# 2-D Orthotropic Material Analysis in WaveFEA

## 1. Introduction

The model shown in Figure 1 is a 20 inch by 40 inch plate with a 4 inch diameter hole in the center of the plate. It is 0.1 inches thick and constructed of a generic 2-D orthotropic material. The property information is shown in Figure 2 and the material values are shown in the Figure 3. Fourteen elements were defined along the length of the model, and seven elements along the height. The hole was defined to have 18 elements around its circumference. This model will be used to show the importance of correctly using material directions in orthotropic material analysis. This same procedure is used to align material angles for composite/laminate analysis.

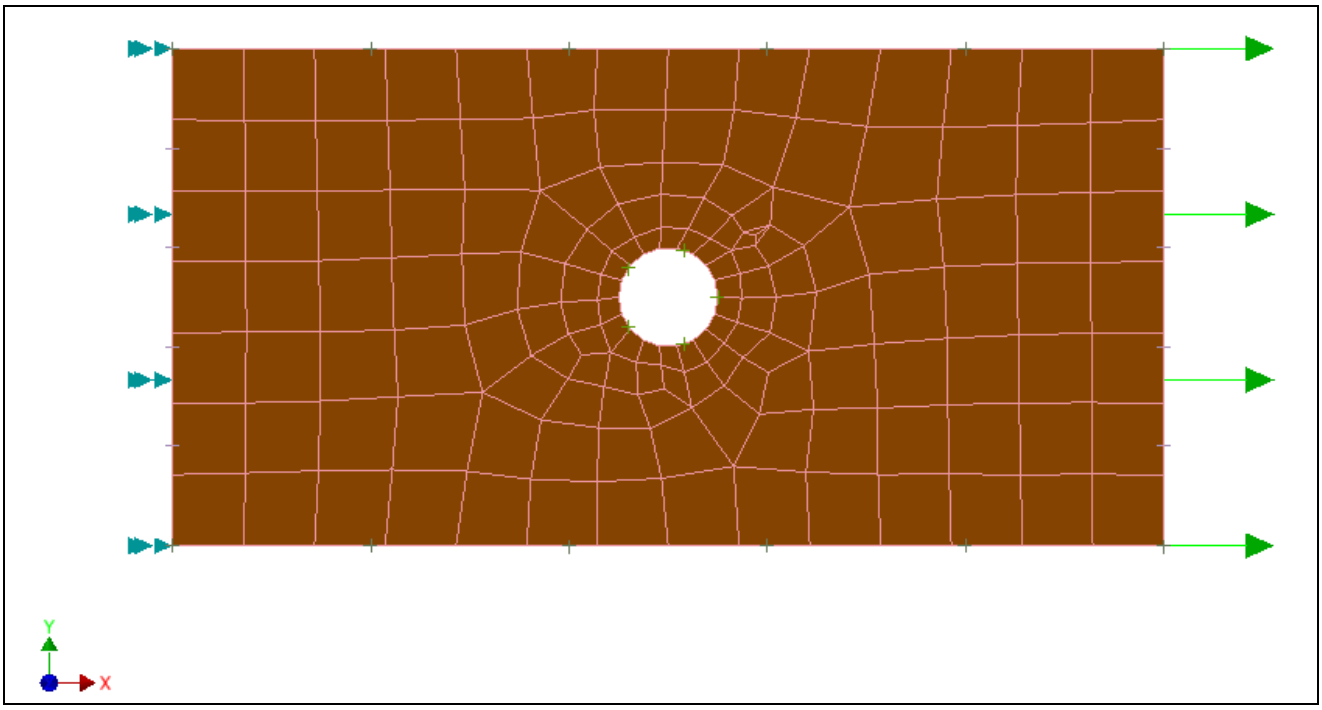



**Figure 1. Plate Model**

## 2. Creating the Model

Here first need to create a part model.

