Journal of Vocational Applied Research and Studies 3, Issue 1 (2026) 54-65

Journal of Vocational Applied Research and Studies

Journal homepage: https://journal.uny.ac.id/v3/jvars/index
ISSN: 3048-1864

Optimization of the Structural Strength of a 130-Ton Hoist Frame Using Finite Element Analysis

Raffi Zidane Maulana[1], Ardani Ahsanul Fakhri[1,*]

[1]Department of Mechanical and Automotive Engineering, Faculty of Vocational, Universitas Negeri Yogyakarta, 55281, Indonesia

ARTICLE INFO

Article history:
Received: 30 April 2026
Received in revised form: 14 May 2026
Accepted: 29 May 2026
Available online: 23 June 2026

Keywords:
Frame hoist; Finite Element Analysis; Safety factor; Haunch plate; Stiffener plate

ABSTRACT

This study aims to: (1) simulate a 130-ton hoist frame structure using the Finite Element Analysis method, (2) analyze the distribution of stress, strain, and deformation occurring in the hoist frame structure based on the simulation results, and (3) determine the safety factor for each design variation and determine the most optimal design based on the simulation results. This study used the Finite Element Analysis (FEA) method using ANSYS Mechanical software to analyze the behavior of the hoist frame structure under a static load of 130 tons. The research stages included three-dimensional modeling of the hoist frame structure, meshing, selecting the material, boundary conditions, static loading, and static structural simulation to obtain stress, strain, deformation, and safety factor distributions. The model variations analyzed included the original frame, a frame with stiffener plate, a frame with haunch plate, and a combination of a haunch plate and stiffener plate. The structural material used SS400 structural steel based on the JIS G3101 standard. The results show that the FEA-based simulation is capable of displaying the stress, strain, deformation, and safety level distributions in detail. The addition of a stiffener plate and haunch plate is proven to reduce the maximum stress and increase the safety factor. The combined haunch and stiffener plate model produces the best performance, with a maximum stress of 127.29 MPa, a maximum strain of 0.00066711 mm/mm, a maximum deformation of 0.7801 mm, and a minimum safety factor of 1.8461.

1. Introduction

Crane hoists are lifting devices widely used in the industrial, construction, port, and water resources infrastructure sectors to move heavy loads safely and efficiently. In dam gate systems, crane hoists serve as key components in the process of raising and lowering gates during both operation and maintenance. The structural reliability of the hoist frame is critical because operational loads are transmitted through this structure. Structural failure can lead to operational disruptions, economic losses, and the risk of workplace accidents. Therefore, strength analysis and structural optimization of the hoist frame are necessary to ensure the safety and reliability of the lifting system.

Advances in computing technology have driven the use of Finite Element Analysis (FEA) as a widely adopted approach for evaluating the strength of engineering structures. This method allows

---

[*] *Corresponding author.*
E-mail address: ardaniahsanulfakhri@uny.ac.id

https://doi.org/10.21831/jvars.v3i1.3195

for a detailed analysis of stress distribution, strain, deformation, and safety factors before a structure is manufactured or modified. FEA can predict stress distributions in crane structures under specific loading conditions [1]. Changes in the structure's dimensions and configuration also significantly affect stress distributions and the safety level of crane construction [2]. These findings indicate that numerical simulations can serve as a tool for evaluating structural performance while reducing the need for costly experimental testing.

Various studies on crane structure optimization have been conducted using different approaches. Reinforcement using stiffeners and reinforcement elements has been reported to increase the strength of crane components based on FEA simulations [3]. Crane structure optimization that accounts for loading effects also plays a role in controlling stress and strain distributions [4]. The finite element method can be used to effectively evaluate the strength of gantry crane structures [5]. Furthermore, the use of haunches at steel structural joints can increase stiffness and improve load distribution in critical joint areas [6].

