

## ME362 Stress Analysis: Computer Assignment

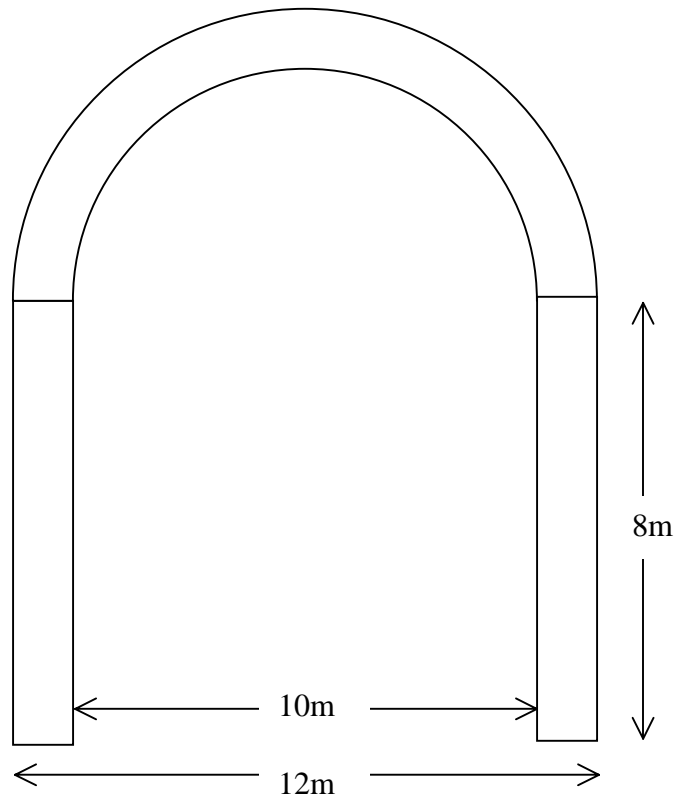
### I. Underwater Tunnel: [15 points]

Analyze the stresses and deformation in the underwater tunnel shown below. The loading on the tunnel can be thought of as uniform pressure on the outside walls, and fixed displacements at the bottom surface.

- (i) What plane approximation is suitable if the tunnel is very long?
- (ii) Pick a suitable material, and obtain the material properties (Google it!).
- (iii) If the tunnel is 1km below the water surface, estimate the pressure load on the tunnel.

Analyze the structure using Ansys.

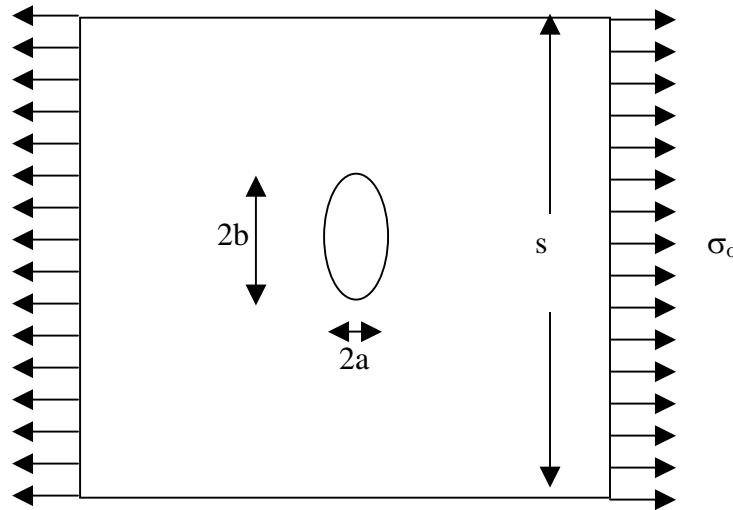
- (iv) Where does the normal stress take its largest value? (obtain this from a contour plot of the first principal stress)
- (v) Plot the deformed shape of the tunnel.



## II. Plate with an elliptical hole: [15 points]

Consider a very large thin square aluminum plate of sides  $s=1\text{m}$  containing an elliptical hole of major axis  $2b = 0.1\text{m}$  and minor axis  $2a=0.05\text{m}$ . The thickness of the plate is  $t = 0.001\text{m}$ . The plate is under a tensile load of  $\sigma_o = 10\text{MPa}$  as shown. Create a suitable Ansys model.

{Hint: You can create the ellipse using Preprocessor:Modeling>Create:Areas:Arbitrary Areas. You will have to think about this a bit.}



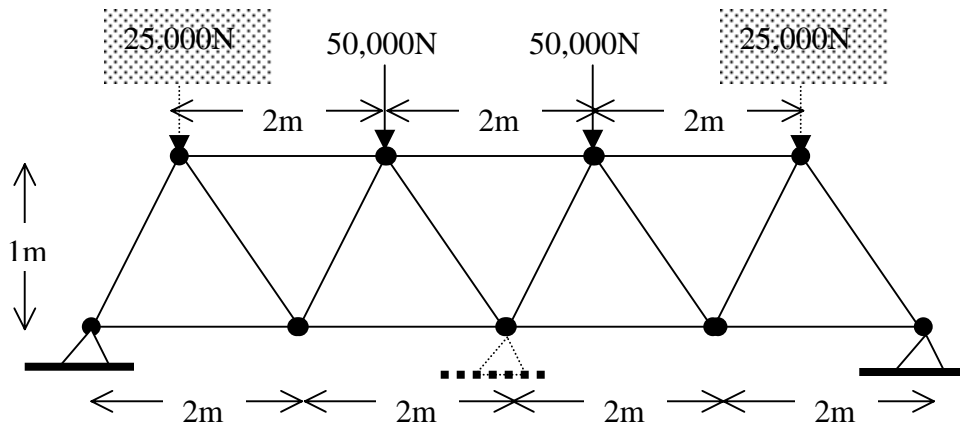
From your Ansys analysis, determine the largest stress near the major axis of the hole. There is an analytical solution to this problem, and it is known that the maximum tensile stress at the ends of the major axis of the hole is given by:

$$\sigma_{\max} = \sigma_o \left[ 1 + 2 \frac{b}{a} \right]$$

That is, there is a stress concentration in the vicinity of the hole, which elevates the maximum tensile stress to a level greater than in a uniform plate without a hole. Check whether this agrees with your Ansys solution.

