Large Strain Analysis of Hyperelastic Material with Coefficients in WaveFEA

1. Introduction

Below is a finite element representation of a hyperelastic material under load. A NEi Nastran nonlinear analysis with load increments will be performed to obtain the deformation of this material under load.

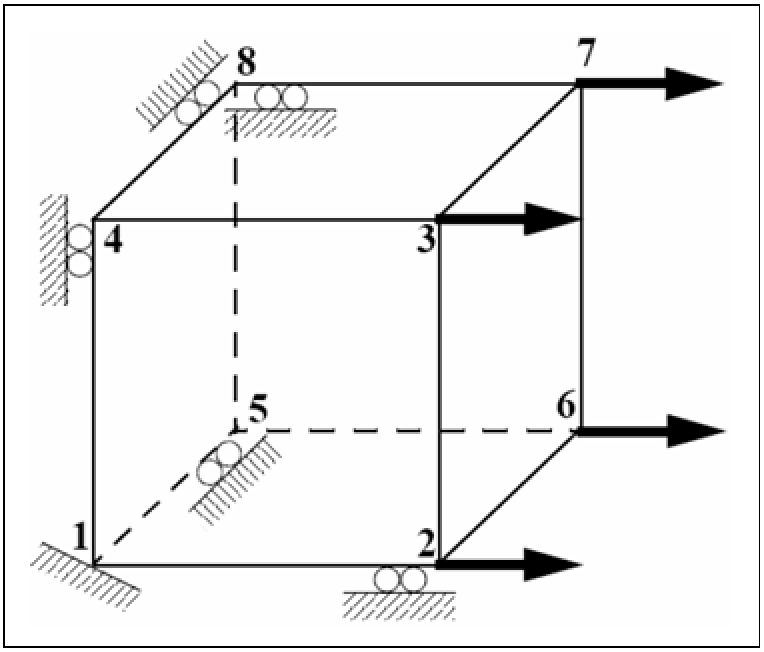


Figure 1. Representation of a Hyperelastic Material under Load

Hyperelastic Properties:

• Distortional Deformation Constants, A10: 90.0

• Distortional Deformation Constants, A01: 0.0

• Bulk Modulus, D1: 11000.0

The hyperelastic material properties are based on the Neo-Hookean model and represent a fully incompressible material.

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 1 in. by 1 in. square. Go to **sketch** select **Front plane** to create a **Square** shown in Figure 2.

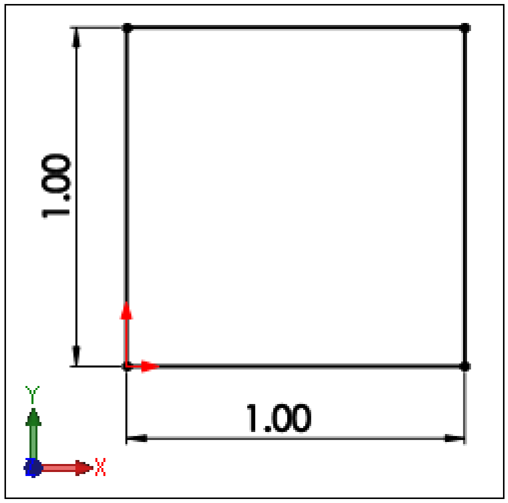


Figure 2. Creating a Square Location

Finish the sketch.

Select **Extruded Boss/Base** command from **Features,** Select the created **Sketch** and **Extrude** up to **1 inch.**

Select **OK** to finish.

The extruded model should look as shown in Figure 3

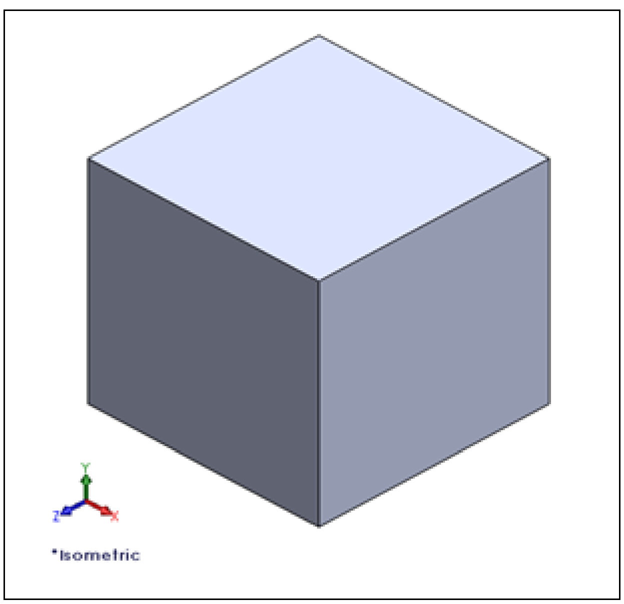


Figure 3. 1_1_1_slab

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

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

3. Pre-Process the Model

You will first define the hyperelastic material, then define the property of the element, create a Physical property for a solid tetra element, apply the constraints and loads, define the nonlinear parameters, and finally run a NEi Nastran Nonlinear analysis.

3.1 Define the hyperelastic 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 **Neo-Hookean,** click on **Type** drop down and select as **'Hyperelastic'**, and change the **Sub-Type** to **'Neo-Hookean'.**
Enter the following values into their respective field as shown in Figure 4.

Your **Material Properties** should look as follows

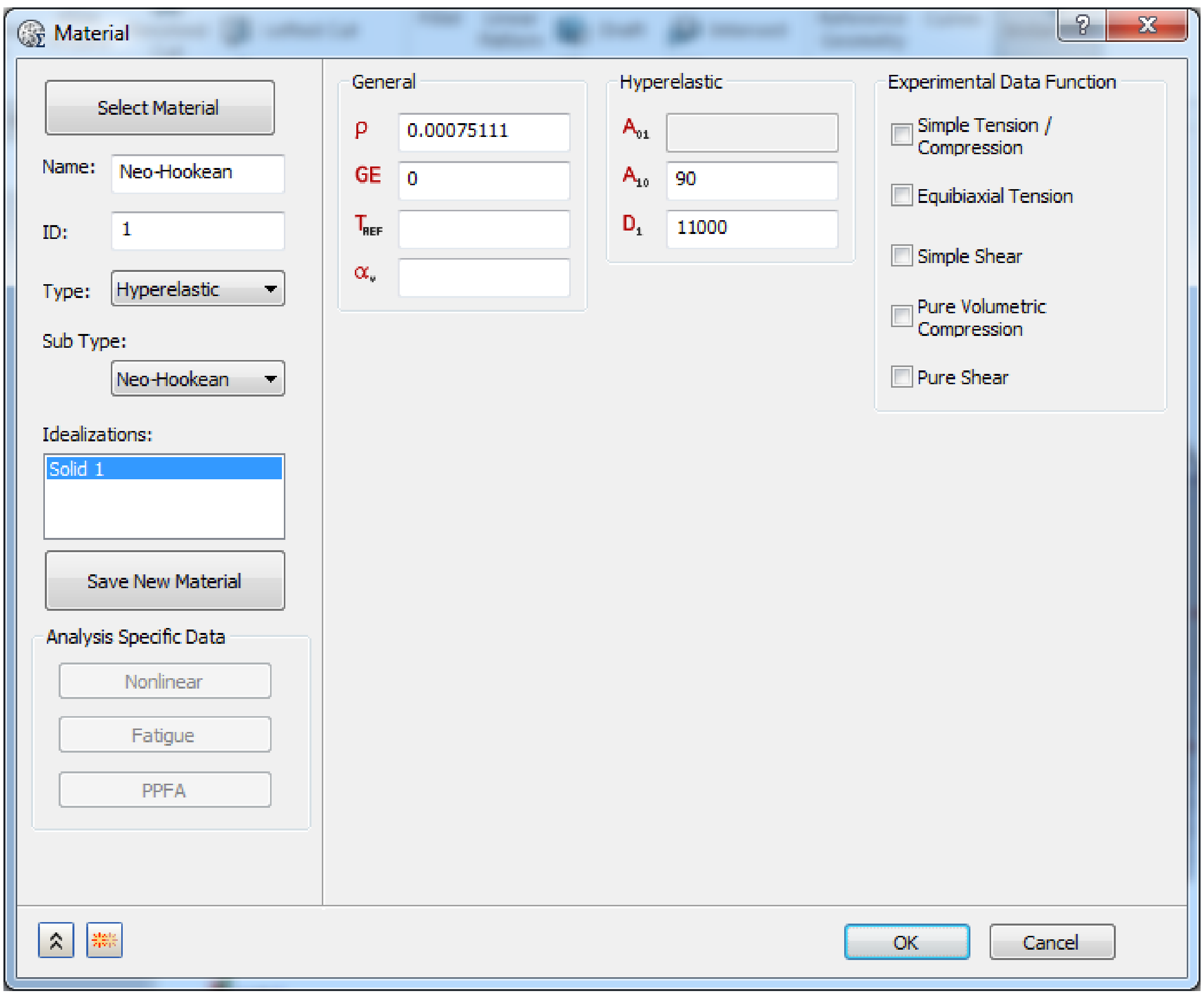


Figure 4. Hyperelastic Material Definition

Click **OK.**

3.2. 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 **NEi Nastran Model Tree** right click on **Physical Properties** and choose **New,** rename the title the material as **Solid Element**, select **Solid Elements** from the **Type** drop down menu, select **Neo-Hookean** from the **Material** drop down menu, click **OK** to define **Physical Property** for **Hyperelastic.**

