# Wind Turbine Tower Design

ME-408 Midterm Project
November 6, 2017

*Pranav Joneja,*
*Mingyang Lee*
*Yueyue Li,*
*Jacob Maarek*

# Table of Contents

# Introduction

The objective of this project is to design a tower structure to support Enercon's upcoming on shore 9.25 megawatt (MW) wind turbine model. The goal for this design is to use Finite Element Analysis as a tool to optimize the tower structure for minimum cost, cross sectional area, and weight. Simulations are used to optimize and find the optimal tower frame geometry and member sizes.

# Design & Model Assumptions

1. All quantities are expressed in U.S. Imperial units (inches, pounds force, Fahrenheit)
2. The structure needs to be designed to withstand the following conditions
    a. Normal weather – operational
    Max wind speed of 60 mph. The structure should not deflect more than L/500 (12 inches).
    b. Extreme weather – non-operational
    Max wind speed of 98 mph. The structure should not deflect more than L/375 (16 inches).
    c. Damaged blade – operational
    In case one of the three blades is damaged during operation, the rotational imbalance will cause a harmonic excitation. The structure's natural frequency should be outside the range of 6-12 Hz to minimize the harmonic response.
3. The structure should withstand temperatures between -10°F and 110°F.
4. Assume the base of the tower structure will be mounted to a steel plate embedded in a concrete pile foundation. Still, the bolted flange connection at the bottom of the tower needs to be designed and simulated in detail.
5. Assume the ground is rigid. In effect, this means that it is safe to assume the tower will not sink into the ground.
6. Assume the guy wires are non-woven and have constant diameter. The effects of this assumption are discussed in the section on cable geometry later.
7. Assume the proposed design can be manufactured.

# Approach

## *Design Concept*

After studying other wind turbines tower structures, we used common features of them to inform our design concept.

Our design calls for a hollow, cylindrical tower with constant thickness. The bottom of the tower is supported by bolted connections through a flange with a fillet. The fillet helps to reduce stress concentrations.

For added support at the middle of the tower's height, there are four guy-wires with pre-tension for added support. The upper half of the tower (above the guy wire) is tapered to a smaller diameter at the top, while the lower half (below the guy wire) remains at a constant diameter. In effect, this means that the added support from the guy-wires allow a thinner tower structure, leading to significant reductions in weight.

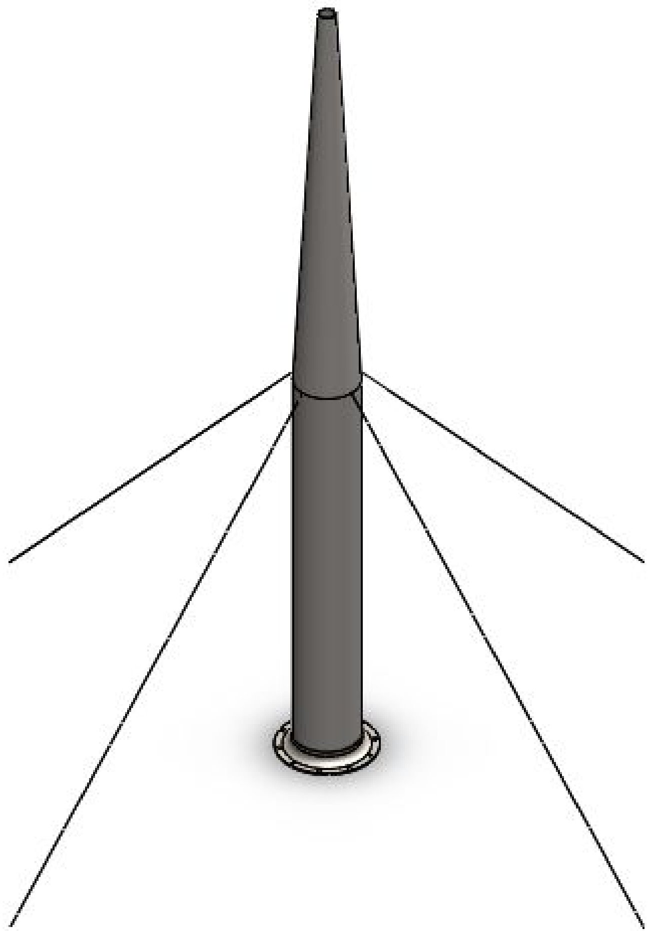

*Figure 1: Illustration of the conceptual design.*

## *Software Selection*

ANSYS Mechanical APDL was the Finite Element Analysis software package chosen for this design project. The benefit of APDL is that it allows more direct control over the element type and contact areas between them. The usefulness of this benefit is explained in the meshing section later. The drawback of APDL is that it is more difficult to draw and make changes to the geometry during the design project

# Material

The entirety of the tower structure and flange is made of A36 Structural Steel (low-carbon).

| | ***A36 Structural Steel*** |
|---|---|
| ***Young's Modulus*** *[psi]* | $2.9\times10^{7}$ |
| ***Poisson's Ratio*** | 0.26 |

| | |
|---|---|
| ***Density*** *[lbm/in$^3$]* | 0.284 |
| ***Coefficient of Thermal Expansion*** *[°F$^{-1}$]* | $6.5 \times 10^{-6}$ |

*Figure 2: Material properties input into ANSYS. All material data from ASTM.*

# Geometry

The dimensions of the tower structure had to be tweaked over several iterations. The figures below present the dimensions that were chosen in the end.

