Effect of Element Size in Finite Element Models

Computer Homework 2

MEMS1047, Finite Element Analysis

Dr. Patrick Smolinski

October 16, 2018

Seth Strayer

Introduction:

The aim of this homework was to consider how element sizing, i.e., mesh sizing, affects the relative accuracy of solutions for finite element models. Consider the model shown below:

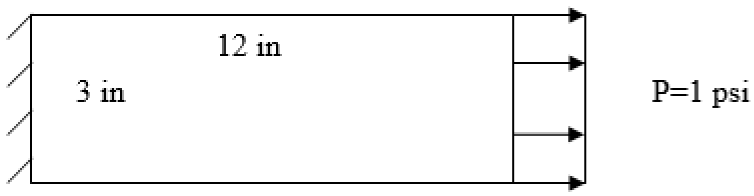

Figure 1: Loading Scenario

We wish to analyze this model using a variety of different element sizes. We will use E = 29e06 psi and Poisson's Ratio, υ, of 0.3. For this problem, we will analyze the model using element sizes of 1, 0.5, 0.25, and 0.1 (shown in Figures 2, 4, 6, and 8, respectively), and analyze the differences in each case. We expect that as we increase the number of elements – i.e., decrease element size – we will receive more accurate solutions. This is because the finite element model uses these elements to solve a series of polynomial equations which ultimately yield the solution for the model. As we decrease the element size, i.e., as the mesh is refined, we expect the finite element solution to approach the "true" solution. Another way of thinking of this is that as the elements become infinitesimally small, the computer can more accurately solve the equations relating to the elements, and thus yield a more accurate solution. We expect that at some point, the elements will become small enough that the accuracy of the solution "plateaus" – i.e., at some specified element size, the elements have become infinitesimally small and we will receive the most accurate solution possible.

Problem Statement:

As stated in the introduction, we wish to consider several different element sizes used for the model shown in Figure 1 and analyze how the element size affects the accuracy of our solution. We first wish to examine if the model has a uniform stress field given the loading scenario shown in Figure 1. Next, we wish to determine which stress - σ_{xx}, stress in the axial direction, σ_{yy}, stress in the y-direction perpendicular to the loading, or σ_{xy}, shear stress in the x-y direction - is largest, and the location of this maximum stress. Furthermore, we will solve the model using four different element sizes, namely 1, 0.5, 0.25, and 0.1, and plot the stresses for each case. We expect that as we increase the number of elements, our solution will become more accurate and perhaps yield a higher value for maximum stress.

For completeness of the results, we will include all of the stresses – that is, σ_{xx}, σ_{yy}, and σ_{xy}, on one plot. This plot will be a graph of stress vs. distance along the beam (which is modeled in Figure 1) and will convey to use how the stress varies as we move along the beam.

Results:

The loading scenario for the model as required by Figure 1 is shown in Figure A below.