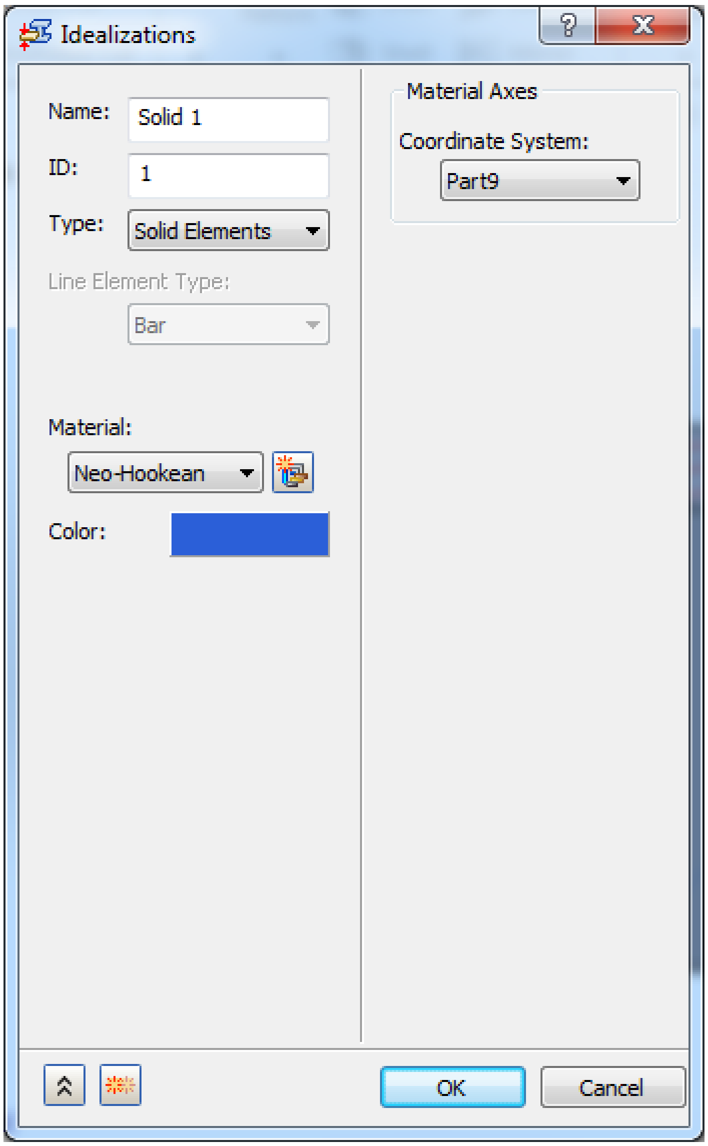


Figure 5. Physical Property Definition

3.3. Creating Constraints

Next, the constraints need to be defined.

From the **NEi Nastran Model r**ight-click on **Constraints** and choose **New,** rename the title as **Fixed**. In the **Selected Entities** select **Vertex1**, as shown in Figure 6 and 7. Click **OK**.

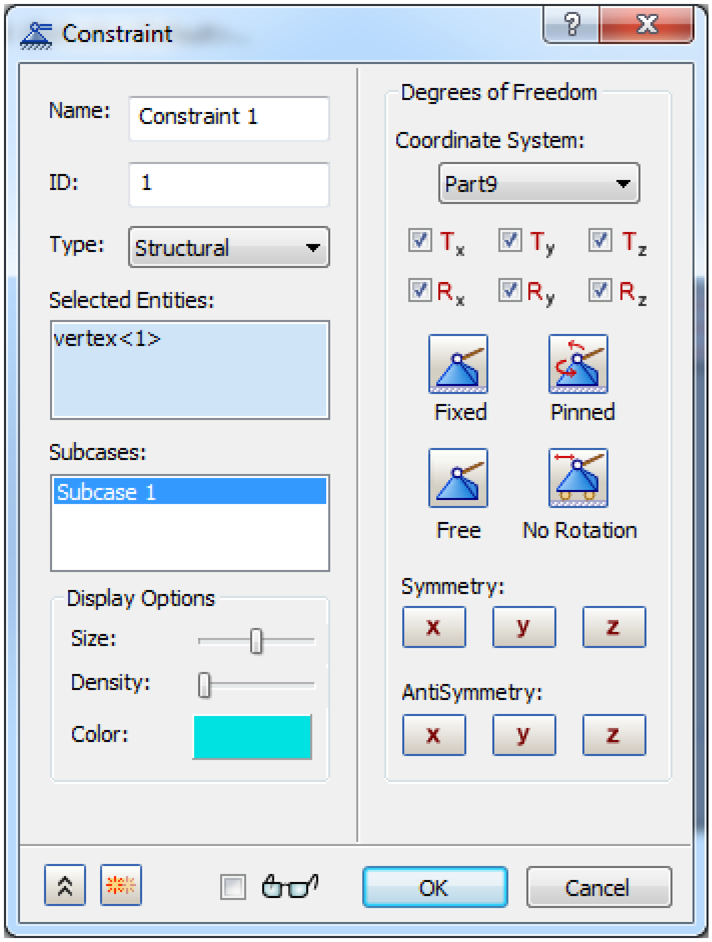


Figure 6. Fixed Constraint Definition

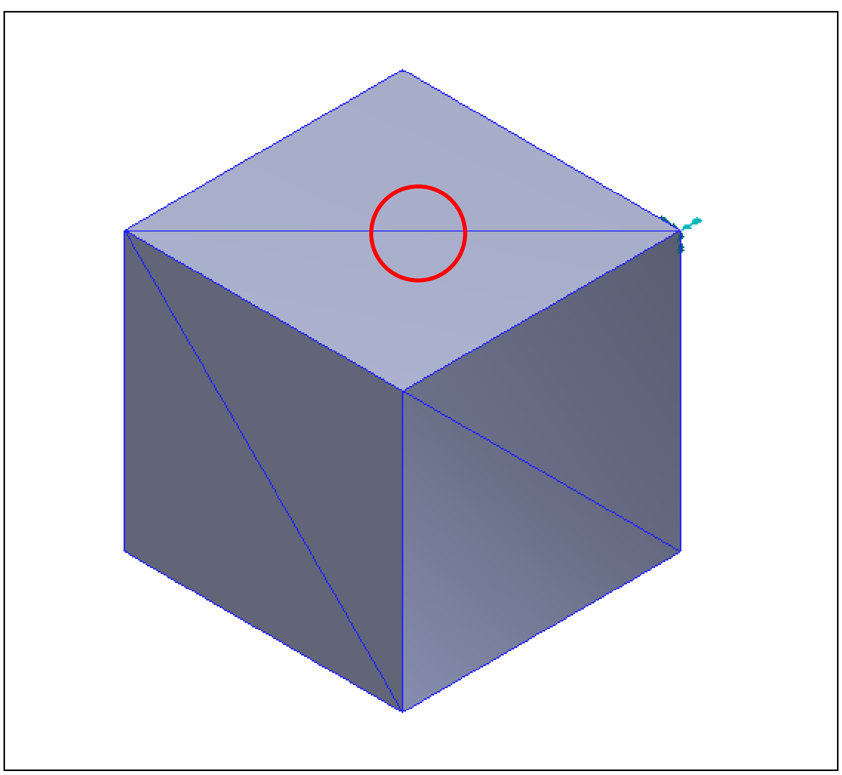


Figure 7. Vertex Selected for Constraint Definition

3.4. Define connector rigid bodies

Now you will create rigid elements to keep all the faces of the solid hex element parallel.
Go to **Model** right click on **Connectors** select **New,** Change the **Name** to **Tx**, From the **Type** dropdown select **Rigid Body**, Make sure in the **Rigid Body** the **Type** dropdown should be **Rigid.**

Select the **Independent Vertex/Point** as Vertex 6 (Constrained vertex), and Select the **Dependent Entities** as shown in the below Figure 8 and 9.

