CAE Simulation #3	Gantry Crane	*21900416 Gyeonheal An*

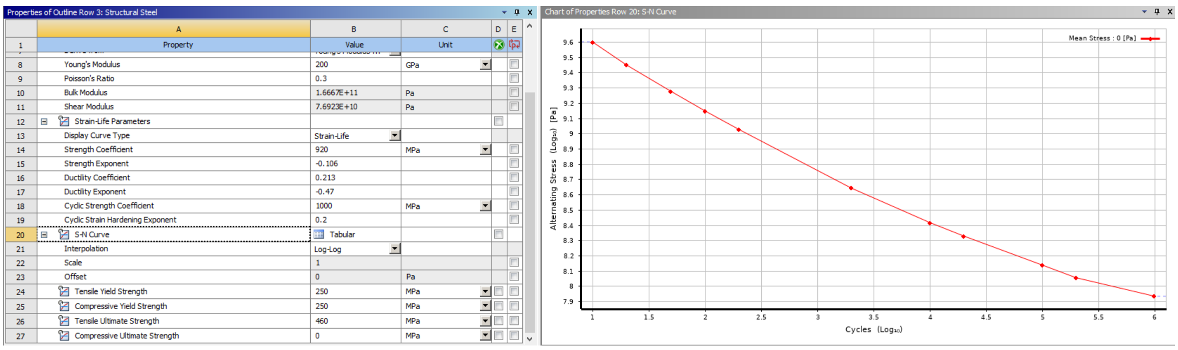
1. Introduction

(Use SI unit by conversion) A gantry crane is designed that must be able to lift 10 tons(use 100KN) as it must lift compressors, motors, heat exchangers, and controls. This load should be placed at the center of one of the main 12-ft-long beams(use 1ft=0.3m) as shown below by the hoisting device location. Weight of the structure is ignored in the analysis. Assume you are using ASTM A36 structural steel(SS400). The crane must be 12 feet long, 8 feet wide, and 15 feet high. The beams should all be the same size, the columns all the same size, and the bracing all the same size. Their cross sections are selected from Appedix F(4th ed.) and shown below. You must verify that the structure is safe by checking the beam's bending strength and allowable deflection. A required safety factor against material yielding of the beam is 3. Verify that the beam deflection is less than L/360 (12/360ft=10mm, downward deflection of the beam center with respect to the ground), where L is the span of the beam. Check yielding and Euler buckling of the long columns. A required factor of safety is 3 against yielding of the column and 5 against buckling of the column.(Ignore local buckling of the horizontal beam) Assume the column-to-beam joints to be rigid while the bracing (a total of eight braces) is pinned to the column and beam at each of the four corners. Use appropriate boundary conditions for the four supports of the gantry crane.

2. Geometry Design (DesignModeler)

Figure 1. Gantry Crane by DesignModeler

3. Material (Structural Steel)



Properties of Outline Row 3: Structural Steel

	A	B	C	D	E
1	Property	Value	Unit		
8	Young's Modulus	200	GPa		
9	Poisson's Ratio	0.3			
10	Bulk Modulus	1.6667E+11	Pa		
11	Shear Modulus	7.6923E+10	Pa		
12	Strain-Life Parameters				
13	Display Curve Type	Strain-Life			
14	Strength Coefficient	920	MPa		
15	Strength Exponent	-0.106			
16	Ductility Coefficient	0.213			
17	Ductility Exponent	-0.47			
18	Cyclic Strength Coefficient	1000	MPa		
19	Cyclic Strain Hardening Exponent	0.2			
20	S-N Curve	Tabular			
21	Interpolation	Log-Log			
22	Scale	1			
23	Offset	0	Pa		
24	Tensile Yield Strength	250	MPa		
25	Compressive Yield Strength	250	MPa		
26	Tensile Ultimate Strength	460	MPa		
27	Compressive Ultimate Strength	0	MPa		