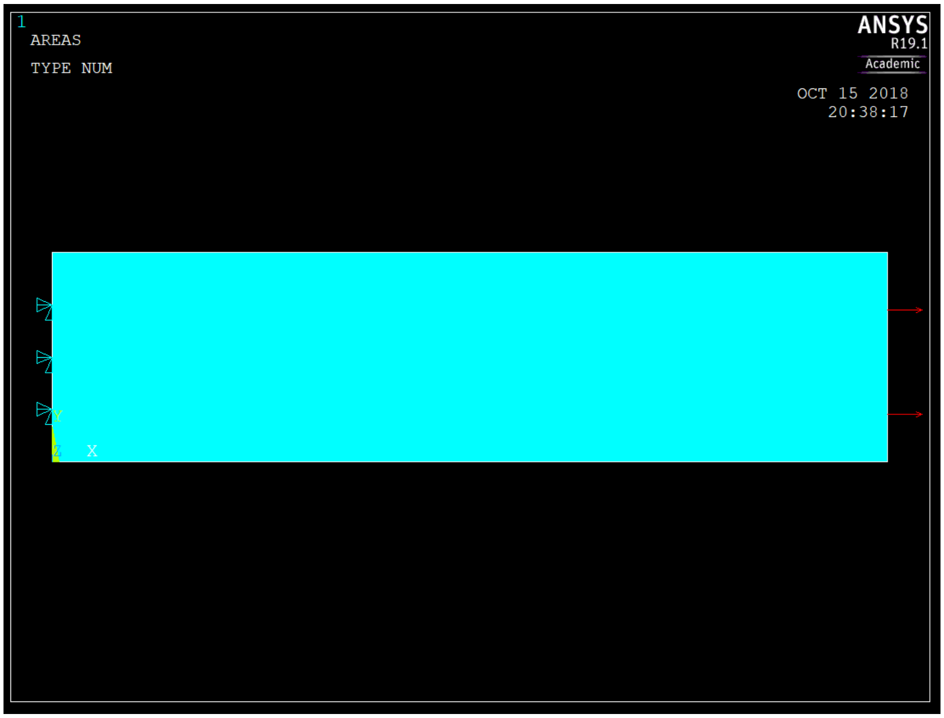

Figure A: Loading Scenario

We see that there is a zero-displacement condition for all degrees of freedom applied on the left side of the beam, and a uniform pressure of 1 psi applied to the right end of the beam. The next several pages will be reserved for the results for various element sizes as described in the above sections.

The model with element size = 1 is shown in Figure 2 below.

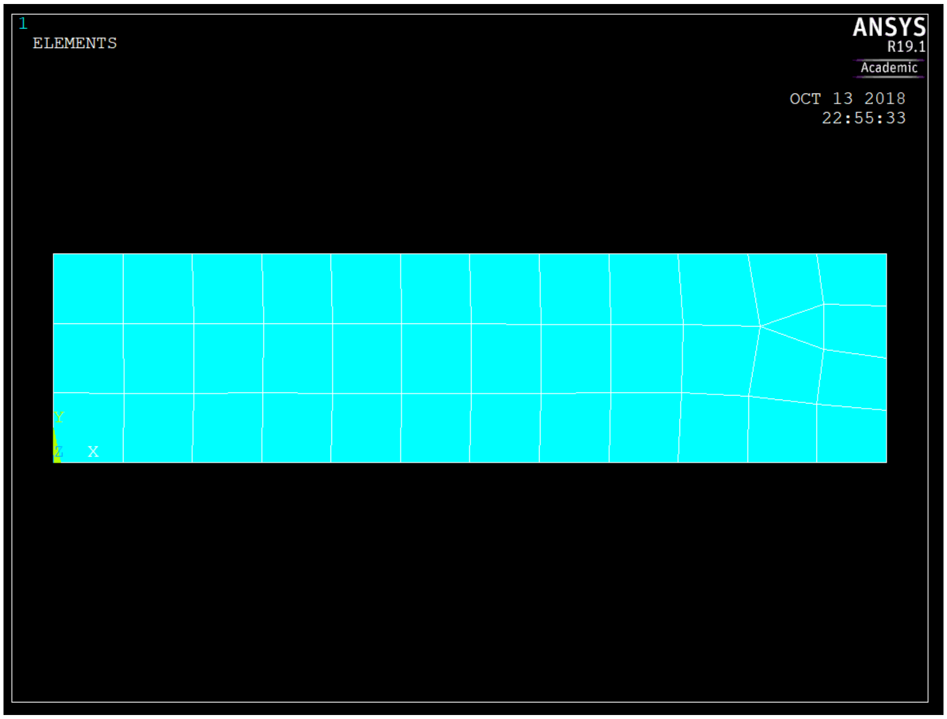

Figure 2: Element Size = 1.0

We see that in this case the mesh is not very well refined. The stresses σ_{xx}, σ_{yy}, and σ_{xy} as a function of distance along the beam are plotted in Figure 3 shown on the following page.