Similar issues were identified in a 130-metric-ton capacity hoist frame structure used as a water gate lifting system. Observation results indicated the presence of cracks and stress concentrations at the joint areas and on the H-beam web, which have the potential to become critical points in the structure. If these conditions are not thoroughly evaluated, the risk of excessive deformation or structural damage may increase as the number of operating cycles rises. Therefore, an analytical approach is needed that can evaluate structural behavior in detail while determining the most effective reinforcement configuration.

Based on the above, this study focuses on developing a safer and more efficient crane hoist structure design through a numerical approach based on FEA. This study aims to analyze the distribution of stress, strain, deformation, and safety factors in a 130-metric-ton hoist frame structure using ANSYS Mechanical, as well as to evaluate the effect of adding haunch plates, stiffener plates, and a combination of both on structural performance. The results of this study are expected to serve as a technical basis for design decision-making and the optimization of hoist frame structures in industrial applications and water resource infrastructure.

2. Method

This study employs a descriptive quantitative approach using the Finite Element Analysis (FEA) method to analyze and optimize the structural strength of a 130-metric-ton hoist frame. The analysis was conducted using ANSYS Mechanical software under static loading conditions. The parameters evaluated include stress distribution, strain distribution, total deformation, and the safety factor. The simulation results from each design variation were compared to determine the most optimal structural configuration for improving the strength and safety of the hoist frame.

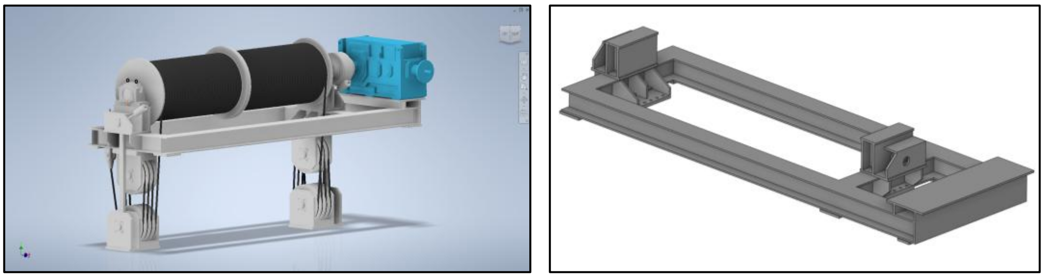
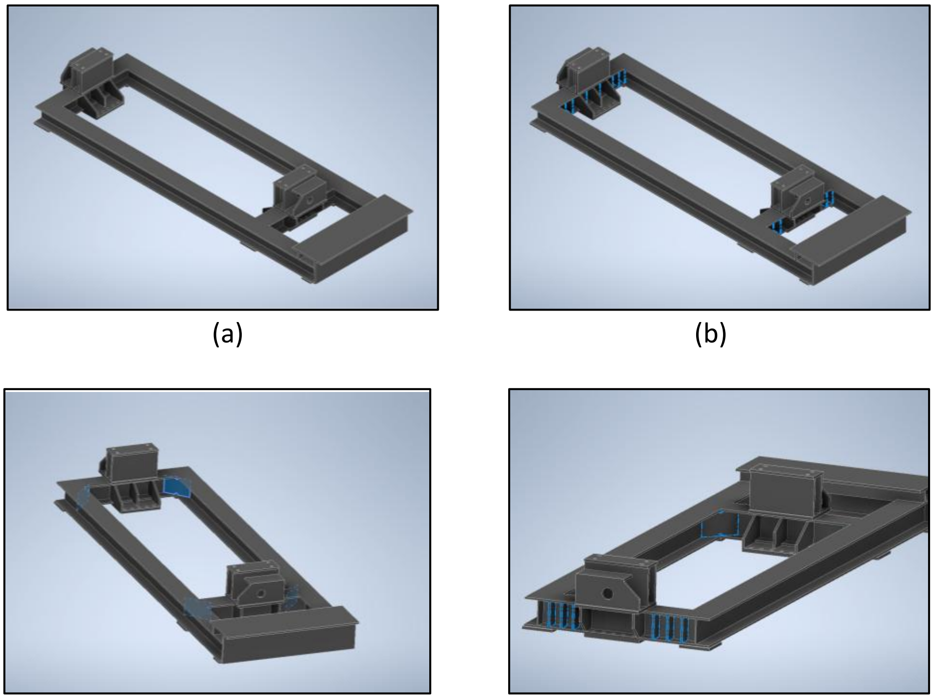


Fig. 1. 3D structure of a hoist frame

The subject of this study is a 130-metric-ton hoist frame structure used as a lifting system for a dam gate. A three-dimensional (3D) model of the structure was created based on its actual geometry using Autodesk Inventor Computer-Aided Design (CAD) software, as shown in Figure 1.

