CONVERGENCE STUDY ON TAPERED AIRCRAFT WING

Introduction:

The purpose of this lab is to go further in depth with PATRAN's function and convergence study. This process include modeling a linearly tapered aircraft wing with a loading condition of linearly varying spanwise distributed load. The necessary dimensions of the model is shown below in Figure 1 and Figure 2. The convergence study will involve three cases on the model: 20, 40, and 80 elements. The effects of dividing the model into smaller and smaller elements will be highlighted in this simulation.

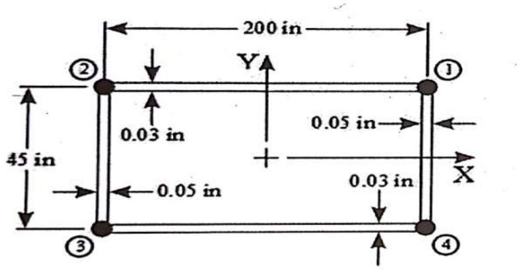

***Figure 1:** Cross Section at Tip*

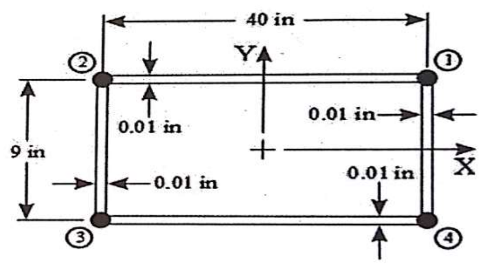

***Figure 2:** Cross Section at Root*

Procedure:

The modeling process begins with creating two nodes, indicating the distance between the root and tip in the z coordinate. Knowing the dimensions of the root and the tip of this tapered wing, students can formulate a function for this model using the root as base. These functions are shown in Table 1 and the beam's material properties be found in Table 2 below:

	Dimension
Height	$200 - z\left(\frac{160}{1000}\right) = 0$
Width	$45 - z\left(\frac{36}{1000}\right) = 0$
Thickness 1 (Vertical)	$0.03 - z\left(\frac{0.02}{1000}\right) = 0$
Thickness 2 (Horizontal)	$0.05 - z\left(\frac{0.04}{1000}\right) = 0$

***Table 2:** Dimensions of the Tapered Aircraft Wing*

Parameter	**Value**
Elastic modulus (E) [psi]	29.5E6
Tension stress limit (SY) [psi]	3.4E4
Poisson's ratio (v)	0.29
Mass density [lbm/in^3]	7.339E-4
Boundary condition at root	Built-in

***Table 1:** Properties of the Tapered Aircraft Wing*

After the material's properties are created with these values, the "Bar Orientation Vector" must be defined as <_ _ _> to define in global coordinate orientation of the cross sections with local Y (upward) vector. This can be found in "Input Properties" of the "Beam" section. Note that *Built-in* listed on Table 2 indicates that node 1 is restrained with displacement and momentum, so that they are <0, 0, 0> for both fields.

With all these steps done, the model is now created and shown as a straight line from node 1 to node 2 as it is viewed as 1D. The next step is to apply a distributed load directly onto this tapered wing where the load is a function of $0 = 250 - z\left(\frac{250}{1000}\right)$. After the load is added, mesh will be created for three different cases in which the model is divided into: 20, 40, and 80 mesh elements.

Results:

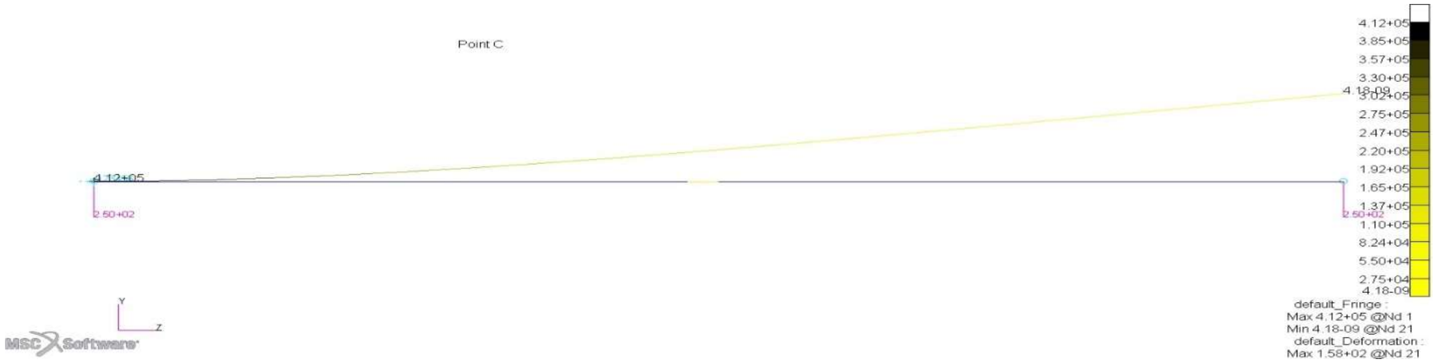

Figure 3: *Deformed Model with 20 Mesh Elements*

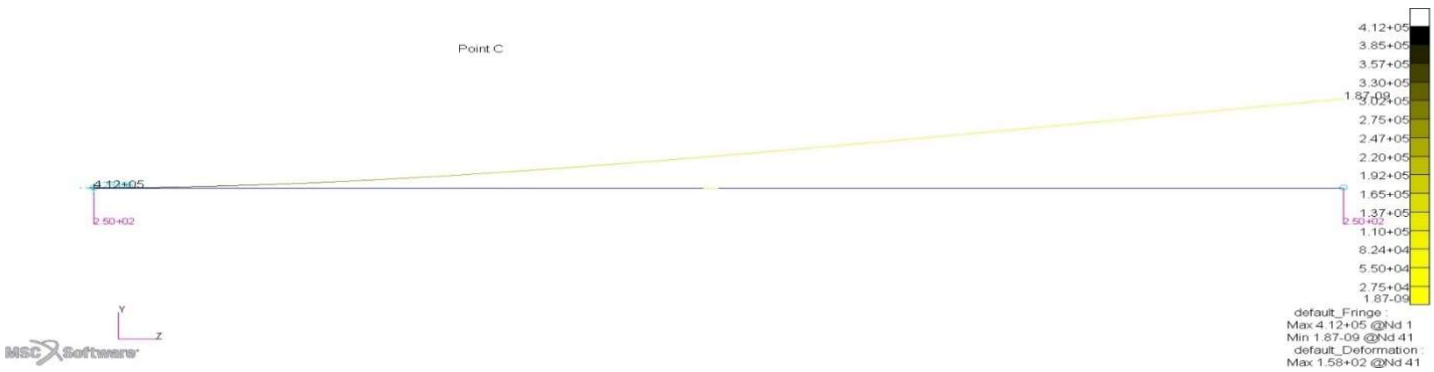

Figure 4: *Deformed Model with 40 Mesh Elements*

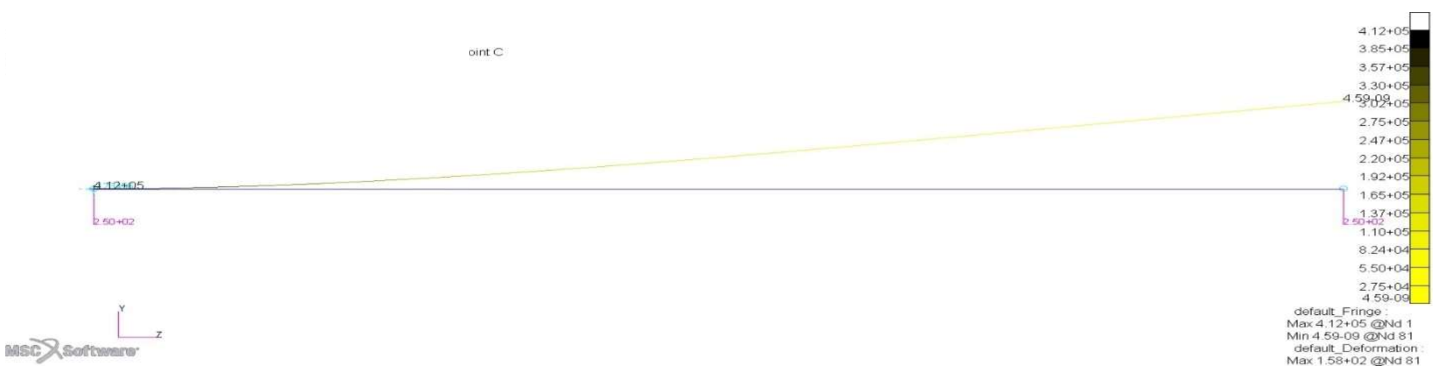

Figure 5: *Deformed Model with 80 Mesh Elements*

	Mesh Elements		
	20	**40**	**80**
Displacement [in] (at the Tip)	1.576846E+02	1.576846E+02	1.085487E-01
Minimum Stress [psi] (at the Tip)	4.175774E-09	1.866773E-09	4.586150E-09
Maximum Stress [psi] (at the Root)	4.121939E+05	4.121939E+05	4.121939E+05

Table 3: *Displacement and Stress of the Tapered Aircraft Wing Model*

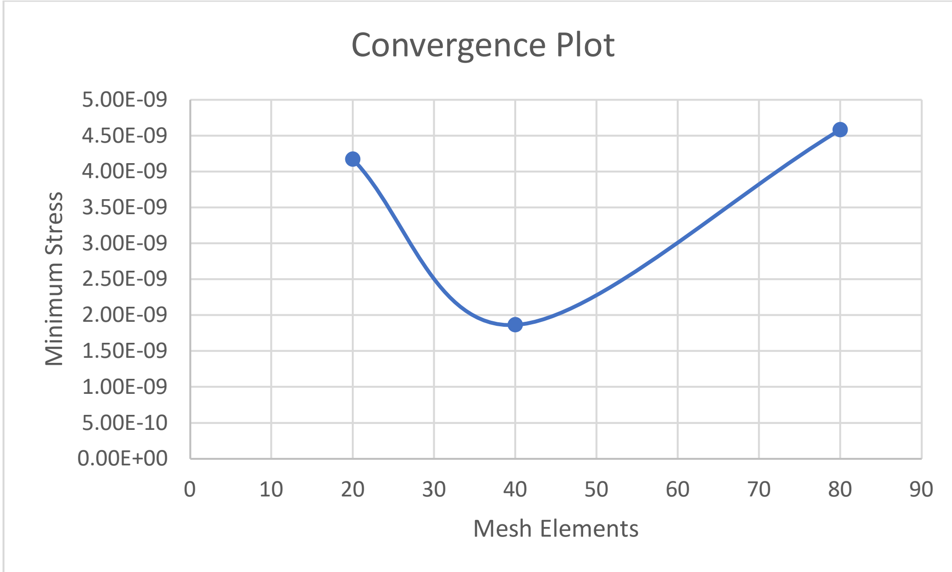

Figure 6: *Convergence Plot of Stress vs. Mesh Elements*

Discussion/Conclusion:

From an overview of the deformation on Figure 3, 4, and 5, the result showed that the displacement is at its greatest at the tip of the beam even though the load is heaviest toward the root. This scenario happens because the root was set with a built-in boundary condition, thus preventing itself from moving. The maximum stress is on the root because the load is a linearly function that starts at the root and decreases to 0 on the z-direction as it reaches the tip of the model. Therefore, the stress at the tip is very minimal. However, since the tip does not have a built-in boundary condition, it experiences a large displacement from the load. From all three figures mentioned earlier, the tip on all of them are shown to have the largest displacement.

Although the lab was successfully conducted, there were still some errors that caused the data to be inaccurate. From Table 3 and Figure 6, the stress on the tip seems to vary by a large margin, this is not how it is supposed to turn out. A correct convergence plot should help pin point a precise valve for the stress as the graph ‘converge’ toward a value rather than moving up and down as shown on Figure 6. The error may have been conceived from a glitch in PATRAN, which interfere with the function of the load when the mesh elements was introduced.