

VIBRATION AND MODAL TRUSS GANTRY CRANE ANALYSIS USING ANSYS

Dionesta Shah Akbar Siregar ¹⁾, Sofia Debi Puspa ^{2)*}, Soeharsono ³⁾, Tono
Sukarnoto ⁴⁾, Faisal Adinegoro ⁵⁾

^{1,2,3,4,5)}Department of Mechanical Engineering/Universitas Trisakti

*Corresponding Author: sofia.debi.puspa@trisakti.ac.id

Diterima: 15 Januari 2025

Revisi: 22 Januari 2025

Disetujui: 22 Januari 2025

Terbit: Februari 2025

ABSTRACT

Cranes are indispensable heavy equipment that can lift and move hefty loads in factories, warehouses, and construction sites with ease. Safe operation of cranes is vital because errors in crane design can make it prone to failures, such as broken trusses, which can cause the crane to collapse and hit people around the crane. This research aims to analyze vibrations and modal Truss Gantry Cranes using ANSYS software to determine natural frequencies, critical points, and *deformations* that can result in damage. To test the Truss Gantry Crane's structural strength with ANSYS software, the Static Structural design program is used to analyze the deformations and stresses experienced by the crane, the Modal design program is utilized to determine its natural frequency, and the Harmonic Response is used to determine its reaction to natural frequencies. Static Structural testing was carried out 30 times, starting from a load of 500 N and increasing to a maximum load of 15000 N, with an interval of 500 N. The test results showed that the Truss Gantry Crane vibrates violently at frequencies of 24 Hz, 68 Hz, and 94 Hz. Further analysis revealed a linear regression analysis for the maximum load and maximum stress that can be applied to the Truss Gantry Crane as follows:

$$y = 0,0091x + 0,1981$$

Keywords: *Deformation, Modal, Maximum Load, Truss Gantry Crane, and Vibration*

I. INTRODUCTION

Cranes are heavy equipment needed in the factory industry, warehousing, construction, and many other industries. The ability of the crane to transport objects with heavy dimensions and weights benefits human needs in moving large and heavy goods (Wibawa, 2020). However, the crane's ability to transport goods is not always safe. Cranes are heavy equipment, and their failure can lead to significant material losses and even loss of life. In 2017 alone, more than five work accidents involving tower cranes were reported. Several factors contribute to these accidents, with 38% occurring due to structural and technical failures. This highlights the importance of the crane's structural integrity and age in ensuring its safe operation (Sunainah & Sutantra, 2018).

Vibration analysis on crane structures needs to be done to determine the natural frequency, deformation, and critical points to overcome the possibility of failure and avoid losses that can result in fatalities. However, analysis does not need to be done directly, along with computing and software technology development (X. Chen & Liu, 2014). Crane analysis can be performed through computer simulation using ANSYS software. ANSYS is widely used for its capability to simulate various engineering aspects, such as fluid dynamics, structural integrity, thermal performance, and more. The

concept for this digital simulation began with John Swanson while he was working at the Westinghouse Astronuclear Laboratory in the 1960s. During that time, engineers conducted finite element analysis (FEA). After Westinghouse rejected Swanson's proposal to automate FEA through software, he left the company in 1969. He then developed his software, initially naming it Swanson Analysis Systems Inc. (SASI), later known as ANSYS (Canonsburg, 2013).

In the study conducted by Augustyn and Barski, the estimation of wind loads on a gantry crane positioned in the open air was examined. Two structures were observed: a box girder and a truss girder. The gantry crane model featuring a full-scale box girder was analyzed using Computational Fluid Dynamics (CFD) simulations based on different Davenport wind profiles. This numerical analysis was performed using the ANSYS software, and the results from the cross-section model's experimental and numerical analyses were compared to the relevant standards. The study highlights the importance of changes in wind direction within the vertical plane, noting that even small variations in the vertical angle, up to 6° , can lead to significant changes in the force required to reverse the gantry crane (Augustyn & Barski, 2024).

In 2020, Chen researched the primary installation technique for a large two-panel segment of a three-girder steel main frame. The research employed a frame-type gantry crane with a lifting capacity of 1,800 tons to elevate the substantial steel frame segment, which consists of four frames and incorporates a high chassis, reversible legs, and three hanging points. A pier-side bracket and a 1,800-ton floating crane were utilized to erect the steel frame segment above the pier, which facilitated enhanced construction monitoring and organization. In the assembly process of the large steel frame featuring two balanced cantilevers, the connections between the members were optimized through theoretical analysis. This analysis informed the assembly sequence, proceeding from the lower chord to the diagonal and concluding with the upper chord, thereby enabling efficient assembly. Moreover, wind-resistant support braces were installed at the apex of the main pylons to improve the wind resistance of the large cantilever steel frame during the construction phase (T. Chen, 2020).

Furthermore, Guan et al. conducted a study on the segmental assembly technique for the main ship channel bridge located on the River Bridge. The dimensions of a typical single segment measure $28 \text{ m} \times 36.2 \text{ m} \times 18.15 \text{ m}$, with a maximum lifting capacity of 1,744 tons. In the frame unit segment assembly construction framework, the frame unit segment is divided into four components: the frame unit, the railway deck block, the road deck block, and the lateral support unit. Two assembly lines have been established, and a variety of gantry cranes and portal cranes are employed to facilitate the assembly process. Upon the horizontal assembly of the main frame on the assembly jig, the frame unit is subsequently rotated 90° for segmental assembly. This operation utilizes an 800-ton gantry crane, while assembly precision is ensured through temporary fixing and adjustment devices (Guan et al., 2016).