Four design variations were analyzed: (a) the original frame, (b) the frame with a stiffener plate, (c) the frame with a haunch plate, and (d) the frame with a combination of a haunch plate and a stiffener plate. These design variations were used to evaluate the effect of reinforcing elements on the structural behavior when subjected to operational loads. The design variations are shown in Figure 2.

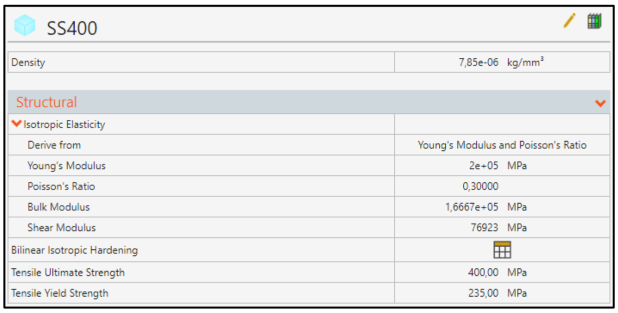


(a) (b)

(c) (d)

Fig. 2. Variations in hoist frame designs

The material used in all simulation models is SS400 structural steel, in accordance with the JIS G3101 standard. SS400 was selected because it is widely used in steel construction that requires a combination of strength, ease of fabrication, and cost-effectiveness. The mechanical properties of the material entered into the simulation include a yield strength of 235 MPa, a Young's modulus of 200 GPa, a Poisson's ratio of 0.30, and an ultimate tensile strength of 400 MPa. These material data were entered into ANSYS Mechanical to represent the material's behavior during the simulation process. The SS400 material data in ANSYS are shown in Figure 3.



SS400

Density	7,85e-06 kg/mm³
Structural	
Isotropic Elasticity	
Derive from	Young's Modulus and Poisson's Ratio
Young's Modulus	2e+05 MPa
Poisson's Ratio	0,30000
Bulk Modulus	1,6667e+05 MPa
Shear Modulus	76923 MPa
Bilinear Isotropic Hardening	▦
Tensile Ultimate Strength	400,00 MPa
Tensile Yield Strength	235,00 MPa

Fig. 3. SS400 material data in ANSYS

The mesh convergence process was performed to determine the optimal mesh size. Face sizes of 50 mm, 45 mm, 40 mm, 35 mm, 30 mm, 25 mm, and 20 mm were tested by comparing the maximum stress and maximum strain from the simulation results. The mesh convergence results show that a face size of 25 mm produces relatively stable values compared to other mesh size variations. A body size of 65 mm was used, taking into account the limitations on the number of nodes and elements in the ANSYS Student Version. Thus, this mesh size was selected because it provides a balance between simulation accuracy, numerical stability, and computational efficiency. The meshing results are shown in Figure 4.

