Project Report

ME 10.320 Section #1: Project #1

20 November 2017

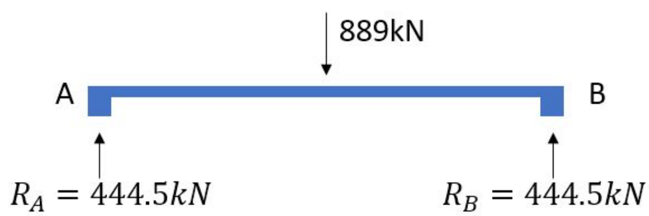
Wind turbine support structure

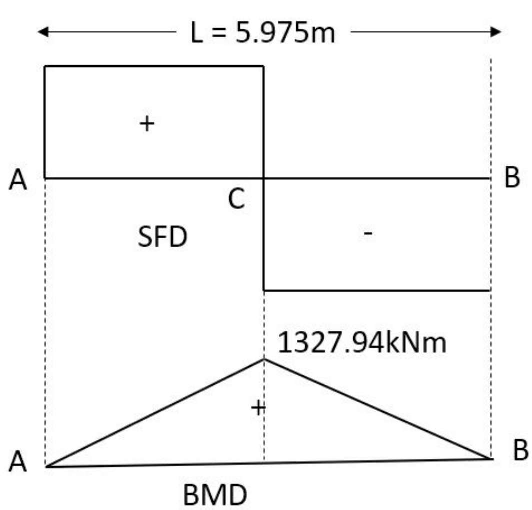
Stephen Glass
Electrical & Computer Engineering, Rowan University

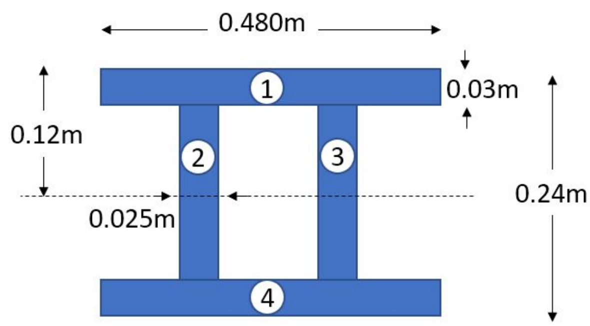
1. Problem Statement

The goal of this project is to design a simply supported beam structure that can withstand the force of a 200-ton turbine. The structure designed should have a factor of safety of two. The values of maximum normal bending stress, strain, and deflection will be calculated in the structure and compared to the simulated results given by Solidworks software. Based on the simulated results, we will also determine proper materials for system components to withstand the applied force.

2. Theoretical Analysis







Stress Analysis

$L = 5.975m$
$W = (W_{total}/2) = (100Tons) = 889kN$

The structure will have zero force in the x-direction. The sum of the forces in the y-direction will be equal to zero. In the simply supported structure there will be two reaction forces.
$\Sigma F_y = 0$
$R_A + R_B = 889kN$

The sum of all moments at point A will equal to zero. We can solve for the reaction forces at point A and point B.I
$\Sigma M_A = 0$

$$R_B \times 5.975m = 889kN \times \frac{5.975m}{2}$$
$$R_A = R_B = 444.5kN$$

Calculate for the maximum total bending moment in the structure.

$$M_{max} = 444.5kN \times \frac{5.975m}{2}$$
$$M_{max} = 1327.9438kN \cdot m$$

To solve for the centroid in the total structure we need to calculate the area and the centroid for each individual part of the structure.
$A_1 = A_4 = 0.480m \times 0.03m = 0.0144m^2$
$A_2 = A_3 = (0.24m - 0.03m - 0.03m) \times (0.025m) = 4.5 \times 10^{-3}m^2$

$\overline{Y}_1 = 0.225m$
$\overline{Y}_2 = 0.012m$
$\overline{Y}_3 = 0.12m$
$\overline{Y}_4 = 0.015m$

Therefore, the total centroid (y-direction) of the structure can be calculated by,

$$\overline{Y} = \frac{A_1 \times \overline{Y}_1 + A_2 \times \overline{Y}_2 + A_3 \times \overline{Y}_3 + A_4 \times \overline{Y}_4}{A_1 + A_2 + A_3 + A_4}$$

$$\overline{Y} = \frac{(0.0144m^2 \times 0.225m) + (4.5 \times 10^{-3}m^2 \times 0.12m) \times 2 + (0.0144m^2 \times 0.015m)}{(2 \times 0.0144m^2) + (2 \times 4.5 \times 10^{-3}m^2)}$$

$$\overline{Y} = 0.12m$$

Calculate for the moment of inertia in the cross-sectional area of the beam.