### 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.

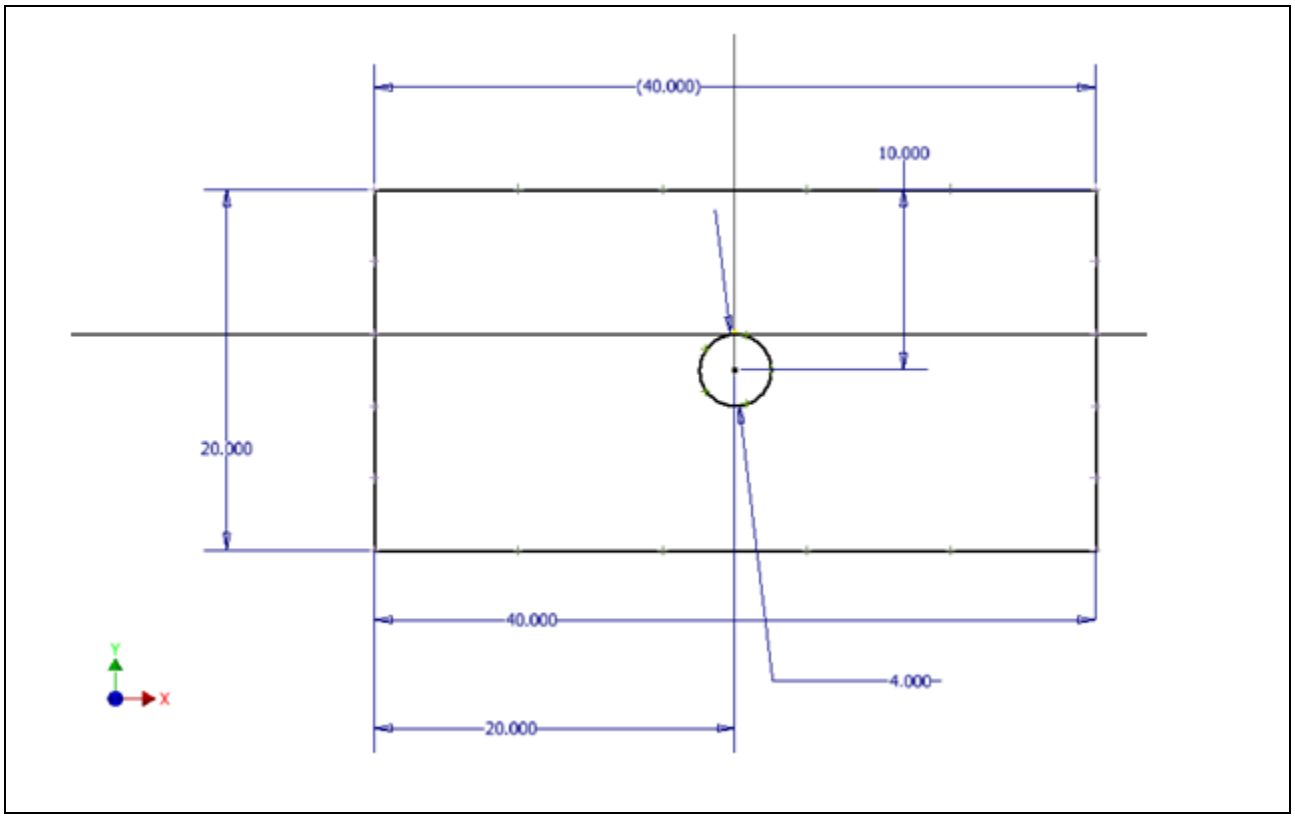



**Figure 2. Creating a Rectangle**

Finish the sketch.

Select **Boundary Patch/Surface** command from **Surfaces.**

Select **OK** to finish.

The extruded model should look as shown in Figure 3.

