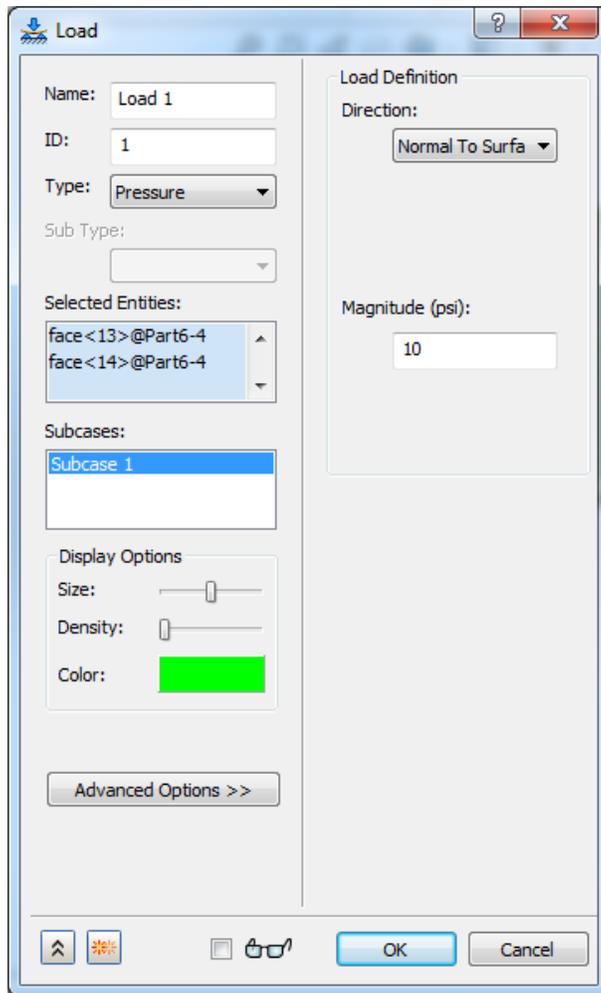


Figure 20. Pressure Load Definition

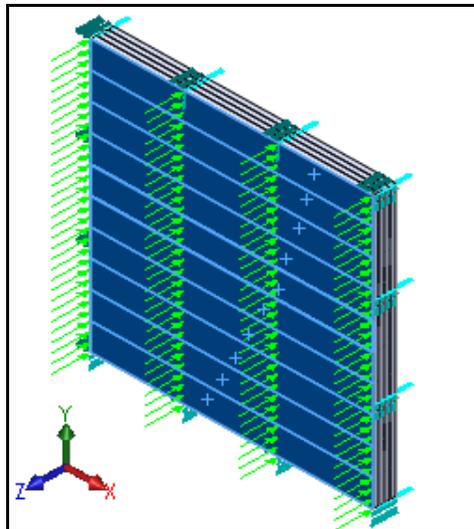


Figure 21. Face Selection for Pressure Load Definition

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 **ON** the **Large Displacements** as shown in Figure 22.

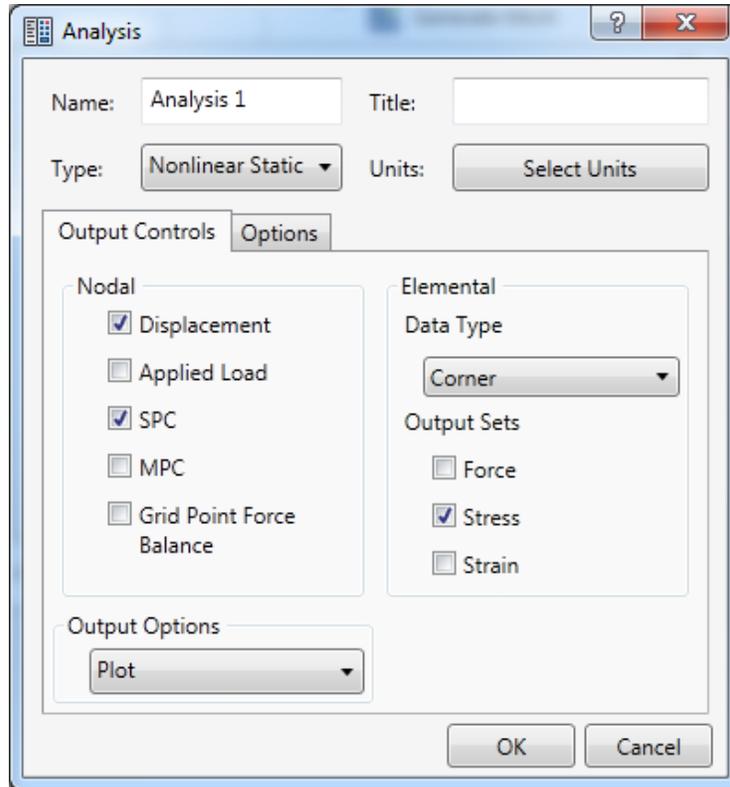


Figure 22. Analysis Dialog

Click **OK**.

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.

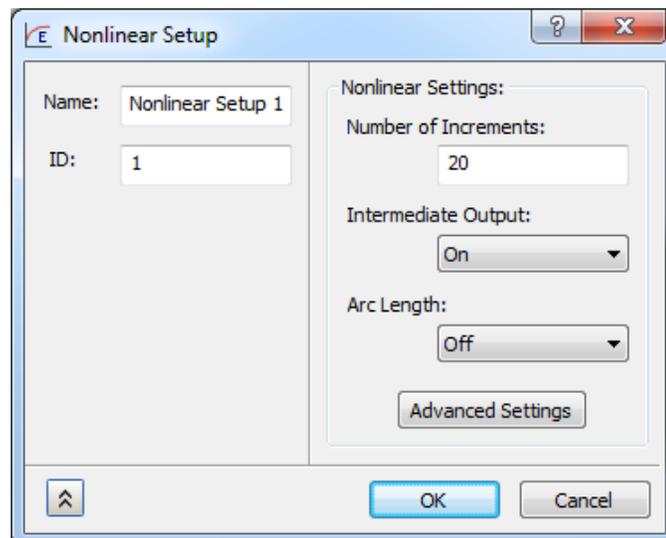


Figure 23. Load Set Options for Nonlinear Analysis

8. Set up Auto surface contact

Right-click on **FE Model** and select **Edit**. In the **Surface Contact** group box select **Auto** radio button, keep **Contact Type** to **Offset Weld**. Click **OK**.