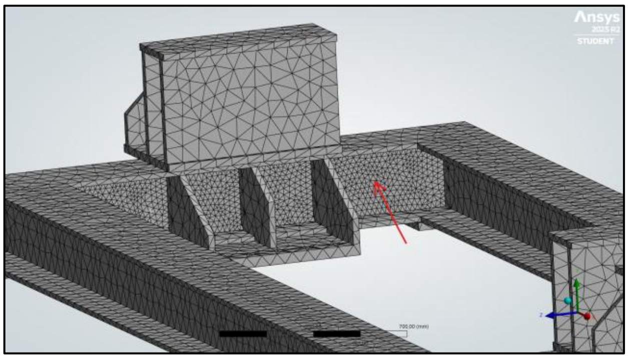
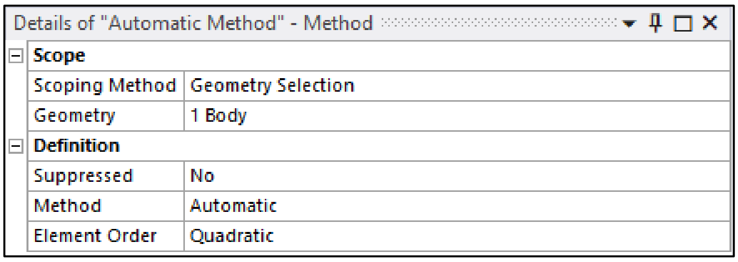


Fig. 4. Hoist frame meshing with a face size of 25 mm and a global mesh size of 65 mm

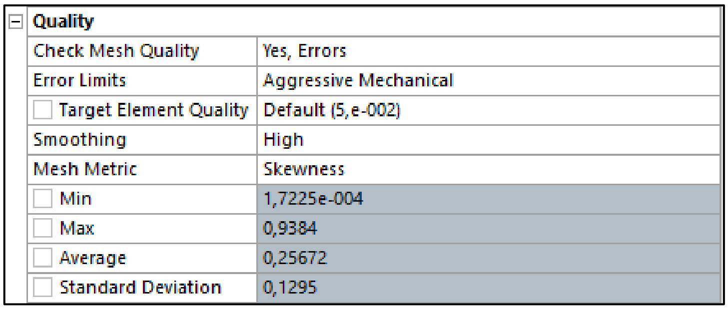
Meshing in this study used the Automatic Method with quadratic element order in ANSYS Mechanical. The Automatic Method was chosen because it is capable of generating a mesh that adapts to the complex geometry of the hoist frame structure, particularly in the areas of joints, haunch plates, and stiffener plates. This method allows the meshing process to be performed efficiently and stably on structural models with a wide variety of shapes and joints. Meanwhile, quadratic elements were used to better represent the distribution of stress and strain compared to linear elements, as they have additional nodes on each side of the element. Details of the Automatic Method are shown in Figure 5.


Details of "Automatic Method" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Automatic
Element Order	Quadratic

Fig. 5. Details of the Automatic Method in ANSYS

Mesh quality was evaluated using the skewness parameter to ensure the suitability of the mesh elements for numerical simulation. The average skewness value obtained was 0.256, with a maximum value of 0.938. Based on the mesh quality criteria in ANSYS Mechanical, this value still falls within the “acceptable” category; therefore, the mesh is deemed suitable for structural analysis using the FEA method. The skewness results from ANSYS are shown in Figure 6.



Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanical
☐ Target Element Quality	Default (5,e-002)
Smoothing	High
Mesh Metric	Skewness
☐ Min	1,7225e-004
☐ Max	0,9384
☐ Average	0,25672
☐ Standard Deviation	0,1295

Fig. 6. Skewness results in ANSYS

The structural analysis accounts for the effect of gravity (self-weight) with an acceleration of 9.81 m/s^2 acting in the direction of the negative Y-axis in ANSYS Mechanical. Gravity was enabled to represent the direction of Earth’s gravity, ensuring that the simulation conditions closely approximate actual field conditions. The effect of self-weight must be considered because the hoist frame structure has large dimensions and mass, meaning that gravity contributes to the distribution of stress, deformation, and loading at the joints and on the H-beam webs. The direction of gravity is shown in Figure 7.