Figure 2. Material Settings

4. Experimental Condition

<table>
<tr><th>Used Material</th><td colspan="3">Structual Steel</td></tr>
<tr><th>Mesh</th><td colspan="3">100mm</td></tr>
<tr><th>Force</th><td colspan="3">100KN</td></tr>
<tr><th rowspan="15">Solution</th><td colspan="3">Total Deformation</td></tr>
<tr><td colspan="3">Directional Deformation</td></tr>
<tr><td colspan="3">Torsional Moment</td></tr>
<tr><td colspan="3">Total Bending Moment</td></tr>
<tr><td colspan="3">Total Shear Force</td></tr>
<tr><th rowspan="4">Beam Tool</th><td colspan="2">Direct Stress</td></tr>
<tr><td colspan="2">Minimum Combined Stress</td></tr>
<tr><td colspan="2">Maximum Combined Stress</td></tr>
<tr><td colspan="2">Maximum Bending Stress</td></tr>
<tr><th rowspan="6">Eigenvalue Buckling</th><td rowspan="6">Total Deformation</td><td>Mode 1</td></tr>
<tr><td>Mode 2</td></tr>
<tr><td>Mode 3</td></tr>
<tr><td>Mode 4</td></tr>
<tr><td>Mode 5</td></tr>
<tr><td>Mode 6</td></tr>
</table>