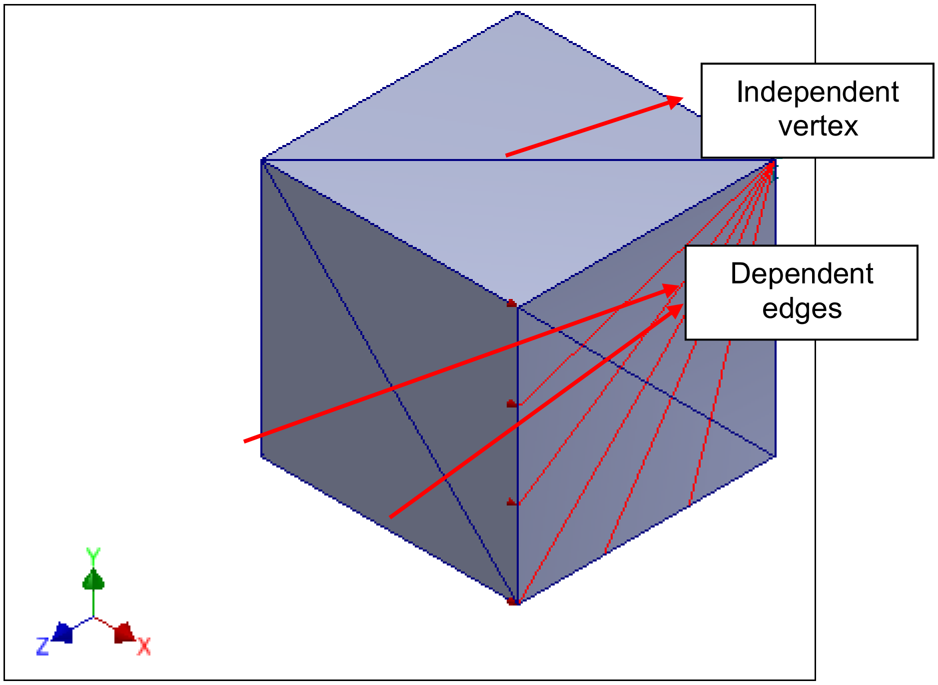


Figure 8. Rigid Element Applied on the Part for Tx

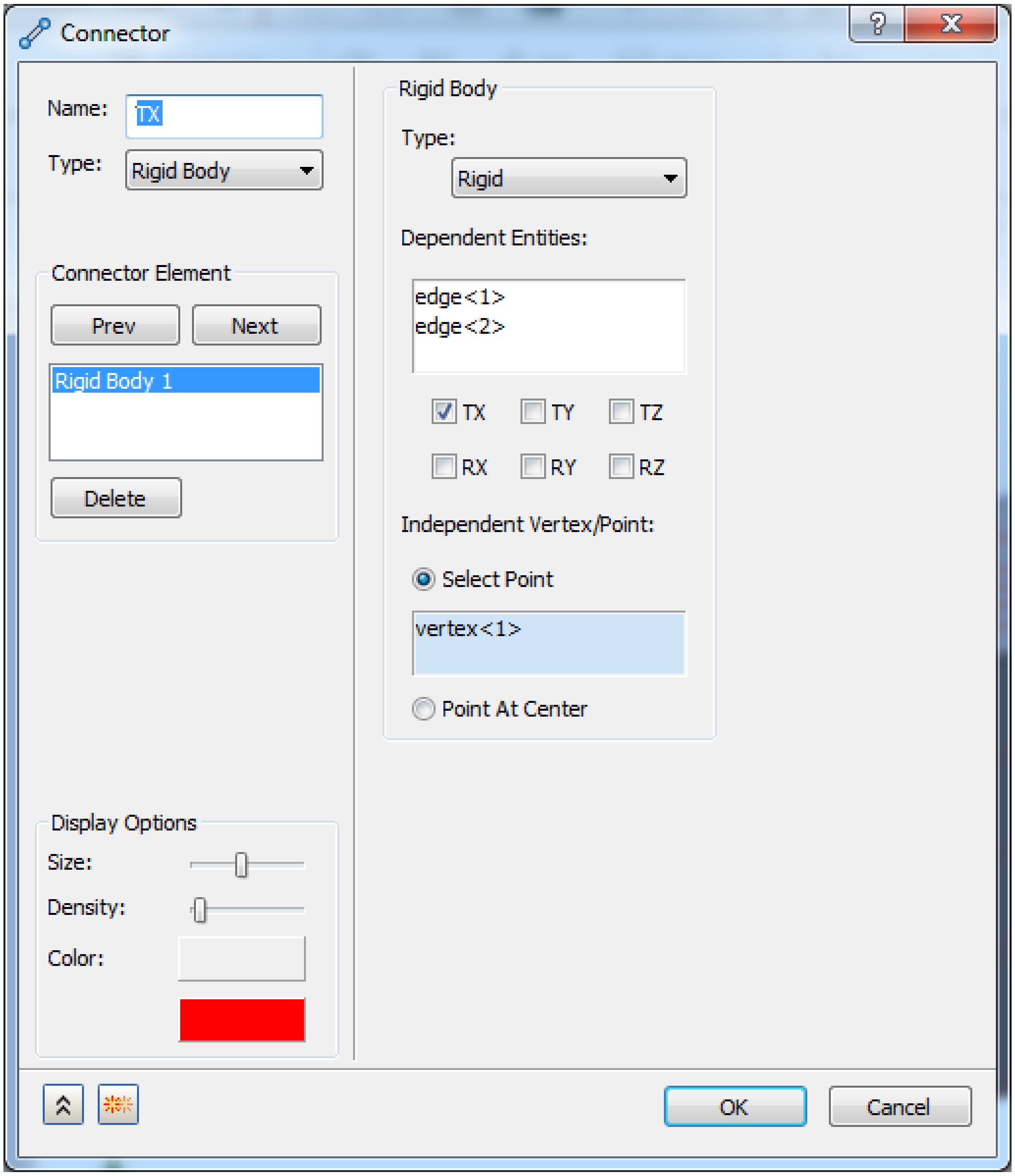


Figure 9. Rigid Element for Tx

Click on **Duplicate** button and clear all selections.

For the second rigid element, change the **Name** to **Ty**, make sure the **Rigid Body Type** dropdown should be **Rigid** and change the **Color** to **Blue.**

Keep the same **Independent Vertex/Point** as Vertex 6 (Constrained vertex), and select the **Dependent Entities** as shown in the below Figure 10 and 11.