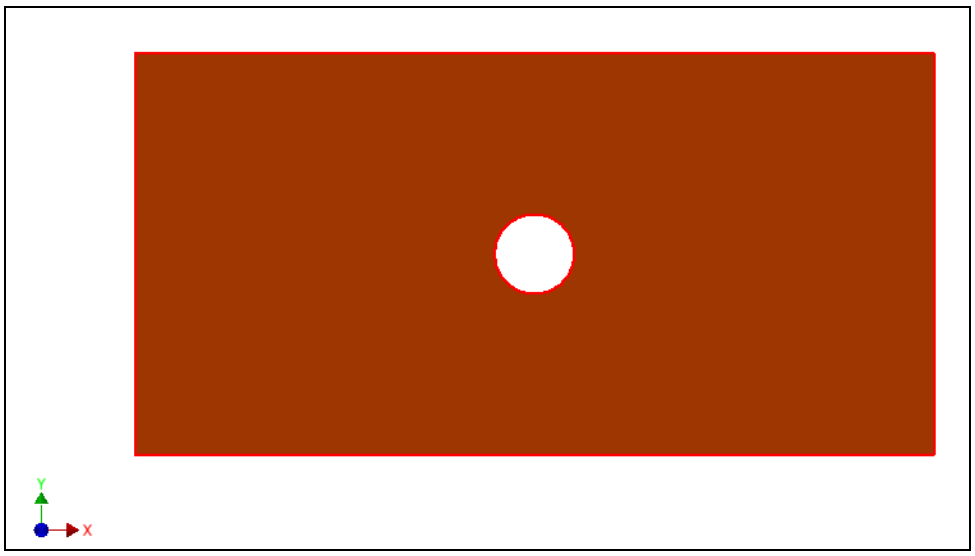



**Figure 3. 2D Surface**

Save the model as **"2D Orthotropic (.sldprt, ,ipt)".**

## 3. Pre-Process the Model

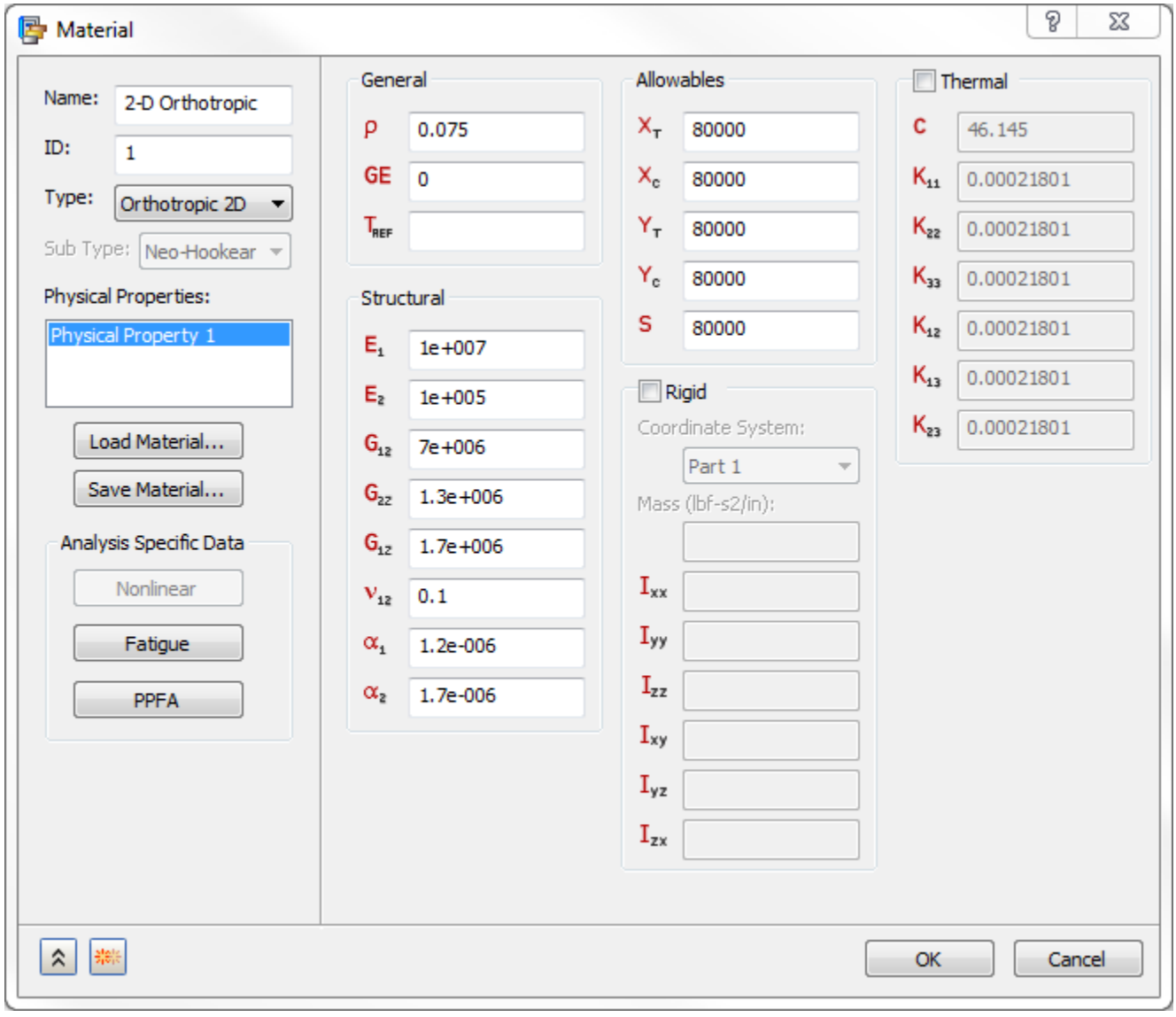
You will first define the material, then define the property of the element, create a Physical property for a quad elements, apply the constraints and loads, and finally run a NEi Nastran Nonlinear analysis.

### 3.1 Define the material and physical property

Define material and physical property as shown In the below images**.**

Enter the following values into their respective field as shown in Figure 4.

Your **Material Properties** should look as follows



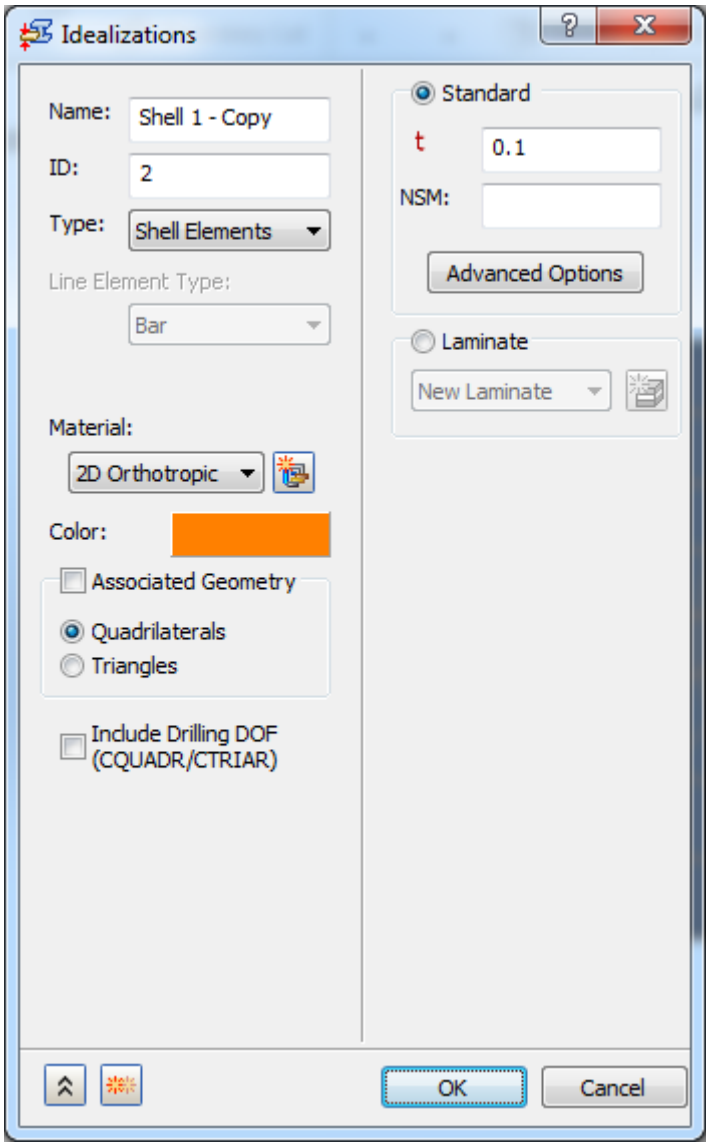



**Figure 4. Material and Physical property definition**

Click **OK.**

## 4. Running the Model with no Changes

Go to **Parameters**, right-click and select Edit. Check Advanced Settings and click on Results Processor Parameters.