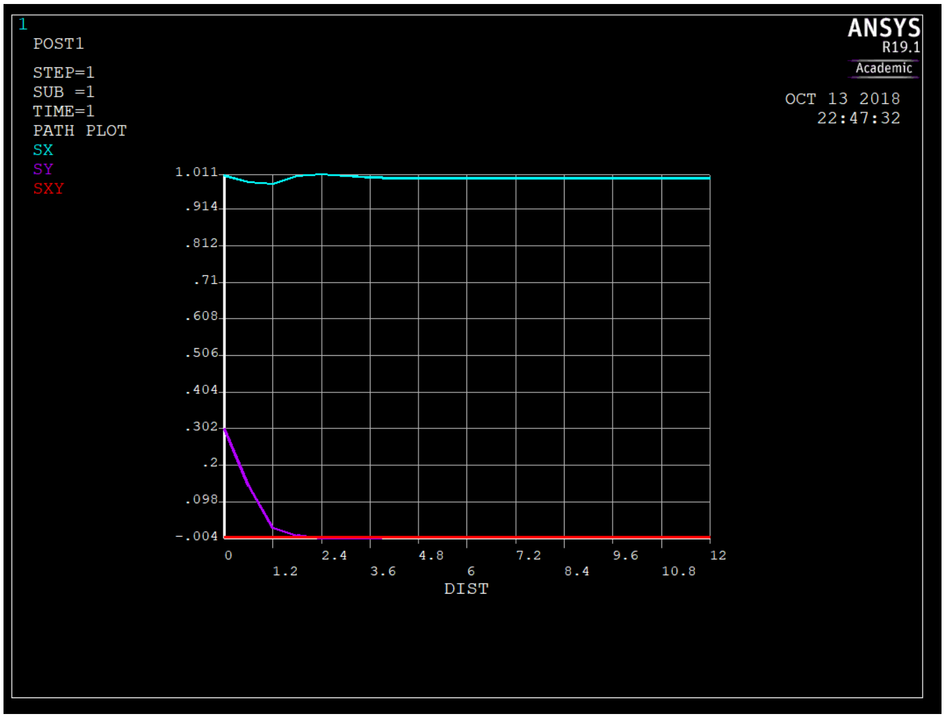

Figure 3: Stress Distribution for Element Size = 1.0

The model with element size = 0.5 is shown in Figure 4 below.

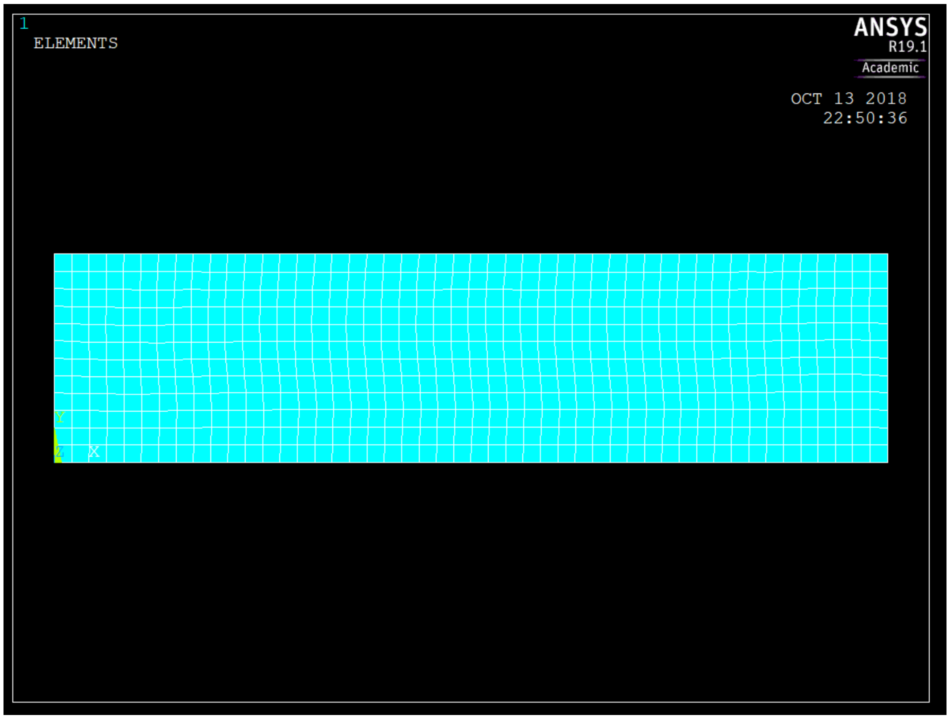

Figure 4: Element Size = 0.5

We would consider this to be a low to intermediate mesh size. The stresses σ_{xx}, σ_{yy}, and σ_{xy} as a function of distance along the beam are plotted in Figure 5 shown below.