Based on the explanation above, this study aims to determine the strength of the Truss Gantry Crane structure in the form of maximum load that can be received and vibration response through ANSYS simulation to avoid damage and failure that can negatively impact users. The type of Crane model used is the Miniature Overhead Truss Gantry Crane, and the crane model material is low-carbon steel with a Yield Strength of 250 MPa.

II. LITERATURE REVIEW

2.1. Truss Gantry Crane

Gantry Crane is heavy equipment that can move and lift loads from one point to another. Gantry Crane is usually used in ports and industries to help with work. A Gantry Crane is equipped with a wire rope that can move up and down, which is wrapped around a roller. Gantry Crane is also equipped with wheels that help the entire crane system to move forward and backward. The load being moved usually has a large weight and large size and is dangerous, so it cannot be handled manually by workers.

A Truss Gantry Crane uses a Truss structure, has a single or double girder welded from steel angles or steel rods, and moves materials ranging from light to heavy. The Truss Gantry Crane is very suitable for outdoor use with wind resistance. Large Truss Gantry Cranes move on rails like train tracks. A Truss Gantry Crane is generally used in outdoor applications.

Truss Gantry Crane moves objects on 3 axes, namely the X, Y, and Z axes. For movement on the X axis, use wheels that generally move only on the X axis but can move back and forth. These wheels usually move on rails like train tracks, but some move without rails. Because these wheels move the entire Truss Gantry Crane system, the wheels are driven using a motor or an engine on large cranes. A lifting device is used to lift goods on the Y and Z axes and move them about the Z axis through the girder on the crane. This lifting device has a steel wire roller connected to a hook at the end of the steel wire to lift materials by hooking the hook on the item to be lifted.

2.2. Modal Analysis

Modal analysis is the starting point of dynamic analysis, which finds a structure's natural frequency. It gives engineers an idea of how the design will respond to various types of dynamic loads. For example, on a bridge, vibrations often occur, which can be caused by people passing by, vehicles passing by, or wind blowing against it. Therefore, Modal Analysis is important for structures likely to experience vibrations or changes in dynamic loads. Figure 1 is the Modal program in ANSYS software, which displays an example of a Static Structural simulation in the form of deformation.

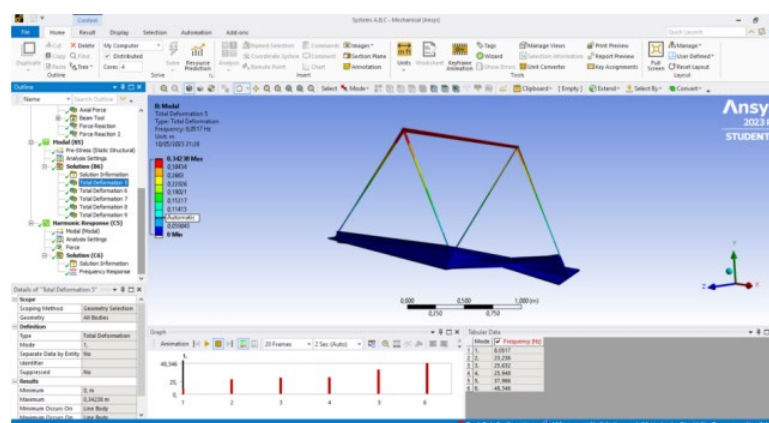


Figure 1. Modal Analysis Program in ANSYS software

2.3. Harmonic Response

Harmonic Response determines how a structure responds when given a specific vibration frequency, especially at frequencies close to its natural frequency. When a

structure is given a vibration frequency, the object will experience oscillation; the magnitude of the oscillation depends on the frequency. Therefore, Harmonic Response analysis determines at what frequency the object experiences the greatest vibration and which parts are likely to fail quickly.

Deformation occurs when an object changes shape due to external influences, such as applied forces or temperature changes. The forces affecting the object can include tensile, compressive, and torsional forces. If deformation leads to a permanent shape change, it is called plastic deformation. In contrast, if the object changes shape but returns to its original form, this type of deformation is called elastic deformation.

2.4. Stress and Strain

Stress is a force applied to an object with a cross-section until the object is deformed. When an object is given force and deformed, its dimensions will change. For example, if a cylinder is given tensile stress, the object will experience changes in length and cross-sectional area; this change is called strain. The formula for strain, is in equation [1].

$$\varepsilon = \frac{\Delta l}{l_0} \quad [1]$$

where:

ε = Strain

Δl = length change (mm)

l_0 = initial length (mm)