## *Tower*

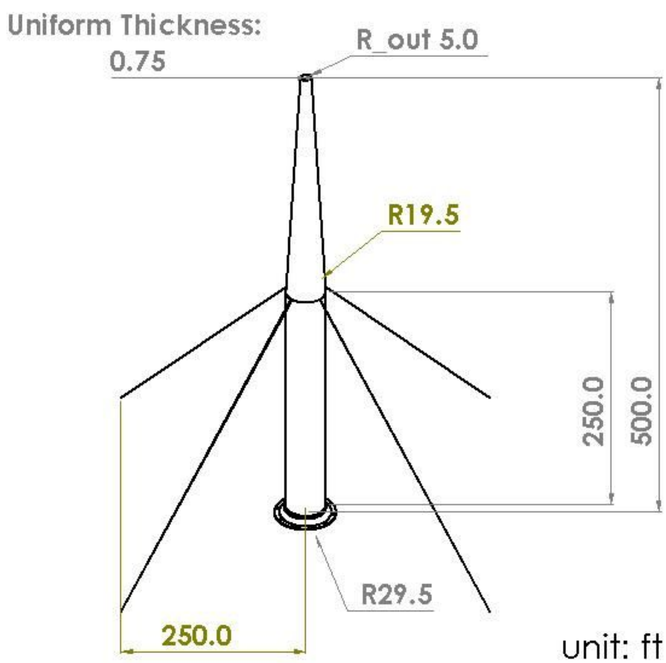


*Figure 3: Isometric view of the tower with dimensions in feet*

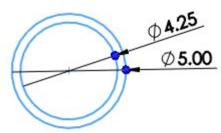


Figure 4: Cross-section of the top of the tower with dimensions in feet

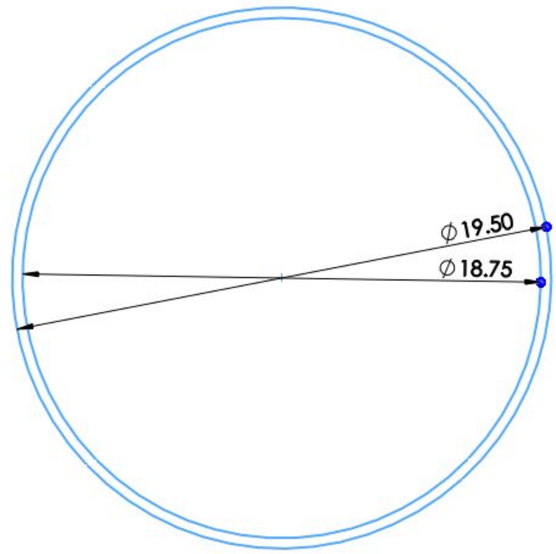


Figure 5: Cross-section of the bottom of the tower with dimensions in feet

## Flange

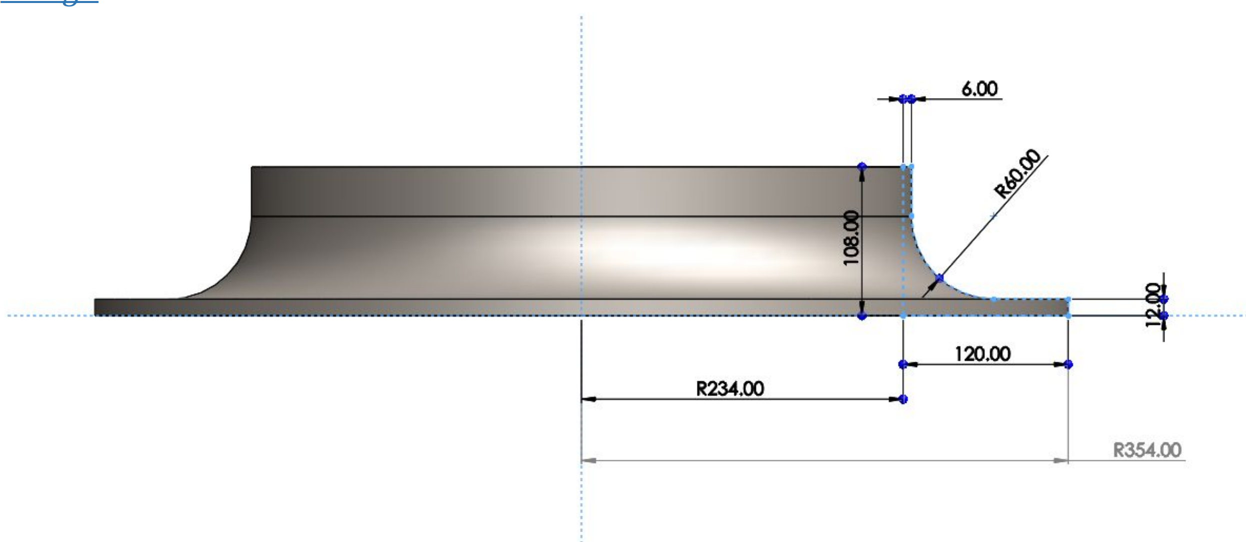


Figure 6: Side view of flange with dimensions in inches

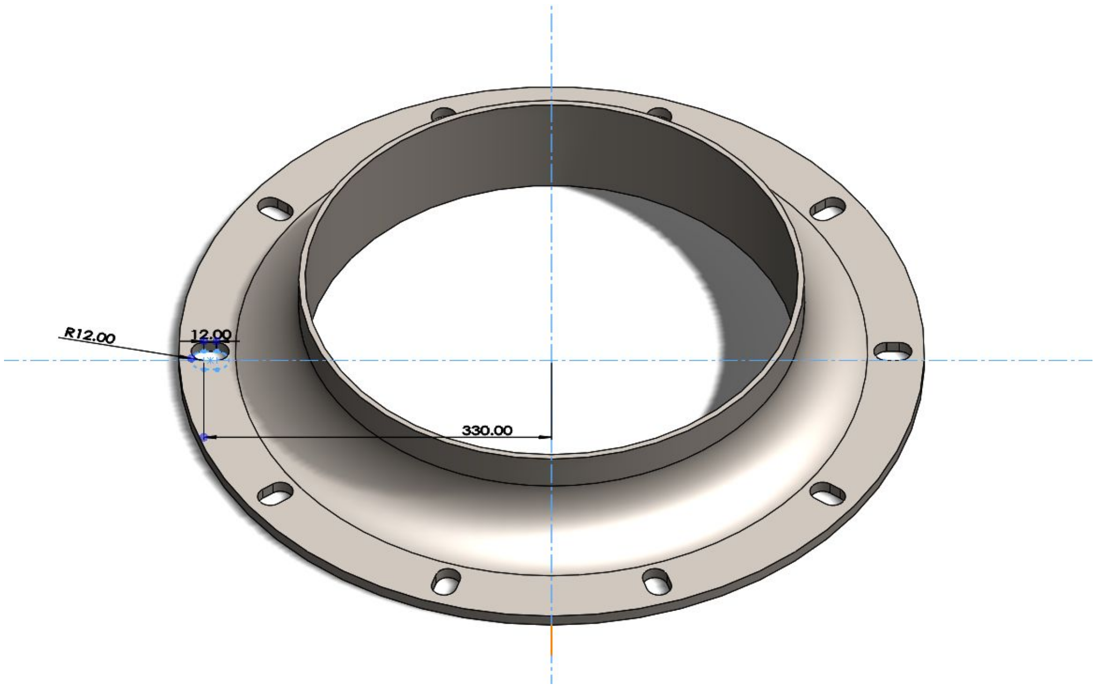


*Figure 7: Isometric view of flange showing dimensions of slots for bolts in inches.*

The slots are 12 inches long in the radially outward direction to facilitate thermal expansion. The flange is connected to the steel plate embedded in the foundation with 10 bolts of 6-inch radius.

### *Cable*

Each of the four cables has a constant diameter of 3 inches. The cross sectional area was chosen by looking at common structural support cables and their max loads.

The cables stretch out in the positive and negative x- and z-axes. The top nodes were placed at 250 feet up the tower to create a support high up the tower without interfering with the blades and the bottom nodes were placed 250 feet away from the center of the tower to create a 45 degree angle with respect to the ground. The reason for this placement is that we know that there cannot be any structures or interfering terrain directly below the turbine's blades. Since the blades are 250 feet long, the cables can also extend a maximum of 250 feet away from the tower.

The cables are modeled without consideration of the woven strands. In real life, it would be necessary to use woven steel cables, but it is not necessary to model the woven strands individually because the simpler, constant diameter model is sufficient to understand its behavior under stress. The simplified model also assumes the cables are tension-only link elements and does take into account its weight.

### *Total Weight*

The total weight of all components of the structure is $1.9\times10^{7}$ lbm

# Meshing

## *Tower*

The tower can be thought of as a thin plate rolled into a cylinder. As such, it is appropriate to model the tower with SHELL181 elements. The shell is given thickness using the lay-up feature, which calculates stress at 7 points through the thickness of the shell without adding a lot of computation time. The thickness was set to 9 inches throughout the tower.

The tower is split into two parts, the lower half with constant diameter, and the upper half with tapering diameter. For the lower half, the maximum element length was 48 inches, while for the upper half, the maximum element edge length as 24 inches.

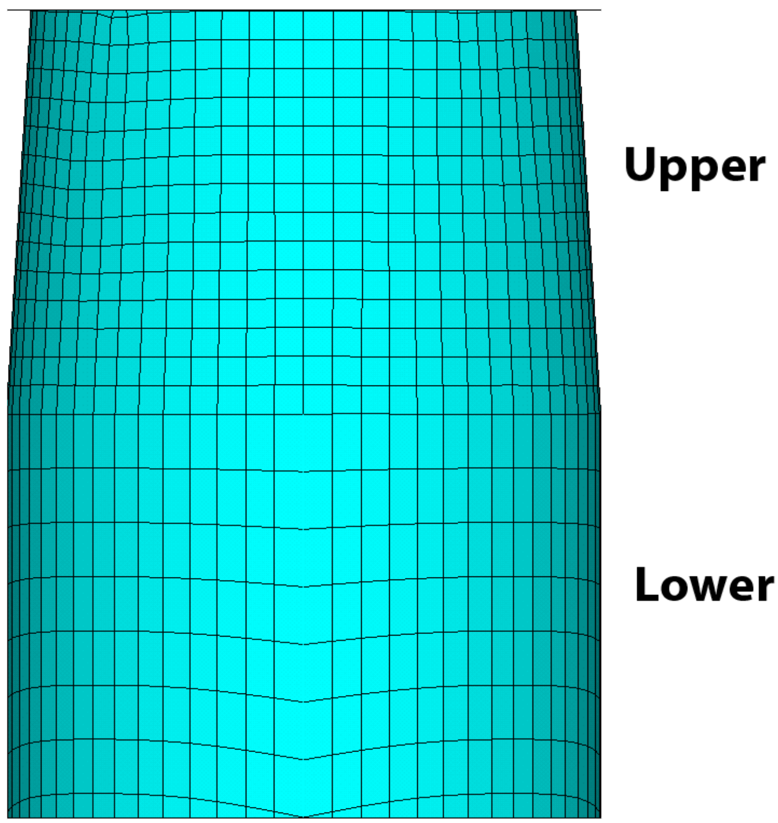


*Figure 8: A zoomed in portion of the tower to show the detail of the mesh*

## *Flange*

The flange is a solid, 3-D volume and so it is most appropriate to model it using SOLID185 elements. The thinnest portion of the flange, the upper lip (see Figure 6), is 6 inches thick. In order to allow for a minimum of 3 elements through this minimum thickness, the maximum element edge length is 2 inches.

The flange volume was strategically sliced into 2 types of meshes, each with 4 pieces, for a total of 8 pieces. The 4 filleted pieces were meshed using the Hexahedral Sweep method. This is ideal for making elements of uniform size, as well as minimizing the total number elements. Meanwhile, the 4 flat, slotted pieces were meshed using the Free method. This is more suited to refining the mesh around the bolt slots, which is necessary to get a good picture of stress at that location. To make sure the 2 types of meshes "talk to each other", the lines at the interface are forced to have 32 divisions each.

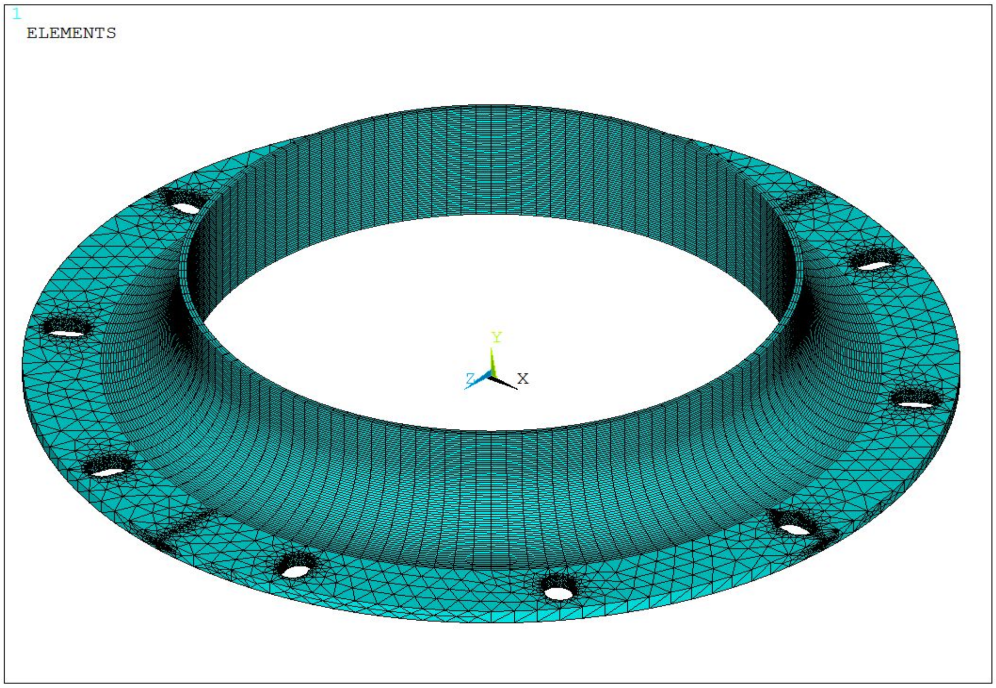


*Figure 9: Fully meshed view of the flange, showing the filleted pieces (inside) meshed using the Hex Sweep method and the flat, slotted pieces (outside) meshed using the Free method.*