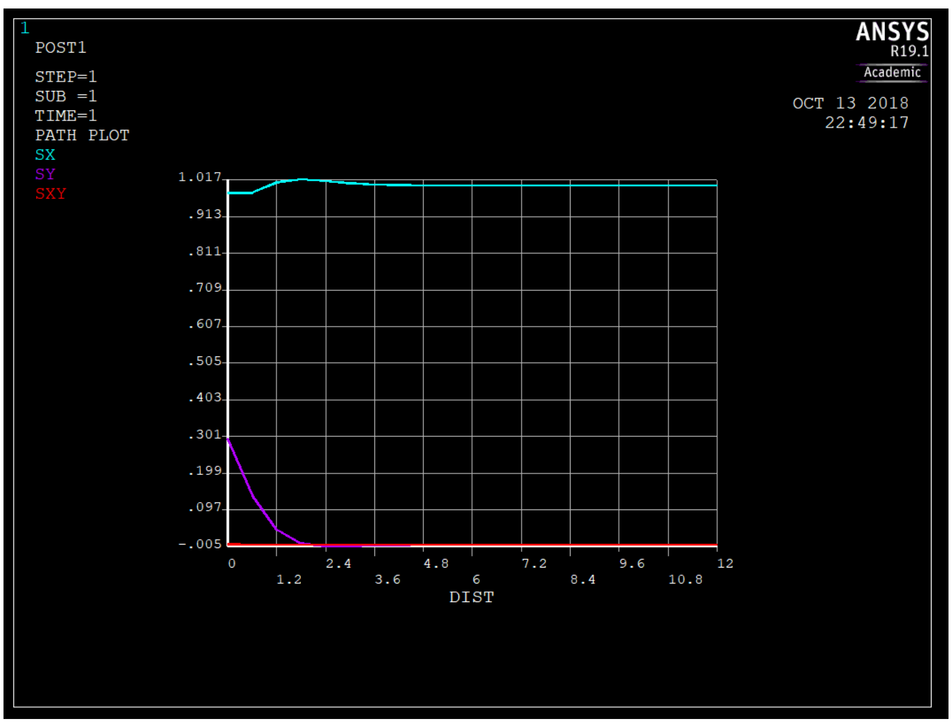

Figure 5: Stress Distribution for Element Size = 0.5

The model with element size = 0.25 is shown in Figure 6 below.