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

The model should look as shown below

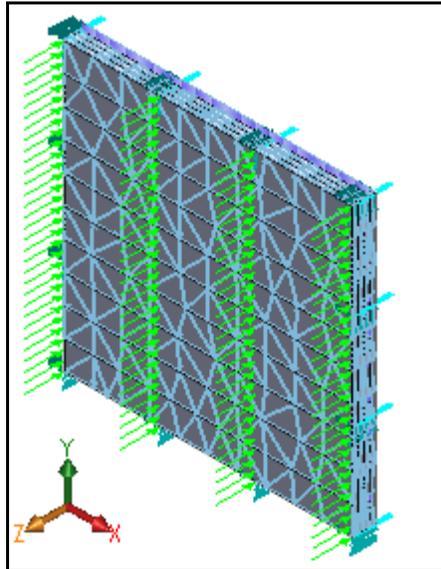


Figure 24. Auto surface contact definition

9. 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.

10. 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 25.

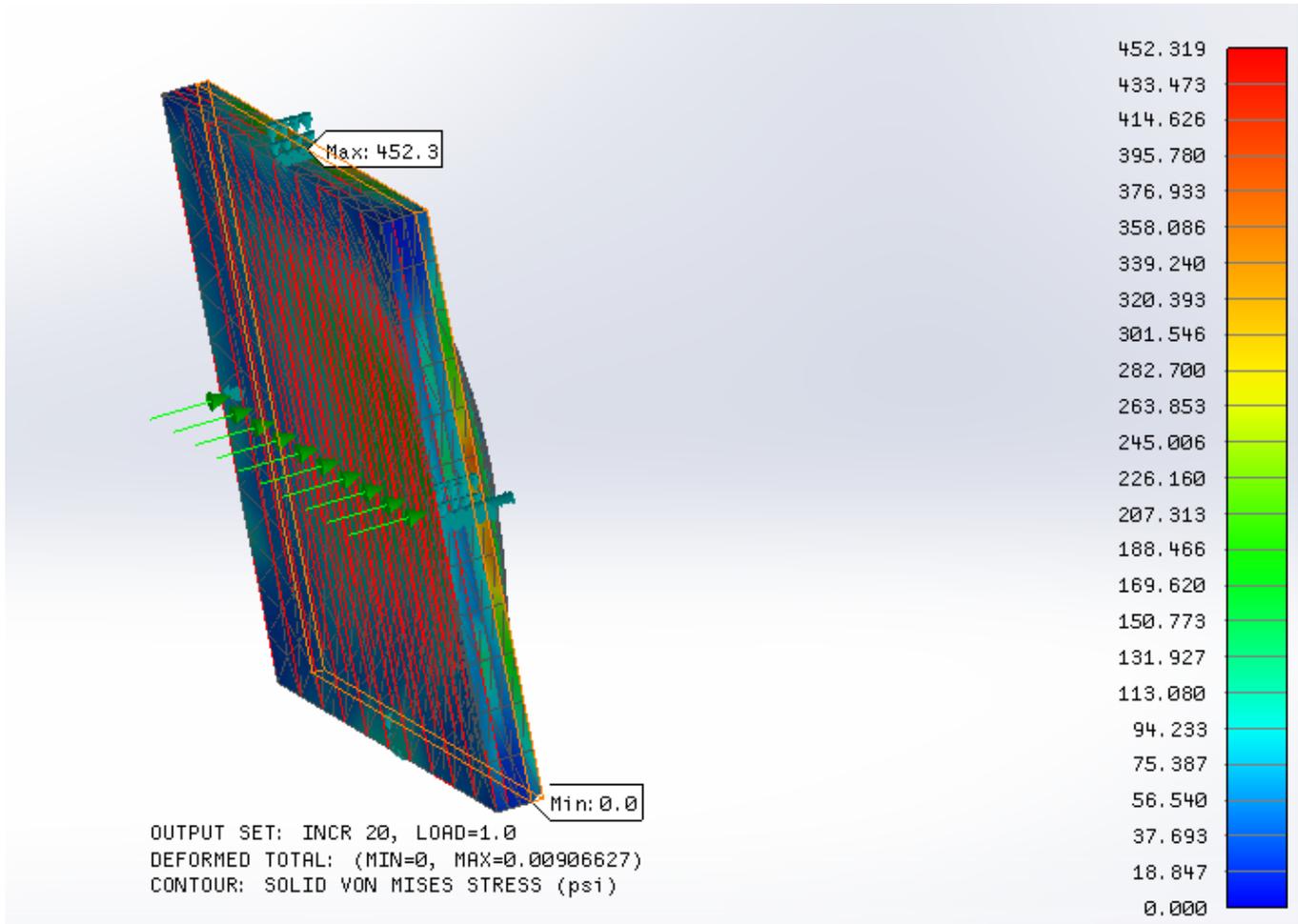


Figure 25. Solid von Mises Stress and Total Translation Results

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 and surface contact as offset-welded in a non-linear analysis to connect dissimilar meshes like beams and solid, as well as solid-solid.