$$I = \frac{bh^3}{12} = \frac{0.48m \times (0.24m)^3}{12} - \frac{(0.48m - 0.025m - 0.025m) \times (0.24m - 0.03m - 0.03m)^3}{12}$$

$$I = 3.4398 \times 10^{-4}m^4$$

Knowing the maximum bending moment, the moment of inertia, and the centroid of the structure we are able to calculate for the total normal bending stress in the object.

$$\frac{M_{max}}{I} = \frac{\sigma_b}{\overline{Y}} = \frac{1327.943 \times 10^3}{3.4398 \times 10^{-4}}$$

$$\frac{\sigma_b}{0.12m} = \frac{1327.943 \times 10^3}{3.4398 \times 10^{-4}}$$

The total normal bending stress in the structure.

$$\sigma_b = 4.632 \times 10^8 Pa$$

Strain Analysis

The material we are using to simulate in Solidworks is:
DIN Steel (Alloyed), 1.6657 (14NiCrMo13-4)
The material has an Elastic Modulus,

$$E = 2.10 \times 10^{11} N/m^2$$

The strain in the structure can be calculated by dividing the total normal bending stress by the elastic modulus of the structure.

$$Elastic\ Modulus = \frac{Stress}{Strain}$$

$$E = \frac{\sigma_b}{\epsilon} = (2.1 \times 10^{11} Pa) \times \frac{4.632 \times 10^8 Pa}{\epsilon}$$

Therefore, the total strain in the structure,

$$\epsilon = 2.205 \times 10^{-3}m$$

Deflection Analysis

The elastic deflection at the midpoint of a beam, loaded its center, supported by two simple supports is given by,

$$\delta = \frac{WL^3}{48EI} = \frac{(889 \times 10^3 N)(5.975m)^3}{48(2.1 \times 10^{11} Pa)(3.4398 \times 10^{-4} m^4)}$$

Therefore, the deflection of the structure is,

$\delta = 0.05469174m$

Factor of Safety Analysis

The factor of safety of the structure can be found by dividing the yield strength of the material we are using by the stress of the structure.

$$FOS = \frac{Yield\ Strength\ of\ Material}{Stress}$$

The yield strength of the DIN Alloyed Steel 1.6657 is **785,593,984 N/m^2**.

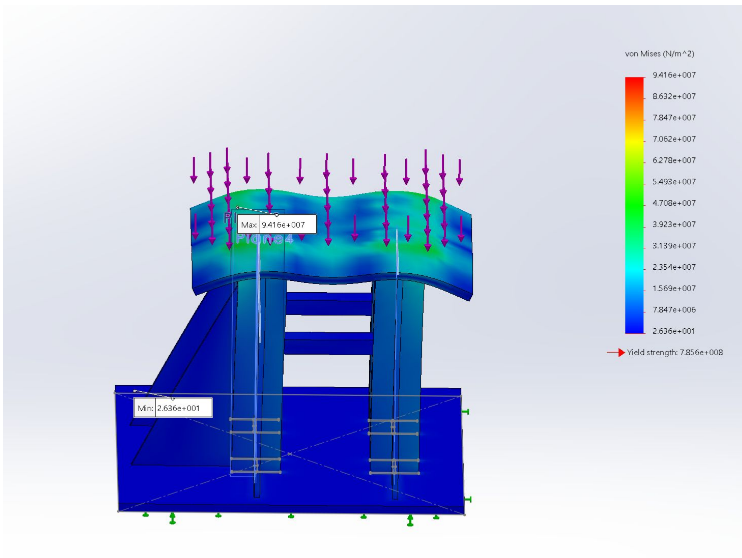
Therefore the factor of safety of the structure,

$$FOS = \frac{785593984 N/m^2}{4.632 \times 10^8 Pa} = 1.7$$

The factor of safety value calculated is close enough to 2 desired for this project. There were many difficulties when trying to increase the factor of safety in the structure.