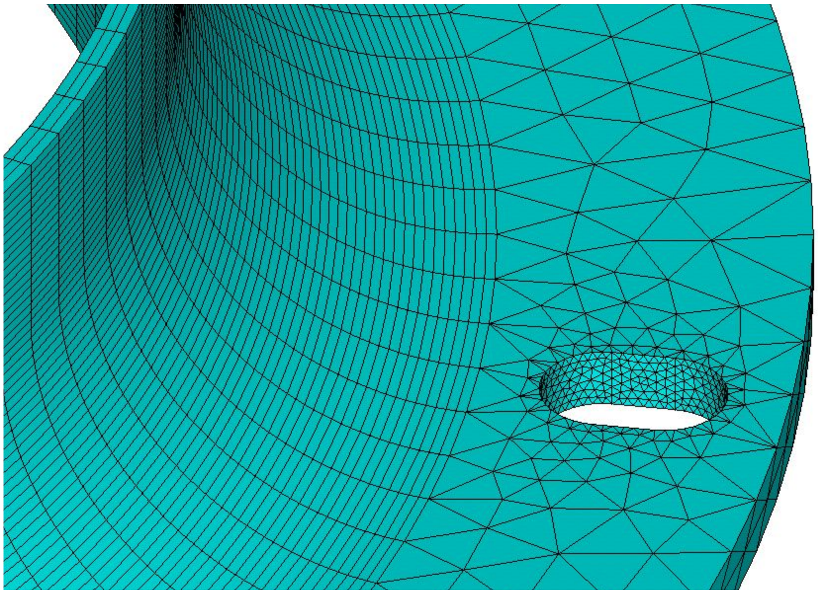

*Figure 10: Zoomed-in view of the slots, showing refinement of the free mesh around the slots. The interface between the swept and free mesh is also emphasized here.*

## Cables

The cables are modeled using LINK180 elements because this element type is well-suited to making tension-only elements. Each cable is a single element. This is appropriate because we do not need to compute stress in the cable with FEA because a simple hand calculation will suffice (see hand calculation section below).

## Summary

The table below summarizes the element types and where they were used to mesh and analyze the tower design.

| Element Type | Usage |
|---:|---|
| SOLID185 | For modeling the flange |
| SHELL181 | For modeling the tower |
| TARGET170 | Created by contact wizard to constrain contact surfaces between the flange and tower |
| CONTACT174 | |
| LINK180 | For modeling the cables |
| MPC184 | For applying boundary conditions at the top of the tower |

# Boundary Conditions & Loading

The boundary conditions (BC's) are the loading and constraints applied to the structure. The BC's include external loads, constraints between different parts of the structure and constraints between the structure and the ground. The BC's are categorized and presented in the sections below.

## *Flange/Bottom of Tower*

The bolts provide cylindrical supports at each of the 10 slots in the flange. To model this bolted connection effectively in APDL, the displacement of the inside surfaces (areas) of the slots were constrained in all degrees of freedom.

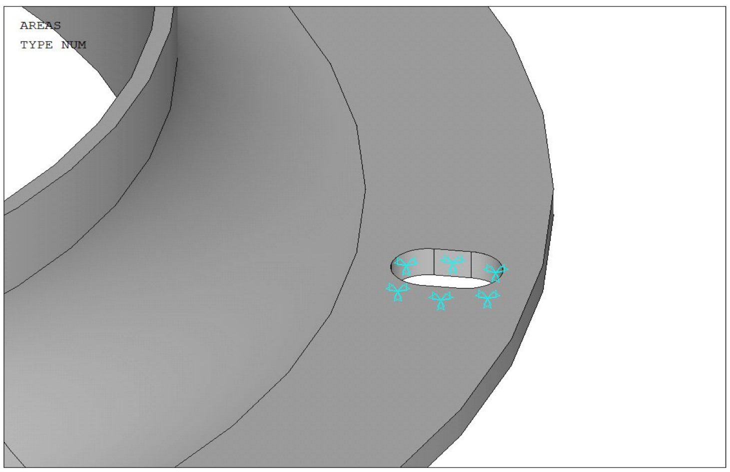


*Figure 11: All DOF displacement = 0 BC applied to the inside surface of the slots. The same BC is applied to all 10 slots.*

In addition to this, the entire bottom surface of the flange needs to be constrained too. This reflects the assumption that the ground is rigid This is modeled as a simple UY = 0 displacement constraint (simply supported).

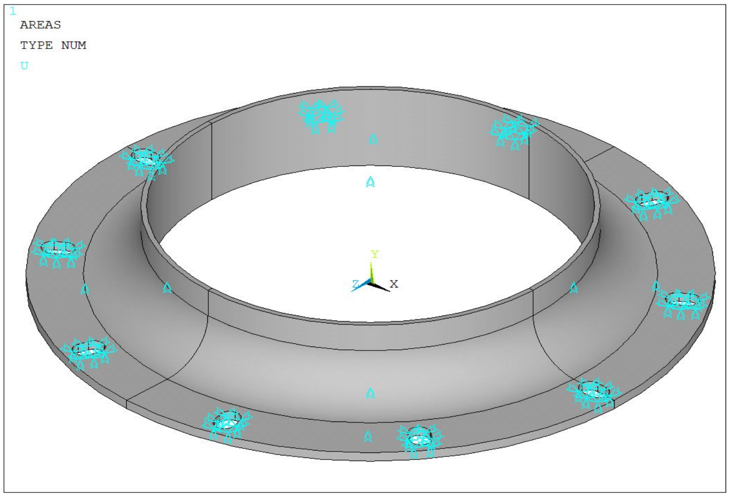


*Figure 12: Illustration of all boundary conditions applied to the flange.*

## *Contact surface between flange and tower*

In APDL, the pair-based contact wizard was used to model the connection between the flange and the tower. The contact was modeled as a surface to surface contact. The flange was chosen as the target volume and the outer surface area of tower as the contact surface. In the optional settings, the close gap option was chosen under the initial gap adjustment to stop rigid body motion. The wizard creates TARGE170 and CONTA174 elements between the two surfaces to constrain the two surfaces relative to each other.

## *Force at top of the tower due to wind pressure on blades*

The wind applies a pressure on the exposed area of the blades. Since the blades are connected at the center through the nacelle, the resultant force is applied at a single point in the x-direction at the top of the tower.

$$F_{blade} = P_{blade}A = \tfrac{1}{2}C_d\rho v^2(3\times l\times w)$$

$$F_{blade} = \tfrac{1}{2}\times 0.8(0.07656\tfrac{lbm}{ft^3})(88\tfrac{ft}{sec})^2(3\times 217\,ft\times 18.08\,ft) = -2.789\times 10^6\ lbf$$

It is important to note that this loading is only applied during the normal weather operational mode. During extreme weather, this loading does not apply because the wind turbine is not operational, so the blades are rotated away.

## *Pressure on tower area due to wind*

The wind also applies a pressure on the exposed area of the tower. For this calculation, the entire tower is considered to be a simple cylinder, with coefficient of drag, $C_d$ = 1.17 [Engineering Toolbox]

$$P_{tower} = \frac{1}{2} C_d \rho v^2 = \frac{1}{2} \times 1.17(0.07656 \frac{lbm}{ft^3})(88 \frac{ft}{sec})^2 (\frac{1ft}{12in})^2 = 2.41 \frac{lbf}{in^2}$$

## *Weight of Nacelle and Blades*

The weight of the nacelle causes a downward force of 185,000 lbf through the central axis of the tower. Meanwhile, the total weight of the 3 blades is 30,750 lbf, but the force is offset 20 feet parallel to the central axis of the tower. The moment due to the blades can be decomposed into a force couple system. In this case, a force through the central axis and a moment acting around the center of the circle at the top of the tower, calculated below:

$$F_{g,blades} = -185000\ lbf - 30750\ lbf = -215750\ lbf$$

$$M = F_{g,\ blades} \times d = -30750\ lbf \times 240 in = -7.38 \times 10^6\ lbf\ in$$

Since the central axis of the tower is empty (hollow cylinder), an MPC164 element was placed at the top to constrain the nodes along the circular edge of the top of the tower together.

## *Cables attached to tower and attached to the ground*

Each of the bottom nodes of the cables were pinned to the ground to constrain displacement but allow for rotation. A problem faced was that due to the initial relaxation of the tower due to gravity effects and loads, the cables were no longer in tension and thus did not accurately model the support loads on the tower. To compensate, we constrained the tower to have zero x-displacement for the two key points located on the z-axes. This would allow for rotation of the tower but more accurately pin the effect of the cables on the tower. While it is recognized that the cables would stretch slightly meaning that x-displacement would not be zero, we felt like it would give a reasonable approximation of the displacements and stresses in the tower.

## *Weight due to structure's own mass*

The model takes the structure's own weight into account with material density in the material model and enabling the inertia model with the gravitational constant, $g = 32.1 \frac{ft}{sec^2}$. The total weight of all components of the structure is $1.9 \times 10^7$ lbm.