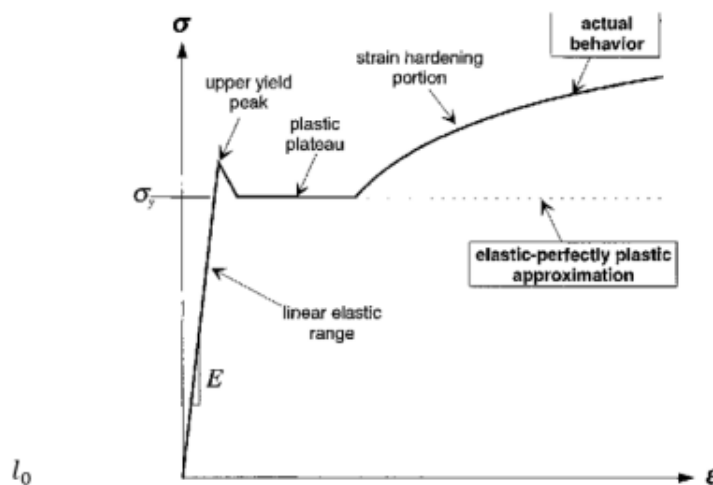


Figure 2. Stress against strain graph of steel

Figure 2 is a graph of stress against strain; metals such as steel have ductile properties, so they have quite large strains. The relationship between stress and strain is:

$$\sigma = E \cdot \varepsilon \quad [2]$$

where:

E = Modulus of Elasticity

σ = Stress

ε = Strain

III. METHOD

The Truss Gantry Crane model research was conducted at the Basic Machine Phenomenon Laboratory at the Faculty of Industrial Technology, Trisakti University, West Jakarta. The research began by collecting data from the dimensions of the Truss Gantry Crane, Structural Static modeling, modal testing, and Harmonic Response using ANSYS. The dimensions of the Truss Gantry Crane model are 1460 mm x 525 mm x 536 mm, with a truss rod diameter of 10 mm and a T-Beam profile as in Figure 3.

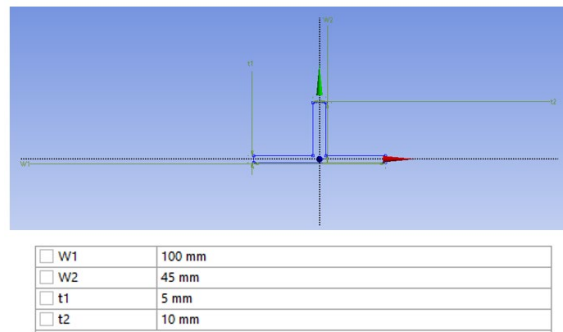


Figure 3. Dimensions of T-Beam

After obtaining the crane's dimensions, the next step is to create a 3D model using ANSYS. The program within ANSYS that will be utilized for this modeling is Design Modeller. The design process begins by establishing coordinate points, which lines will connect; these coordinate values are derived from the measurement results. Figure 4 shows a point-and-line design resembling a Truss Gantry Crane.

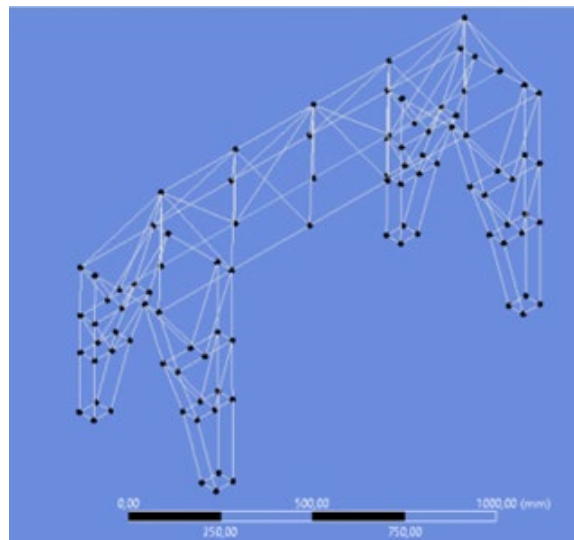


Figure 4. Coordinate Points

The Truss Gantry Crane model has two different cross-sections, namely solid circles and T-Beam. Therefore, the lines representing them must be separated to assign different cross-sections to each. The "Add Frozen" feature is used to distinguish lines with different cross-sections. Figure 5 shows the line that will be given a solid circle

cross-section (left) and a line that will be given a T-Beam cross-section (right). The final result of the Truss Gantry Crane modeling can be seen in Figure 6.

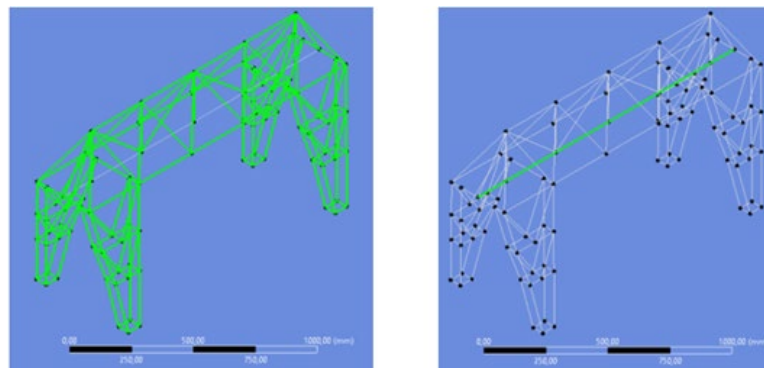


Figure 5. The line will have a solid circular cross-section (left) and a T-Beam cross-section (right)