Table 1. Simulation Conditions

5. Result

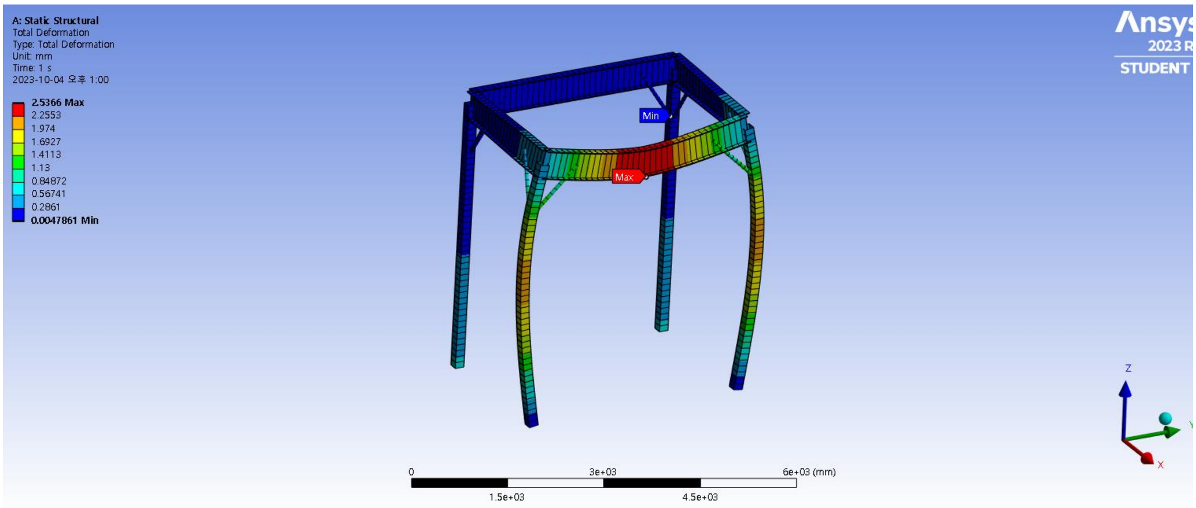


Figure 3. Total Deformation

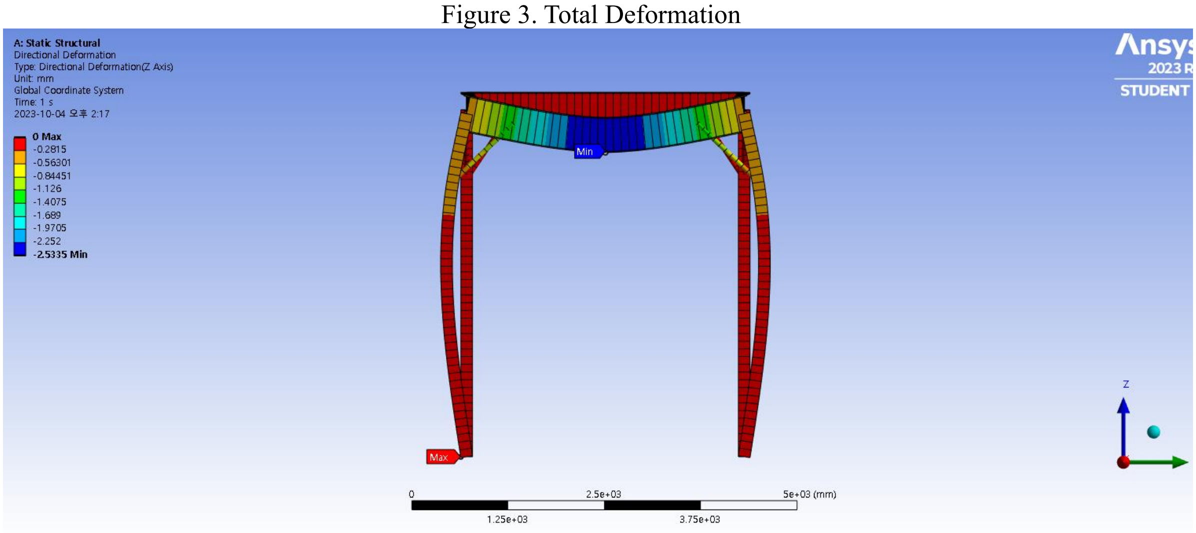


Figure 4. Directional Deformation