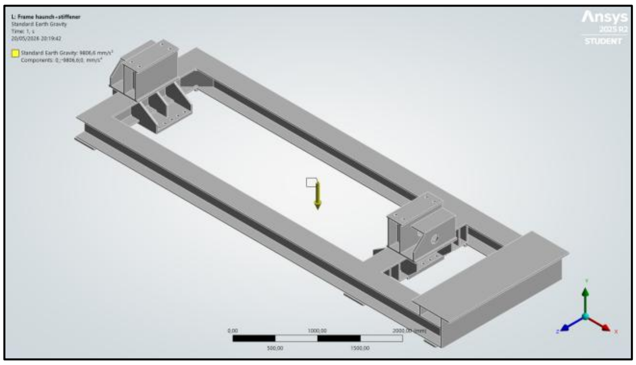
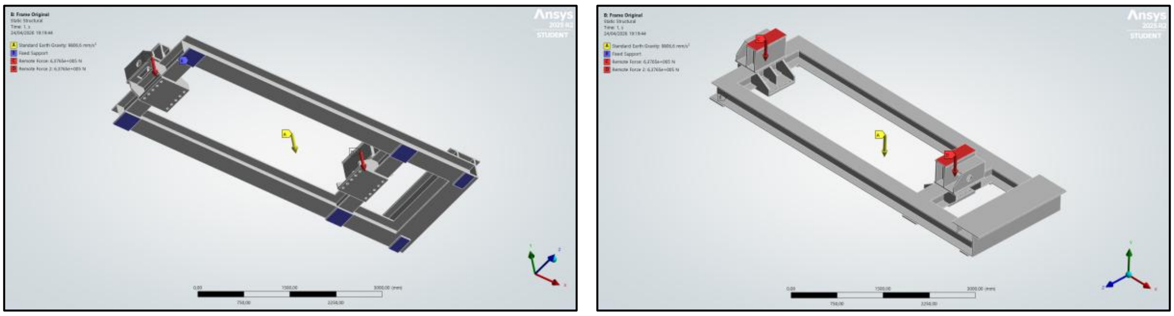


Fig. 7. Standard Earth gravity in ANSYS

Boundary conditions were applied by providing fixed supports at the structure's support points to represent the actual installation conditions of the hoist frame. The loading was based on a maximum capacity of 130 metric tons, which was converted into forces using the acceleration due to gravity. The total force obtained is 1,275,300 N and is used as the primary load in the simulation. The load distribution must account for the lifting system's configuration to ensure more accurate analysis results [7]. The lifting system uses two wire ropes, so the load is assumed to be evenly distributed across the two loading points. Each point receives a load of 637,650 N, applied at the lifting locations according to the structure's operating conditions. The conditions for (a) fixed support and (b) loading are shown in Figure 8.



(a) (b)

Fig. 8. Boundary condition for the frame hoist

The simulation was performed using the Static Structural module in ANSYS Mechanical to analyze the response of the hoist frame structure to a static load of 130 metric tons based on the FEA method. The simulation results—including the distribution of von Mises stress, strain, total deformation, and safety factor—were used as parameters for evaluating the structure's performance. The analysis was conducted by comparing the maximum stress to the yield strength of SS400 material and evaluating the levels of strain, deformation, and safety margin for each design variation. Subsequently, the simulation results were quantitatively compared to determine the most optimal structural configuration based on its ability to reduce stress, minimize deformation, and improve the safety factor.