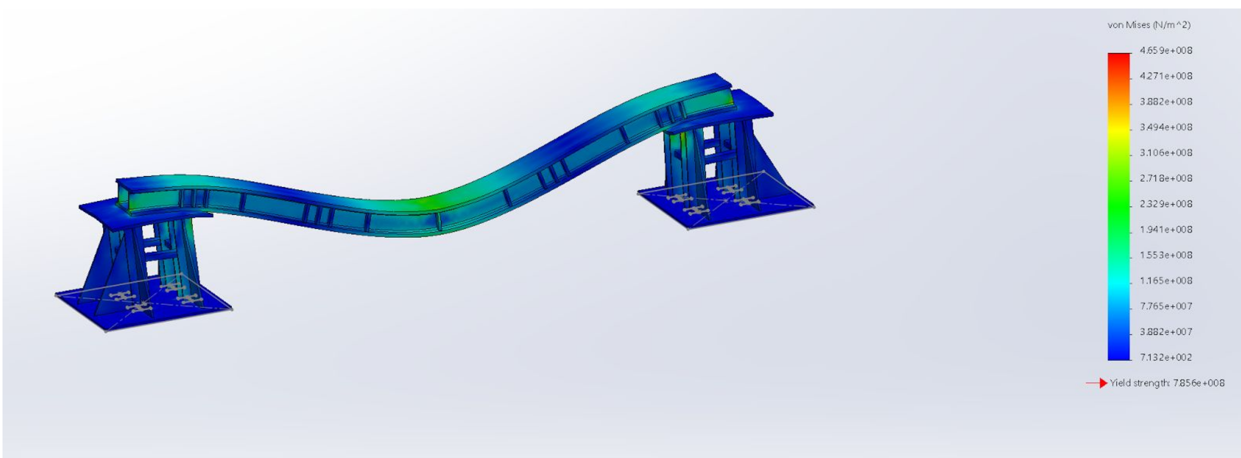
3. Simulation Results

The designs of the supports and beams were modeled in Solidworks CAD design. First, we would like to see the stress analysis on the support structure alone. The support is subject to a distributed force of 444,822N (50 tons).

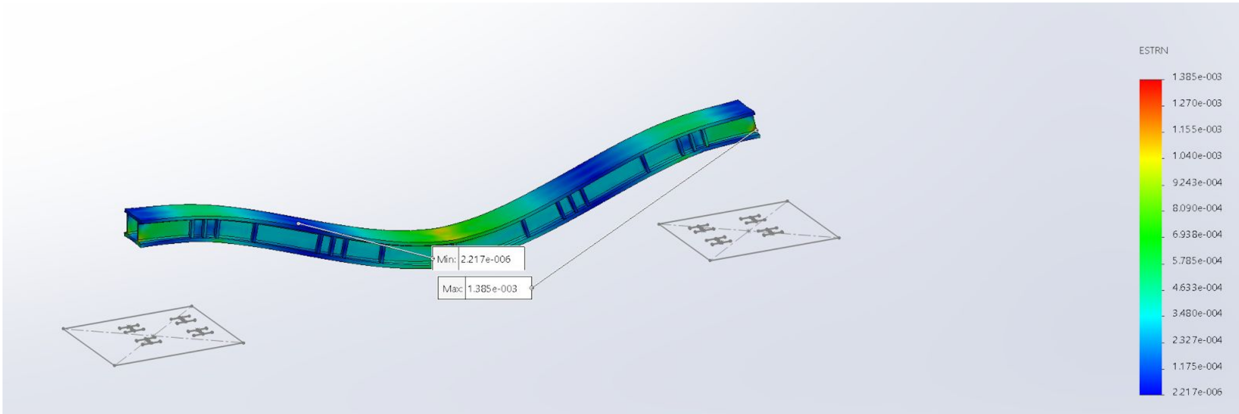


Parameter	Value
Stress	9.41×10^{11} Pa

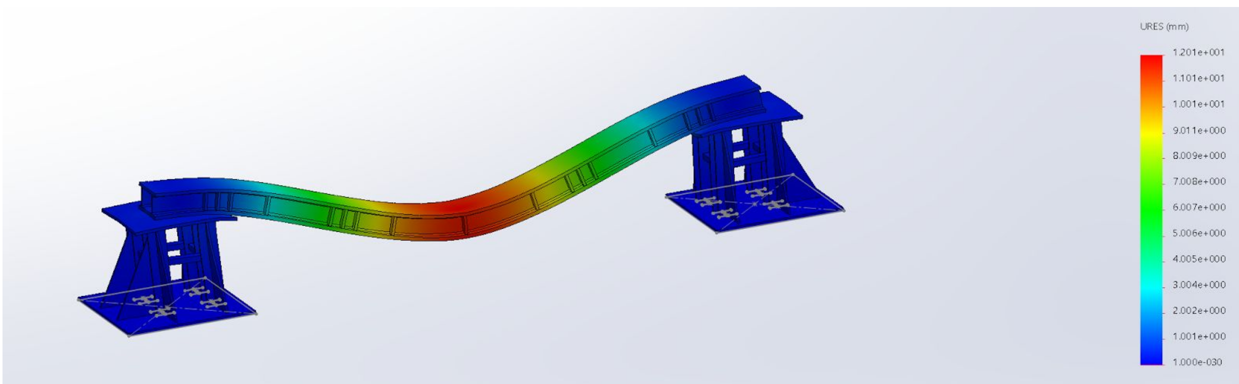
By visual inspection, the support looks able to withstand the force of the turbine. The support and beam can now be joined in a complete assembly for simulated analysis. The assembly is subject to a 889kN force in the center. The plot of the stress plot in the structure is below,