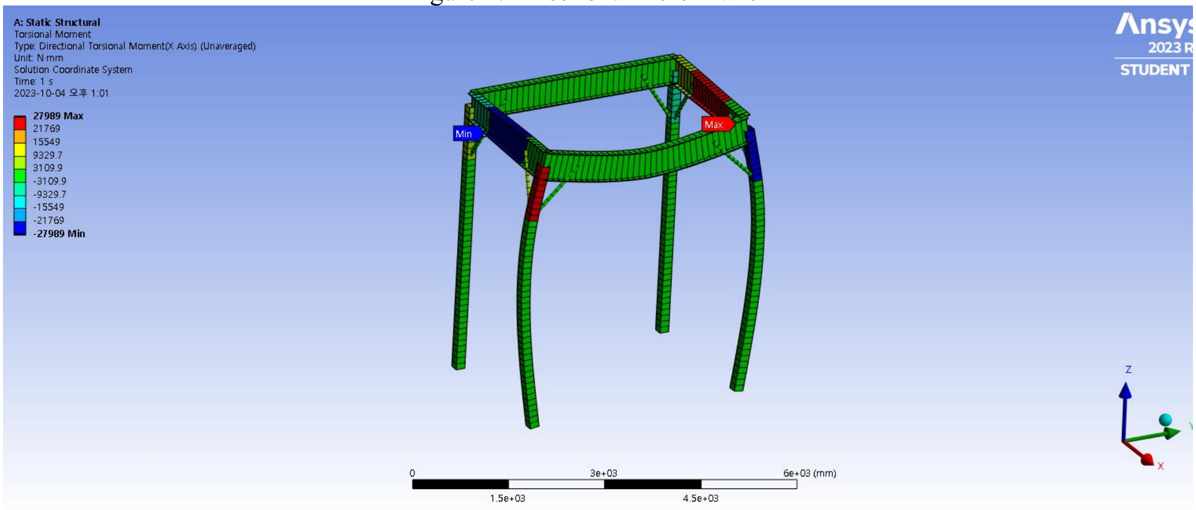


Figure 5. Torsional Moment

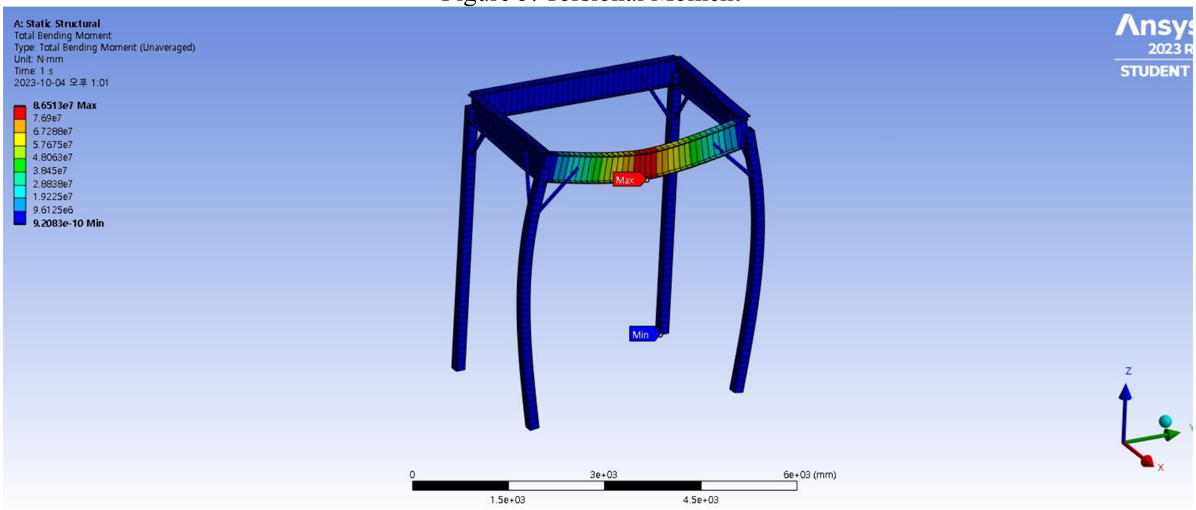


Figure 6. Total Bending Moment

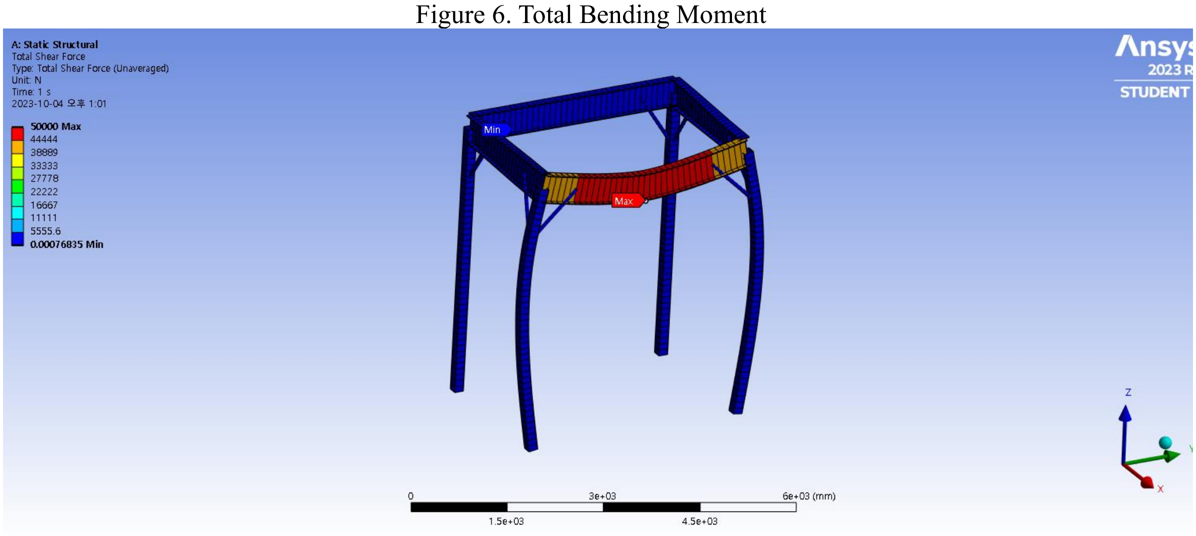