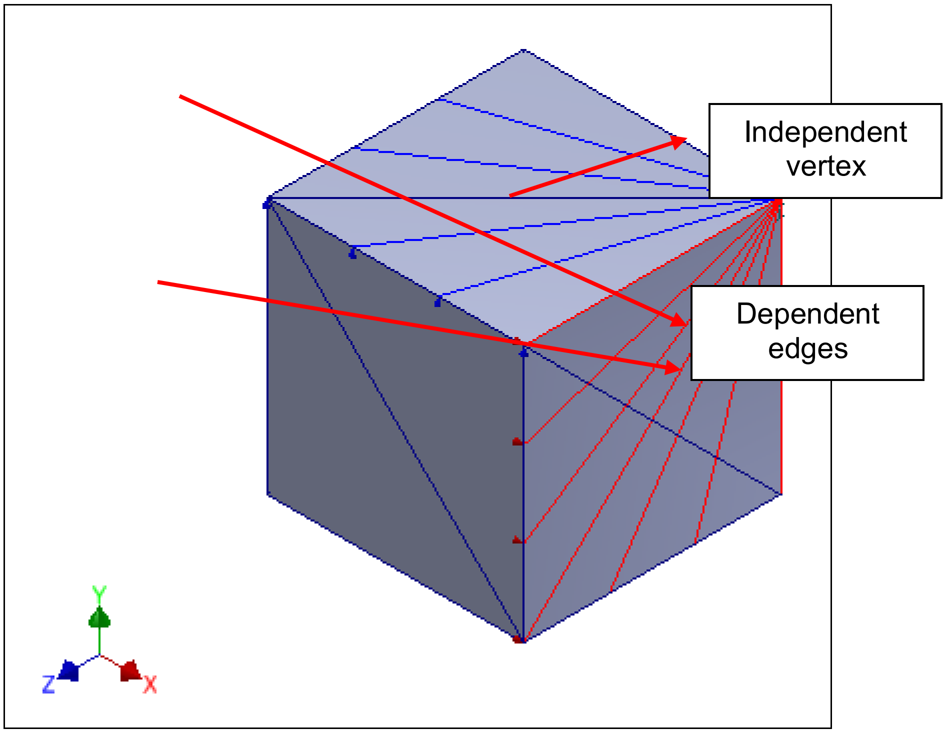


Figure 10. Rigid Element applied on the part for Ty

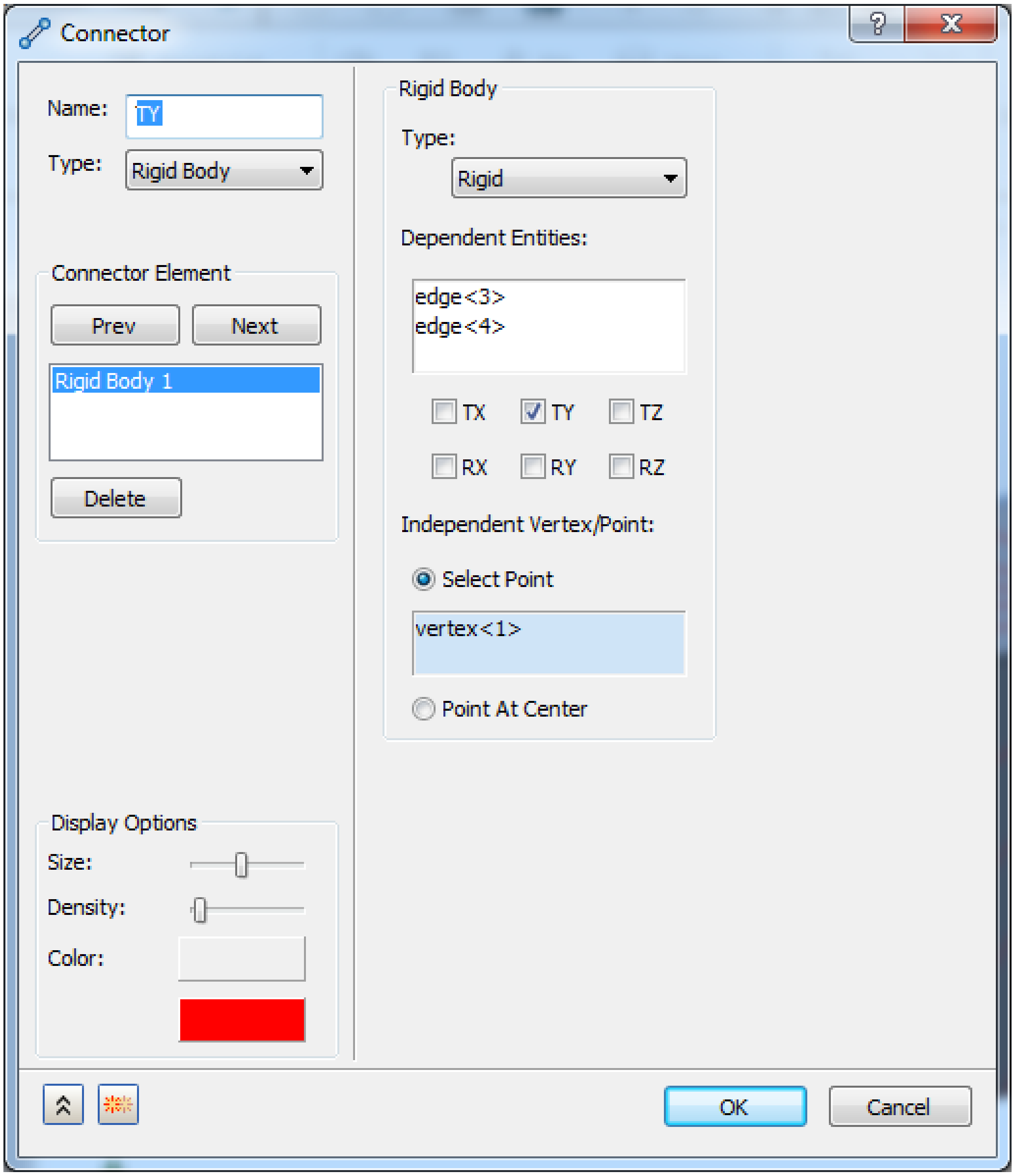


Figure 11. Rigid Element for Ty

Click on **Duplicate** button and clear all selections.

For the second rigid element, change the **Name** to **Tz**, make sure the **Rigid Body Type** dropdown should be **Rigid** and change the **Color** to **Green.**

Keep the same **Independent Vertex/Point** as Vertex 6 (Constrained vertex), and select the **Dependent Entities** as shown in the below Figure 12 and 13.