3. Results and Discussion

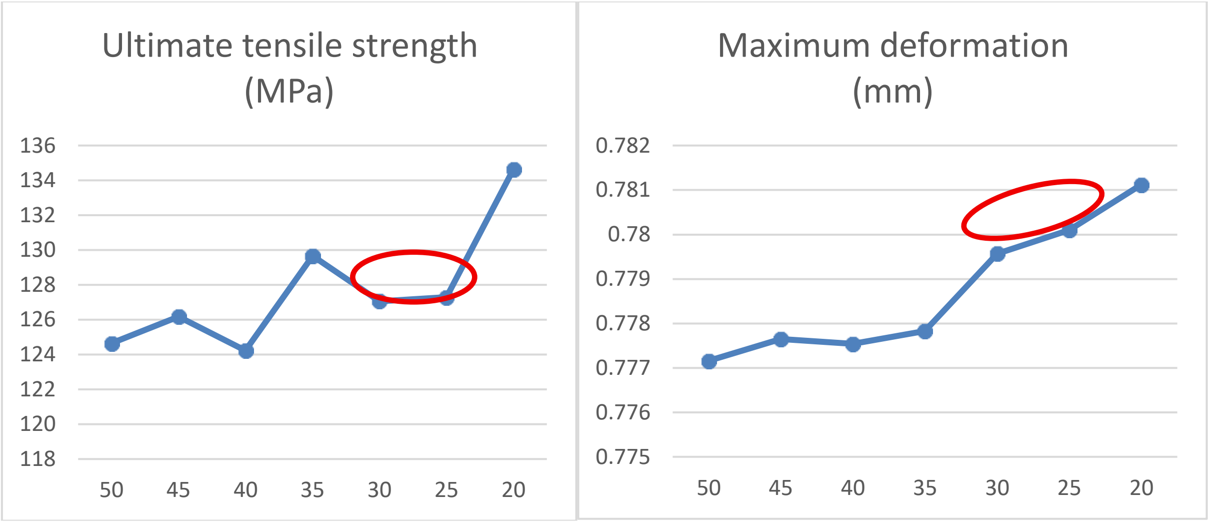
3.1. Results

3.1.1. Mesh convergence results

Mesh convergence testing was conducted to determine the mesh size that provides a balance between the accuracy of the simulation results and computational efficiency. Several variations in mesh size were analyzed by comparing the maximum stress and maximum strain. The test results showed that a face size of 25 mm was in a relatively convergent state, as indicated by small changes in values and stable simulation results. Therefore, a face size of 25 mm was selected as the mesh size for all simulations because it was deemed capable of producing accurate and consistent results. The mesh convergence data are presented in Table 1.

Table 1. Mesh convergence results

Face sizing (mm)	Ultimate tensile strength (MPa)	Maximum deformation (mm)	Number of elements	Number of nodes
50	124.64	0.77716	43021	84511
45	126.18	0.77765	43420	85267
40	124.23	0.77754	44659	87557
35	129.66	0.77783	46090	90145
30	127.06	0.77957	48019	93827
25	127.29	0.78010	52302	102011
20	134.63	0.78112	59935	115406



(a) (b)

Fig. 9. Graphs showing mesh convergence results: (a) the relationship between face sizing and maximum stress. and (b) the relationship between face sizing and maximum strain

3.1.2. Stress distribution results

The simulation results show that each design variation produces a different maximum stress. The original frame model produces a maximum stress of 182.54 MPa. while the model with a stiffener plate produces 143.96 MPa. In the model with a haunch plate. the maximum stress decreased to 129.28 MPa. The model combining a haunch plate and a stiffener plate produced the lowest maximum stress of 127.29 MPa. Thus. the combined model was able to reduce the maximum stress by 30.27% compared to the original model. The stress distribution data are presented in Table 2.

Table 2. Stress Distribution Results

No.	Model variations	Maximum stress (MPa)	Minimum stress (MPa)	Difference in the decline (MPa)	Percentage decrease (%)
1.	Frame original	182.54	0.011735	-	-
2.	Frame with stiffener plate	143.96	0.018308	38.58	21.13
3.	Frame with haunch plate	129.28	0.006722	53.26	29.18
4.	Frame with haunch plate dan stiffener plate	127.29	0.005399	55.25	30.27