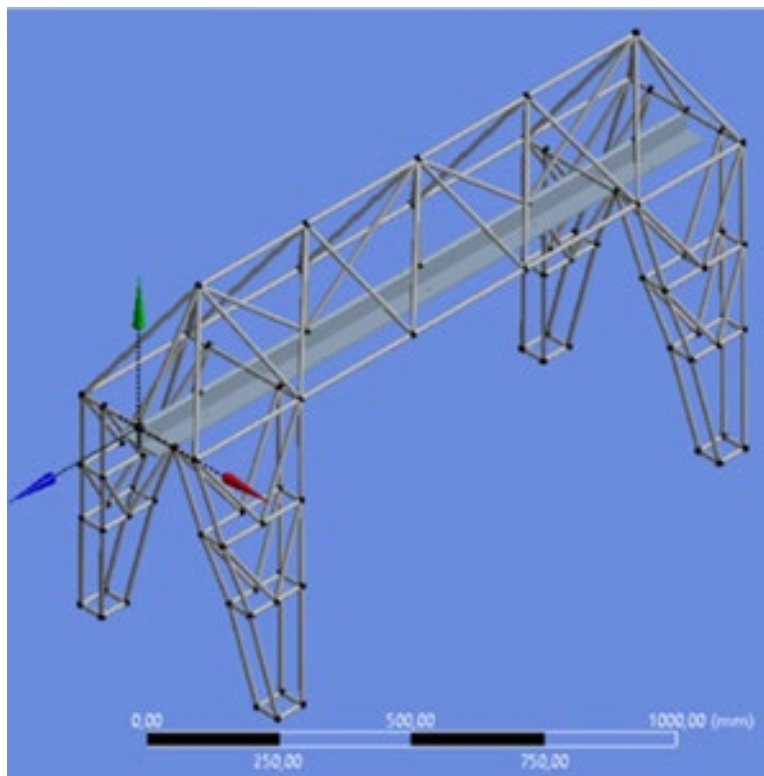


Figure 6. The Final Result of Truss Gantry Crane Modeling

After completing the modeling step, the next phase is to test the design of the Truss Gantry Crane using the Mechanical program. This program conducts a Static Structural analysis to gather data, including deformation, shear force, bending moment diagrams, support reactions, and the maximum loads the crane can withstand. Following this, a Modal analysis is performed to determine the natural frequency across six modes. A Harmonic Response analysis is conducted to assess the frequency response based on the natural frequency range obtained from the Modal test.

The study has research limitations and conditions, which are shown in Table 1. The temperature conditions are 22°C with a pressure of 1 atm, assuming that the metal and joints are defect-free.

Table 1. Limitation Research

Material	Structural Steel
Density	7850 kg/m ³
Crane dimensions	1460 mm x 525 mm x 536 mm
Mesh	100 mm
Mass	32.86 kg
Truss cross section	Solid Circle with Diameter 10 mm
Safety Factor	2
Yield Strength	250 MPa
Ultimate Tensile Strength	450 MPa

III. RESULTS AND DISCUSSION

In the Static Structural analysis, the crane design is given a point support in the form of Simply Supported on 16 legs, as in Figure 7. Then, a concentrated load at the midpoint of the Truss Gantry Crane on the negative Y axis and the earth's gravitational force with a value of 9.806 m/s² on the negative Y axis are given.

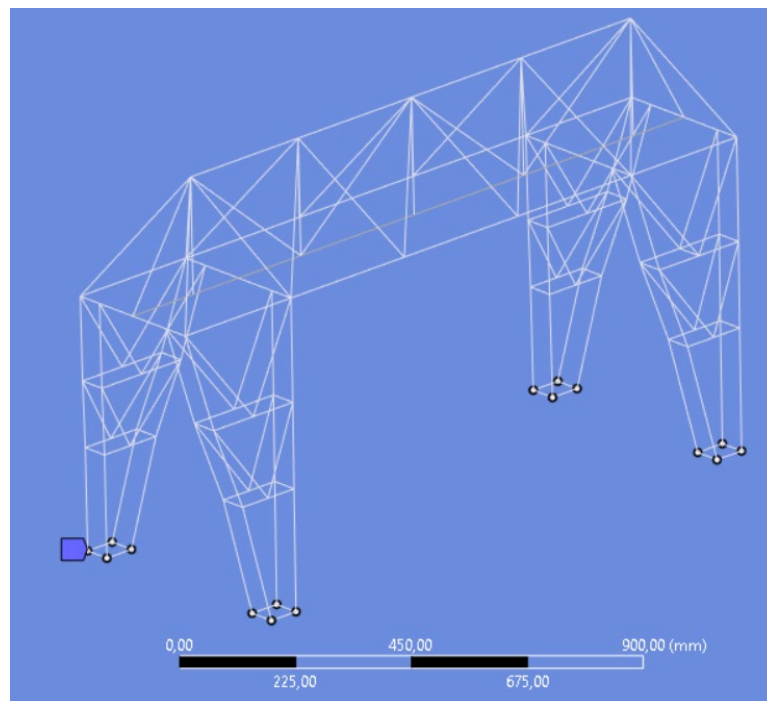


Figure 7. Point Support on Truss Gantry Crane

3.1. Maximum Load and Static Structural Analysis

In the Truss Gantry Crane design, there are two lines with different cross-sections: the line for the Truss Gantry Crane, which has a circular cross-section, and the line for the girder, which has a T-Beam cross-section. Both lines are separate parts in

the mechanical program, so analysis cannot be performed. Therefore, both lines need to be connected using the Connection feature. After inputting the support, load, and gravity data, the next step is to find the deformation and stress experienced in the crane design and input the Beam Tool data, which has features to find Direct Stress, Minimum Combined Stress, and Maximum Combined Stress. In the Truss Gantry Crane structure, the stress received is compressive stress and tensile stress. Therefore, the value of Direct Stress will be used to find the maximum load.