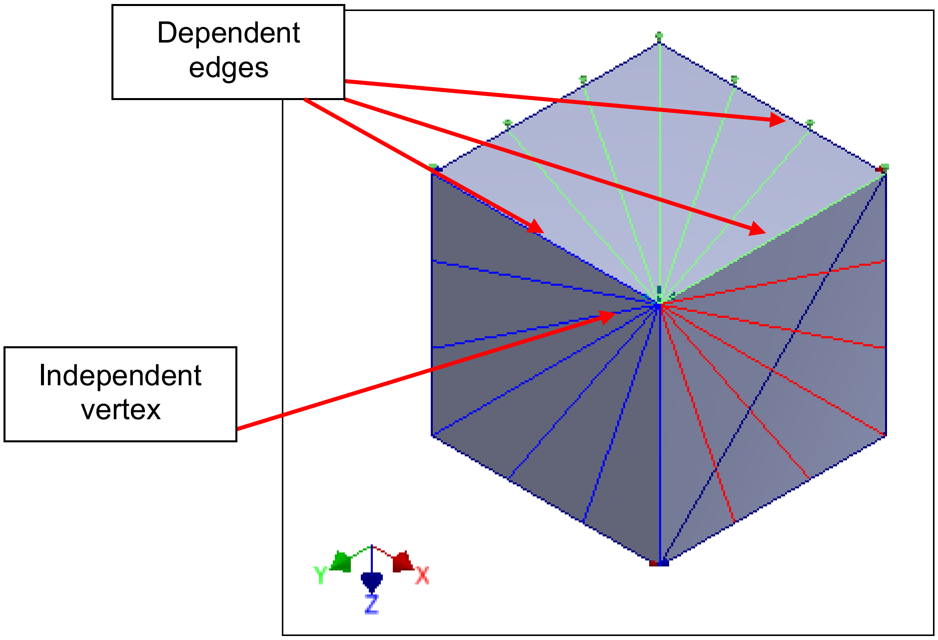


Figure 12. Rigid Element applied on the part for Tz

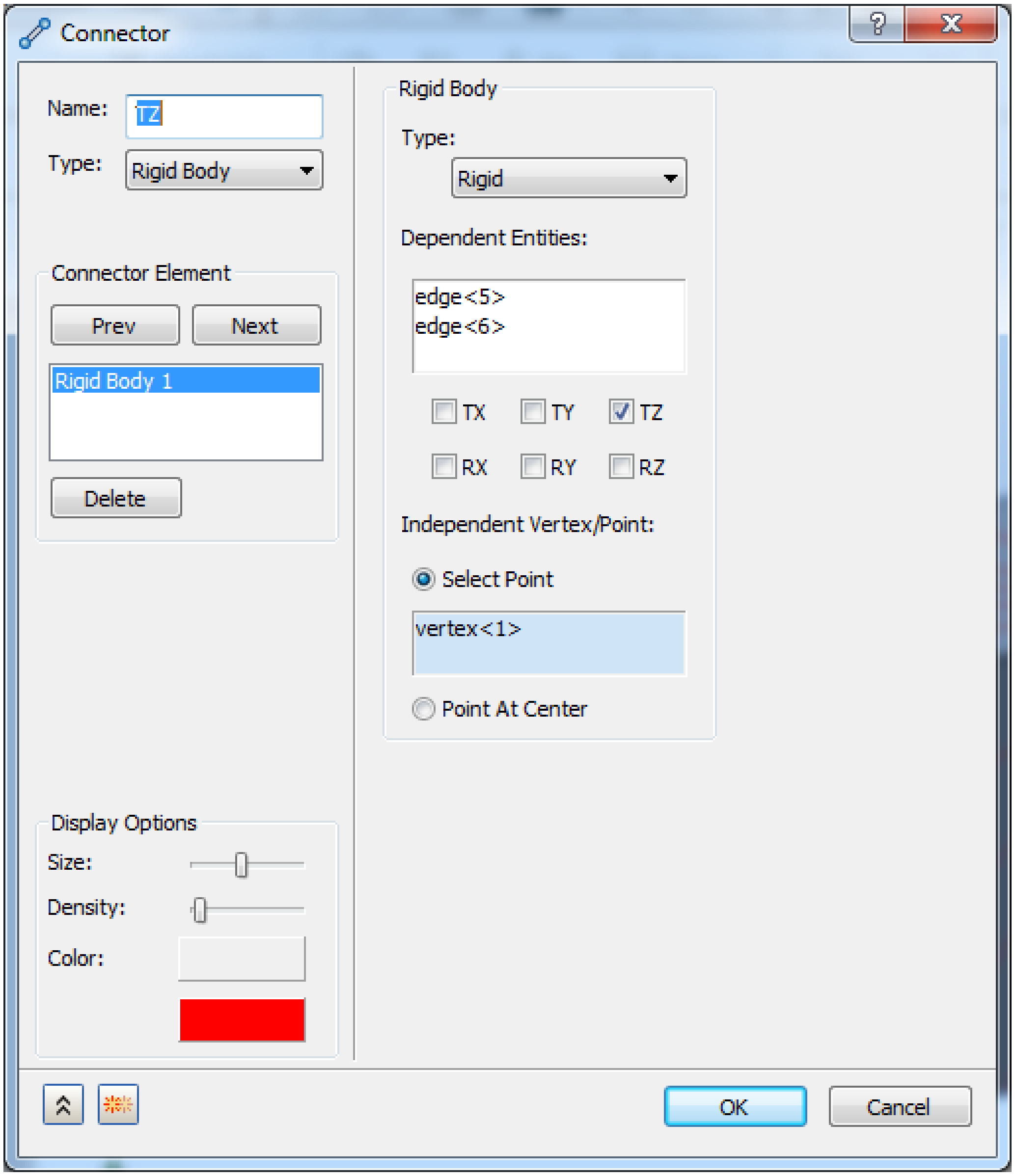


Figure 13. Rigid Element for Tz

3.5 Creating the Enforced Displacement Load

Create the **Enforced load,** go to WaveFEA **Model Tree** right-click on **Loads** and choose **New,** rename the Title as **Enforced Load**, select **Enforced Motion** from the **Type** drop down menu, and select **Displacement** from the **Enforced Motion Types** drop down menu. Enter **Tx** load of **-6** in the **Magnitude (in)**, In the **Selected Entities** select the **Front surface** as shown in Figure 14. Click **OK,**

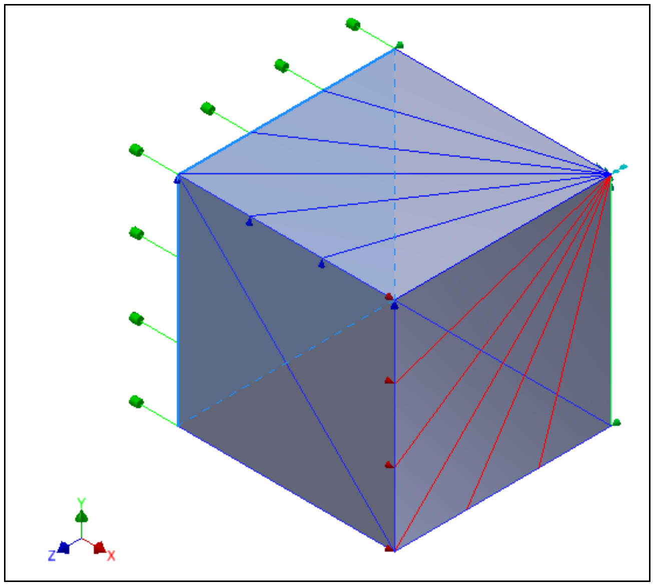


Figure 14. Face Selection for Enforced Displacement Load Definition

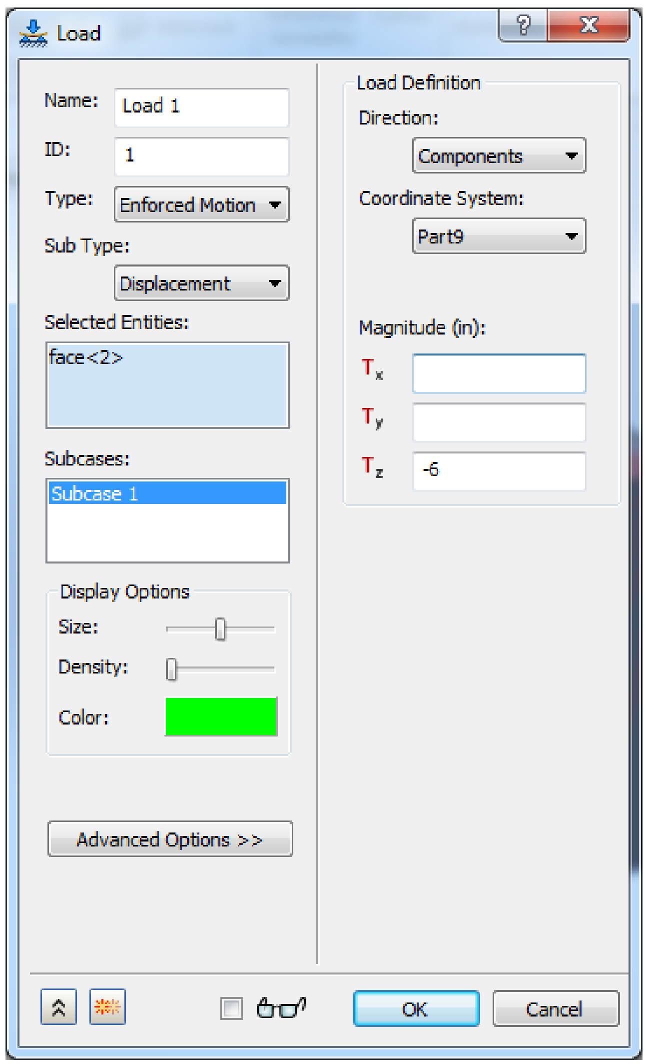


Figure 15. Enforced Displacement Load Definition

Click **OK** to create Enforced Displacement Load.

3.6. 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 16. Click **OK.**

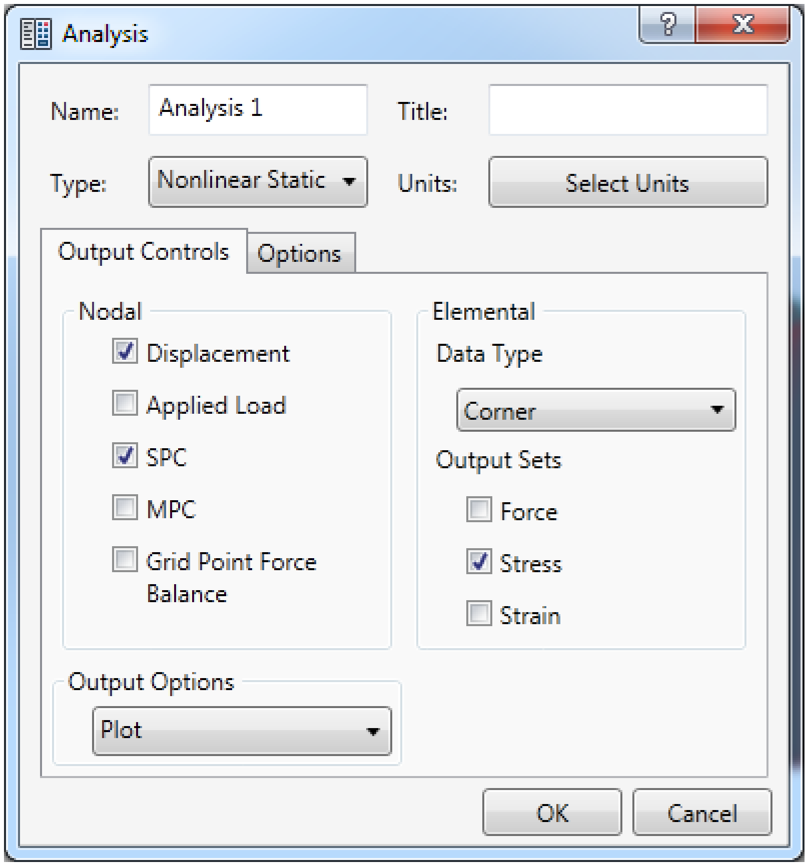


Figure 16. Analysis dialog

Now, from the **Subcase 1**, right-click on **Nonlinear setup 1** and select **Edit.** In the **Number of Increments** field, enter **48** Keep **Intermediate Output** as **ON** as shown in Figure 17.