## Operating Condition – Normal Weather Loading

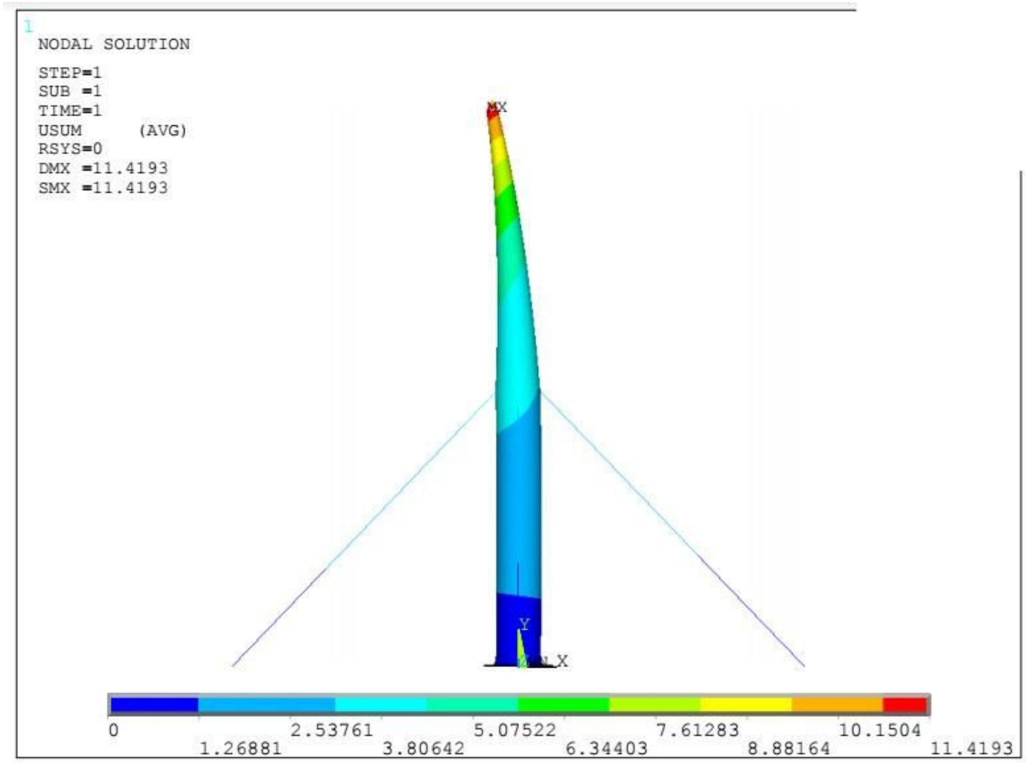


*Figure 13: Nodal deflection, units are inches*

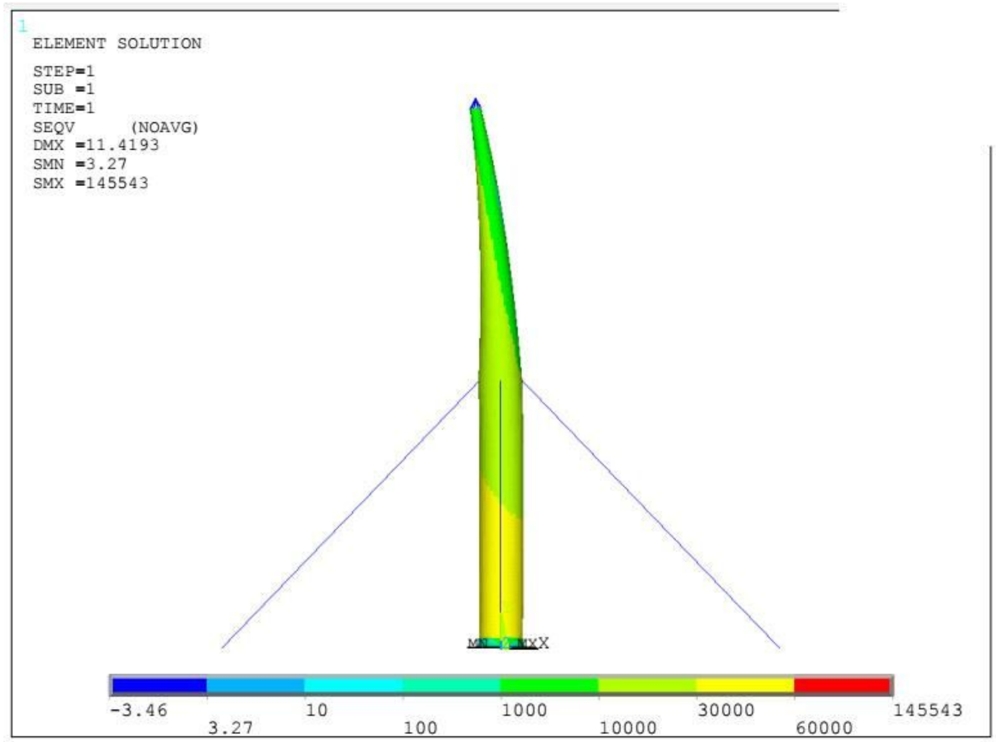


*Figure 14: Element stress, units are psi*

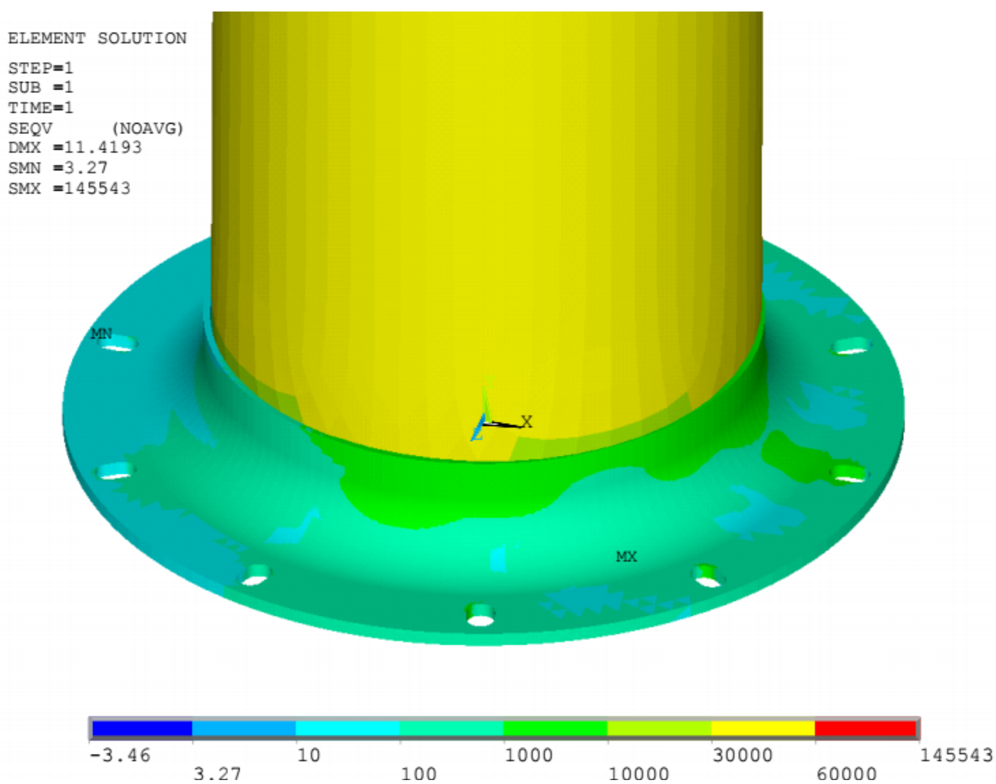


*Figure 15: Element stress at flange, units are psi*

### *Discussion – Normal Weather Loading*

Figure 13 shows the maximum nodal deflection is located at the top of the tower with a value of 11.4 inches in the negative x-direction. This meets the normal weather operation design criteria of ensuring maximum deflection is less than 12 inches.

Figure 14 shows the maximum element stress is located at the base of the tower, near the connection between the tower and flange. Figure 15 shows a zoomed-in view of the stress in this area. The maximum element stress value is 145,543 psi. This exceeds the yield strength and even above the ultimate strength of the structural steel proposed to make this part. A discussion of why the simulation returns this extremely high stress is included in the evaluation section.

Figure 15 also shows that the stress at the base of the tower (just above the connection to the flange) is in the range of 30,000 psi. This is the area with the second-greatest stress. This value is closer to the yield strength of structural steel (36,000 psi), but without any room for factor of safety.