Notice, **ELEMRSLTCORD** is set by default to MATERIAL, change it to **ELEMENTAL**; to know the importance of keeping it default to MATERIAL.

Run the analysis with above settings. The result of this analysis run are shown in Figures 5 and 6. Clearly these results are not valid.

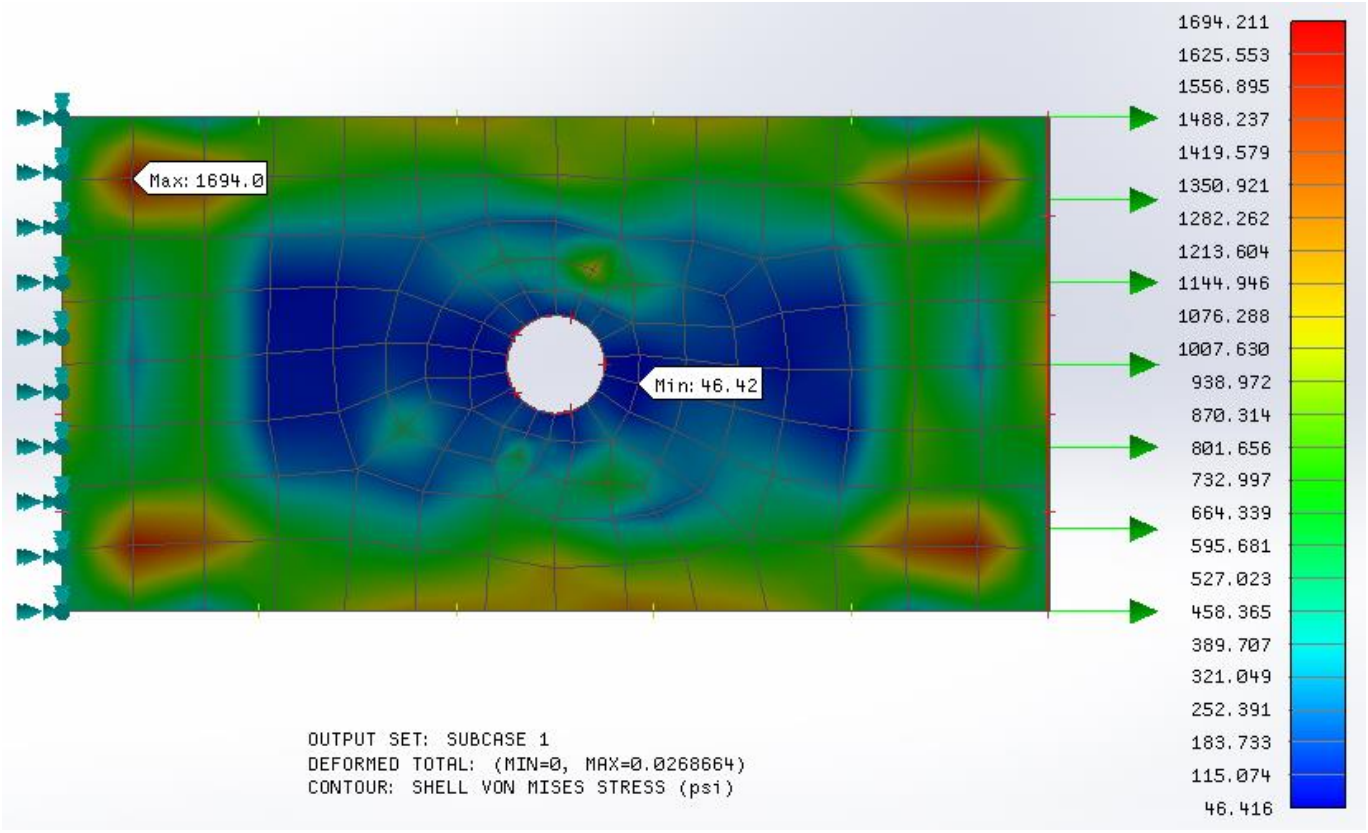



**Figure 5. VonMises Stresses with Basic Element/Material Directions.**

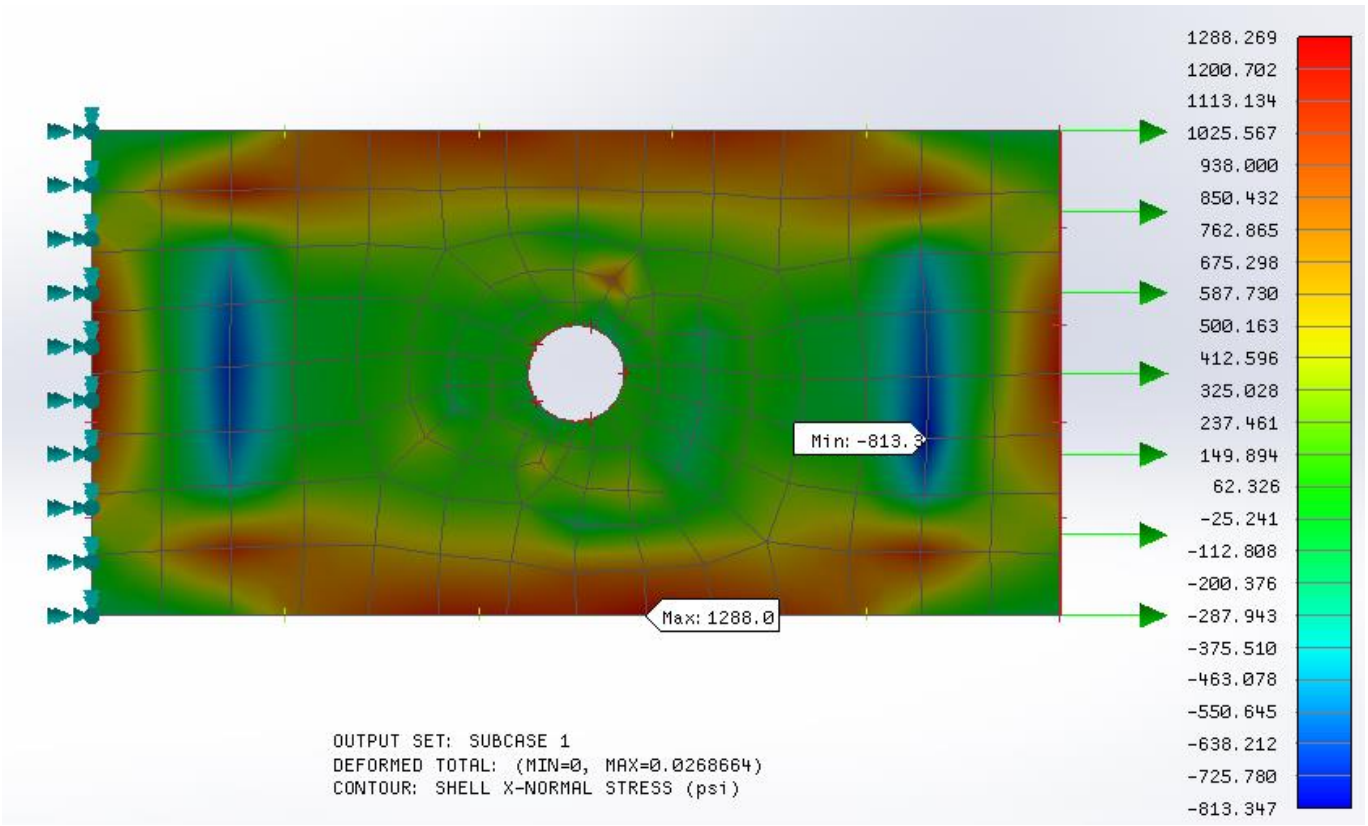
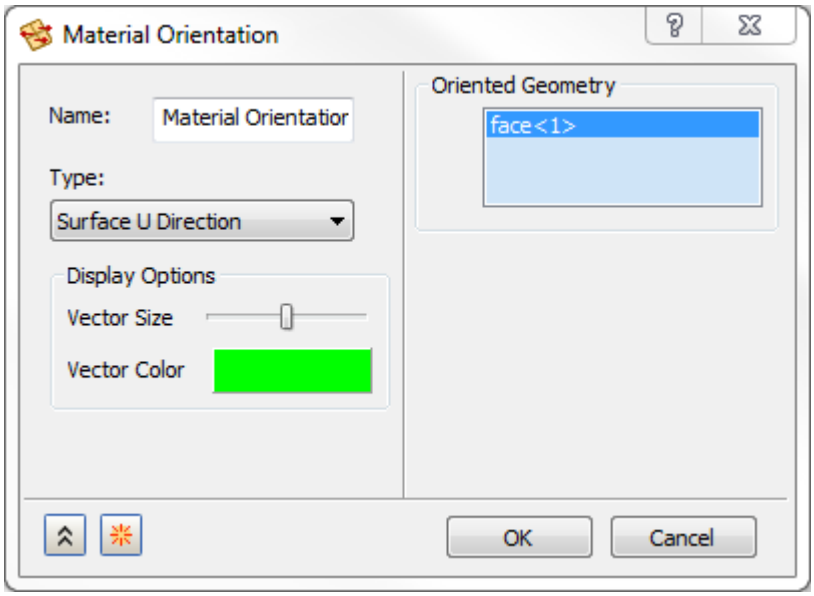



**Figure 6. Normal X-Stress with Basic Element/Material Directions.**

## 5. Updating Material Direction

Because a 2-D orthotropic material is used, the material direction must be set. This is accomplished by using Material Orientation, right-click **FE Model** and select **Add Material Orientation** and select the type as **Surface U Direction** and select the face.