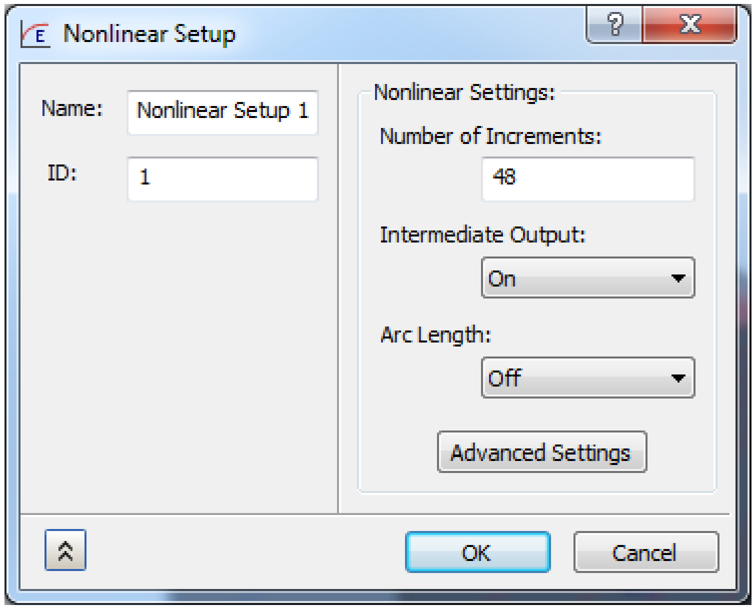


Figure 17. Load Set Options for Nonlinear Analysis

3.7. Mesh the Model

Right-click on **Mesh Model** and choose **Edit**. Set the **Element Size** to **1.7**, from the **Element Order** drop down menu select **Linear** click **OK**.

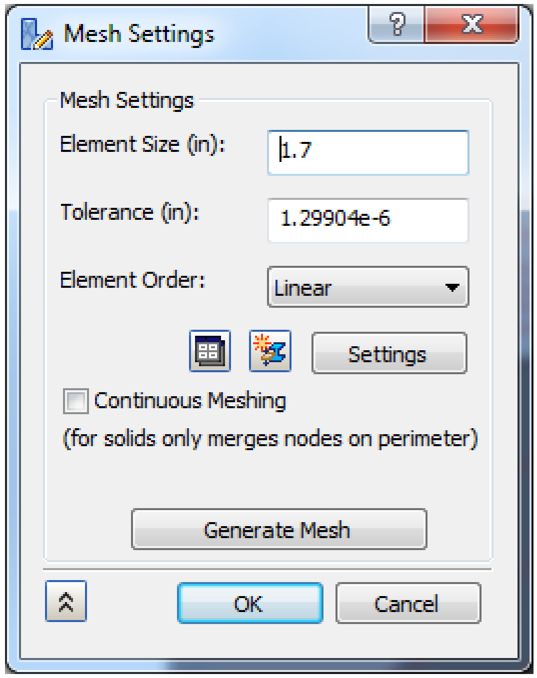


Figure 18. Mesh size

After updating the mesh, now your model should look like the following.

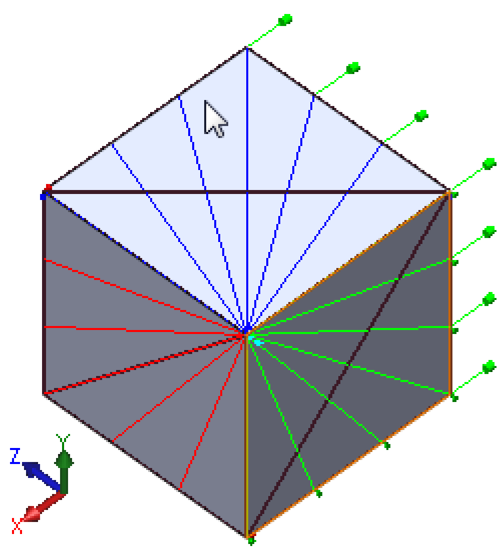


Figure 19. FEA Entities Applied for the Part

Save the model.

4. 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 **NEi Nastran** solution completes**,** click **OK** to close that message window.

5. Results review

In this step, you will view the final deformation and animate the deformation from 0% to 100% load.

Right-click on **Result** and select **Edit**, from the **Subcase** drop box select **'Increment 48, Load=1.0',** select **Displacement** under **Contour options**, select **Displacement** under **Deformed options,** keep **Actual** in the **Deformation Scale**, click **Display.**

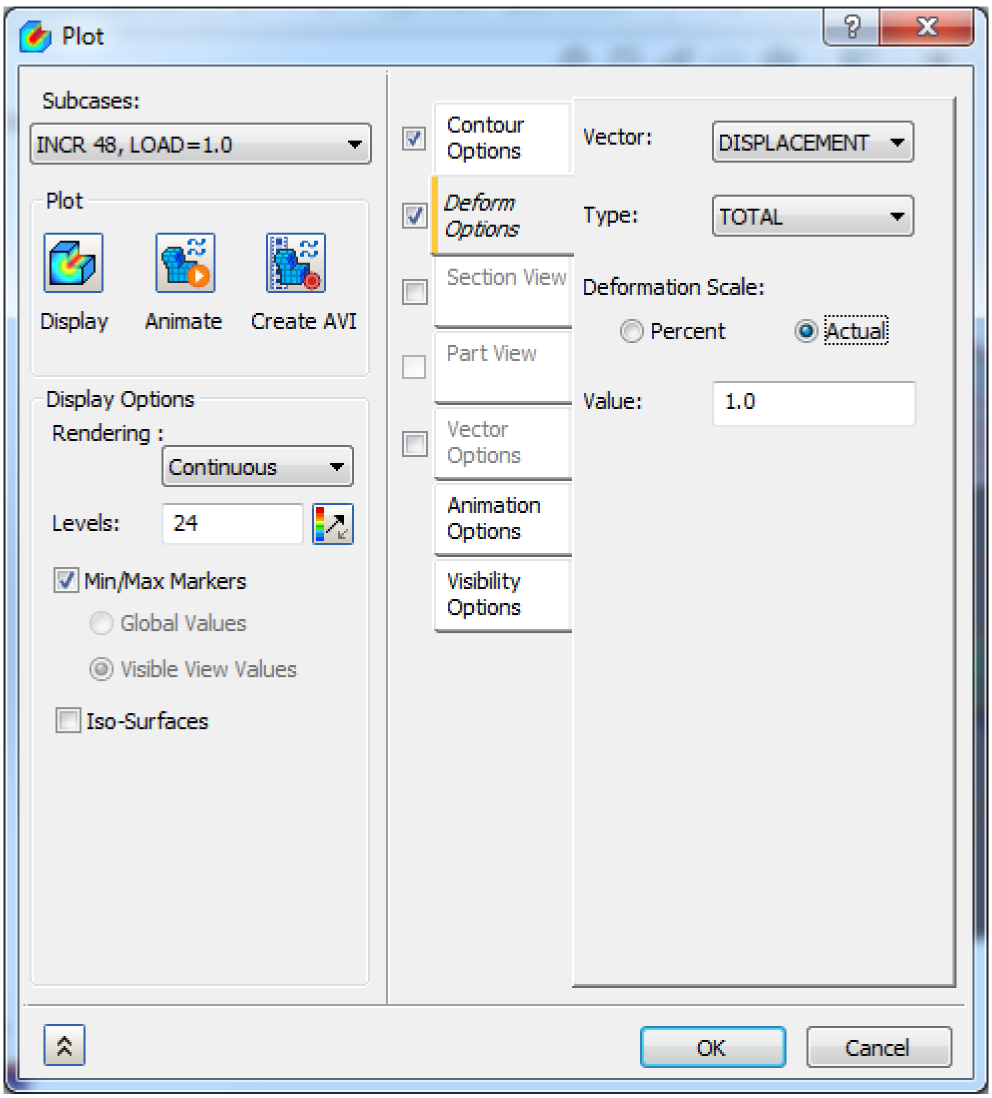


Figure 20. Displacement plot settings

Your deformed displacement contour should look like the following:

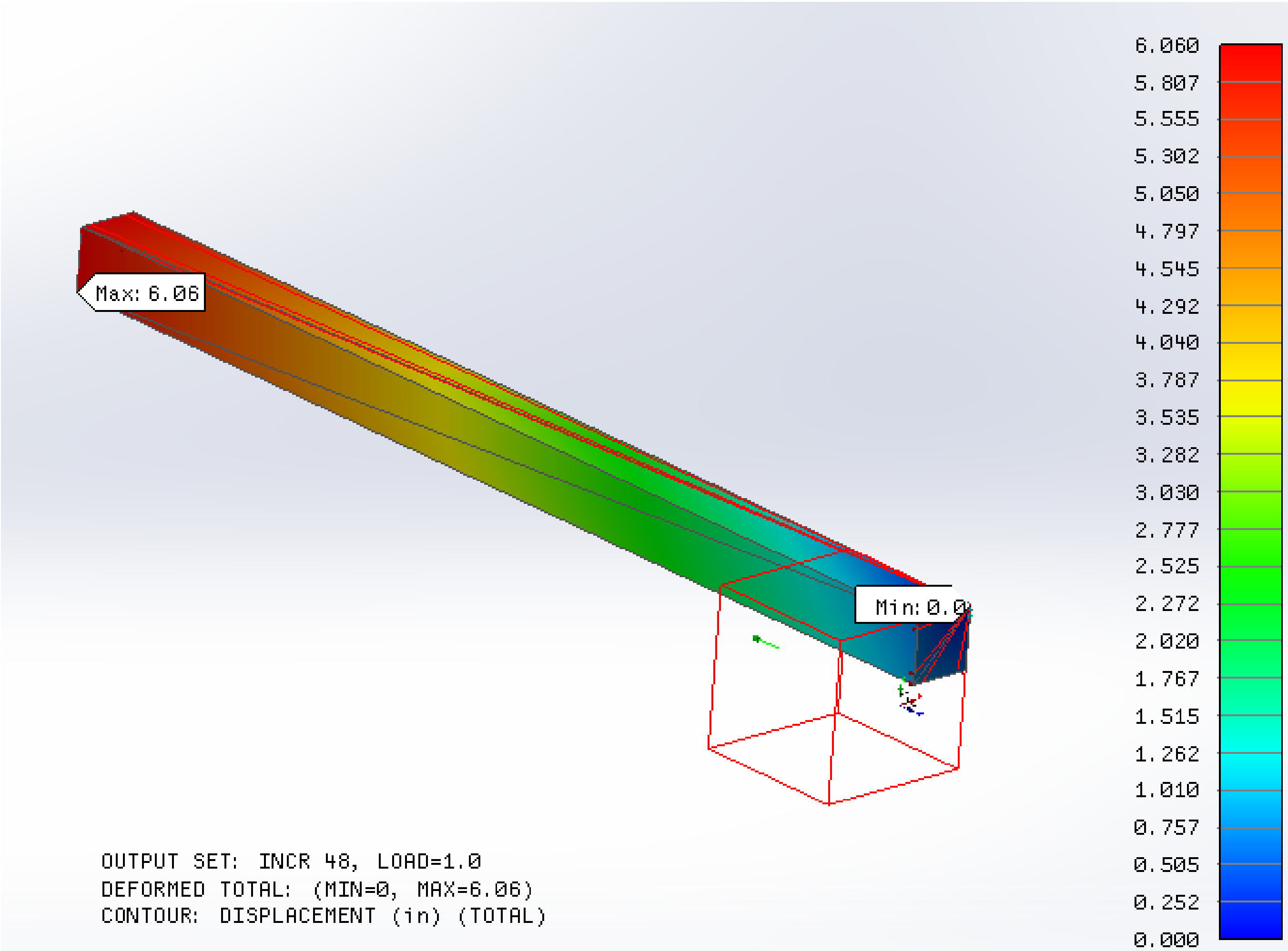


Figure 21. Displacement Total Translation Results

The shape of the hyperelastic element has changed greatly. The height and width of the element shrinked to less than half of their original dimensions to compensate for the deformation in the x-direction.

To **Animate** the deformation from 0% to 100%.

Select **Animate-MultiSet** as follows:

Right-click on **Result** and select **Multiset Animation Settings**, from the **Output Set,** click on **Start Set** drop box and select **'Increment 1, Load=0.0208333',** click on **End Set** drop box and select **'Increment 48, Load=1.0'.** Select **Displacement** under **Contour options,** also select **Displacement** under **Deformed options,** keep **Actual** in the **Deformation Scale**, and click **Animate**.