# Non-operating Condition – Extreme Weather Loading

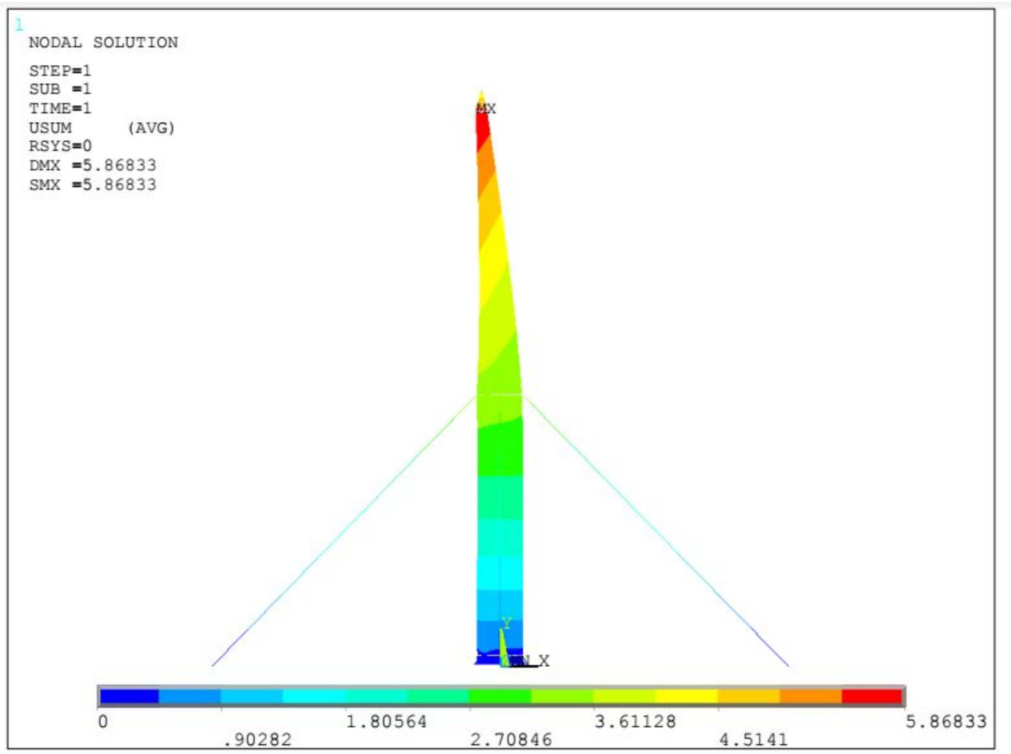


*Figure 16 Nodal deflection, units are inches*

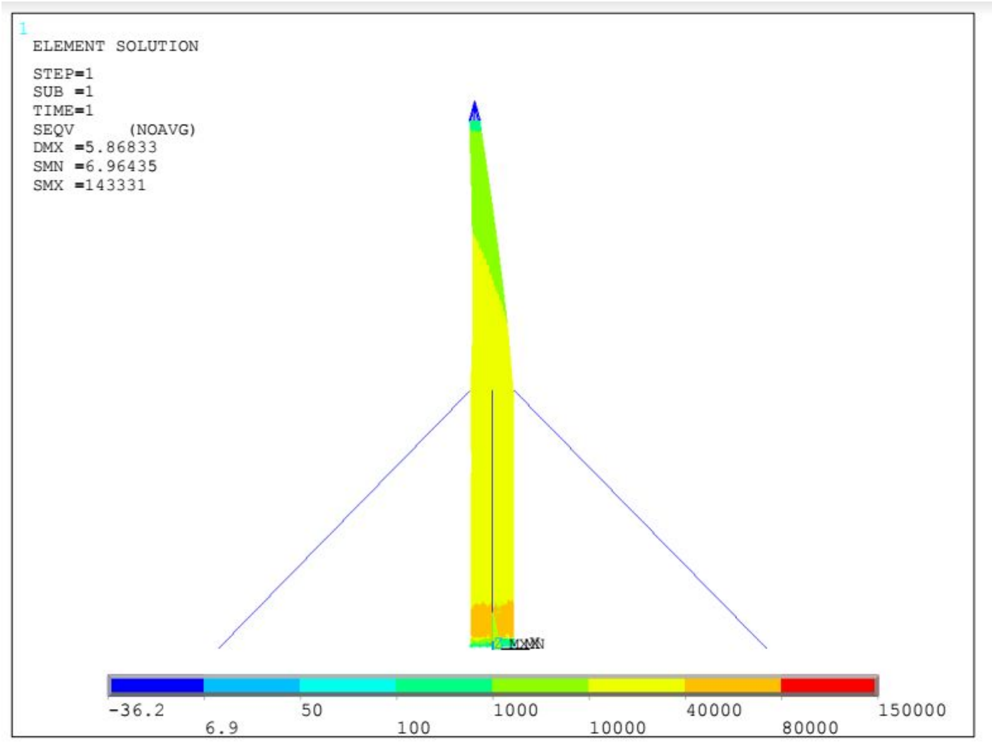


*Figure 17: Element stress, units are psi*

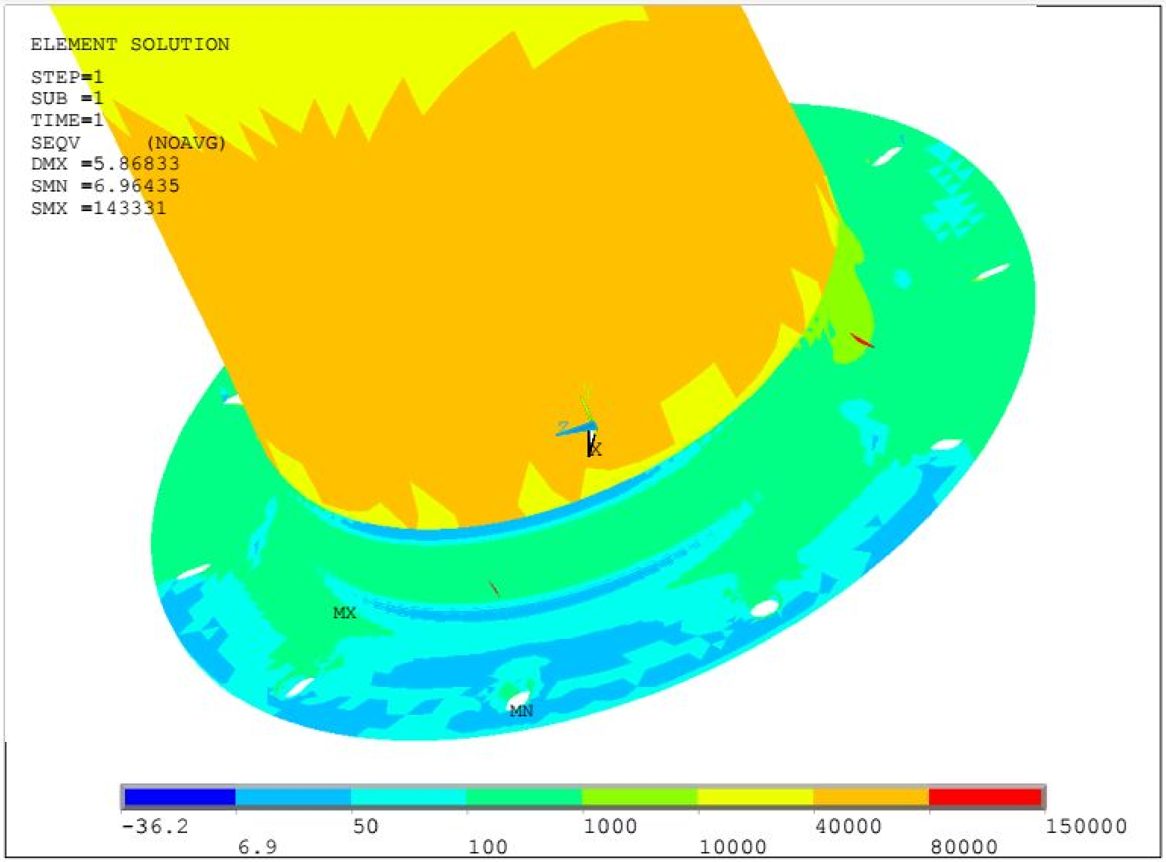


*Figure 18: Element stress at flange, units are psi*

## Discussion – Extreme Weather Loading

Figure 16 shows the maximum nodal deflection is located at the top of the tower with a value of 5.9 inches in the negative x-direction. This is lower than the maximum deflection under normal weather loading because the model assumes that the wind turbine is non-operational, and so the blades have been rotated away from the incoming wind (the force due to wind pressure on the blades is minimized). This maximum deflection meets the extreme weather design criteria of ensuring the maximum deflection is less than 16 inches.

Figure 17 shows the maximum element stress is located at the base of the tower, near the connection between the tower and the flange. Figure 18 shows a zoomed-in view of the stress in this area. The maximum element stress value is 143,331 psi. Once again, this exceeds the yield strength and even ultimate strength of structural steel. [Refer to discussion in the evaluation section of this paper].

Figure 18 also shows that the stress at the base of the tower, but above the connection to the flange, is in the range of 40,000 psi. This is the area with the second-greatest stress.

This value is still above the yield strength of structural steel (36,000 psi), but not exceedingly high stress.

## Modal Analysis

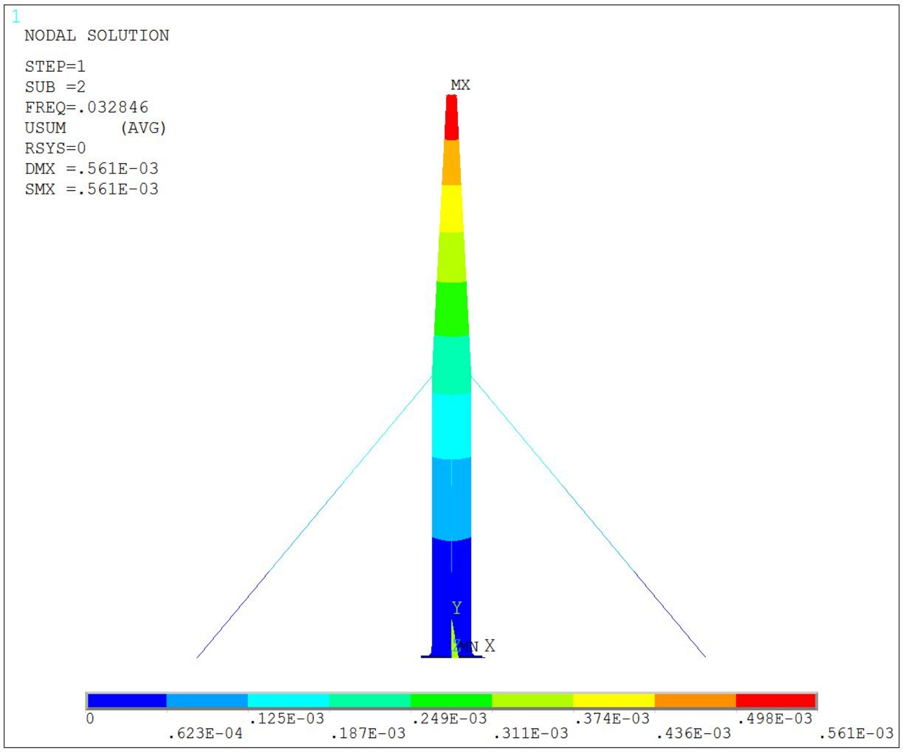


*Figure 19: Nodal deflection at the first mode of vibration, units are inches*

The table below details the first five modes of the system and their corresponding frequency

| Mode Number | Frequency [Hz] |
|---|---|
| 1 | 0.032 |
| 2 | 0.082 |
| 3 | 0.118 |
| 4 | 0.192 |
| 5 | 0.199 |

The analysis shows that the natural frequency of the proposed tower structure is 0.032 Hz. In comparison, the wind turbine operates in the range of 6 to 12 rpm or 0.1 to 0.2 Hz. Since the natural frequency is outside this range, it is safe to assume that harmonic excitation due to the normal operation of the turbine will not cause significant resonance. Vibration will not cause excessive deflection or strain.