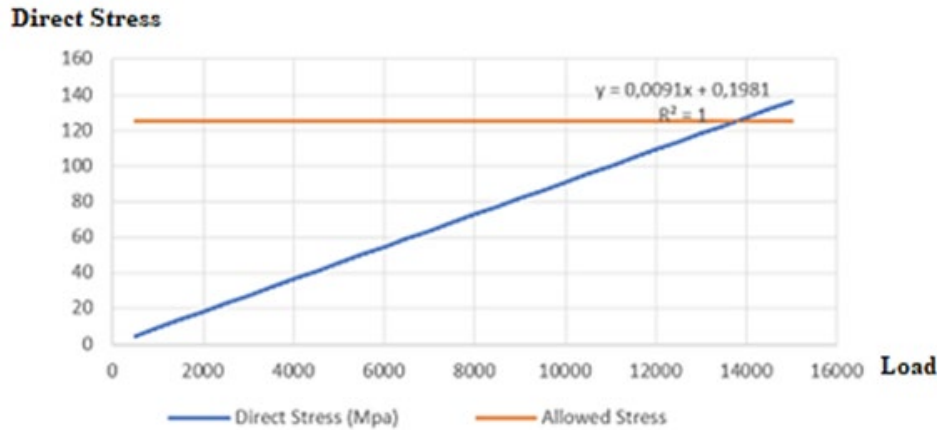


Figure 8. Graph of Stress against Load

In determining the maximum load, direct stress testing is carried out 30 times from 500 N to 15000 N to obtain the stress produced by the test; with a safety factor value of 2, the permissible stress is 125 MPa. In Figure 8, the graph displays the relationship between stress and load. The vertical axis represents the load variable, while the horizontal axis represents the stress variable. The results indicate that the stress value increases linearly as the load value increases. The calculated R^2 value is 1, indicating a strong relationship between the load and stress variables. The resulting linear equation is $y = 0,0091x + 0,1981$ where x is load (N) and y is stress (MPa).

The maximum load can be determined using the linear equation. Since the maximum allowable stress is 125 MPa, the maximum allowable load is 13714.5 N. This maximum load value will be used for further simulations.

Deformation testing is conducted to assess the deformation in the Truss Gantry Crane under maximum load. Testing is carried out on the X, Y, and Z axes. Figure 9 shows the results of deformation testing on the X-axis. The red part is the deformation on the positive X-axis with a value of 0.31741 mm, and the blue part is the deformation on the negative X-axis with a value of -0.3173 mm.

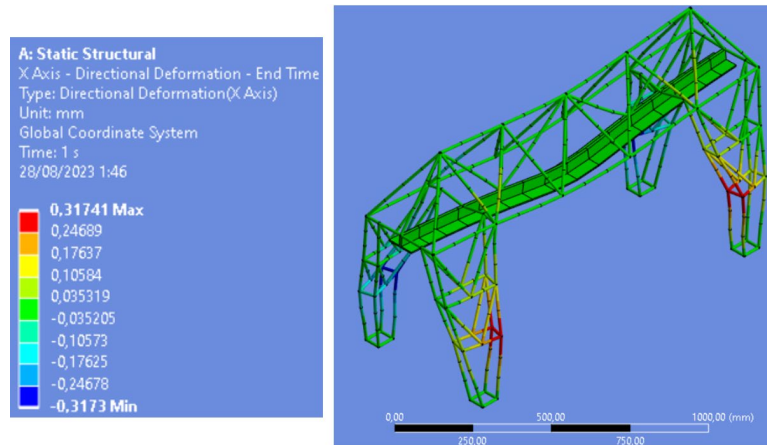


Figure 9. X-Axis Deformation

Figure 10 illustrates the deformation on the Y-axis with the red part being the deformation on the positive Y-axis with a value of 0.046201 mm and the blue part being the deformation on the negative Y-axis -0.62468 mm.

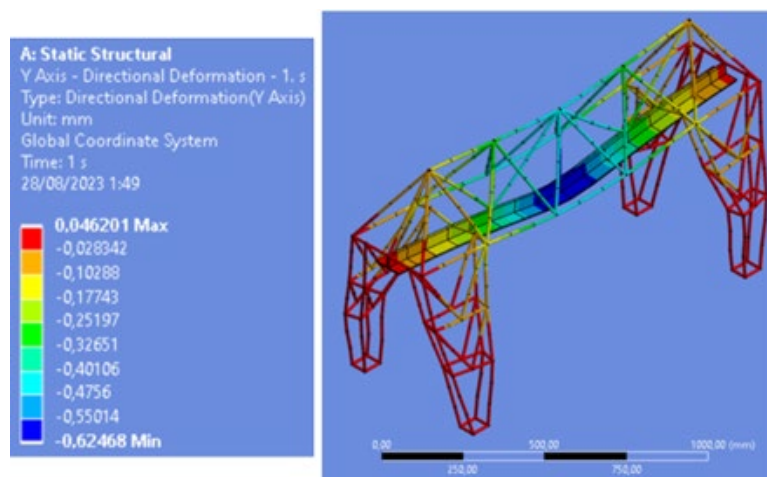


Figure 10. Y-Axis Deformation

Figure 11 illustrates deformation along the Z-axis. The red section represents deformation in the positive Z direction, measuring 0.17547 mm, while the blue section indicates deformation in the negative Z direction, measuring -0.16837 mm.