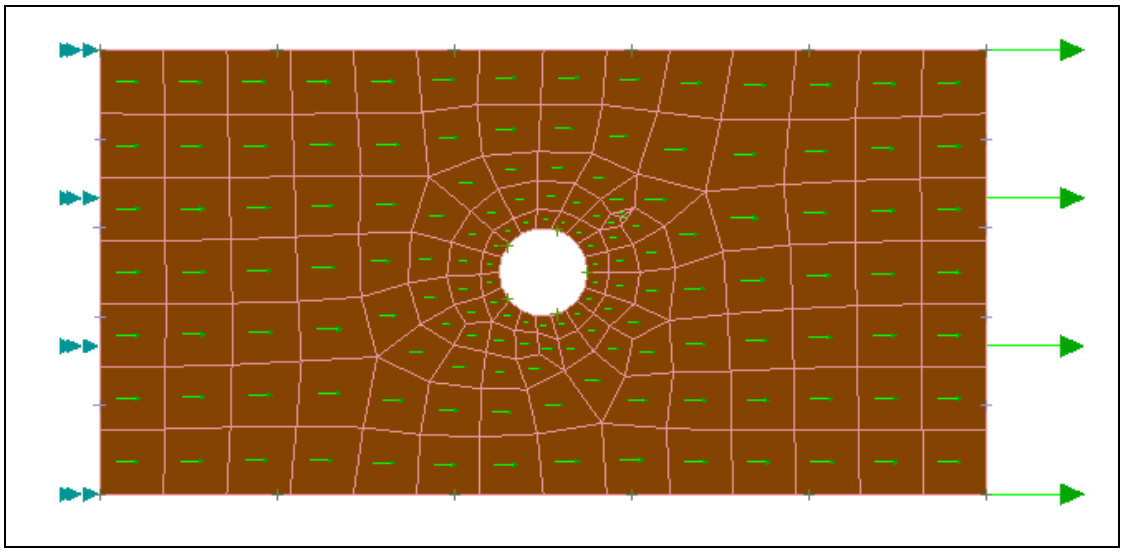



**Figure 6. Normal X-Stress with Basic Element/Material Directions**

## 6. Re-run the Model

Re-running this model produces the following results in Figure 7 and 8. The von Mises stress plot is now correct; however, the normal X-stress is not.