# Buckling Analysis

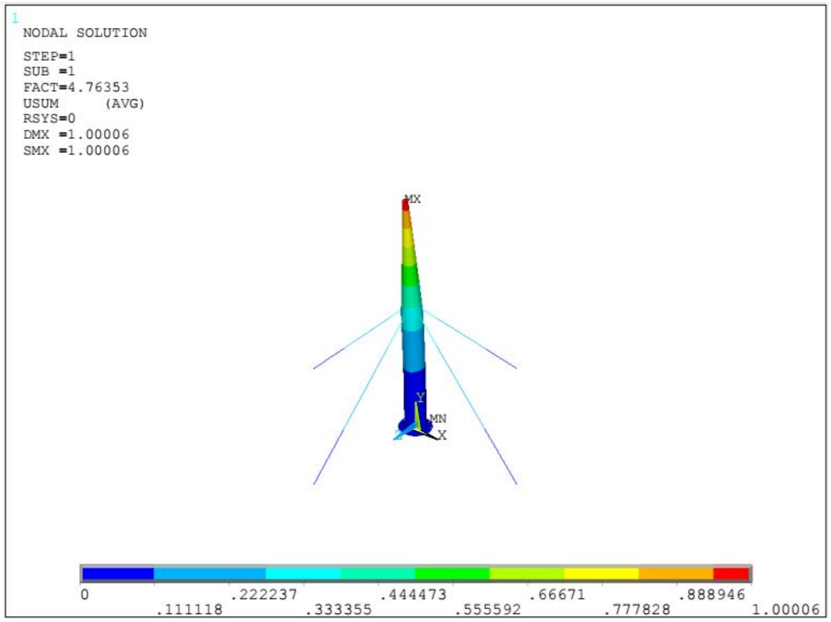


*Figure 20: First-order buckling analysis factor is 4.76*

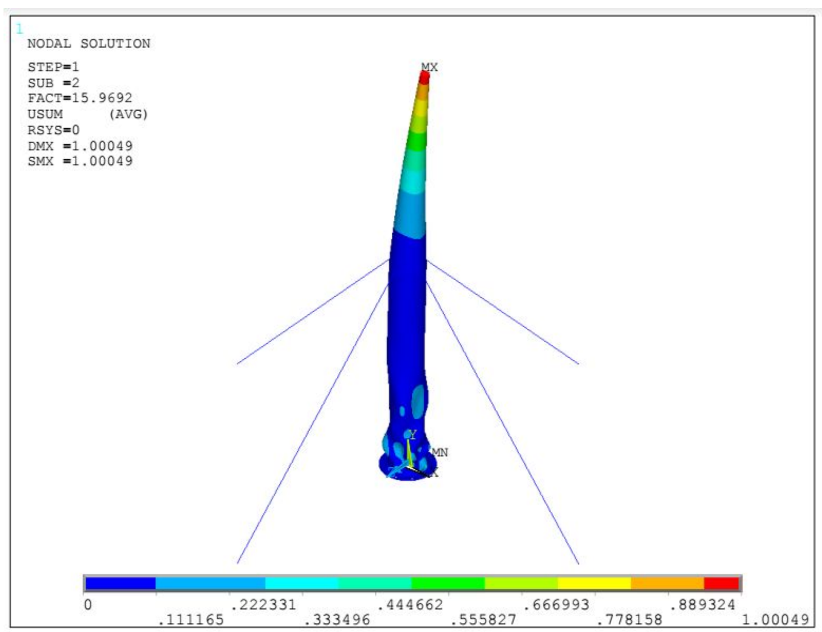


*Figure 21: Second-order buckling analysis factor is 15.97*

The loading applied to the model during eigenvalue buckling analysis included all axial loads (y-direction), including weight of the nacelle, blades and total weight of the tower structure itself. The loading for buckling analysis is the same for both normal and extreme weather conditions. The results show that the first factor of buckling is 4.76 and the second factor of buckling is 15.97. Since both of these values are greater than the safety factor of 1.75, it is safe to assume that the tower structure will not buckle under any axial loading that the tower can expect to face.

## Thermal Expansion

To model the thermal behavior of the proposed tower design, the material's coefficient of thermal expansion must be taken into account. For the A36 structural steel proposed for this tower, the thermal expansion coefficient is $6.5\times10^{-6}\ {}^{\circ}F^{-1}$ [AmesWeb Material Data].

In APDL, the reference temperature is set to -10°F and the universal temperature is set to 110°F. The nodal deflection results in Figure 22 show the structure's height is expected to increase by 4.7 inches vertically due to thermal expansion.

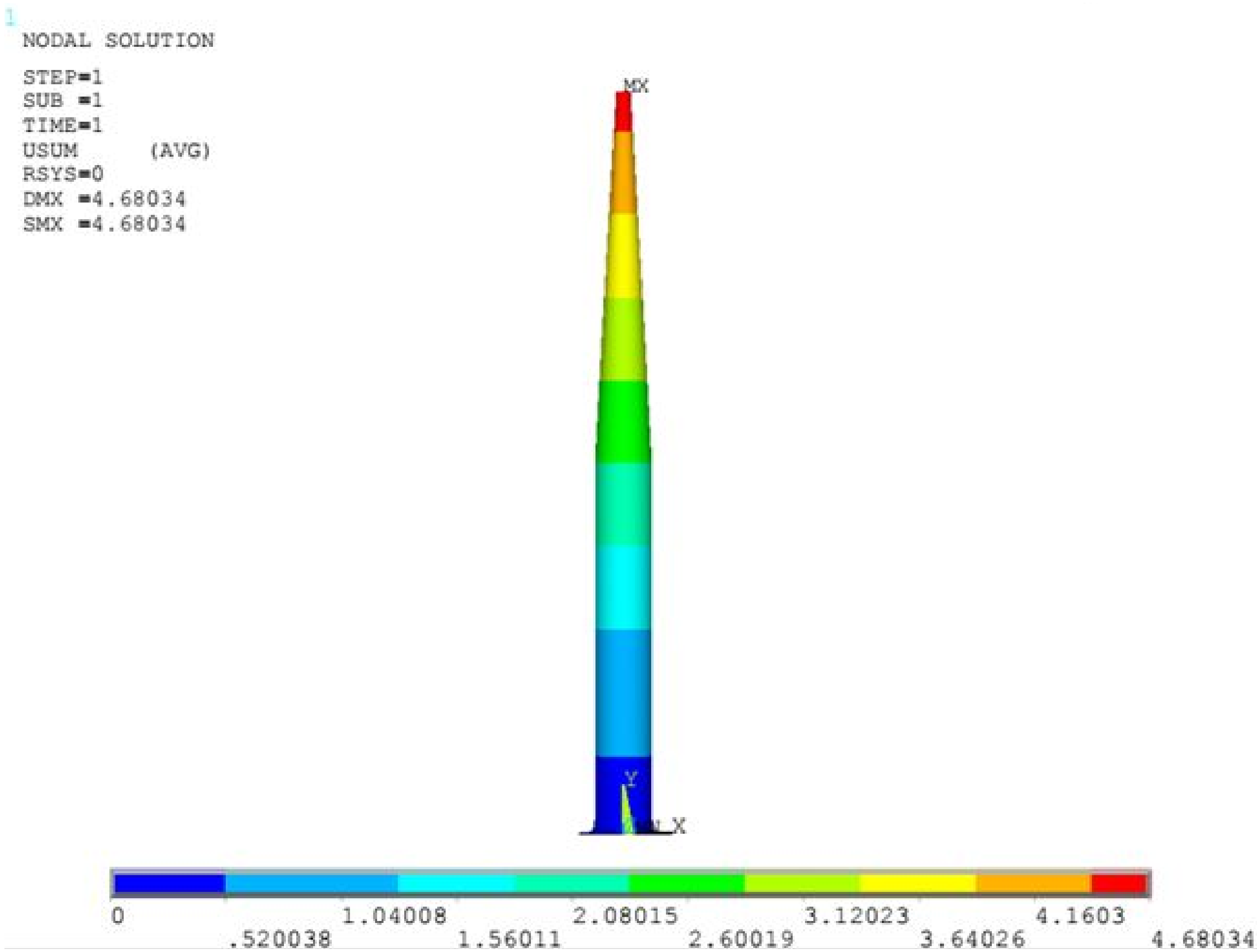


*Figure 22: Change in height due to thermal expansion from -10°F to 110°F. Units are inches*