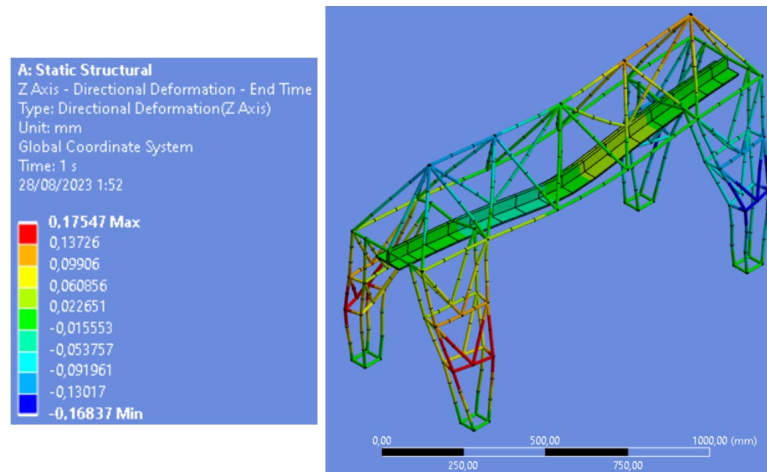


Figure 11. Z-Axis Deformation

Direct Stress refers to axial Stress in a truss rod, manifested as compressive or tensile Stress. The red section of the rod experiences the highest Stress, measured at 124.81 MPa, while the blue section experiences the lowest Stress, recorded at -73.16 MPa. Figure 12 illustrates the design of the Truss Gantry Crane, which has been subjected to direct stress testing.

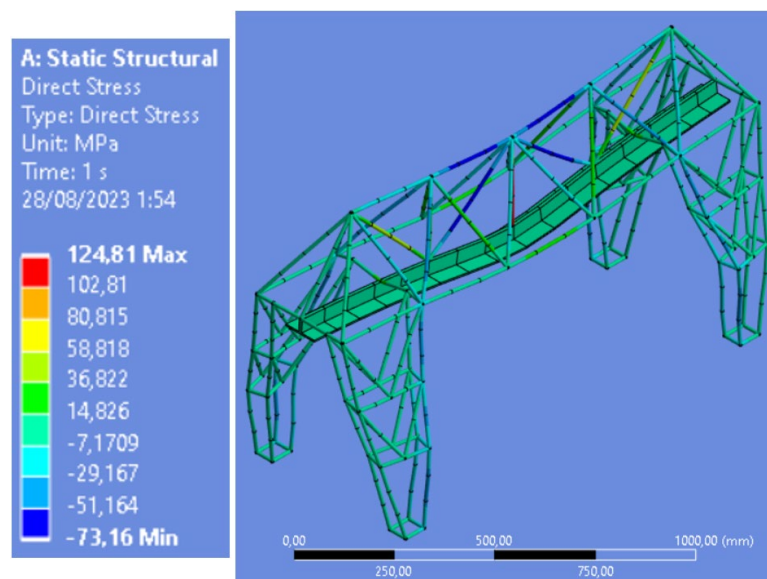


Figure 12. Direct Stress Test Results

Figure 13 shows the force reaction simulation to find the support reaction in the design. The largest support reaction produced occurs on the Y axis, with a value of 14058 N. The support reaction on the X axis has a value of $6,0635 \times 10^{-10}$ N, and the Z axis has a value of $2,5644 \times 10^{-9}$ N. Due to their minimal magnitude, these numbers are considered negligible and, therefore, not taken into account.

The graph in Figure 14 shows the shear force, bending moment, and Deflection diagram on the T-Beam bar. The leftmost part of the diagram is the zero point of the bar, while the rightmost part is the end point of the bar. The simulation results are

calculated at 16 points; the meshing size influences the number of these points. Shear force is the force applied vertically to a bar. The greatest shear force occurs at a distance of 938.33 mm upwards with a value of 2016.7 N. Meanwhile, the minimum value of shear force is 426.65 N at a distance of 350 mm downwards.

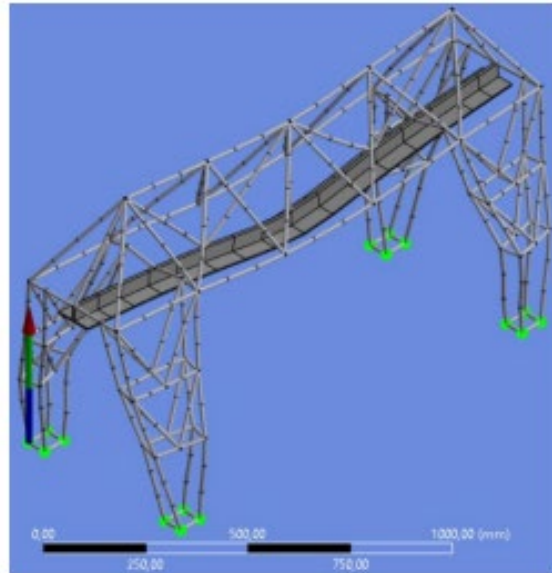


Figure 13. Force Reaction Simulation Results

A bending moment is the moment that occurs due to the support on the bar that produces a moment. The minimum value is $1,6806 \times 10^{-5}$ Nmm upwards, for a maximum value of 3.7931×10^{-5} Nmm. Furthermore, Deflexion is the bending Deformation that occurs in a bar, and the value obtained is the distance between the Deformation and the straight line. The largest value is 0.62468 mm at a distance of 745 mm downwards.