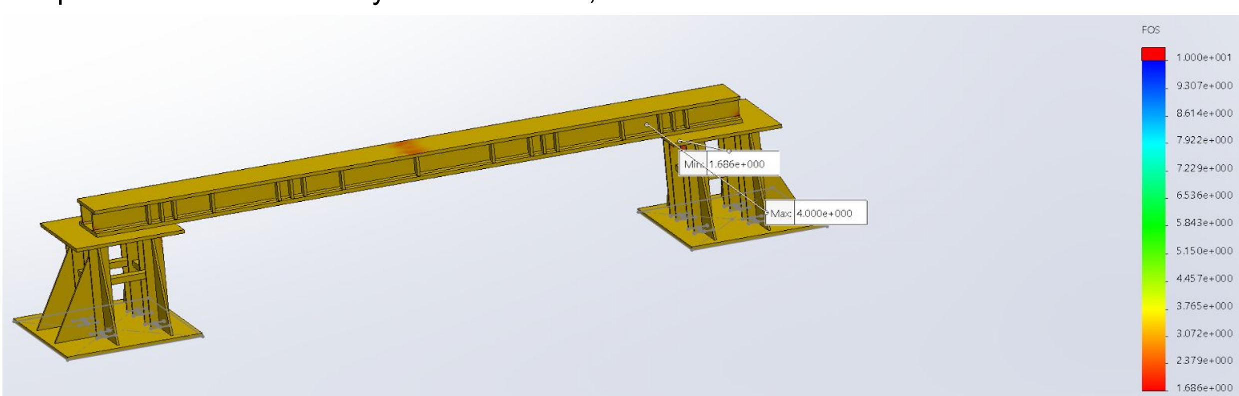
The plot of the strain in the structure,



The plot of the displacement in the structure,



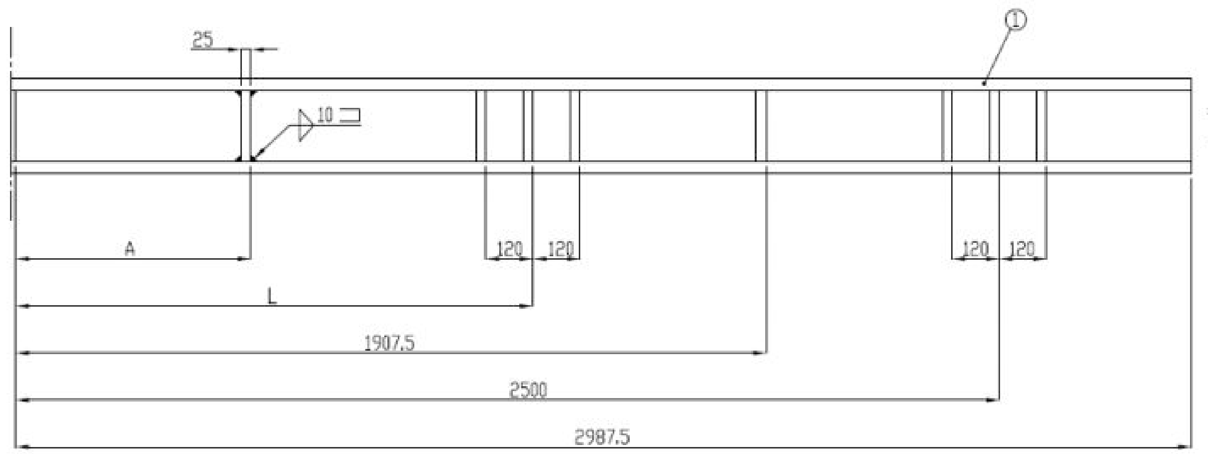
The plot of the factor of safety in the structure,



Parameter	Value (simulation)	Value (theoretical)	% Difference
Stress	3.409×10^{8} Pa	4.632×10^{8} Pa	26%
Strain	1.385×10^{-3} m/m	2.200×10^{-3} m/m	37%

Deflection	0.0120142m	0.0546917	78%
Factor of Safety	1.7	1.7	0%

7. Optimizations



To obtain the greatest factor of safety in the beam we chose the following parameters for **A** and **L**:

A = 120mm
L = 850mm

These values put the supports in the beam close to the center where most of the stress occurs. The material of the beam chosen to be **DIN Steel (Alloyed), 1.6657 (14NiCrMo13-4).**
The yield strength of the material is 785593984N/m^2 and the Elastic Modulus is
$E = 2.10 \times 10^{11} N/m^2$
Using this material we were able to receive a factor of safety of **1.7** in the structure.

8. Conclusion

The structure to support a 200-ton wind turbine was successfully designed and simulated using Solidworks. The simulation shows the structure has a factor of safety of 1.7 (which we will round to 2). We were able to use hand analysis to show that the simulated results were correct. Although the hand calculated results did not match the simulated results exactly, that is to be expected. Solidworks is much more complex and takes many more input variables when simulating.