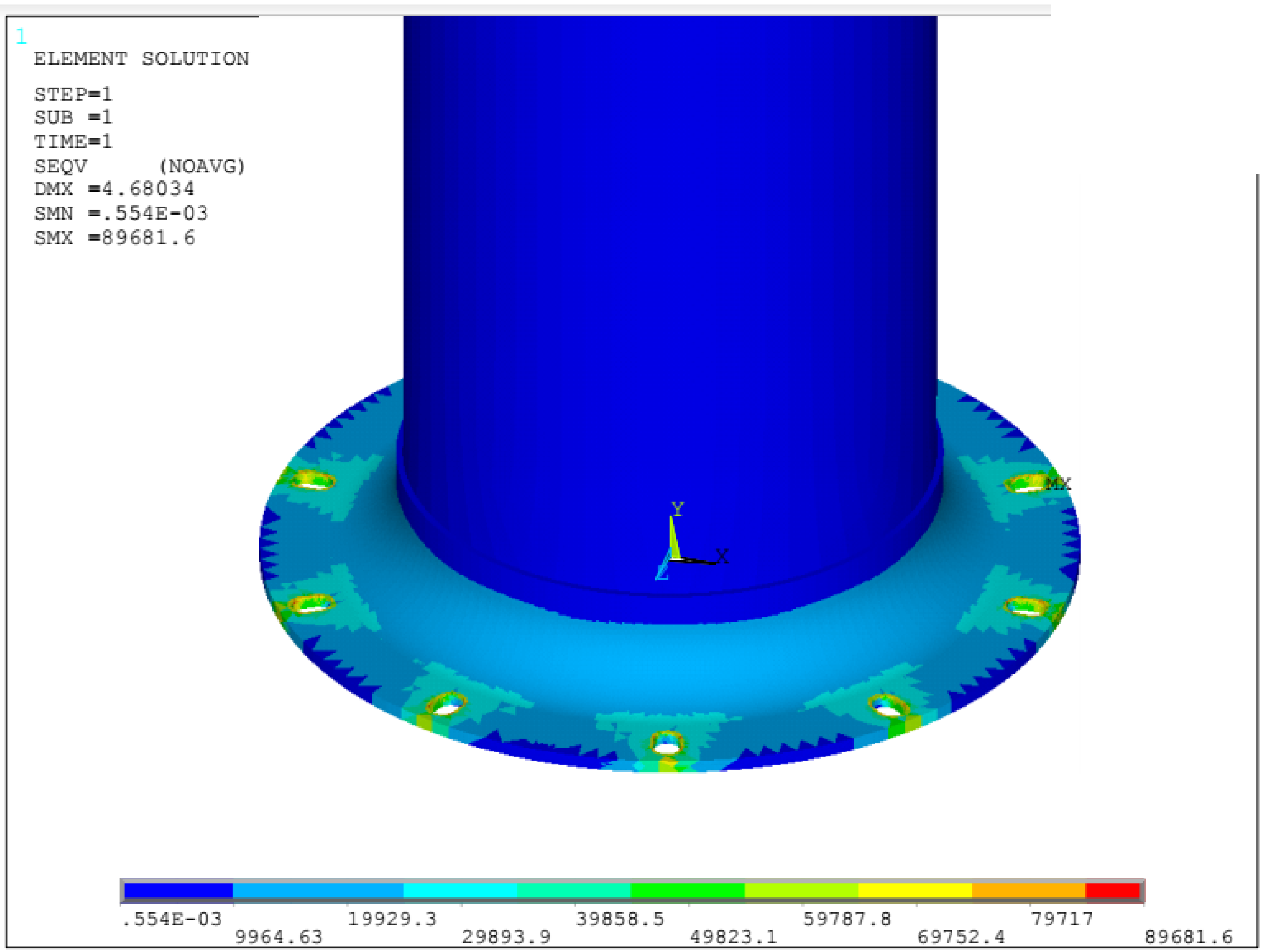


*Figure 23: Element stress in the flange due to thermal expansion from -10°F to 110°F. Units are psi*

The flange is designed to have a straight slot bolt hole, which allows the flange to expand radially outward during thermal expansion. The slot is 12 inches long, which is much longer than any anticipated expansion of the flange.

The element stress results in Figure 23 show the stresses in the flange reach a maximum of 90 ksi, which exceeds the ultimate strength of the steel. The shape of the stress distribution around the bolt holes shows that the model behaves correctly. However, it is hypothesized that this large magnitude of stress is <u>not</u> accurate, but is instead a result of over-constraining the model. In the model, the displacements of the inner surfaces of the bolt holes are constrained in all degrees of freedom. This is not a realistic or effective constraint. Instead, the correct way to constrain the flange is as follows:

- Constrain the 'straight' sections of the inner surface of the slot to zero displacement in the tangential direction (normal to these surfaces). (But, it is crucial to allow radial displacement of these surfaces)
- Constrain the surface in the immediate vicinity of the bolt holes to zero y-displacement, effectively modelling the reaction force due to the bolt.

- Constrain the whole bottom surface of flange to zero y-displacement (same as before).

## Mesh Sensitivity

One way to evaluate the accuracy of the results of the finite element analysis is to check the extent to which the model's mesh affects the results. In order to test the result's dependence on mesh, we increased the number of elements (decreased the maximum element edge length) and ran the same analysis again to compare results.

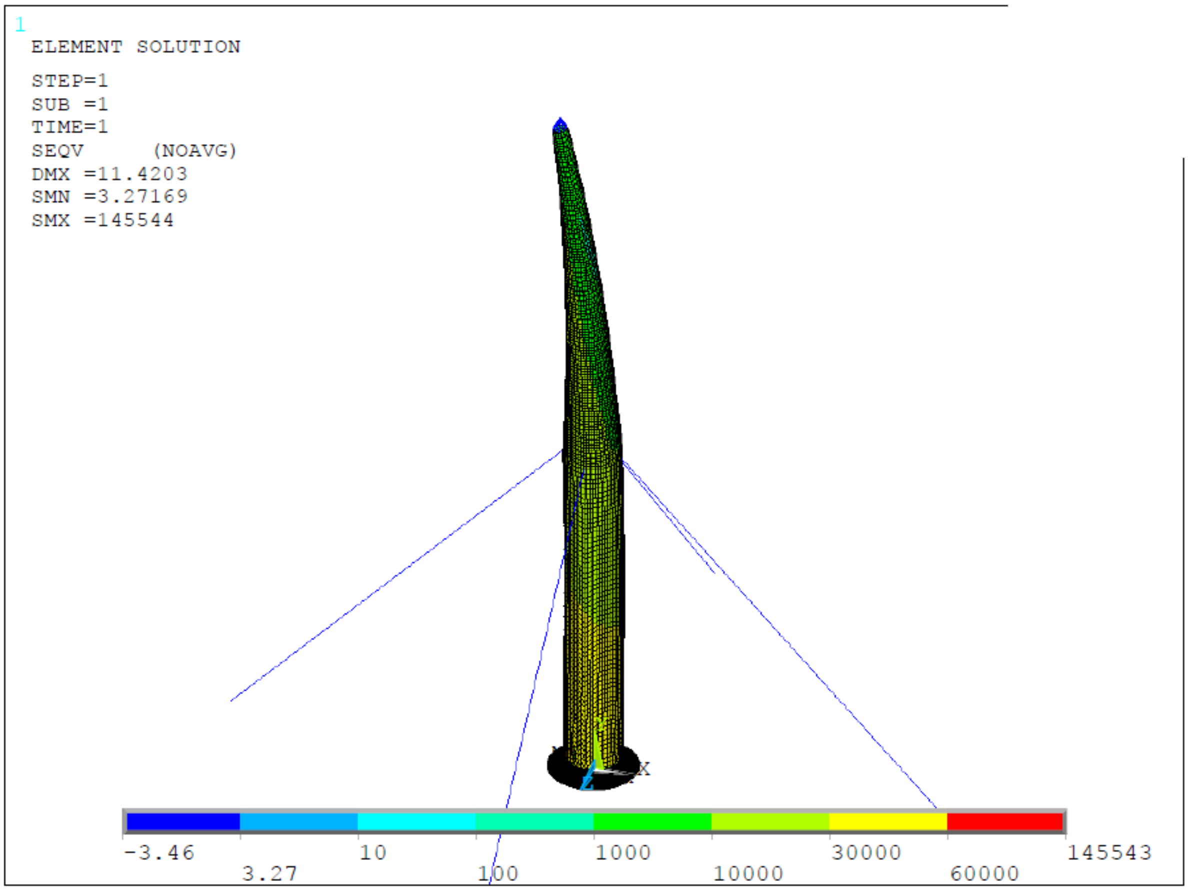


*Figure 24: Same model with increased mesh density (increased number of elements). Figure shows element stress in psi.*

The table below shows the comparison of key results of the normal weather loading condition on both models.

| | **Original Mesh** | **Finer/Denser Mesh** |
|---|---|---|
| **Max. Nodal Deflection [in]** | 11.4193 | 11.4203 |
| **Max. Element Stress [psi]** | 145,543 | 145,544 |

Since the results of both maximum nodal deflection and maximum element stress differ by less than 0.01%, we can conclude that the solution is not dependent on the mesh (or on the size of the elements). The original mesh sufficiently models the structure's behavior.

# Hand Calculations

## *Structural Analysis*

Hand calculations for structural analysis were done based on elementary beam theory and did not take into account the effect of the 4 cables. Also, during stress and deflection calculation, the tower is assumed to have uniform radius and thickness (simple hollow cylinder).