Figure 7. Total Sheer Force

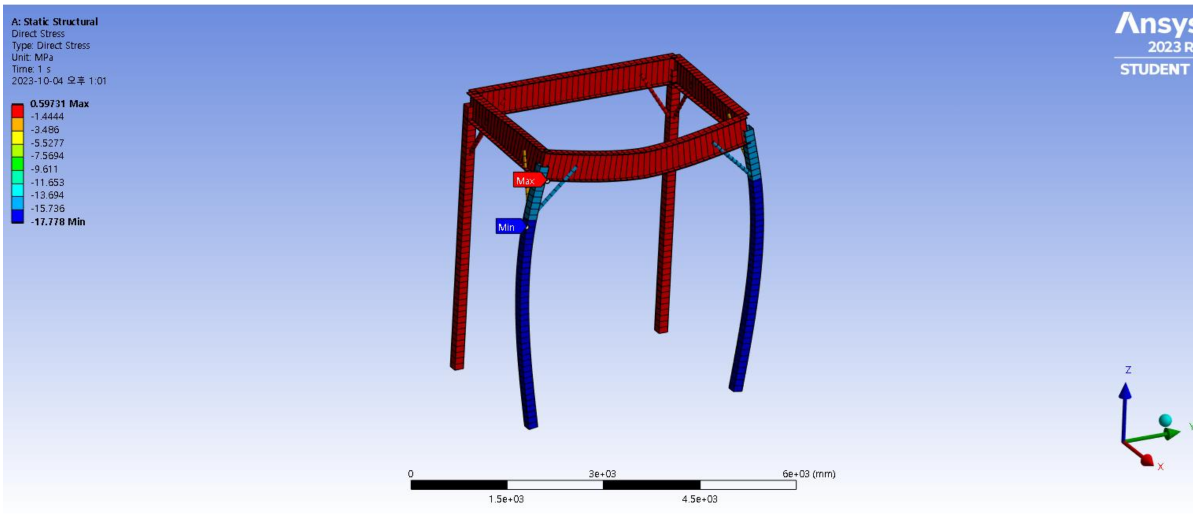


Figure 8. Direct Stress

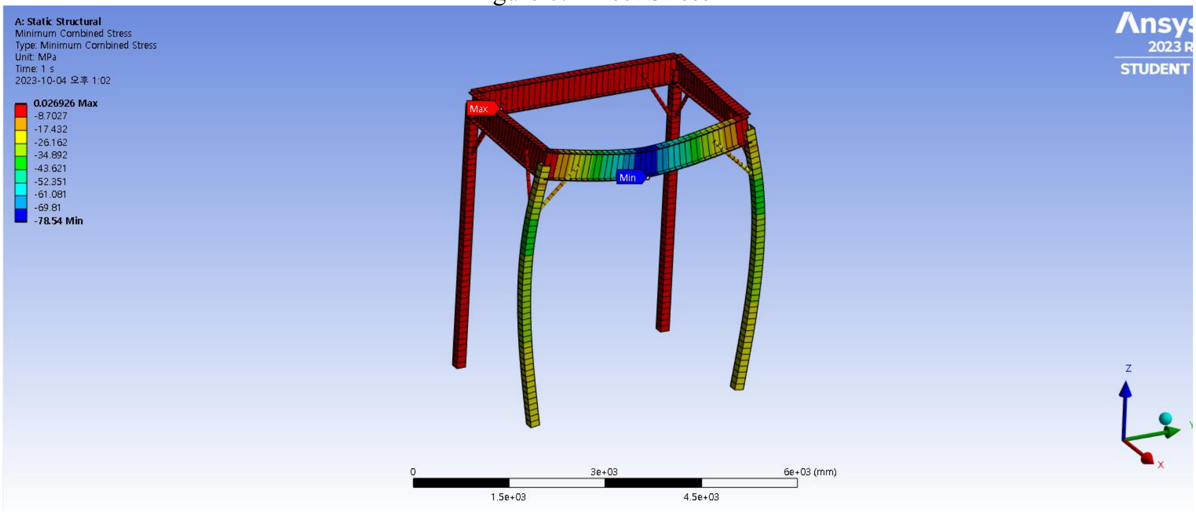


Figure 9. Minimum Combined Stress

