FEA ANALYSIS OF A JOINT

Introduction:

In this lab, I will go further in depth with PATRAN's function and analyze the stress on a joint through FEA analysis and convergence study. This process includes modeling a thin plate with a loading condition shown on Figure 1. The eyelet is subjected to a downward force of 1000 lbf. The load is distributed along the bottom half of the eyelet. The convergence study will involve a variety of runs with different global edge length for the necessary results. Students will get to see the effects as the model being divided into smaller and smaller elements.

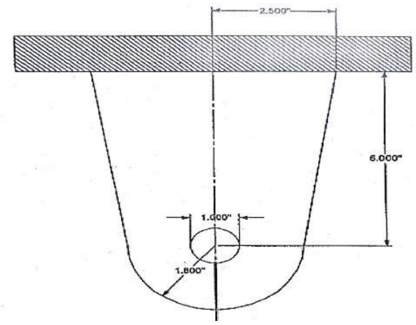

Figure 1: *The Joint Subject to be Analyzed*

Procedure:

The modeling process begins with creating a semicircle curve through using "Arc3Point", which connects three nodes from the beginning to end of the semicircle curve. This same process will then be used to complete the eyehole by creating four nodes and connect two semicircles (formed from top three nodes and bottom three nodes, where 2 nodes will be used twice) to make a complete circular hole. From the initial semicircle, draw a line and connect it to the base as shown on Figure 1 by using "TanPoint" and trim the unnecessary excess formed from the curve. After the basic model is formed, a surface will be added onto these curves and replaced them. The material and thickness of this surface is listed on Table 1 below.

Thickness	Elastic Modulus	Yield Strength	Poisson's Ratio
1/8"	29.5E6 psi	3.4E4 psi	0.3

Table 1: Material Properties for the Joint Subject

The displacement condition is set as "Element Uniform" with 2D element rather than "Nodal" because the joint subject is held down along its base. The load will then be added as "CID Distributed Load" targeting the bottom lip of the eyelet with a downward force of 1000 lbf. After everything is correctly set up, a mesh will be created for the analysis. Each mesh creation represents a run for the convergence study; thus, it will have a different value on the "Global Edge Length" for each time the test is conducted. For this experiment, the tests will run on the values: 0.05, 0.1, 0.5, 1, and 2.

Result:

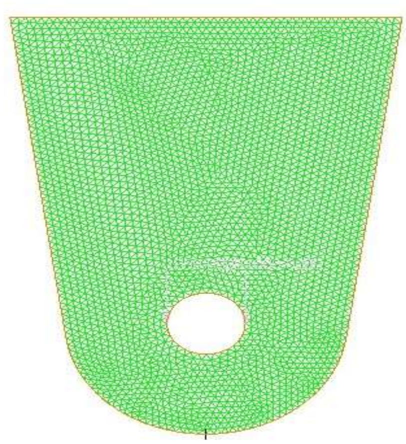

Figure 2: *Undeformed Structure of the Joint Subject with Mesh Added*

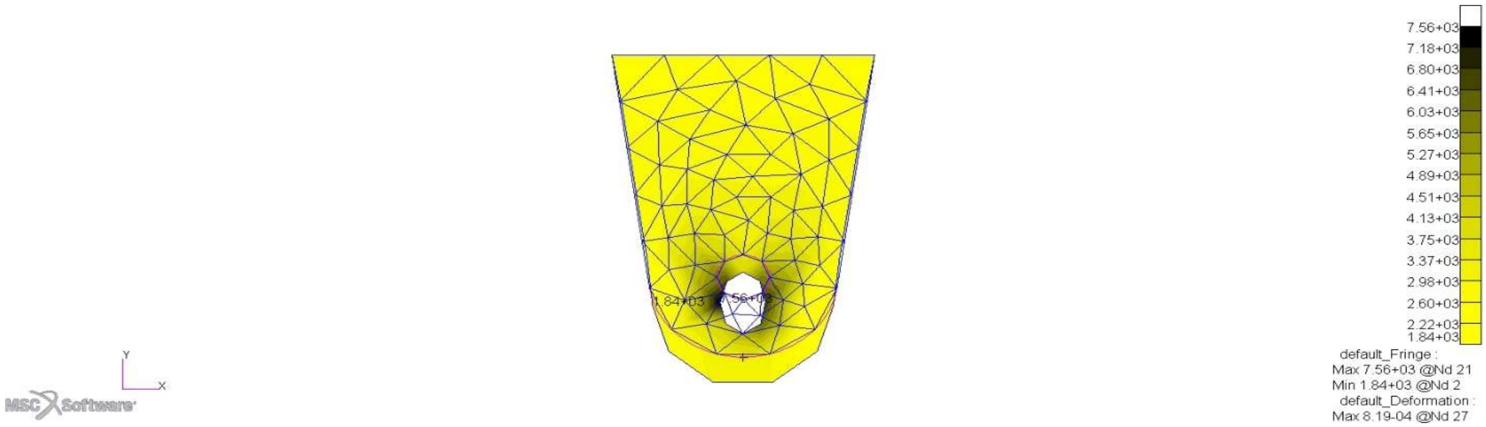

Figure 3: *Deformed Structure of the Joint Subject at 0.1 Global Edge Length*

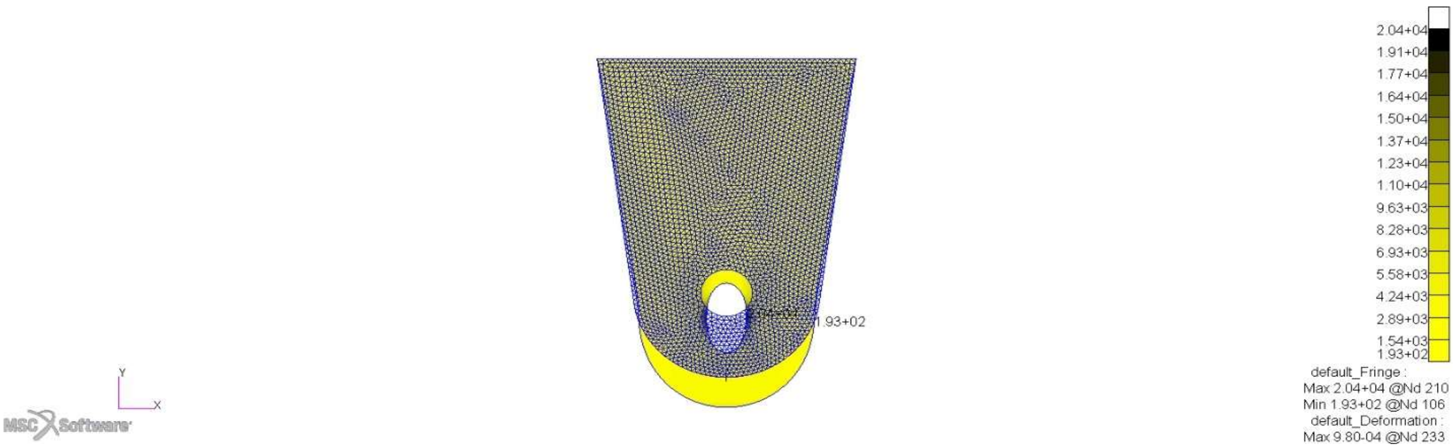

Figure 4: *Deformed Structure of the Joint Subject at 1.0 Global Edge Length*

	Global Edge Length				
	0.05	**0.1**	**0.5**	**1**	**2**
Displacement	9.89E-04	9.80E-04	8.85E-04	8.19E-04	8.13E-04
Minimum Stress	1.05E+02	1.93E+02	9.99E+02	1.84E+03	2.35E+03
Maximum Stress	2.46E+04	2.04E+04	8.77E+03	7.56E+03	9.06E+03

Table 2: *Collected Data for the Trials*

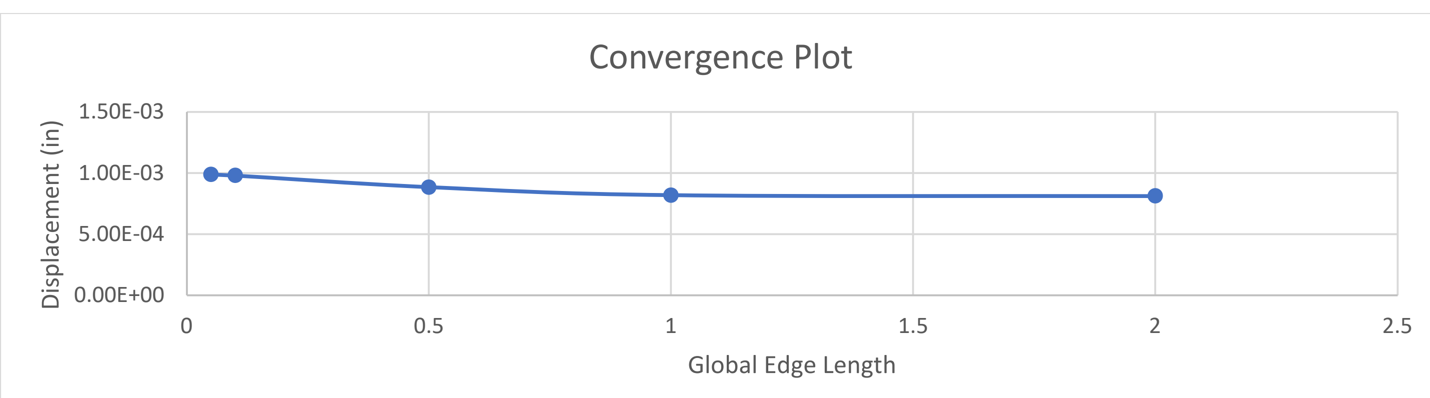

Figure 5: *Convergence Plot of the Joint Subject with Global Edge Length vs. Displacement*

Discussion/Conclusion:

From an overview of the deformation on Figure 3, and Figure 4, it can be seen that as the values of Global Edge Length increase, the number of elements decrease to accommodate the size of global length. Thus, the 'mesh' seems to be sparse and thinner. The stress region on Figure 4 appears to be larger compared to the one on Figure 3. This is due to a wider mesh, thus giving the stress region a larger size. This amplification may not be accurate, which is why there are multiple runs conducted for this convergence study.

In a way, the smaller the Global Edge Lengths get, the more accuracy they can generate. So, the plot in Figure 5 should be perceived as the graph moves toward zero, which means the displacement is converging closer to 1.00E-03 rather than 8.00E-04. Using Figure 3 as an example, the stress region is darker on the sides of the eyelet. This situation indicate that these sides are under a larger stress, and therefore, the damage will begin to occur in these areas when more stress is loaded onto this joint subject.