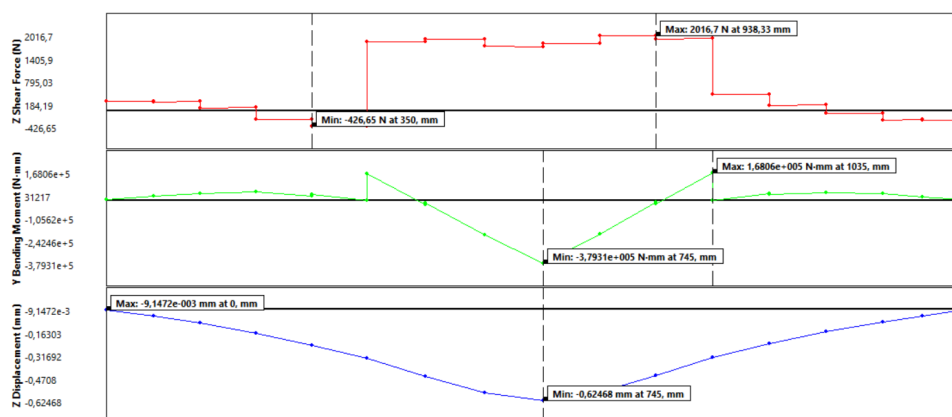


Figure 14. Shear Force, Bending Moment, and Deflection Diagram

3.2. Modal Analysis

After performing the analysis on Static Structural, the next step is to perform Modal analysis. This analysis is performed to find the natural frequency of the design.

This analysis is performed on the Y axis, with six modes, to find the frequency in each mode. Data such as material, load, and support use data from Static Structural. Figure 15 illustrates a sample result from one of the modes in the Modal test. The results of the Modal test on six Modes are shown in Table 2.

Table 2. Modal Test Results on 6 Modes

Mode	Frequency (Hz)
1	23,94
2	33,428
3	50,214
4	58,526
5	68,202
6	94,43

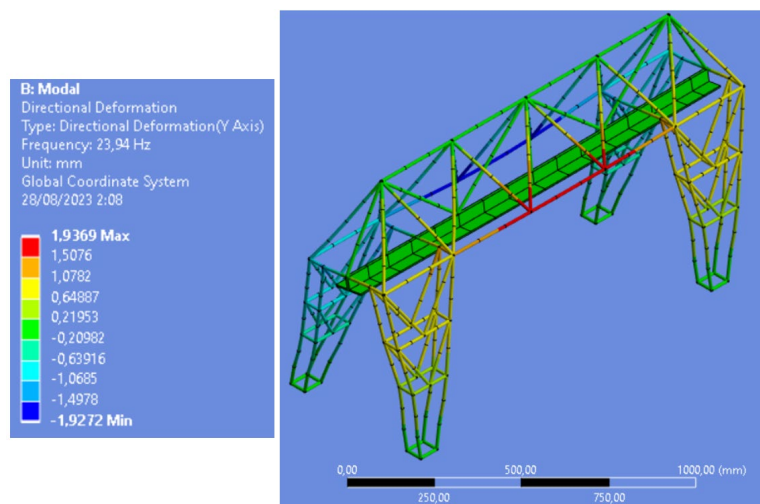


Figure 15. Modal Test Results in Mode 1

3.3. Harmonic Response Analysis

After obtaining the natural frequency values from the modal simulation, the next step is to determine the vibration response of the Truss Gantry Crane for a given natural frequency. The analysis data from the modal test will be reused in the harmonic response analysis. A concentrated load is applied at the midpoint of the Truss Gantry Crane in the negative Y-axis direction (downward). The modal test revealed a frequency range from 23.94 Hz (the lowest value) to 94.43 Hz (the highest value). Therefore, the frequency range used for the analysis is from 0 Hz to 100 Hz, resulting in 100 calculated solutions. Figure 16 illustrates the results of the frequency response test on each of the X, Y, and Z axes. The frequencies with the highest amplitudes on the X, Y, and Z axes are 24 Hz, 68 Hz, and 50 Hz, respectively. The combined Amplitude versus Frequency graph or phase angle graph is shown in Figure 17.