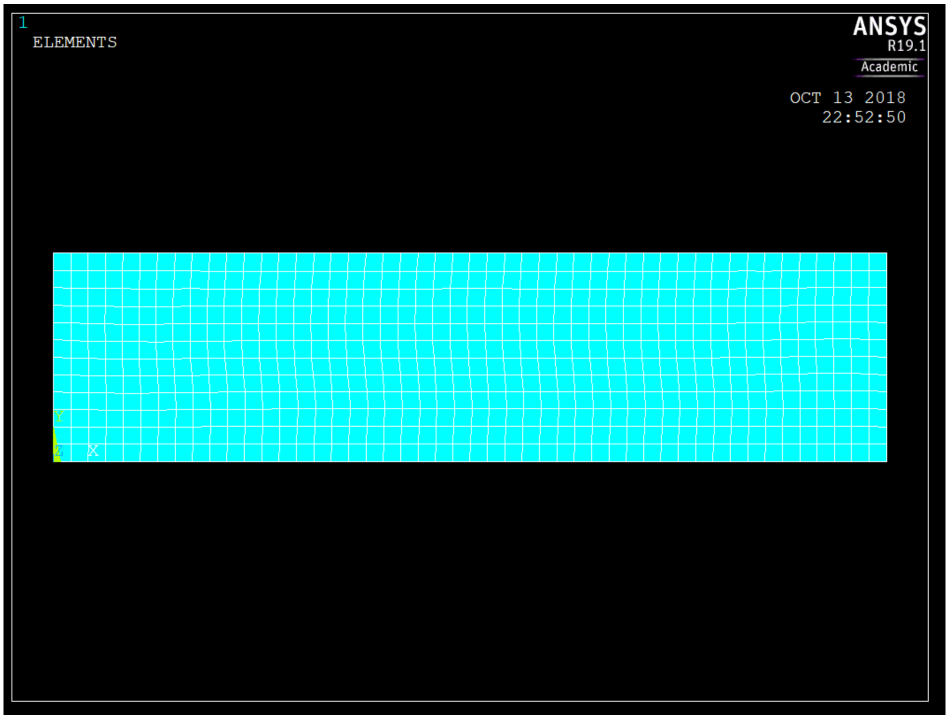

Figure 6: Element Size = 0.25

We would consider this to be an intermediate mesh size. The stresses σ_{xx}, σ_{yy}, and σ_{xy} as a function of distance along the beam are plotted in Figure 7 shown below.

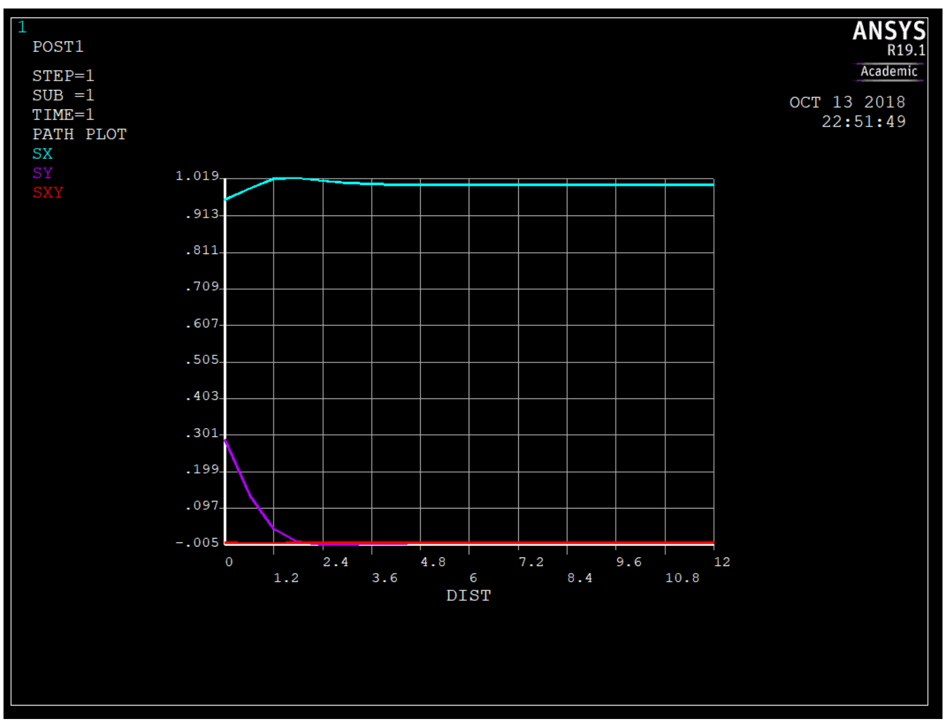

Figure 7: Stress Distribution for Element Size = 0.25

The model with element size = 0.1 is shown in Figure 8 below.