### *Loads*

For this hand calculation, it was assumed that the tower structure acts like a cantilevered beam. The input parameters, design constraints and geometry are detailed in the tables below.

| Design constrain | | | | |
|---|---|---|---|---|
| Height | 500 | ft | 500ft | |
| offset | 20 | ft | 20ft | |
| Weight | | | | |
| nacelle | 5749.9845 | lb | 185000 | lbf |
| blade | 318.58022 | lb | 10250 | lbf |
| Combined Weight | | 6705.72512 | lb | |
| Blade geo | | | | |
| ratio | | 12.00 | | |
| cg | | 0.333 | | |
| Cd_Cylinder | | 1.17 | | |
| Cd | | 0.8 | | |
| Operating range | | | | |
| 6 to 12 | | rpm | | |
| Wind Speed | | | | |
| Peak | 98 | mph | | |
| | 143.73366 | fps | | |
| Max | 60 | mph | | |
| | 88.0002 | fps | | |
| Temperature | | | | |
| min | -10 | F | | |
| max | 110 | F | | |
| Factor of Safety | | | | |
| 1.67 | for strength | | | |
| 4 | for connection | | | |
| Expansion Coefficient | | | | |
| 0.0000065 | | in/F | | |
| 5.41667E-07 | | ft/F | | |

| Design Geometry | | | |
|---|---|---|---|
| Outer Radius top | | 6 | ft |
| Outer Radius bottom | | 19.5 | ft |
| Thickness | | 0.75 | ft |
| Blade Length | | 200 | ft |
| Blade Width | | 16.66666667 | ft |
| V | | 45060.89063 | ft^3 |
| m | | 22503048.29 | lb |
| | A | 90.12178125 | ft^2 |
| | I | 16459.19494 | ft^4 |

| material property for stainless steel | | |
|---|---|---|
| Modulus | 4176000000 | lbf/ft^2 |
| Yeild | 4492800 | lbf/ft^2 |
| Ultimate | 10540800 | lbf/ft^2 |
| density | 499.392 | lb/ft^3 |
| | | |
| **Property for Air** | | |
| Density | 0.0765 | lb.ft^3 |

The second area moment of inertia for the ring cross-section:

$$I = \frac{\pi}{4}(D_{outer}^4 - (D_{outer} - t)^4) = \frac{\pi}{4}(19.5^4 - (19.5 - 0.75)^4) = 16488.5\, ft^4$$

Since the blades are 20 feet offset from the central axis, the weight of the blades is modeled as a moment applied on the center ring at the top of the tower. The moment from the blades is given by:

$$M = (3\times F_{g,blade})\times 20 = 615000\ lbf$$

Next, the wind pressure on the tower itself is calculated for each loading condition (normal and extreme weather) for each part of the tower structure ( $D_{bottom,outer} = 19.5\,ft$, and $D_{top,outer} = 6\,ft$ )

A sample calculation for wind pressure at the bottom due to normal loading condition:

$$P_{wind,tower} = \tfrac{1}{2}C_d\rho v^2\tfrac{D}{12} = \tfrac{1}{2}(1.17)(0.0765\tfrac{lbm}{ft^3})(88\tfrac{ft}{sec})^2\,(\tfrac{19.5}{12}ft) = 563.2\tfrac{lbf}{ft^2}$$

Summarized in the table below:

| Wind Pressure | | | |
|---|---|---|---|
| During peak wind speed | Lower part | 1502.407 | lbf/ft^2 |
| | Upper part | 462.2791 | lbf/ft^2 |
| During normal operation | Lower part | 563.168 | lbf/ft^2 |
| | Upper part | 173.2825 | lbf/ft^2 |

The total force on the top of the tower due to wind pressure applied to the blades:

$$F_{blade} = P_{blade}A = \tfrac{1}{2}C_d\rho v^2(3\times l\times w)$$

$$F_{blade} = \tfrac{1}{2}\times 0.8(0.07656\tfrac{lbm}{ft^3})(88\tfrac{ft}{sec})^2(3\times 217\,ft\times 18.08\,ft) =-2.789\times 10^6\ lbf$$

## *Deflection*

Tower deflection is calculated using elementary beam theory for a cantilevered beam. The total deflection is the summation of the deflections caused by each different load component, the wind pressure, blade moment, and wind force. During peak wind speed (non-operational), we assumed that the blade could be adjusted such that a minimum force will be applied on the blade. Meanwhile during operation, the assumption was that all of the wind force on blades is included in the calculation of the tower deflection.

$$\delta_{working} = \frac{w_{top}(\frac{l}{2})^4}{8EI} + \frac{w_{btm}(\frac{l}{2})^4}{8EI} + \frac{P_{blade}(l)^3}{3EI} + \frac{M_{blade}(l)^2}{2EI}$$

| Total Deflection | | |
|---|---|---|
| Operation | 1.486451276 | ft |
| Extreme Weather | 0.759471308 | ft |

### *Bending Moment*

There are three individual bending moments applied on the tower. The moments are applied around the point at the center of the base of the tower (where the central axis meets the ground).

| Bending Moment during operation | | |
|---|---|---|
| From the weight of the blade | 615000 | lbf-ft |
| From wind force on the blade | 1184837386 | lbf-ft |
| From wind pressure on the tower | 23014077.7<br>3 | lbf-ft |

### *Maximum Stress*

From elementary beam theory:

$$\sigma = \frac{Mc}{I} = (615000 + 1184837386 + 23014077.73) * \frac{19.5}{16488.5} = 7.30\times10^{4}\, lbf/ft^{2}$$

### Frequency/Modal Analysis

The natural frequency of the tower is given by:

$$f_n = \frac{1}{2\pi}\sqrt{\frac{3EI}{(0.23*m_{tower}+m_{nacelle})L^3}} == 0.089$$

### End Constrained Buckling Analysis

The critical load on the tower before buckling is given by

$$P_{cr} = \frac{\pi^2 EI}{(2L)^2} = 8.8710\times10^{8}\, lbf$$

### Thermal Expansion Analysis

The thermal expansion coefficient of structural steel is $\alpha = 6.5\times10^{-6}\,{}^{\circ}F^{-1}$. The low temperature -10°F is taken as the baseline and the high temperature 110°F is the applied universal temperature. The change in the tower height and radius is calculated and summarized in the table below. A sample calculation for tower height is also shown.

$$L_{expanded} = L_0\times(\alpha\times(T_{high} - T_{low}) + 1)$$

$$L_{expanded} = (6000\ in)(6.5\times10^{-6}\,{}^{\circ}F^{-1}\times(110{}^{\circ}F - (-10{}^{\circ}F)) + 1) = 6004.68\ in$$

| Thermal Expansion | | |
|---|---|---|
| Radius at -10F | 234 | i<br>n |
| Radius at 110F | 234.18252 | i<br>n |
| Height at -10F | 6000 | i<br>n |
| Height at 110F | 6004.68 | i<br>n |

Thus, the total change in height according to hand calculation is 4.68 inches.

### *Comparison between hand calculation and simulated result*

The table below compares key values under normal weather loading condition only.

| Key Value | Hand Calculation | Finite Element Analysis Simulation |
|---|---|---|
| Max. Deflection [in] | 17.76 | 11.4193 |
| Max. Stress [ksi] | 73 | 30 |
| Natural Frequency [Hz] | 0.09 | 0.03 |
| Height change due to thermal expansion [in] | 4.68 | 4.68 |

The results for the change in height due to thermal expansion are exactly the same. Meanwhile, the results for maximum deflection, maximum stress and natural frequency are within the same order of magnitude. The hand calculations for these key values verify the corresponding FEA results. This agreement of key results between the hand calculation and FEA model shows that the model is well-conceived and behaves according to theory.

# Key Results

All key results are from the FEA analysis only.

## *Size, mass and footprint*

| Property | Value | Units |
|---|---|---|
| Total Mass | $1.9\times10^{7}$ | lbm |
| Cross-sectional area at bottom of flange | 2776 | $ft^2$ |
| Cross-sectional area at base of tower | 90.12 | $ft^2$ |
| Cross-sectional area at top of tower | 26.51 | $ft^2$ |
| Total installed footprint (land area with guy wires) | 196,349 | $ft^2$ |

## *Loading Response of Proposed Design*

| | Normal Weather Loading | Extreme Weather Loading | Units |
|---|---|---|---|
| **Max. Deflection** | 11.4 | 5.9 | in |
| **Max. Stress** | 143 | 145 | ksi |

## *Overall Performance Characteristics of Proposed Design*

| Property | Value | Units |
|---|---|---|
| Natural Frequency | 0.03 | Hz |
| Factor of safety against buckling | 4.76 | - |
| Height change due to thermal expansion | 4.68 | in |

# Evaluation

## *Stress in the Model*

The element stress results are shown in this report on Figures 14-15 and Figures 17-18 on pages 15-18. The contour plots seem to show that large areas of the base of the tower have values of stress that exceed the yield strength of structural steel. It is hypothesized that these results are artifacts of the way the tower and flange are connected. The contact wizard that made this connection calls for a contact surface on one body and a target surface on the other body. Details of how this connection was made are shown on page 13.

We originally tried to model the tower as BEAM188 elements with ring-shaped cross-section. We learned how to use the tapering function and how to apply the LINK8

The big lesson learned from examining stress in this model is that the constraints and boundary conditions on the model play the greatest role in the stress behavior within the model.

## *Bolted Connection in the Flange*

The bolted flange connection has only

## *Design Time Estimate*

| | **Hours** |
|---:|---|
| Approach and Team Organization | 6 |
| Geometry & Meshing | 42 |
| Applying BC's & Contact Surfaces | 48 |
| Results Analysis and Iterating Design | 20 |
| Hand Calculations | 6 |
| Report | 12 |
| **TOTAL** | 134 |