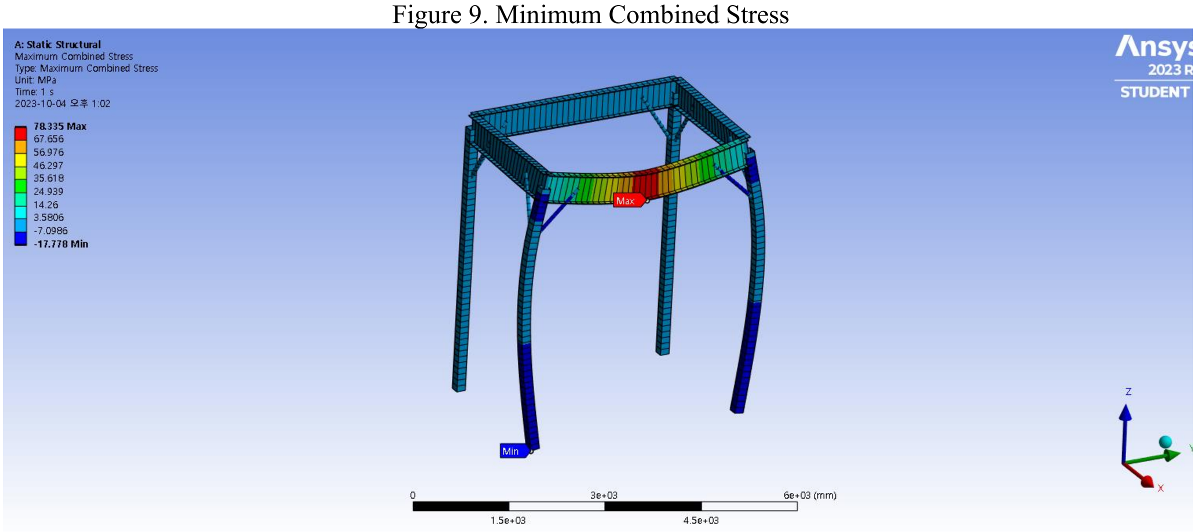


Figure 10. Maximum Combined Stress

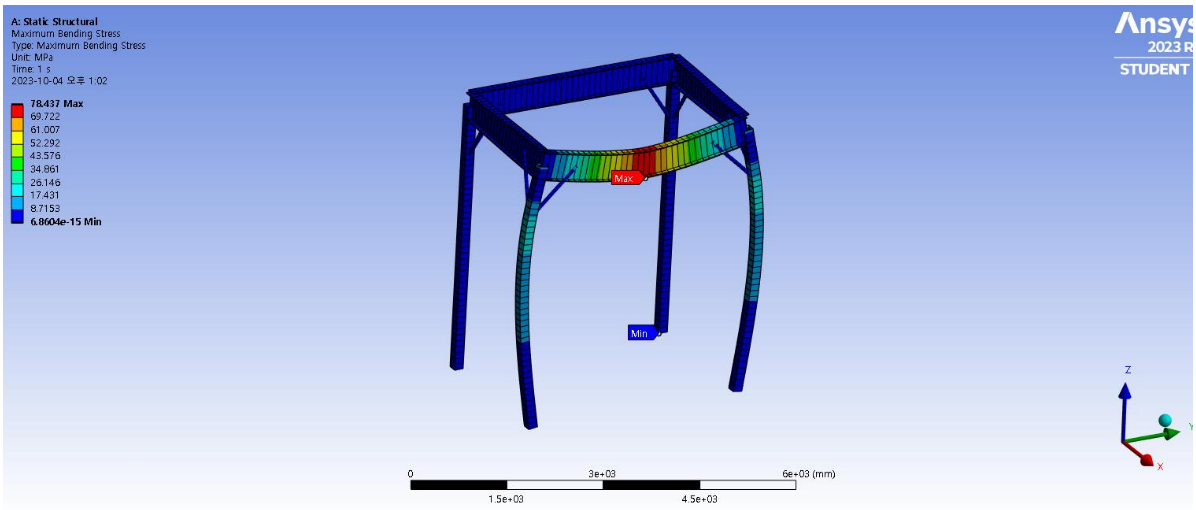
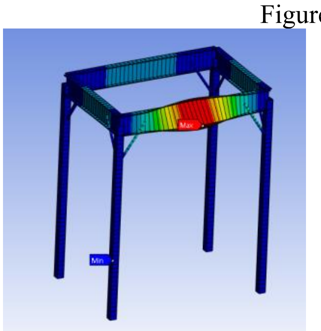
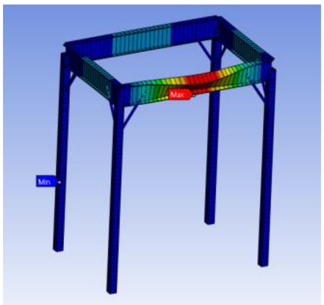