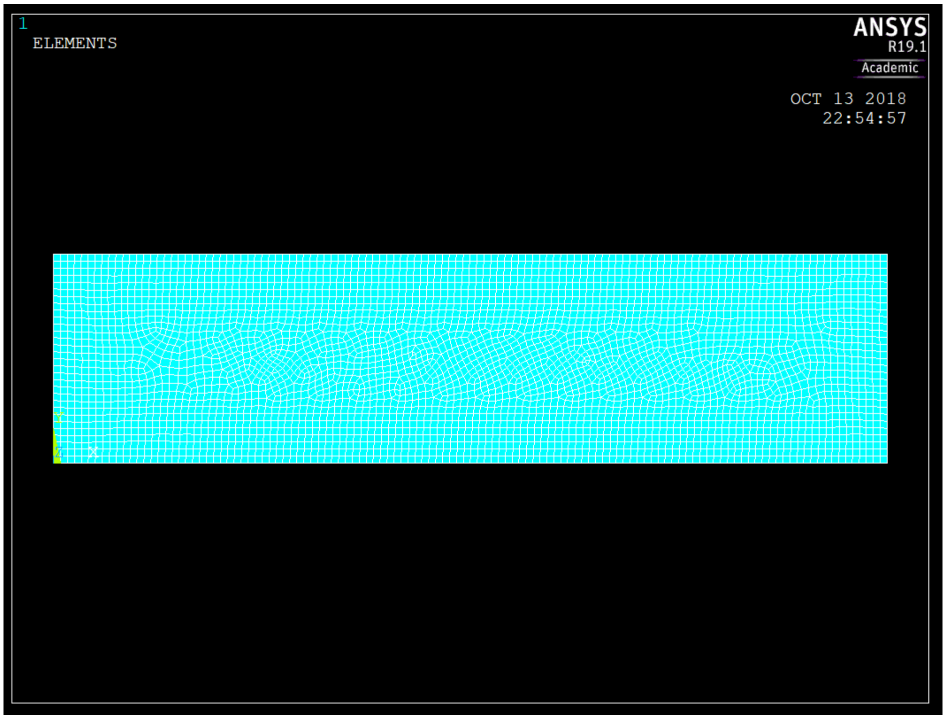

Figure 6: Element Size = 0.1

We would consider this to be a refined mesh size. The stresses σ_{xx}, σ_{yy}, and σ_{xy} as a function of distance along the beam are plotted in Figure 9 shown below.

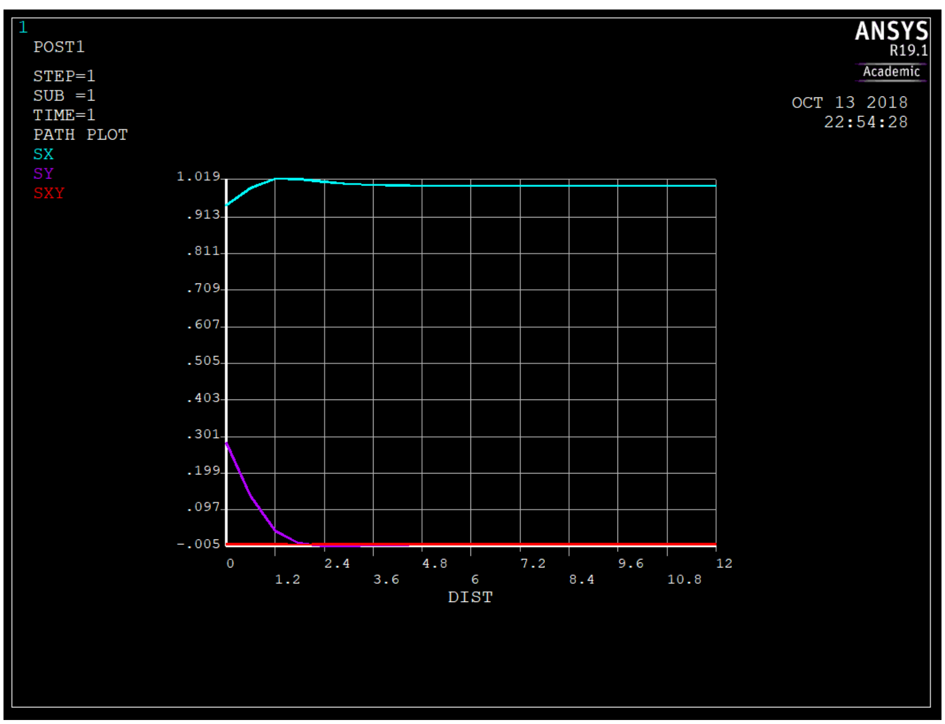

Figure 9: Stress Distribution for Element Size = 0.1

Discussion:

This section will discuss, in detail, the results obtained from our analyses and answer the questions as provided in the problem statement. We first note that the maximum value of stress occurs in the axial direction, namely σ_{xx}. This result is verified considering that we are only loading the model in the axial direction. We can see (from any of the stress distribution figures) that there is an average axial stress of

$$\sigma_{xx,avg} = 1.0\, psi$$

This result can be easily verified with hand calculations (although they have been omitted from this report). With poison's ratio present, we also expect to see a stress in the y-direction, σ_{yy}, given by the following equation:

$$\begin{bmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ 2\epsilon_{yz} \\ 2\epsilon_{zx} \\ 2\epsilon_{xy} \end{bmatrix} = \begin{bmatrix} \frac{1}{E_x} & -\frac{\nu_{yx}}{E_y} & -\frac{\nu_{zx}}{E_z} & 0 & 0 & 0 \\ -\frac{\nu_{xy}}{E_x} & \frac{1}{E_y} & -\frac{\nu_{zy}}{E_z} & 0 & 0 & 0 \\ -\frac{\nu_{xz}}{E_x} & -\frac{\nu_{yz}}{E_y} & \frac{1}{E_z} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{yz}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{zx}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{xy}} \end{bmatrix} \begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{yz} \\ \sigma_{zx} \\ \sigma_{xy} \end{bmatrix} \tag{1}$$

Equation (1) represents Hooke's Law in three-dimensions for an isotropic material. Let us consider Hooke's Law in two dimensions, i.e., the x and y directions. At the base, x = 0 in., we have constrained the model such that $\epsilon_{xx} = 0$ and $\epsilon_{yy} = 0$. Then, from (1), we get

$$-\frac{\upsilon}{E}\sigma_{xx} = \frac{1}{E}\sigma_{yy} \tag{2}$$

Thus, from (2)

$$-(0.3)(1) = \sigma_{yy}$$

$$\sigma_{yy} = 0.3\, psi\ (C)$$

which agrees with the figures. It is a compressive stress since the model is expanding in the axial direction and thus contracting in the transverse direction.

Let us consider the stress field associated with an element size equal to 0.1, given by Figure 9. We see that there is a non-uniform stress field near x = 0 in. As we move away from this point, we first see a spike in the axial stress followed by a uniform distribution of stress as we move towards the end of the beam, x = 12.0 in. There are several ways that we can explain this. One, using our boundary conditions; at x = 0 in., the model is constrained, and there is zero strain at this point. This result is given by equation (2). As we move away from the point x = 0 in., we allow strain in the x and y directions. We will assume that the strain (and stress) in the y-direction is very small considering the loading scenario. Then σ_{xy} will approach 0 and σ_{xx} will approach its average value of 1 psi. Two, we can say that there is a stress concentration at the wall, due to the constraint of our boundary conditions, that causes a spike in the stress near x = 0. These are the two most relevant possibilities.

We observe that the maximum stress (i.e., maximum axial stress) occurs with element sizes of 0.25 *and* 0.1. The maximum value occurs at x ≈ 1.2 in., with a value

$$\sigma_{xx,max} = 1.019\, psi$$

We note that as we increase our element size, our maximum stress increases. This agrees with our hypothesis made in the introduction that, as we increase our element size, we receive more accurate solutions and our model yields a higher stress. We also note that the maximum stress is the same for element sizes of 0.25 and 0.1. This phenomenon was explained in the introduction by the fact that, at some point, the elements will become small enough that the accuracy of the solution "plateaus" – i.e., at some specified element size, the elements have become infinitesimally small and we will receive the most accurate solution possible.

References

"Multiphysics Cyclopedia." COMSOL, 6 Jan. 2016, www.comsol.com/multiphysics/mesh-refinement.