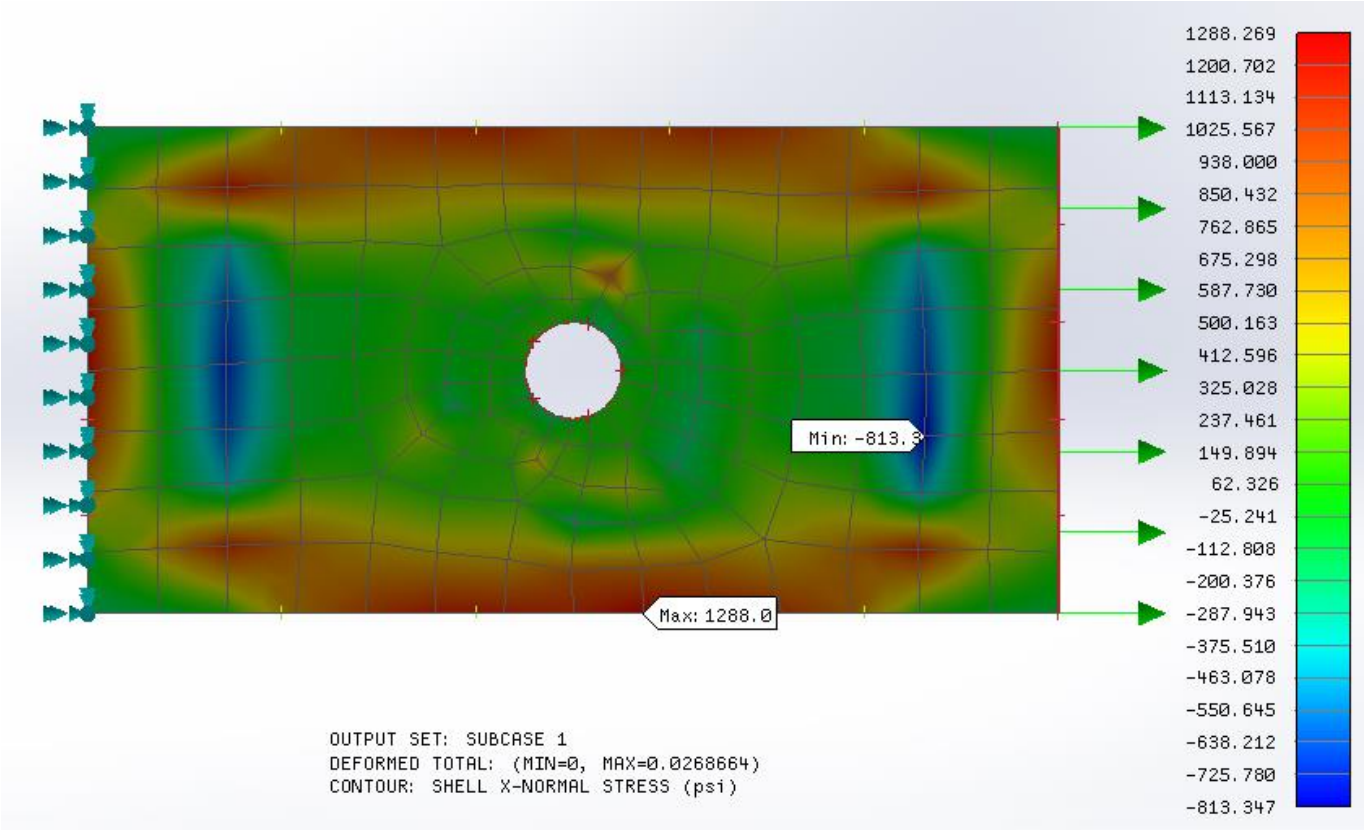



**Figure 7. von Mises Stresses after Material Angle