

ME 477  
Finite Element Analysis  
Fall 2007

**ANSYS Assignment #1**  
**Structural Analysis of a Transmission Tower using 3D Beam Elements**

Shown in the figures below is a transmission tower. The tower is designed to support electrical transmission cables at the ends of the four arms at the top of the tower, as well as wind loads that may be distributed over the entire frame. The tower is 60 ft high, 18 ft square at the base, 4 ft square at the top, and has 4 cable arms that extend out 6 ft from the main tower. The tower is constructed of round steel tubing ( $E = 30 \text{ Msi}$ ,  $\nu = 0.30$ ). The main vertical and horizontal members in the tower, and the cable arms, are constructed of tubing with an OD of 2.5 in and a wall thickness of 0.25 in. All diagonal bracing (triangulating) members are constructed of tubing with an OD of 2 in and a wall thickness of 0.25 in. The effective weight of the cables at the end of each arm is approximately 1,000 lb. In addition, the wind loads acting on the tower in any direction may reach levels as high as 10,000 lb, distributed over the entire tower.

Create an FE model of this tower in ANSYS using 3D beam elements (Beam4), and perform a structural analysis to determine the deflections and stresses in the tower under the anticipated cable and wind loads. For this analysis, model the cable loads as point forces at the end of each arm. Assume the wind loading is directed at an angle to the tower, producing a total force of 5,000 lb in the x-direction and 10,000 lb in the negative z-direction. Although the wind loads should physically be distributed over every structural member, it is permissible for this simplified analysis to distribute the wind loads at selected keypoints along the front and side surfaces of the tower. Assume the four base points of the tower are rigidly fixed to ground.

Submit a brief report describing the objectives of the analysis, the details of the FE model (i.e., assumptions and simplifications made, how and where the boundary conditions and loading were applied), and the results of the analysis. The results should include (at a minimum) the deformed geometry of the tower, a contour plot of the maximum stresses in the elements, and a summary of the maximum deflections and stresses (and their locations) in the tower. Also, provide some interpretation of the results; e.g., do the results seem reasonable (qualitatively or quantitatively), will the tower withstand the loading, what should be the minimum yield strength of the steel (assuming an appropriate safety factor), should any modifications be made to the tower to reduce deflections or stresses?

A convenient method for dividing a line into equally spaced increments is to use the following command:

**Preprocessor** → Modeling – Operate → Booleans – Divide → Line into N Ln's → Pick the Line → Click OK → Check Line No. and set NDIV to the desired number (Existing Line will be modified)

This operation divides the original line into smaller segments, and automatically defines new keypoints between each segment.