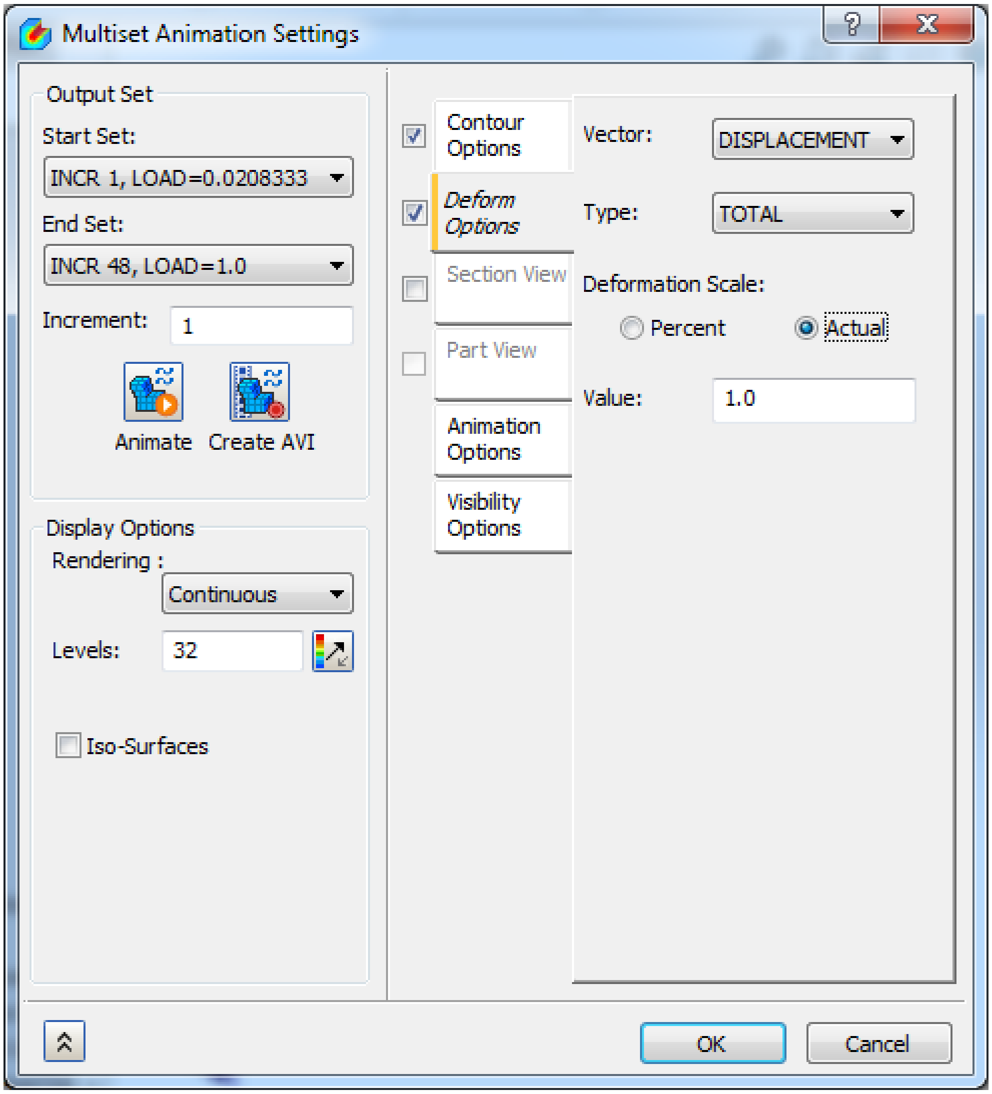


Figure 25. Multiset Animation settings

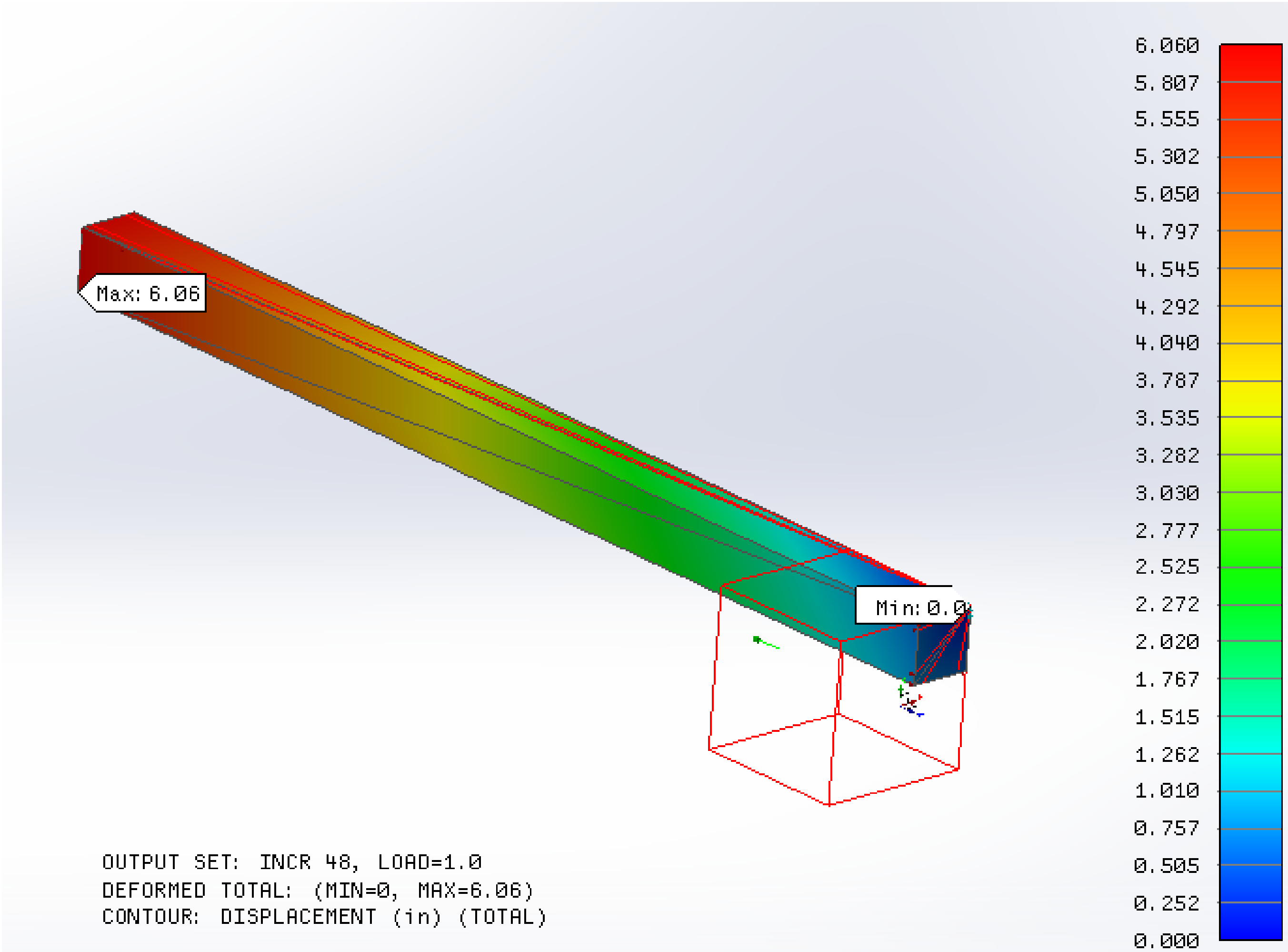


Figure 26. Displacement Total Translation Animation

6. Explanation of Output Results

The nonlinear stress output for the hyperelastic elements differs from other material nonlinear stress output. The following quantities are output at the Gauss points:
Cauchy stresses

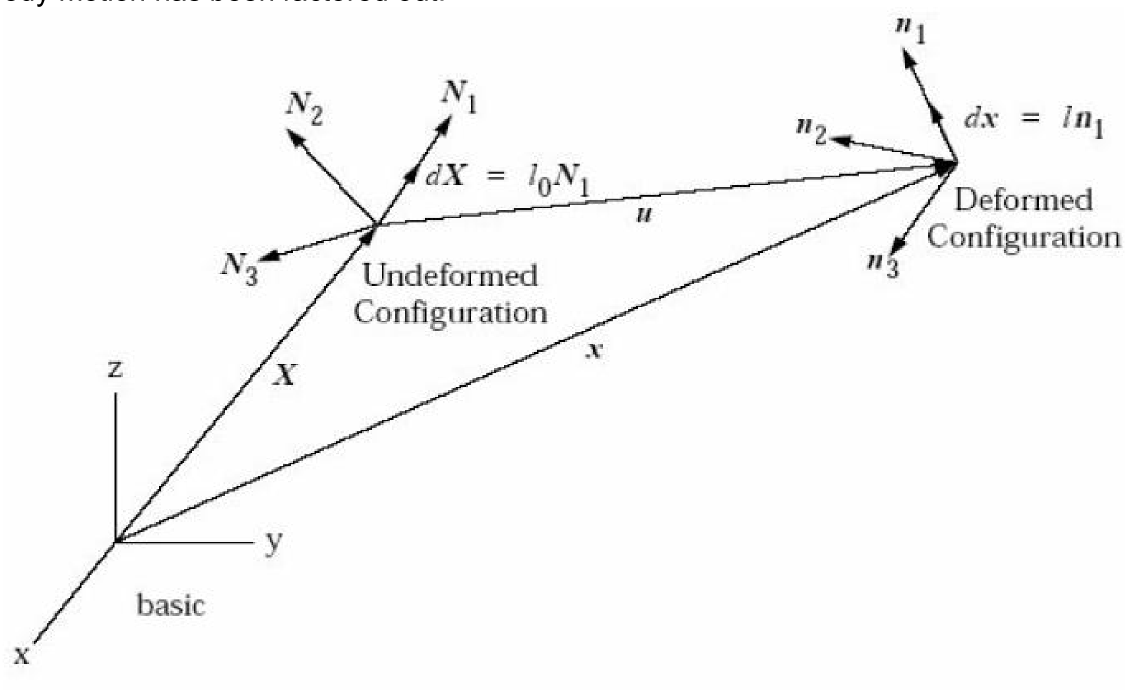
$\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{yz}, \tau_{zx}$

Logarithmic strains, the components of

$$\sum_{l=1}^{3} \ln \lambda_l N_l N_l^T$$

Where:

- λ_l are principal stretches l/l_0 in the principal directions N_1.
- N_1 equals principal directions in undeformed configuration; $n_l = RN_l$ =principal directions in deformed configuration, obtained from N_l by a rigid body rotation. The deformation along N_l is a pure stretch, after the rigid body motion has been factored out.



This output is obtained with the STRESS (or ELSTRESS) Case Control command.

7. Conclusion

Hyperelastic model setup, it is highly nonlinear material and large stain element is used to defined the nonlinearity. This case shows WaveFEA versatility for handling such a problem which has hyperelastic material. It also shows the capability of WaveFEA to handle rubber materials with different options like Neo-Hooken, Mooney Rivilin, Ogden, Yeoh and Polynomial. Also, support for different experimental data functions.