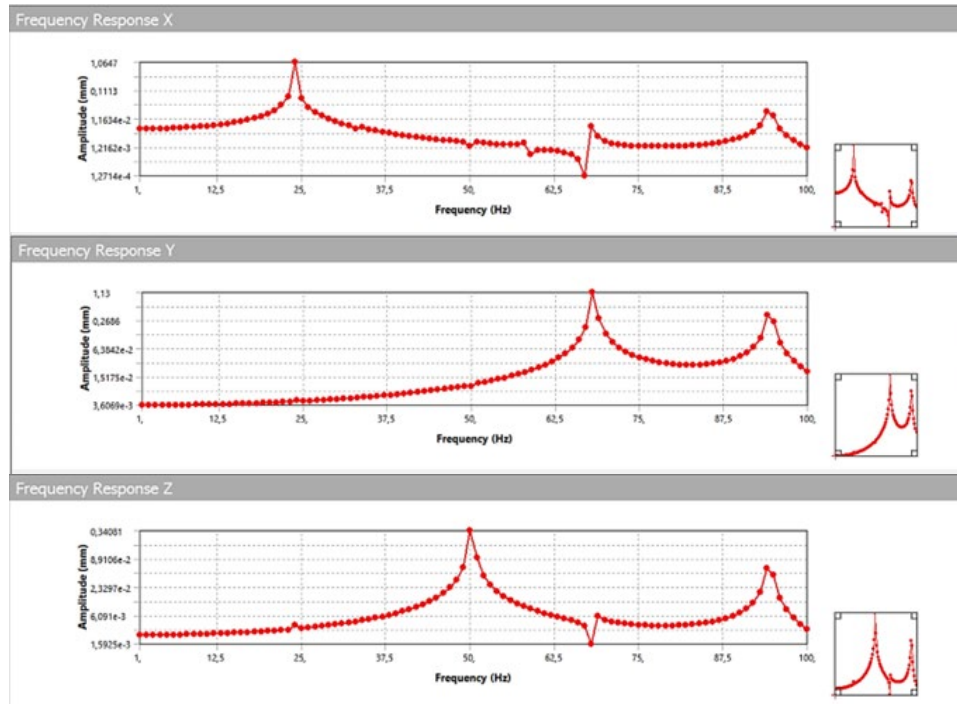


Figure 16. Frequency Response Test Results

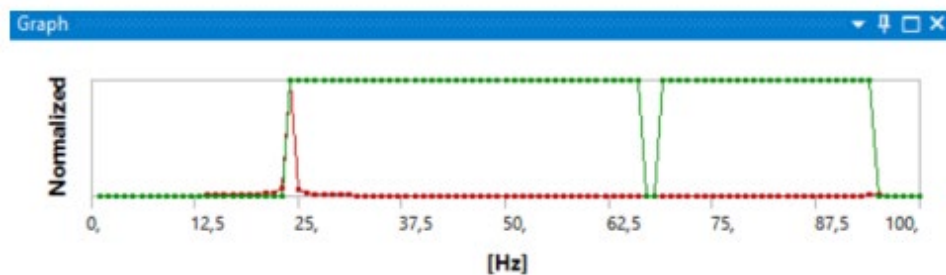


Figure 17. Graph of Normalization against frequency

IV. CONCLUSION

Based on the results of the analysis that has been carried out, there are several conclusions obtained:

1. The Static Structural results found that the maximum load permitted by this design is 13714.5 N or around 1400 kg. At a load of 13714.5N, or around 1400 kg, the stress in the Truss Gantry Crane is 125 MPa. Therefore, it is safe because the maximum stress allowed is 125 MPa with a Safety Factor value of 2 and does not reach the yield strength of the metal used, which is 250 MPa.
2. The modal analysis and Harmonic Response simulation reveal that the Truss Gantry Crane's operation should not exceed 24 Hz, 50 Hz, 68 Hz. Operating at these frequencies risks causing resonance, which can potentially damage the crane. This finding underscores the need for caution during operation.
3. The addition of load results in a linear increase in tension experienced by the crane, as described by the equation $y = 0,0091x + 0,198$. This understanding of the crane's behavior under load is crucial for safe and efficient operation.

REFERENCES

- Augustyn, M., & Barski, M. (2024). Estimation of the Wind Load Required to Cause the Overturning of a Gantry Crane, Comparing Different Structures of the Main Horizontal Girder. *Applied Sciences (Switzerland)*, 14(3). <https://doi.org/10.3390/app14031092>
- Canonsburg, T. D. (2013). ANSYS Mechanical APDL Structural Analysis Guide. ANSYS, Inc, 3304, 724-746. <http://www.ansys.com>
- Chen, T. (2020). Key Erection Techniques for Two-Panel Large Segments of Three-Main-Truss Steel Girder. *Bridge Construction*, 50(6).
- Chen, X., & Liu, Y. (2014). Finite element modeling and simulation with ANSYS workbench. In *Finite Element Modeling and Simulation with ANSYS Workbench*. <https://doi.org/10.1201/b17284>
- Guan, Y., Chen, T., Tang, Y., & Song, H. F. (2016). Techniques for segmental assembling of main ship channel bridge of Hutong Changjiang River Bridge. *Bridge Construction*, 46(6).
- Sunainah, A., & Sutantra, I. N. (2018). Analisis dan Redesign Kekuatan Struktur pada Girder Overhead Crane 6.3 Ton. *Jurnal Teknik ITS*, 7(1).
- Wibawa, L. A. N. (2020). DESAIN DAN ANALISIS TEGANGAN STRUKTUR CRANE KAPASITAS 10 TON MENGGUNAKAN METODE ELEMEN HINGGA. *Jurnal Muara Sains, Teknologi, Kedokteran Dan Ilmu Kesehatan*, 4(2). <https://doi.org/10.24912/jmstkik.v4i2.7006>