Update**

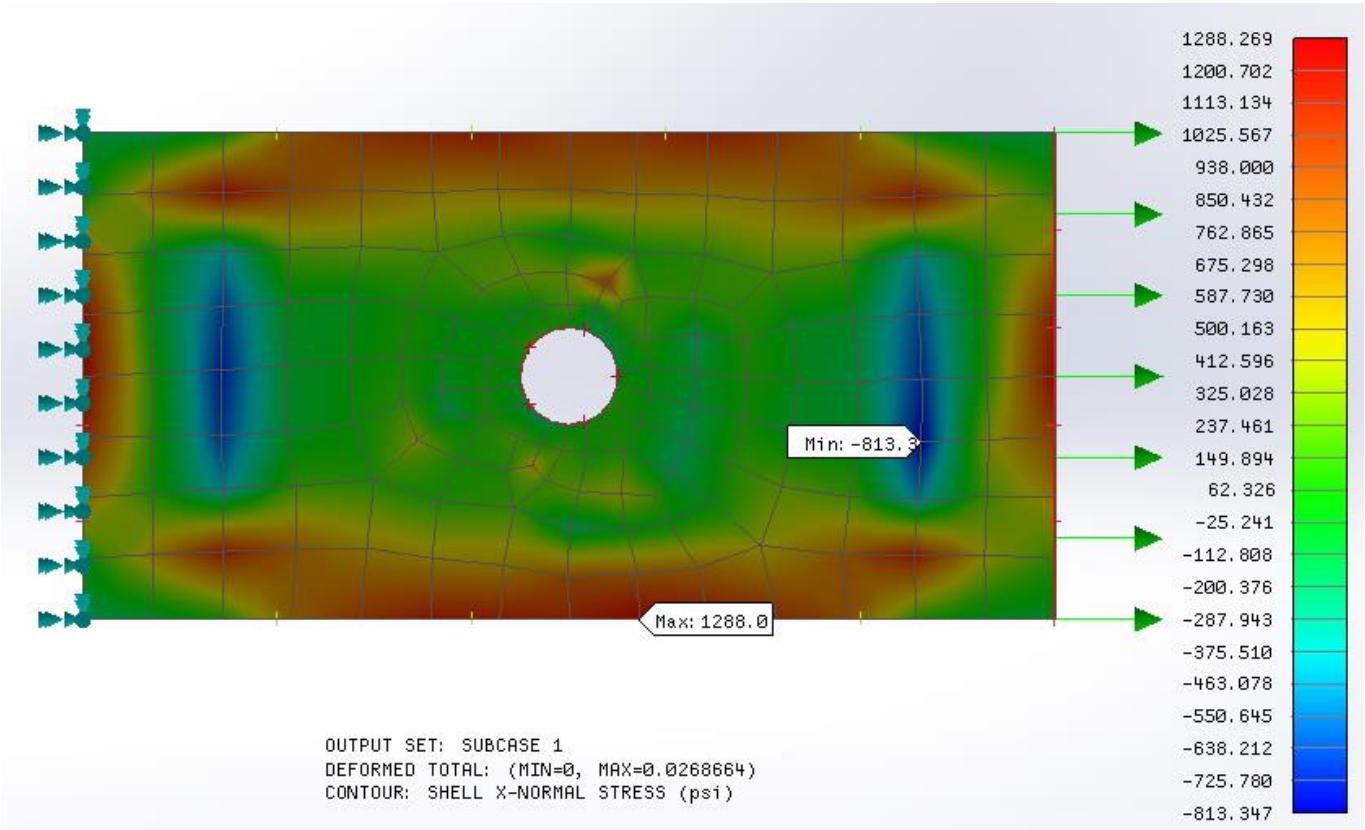
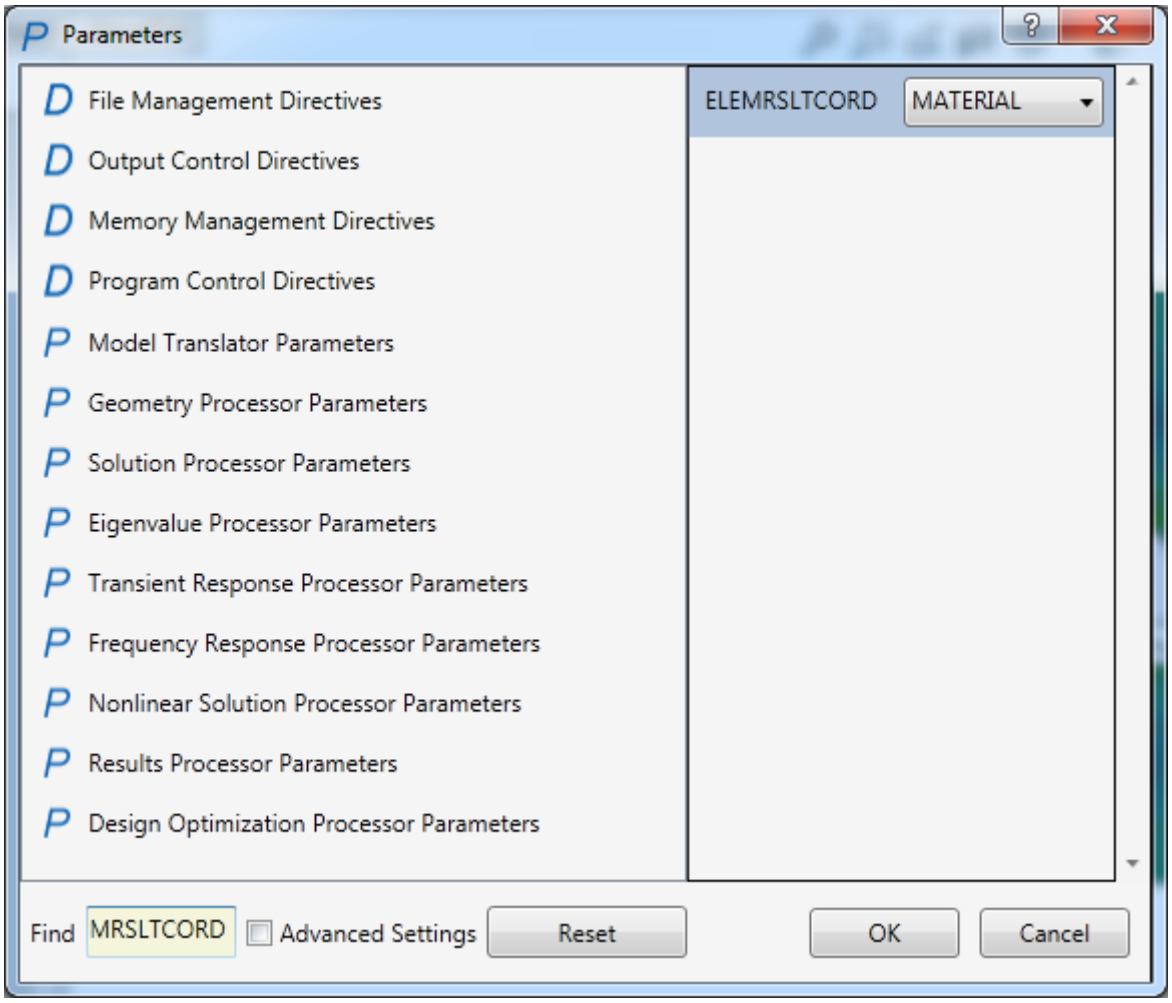