Figure 11. Maximum Bending Stress

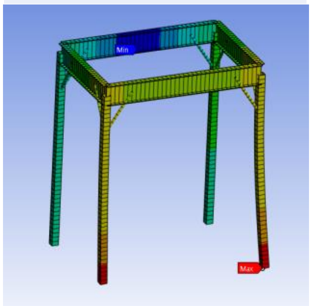
1



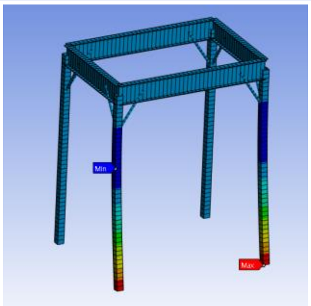
2



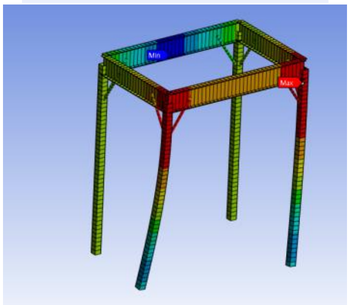
3



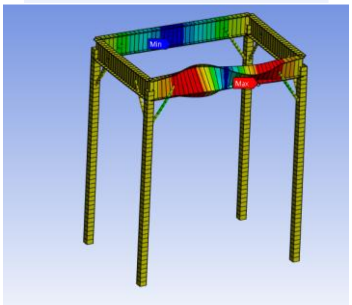
4



5



6



Tabular Data

	Mode	Load Multiplier
1	1.	-4.2233
2	2.	4.1995
3	3.	6.0991
4	4.	6.2671
5	5.	6.4117
6	6.	8.3688

Figure 12. Eigenvalue Buckling (1~6)

6. Analysis

① Generate an ANSYS beam model and get safety factors and deflections

$$SF = \frac{Yielding\ Strength}{Maximum\ Bending\ Stress} = \frac{250}{78.437} = 3.19$$

$$(z - axis)\ Deflections = \ -2.5335mm$$

$$(y - axix)\ Deflections = 2.0333mm$$

② Compare simulation results with appropriate theoretical results (stresses and downward deflections of the horizontal beam, critical buckling load of the columns)

$$I_{Ibeam} = I_{z1} - 2 * I_{z2} = \frac{152.78 \times 454.66^3}{12} - 2 \times \frac{72.3895 \times 427.99^3}{12} = 250.73 \times 10^{-6} m^4$$

$$I_{RecTube} = I_{z1} - I_{z2} = \frac{152.4 \times 152.4}{12} - \frac{142.875 \times 142.875}{12} = 10.23 \times 10^{-6} m^4$$

- **Bending stress of the horizontal beam**

$$M = \frac{FL}{4} = \frac{10^5 \times (12 \times 0.3)}{4} = 90 \times 10^3\ Nm$$

$$\sigma = \frac{MC}{I} = \frac{90 \times 10^3 \times \frac{454.66}{2} \times 10^{-3}}{250.73 \times 10^{-6}} = 81.6\ Mpa\ (\cong simulation：\ 78.44\ Mpa\ )$$