Try ellipses with different aspect ratios ranging from  $b/a = 1$  to 5 and compare with the analytical result.

III. **Two-dimensional Truss:** Use Ansys to analyze the two-dimensional truss below:



Material=steel, cross-sectional area of all elements  $A=0.1\text{m}^2$ .

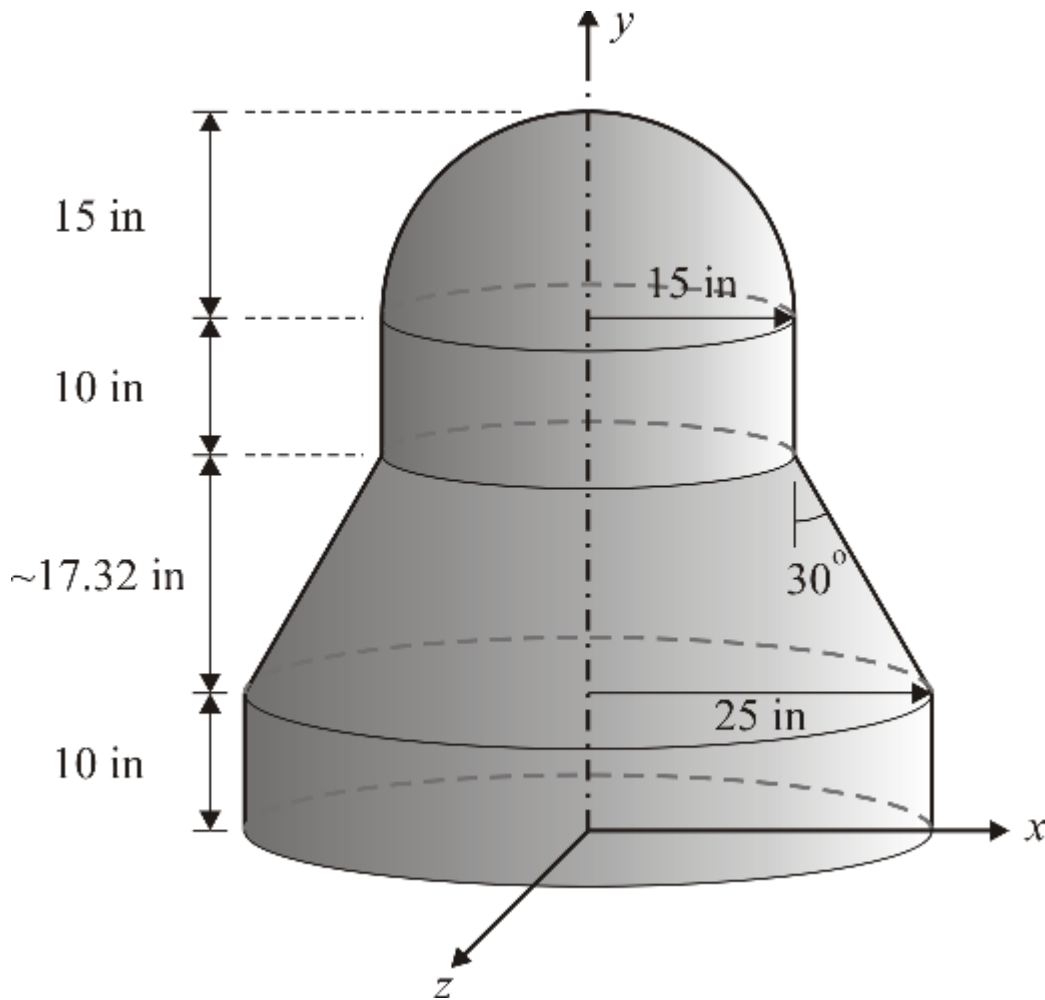
(i) [5 points] First consider the two dimensional truss shown in the figure above without the ghosted out mid-point support and without the ghosted out loads at both ends at the top. Determine the elements which have the largest tensile and largest compressive stresses.

(ii) [5 points] Upon learning that the truss is expected to carry an increased load (as shown by the ghost arrows), the truss designer panics and decides to add the ghosted out midpoint support. Modify the above Ansys program to solve this structure, and determine if the largest tensile and compressive stresses in any element exceed those in part (i). How would you ensure that the *stresses* in all the elements are no larger than those found in part (i)?

#### IV. Shells: [10 points]

An axisymmetric pressure vessel is shown in the figure. The thickness of the walls is  $t=0.25\text{in}$  everywhere. The internal pressure is  $300\text{psi}$ . The material has a Young's modulus  $E=10 \times 10^6\text{psi}$  and  $\nu=0.3$ . Use axisymmetric shell elements ("2D axisymmetric 51") for this problem.

Plot the deformed shape of the vessel.



From "The Finite Element Method and Applications in Engineering Using Ansys"  
Erdogan Madenci & Ibrahim Guven, Springer.