**Figure 8. Normal X-Stresses after Material Angle Update**

## 7. Correcting the Normal X-Results

Re-run the same model with the material angle changed created above in WaveFEA and change the **ELEMRSLTCORD = MATERIAL** in parameters dialog. Notice, the normal X-results are now correct as shown in Figure 9. The **ELEMRSLTCORD** parameter changes the output coordinate system.



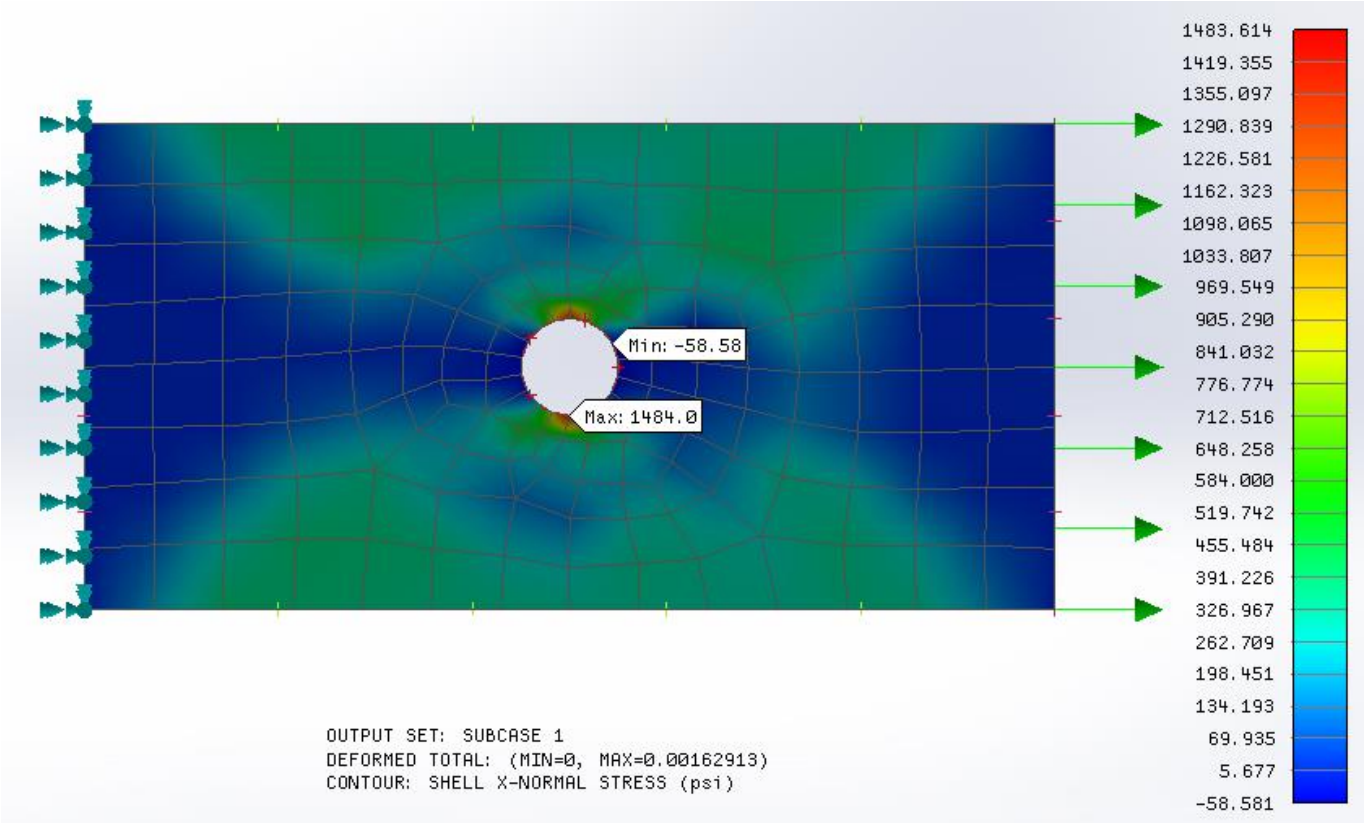



**Figure 9. Correct Normal X-Stress Results**

## 8. Conclusion

To generate the correct results when using a 2-D orthotropic material, or when using a composite/laminate it is important to update the material angle in WaveFEA and to request the output in the Material coordinate system, which is taken care in WaveFEA by default.