- **Downward deflections of the horizontal beam**

$$\delta = \frac{FL^3}{48EI} + \frac{FL}{EA}$$

$$= \frac{10^5 \times (12 \times 0.3)^3}{48 \times 200 \times 10^9 \times 250.73 \times 10^{-6}} + \frac{\frac{10^5}{2} \times (15 \times 0.3)}{200 \times 10^9 \times 2812.49 \times 10^{-6}} = 2.34mm\ (\cong simulation : 2.53mm)$$

- **Critical buckling load of the columns (free-fix)**

$$P_{cr} = \frac{n\pi^2 EI}{L^2} = \frac{n \times \pi^2 \times 200 \times 10^9 \times 10.23 \times 10^{-6}}{(15 \times 0.3)^2} = 249299\ (n = 0.25)$$

When you watch the Buckling Animation in Ansys, for the Eigenvalue Buckling analysis we need to perform, the modes are 1 and 2. In ultimate buckling, the critical buckling load is equal to the Load Multiplier multiplied by the magnitude of the applied force during analysis.

$$P_{cr} = 4.2233 \times 50000 = 211165N$$

$$P_{cr} = 4.1995 \times 50000 = 209975N$$

Therefore, we can observe that the simulation value of the critical buckling load is very similar to the theoretical value.

③ **Discuss on boundary conditions of the four supports**

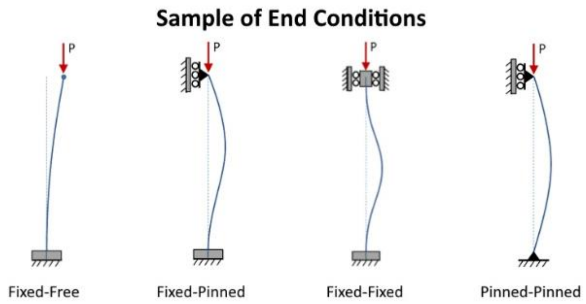


Figure 13. Boundary Conditions

$$P_{cr} = \frac{n\pi^2 EI}{L^2}$$

Connection	N
Fixed-Free	**0.25**
Fixed-Pinned	**1**
Fixed-Fixed	**4**
Pinned-Pinned	**2.046**

In buckling analysis, when the connection methods at both ends change, the value of the multiplying constant varies, resulting in different critical buckling loads. The highest critical buckling load is observed in the case of both ends being fixed-fixed. Following that, it is in the order of pin-to-pin, fixed-to-pinned, and fixed-to-free connections.

④ **What do you think the function of the braces are?**

A. Load Distribution: Bracing serves to distribute the load and transmit it to various parts of the structure. This allows the structure to evenly bear the load and maintain stability. This is particularly important when lifting heavy objects, such as cranes.

B. Motion Control: It contributes to controlling the motion (deformation or sway) of the structure. Large structures may experience motion when subjected to wind, earthquakes, or when lifting and moving objects. Bracing restricts such motion and ensures stability.

C. Buckling Prevention: Bracing prevents the buckling of columns and enhances their safety. When using tall columns, as in the case of cranes, column buckling can be a critical safety concern. Bracing limits column buckling, preventing the column from bending or collapsing.

D. Wind Resistance: Tall structures like cranes are exposed to the force of the wind. Bracing helps control wind-induced swaying and maintains the stability of the structure.

E. Overall Structural Strength Improvement: Bracing contributes to improving the overall strength of the structure. By transferring and distributing loads among various parts of the structure, it enables the structure to support heavier loads.

⑤ How many number of elements is required to accurately solve this problem?

When the default mesh size is set to 311.15mm, there are 132 elements, and with a mesh size of 100mm, the element count increases to 332. However, when the mesh size is reduced to 50mm, the element count further increases to 632. Analyzing the results from Ansys Simulation, it is evident that as we continue to reduce the mesh sizing, deformation or stress stabilizes at a certain point of change. Therefore, for this analysis, it is necessary to maintain a mesh sizing of 50mm or less in order to achieve an element count of 332, which seems to be the most suitable.