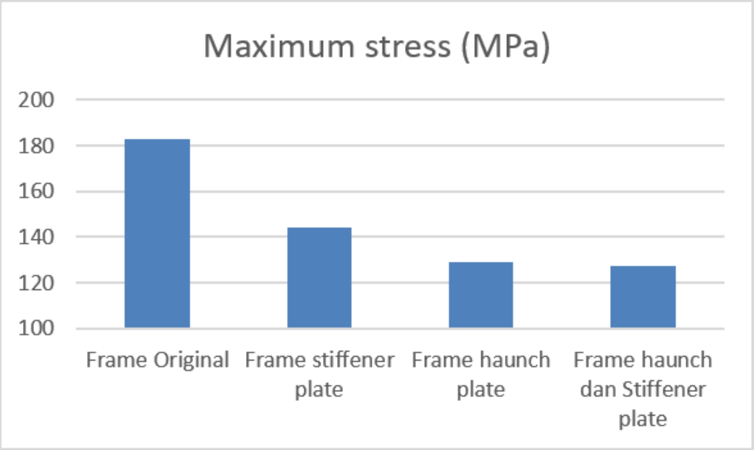
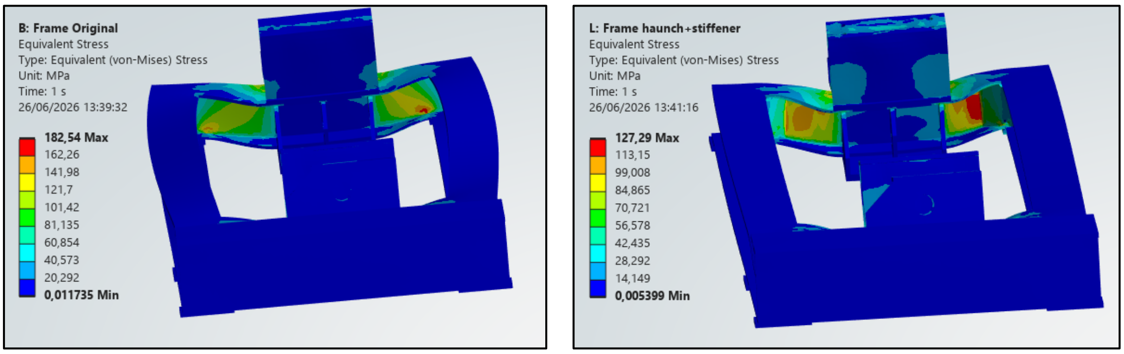


Fig. 10. Graph of the stress distribution results



(a) (b)

Fig. 11. Stress distribution results: (a) original frame and (b) frame combining a haunch plate and a stiffener plate

3.1.3. Strain distribution results

The maximum strain value showed a decreasing trend across all design variations. The original frame model produced a maximum strain of 0.00091612 mm/mm. The addition of a stiffener plate reduced the maximum strain to 0.00072120 mm/mm. while the use of a haunch plate resulted in a maximum strain of 0.00066977 mm/mm. The lowest strain value was obtained in the model combining a haunch plate and a stiffener plate. at 0.00066711 mm/mm. These results indicate that structural modifications can increase stiffness. resulting in a more even strain distribution. The strain distribution data are presented in Table 3.

Table 3. Strain distribution results

No.	Model variations	Maximum strain (mm/mm)	Minimum strain (mm/mm)
1.	Frame original	0.00091612	1.3365e-7
2.	Frame with stiffener plate	0.00072120	1.0978e-7
3.	Frame with haunch plate	0.00066977	1.0912e-7
4.	Frame with haunch plate dan stiffener plate	0.00066711	8.049e-8

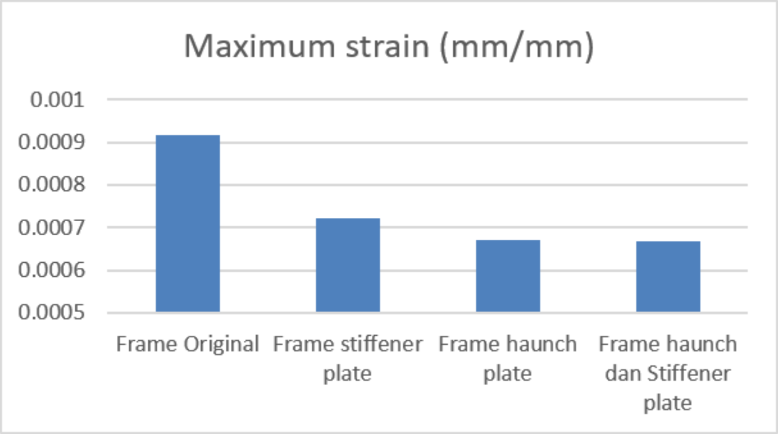
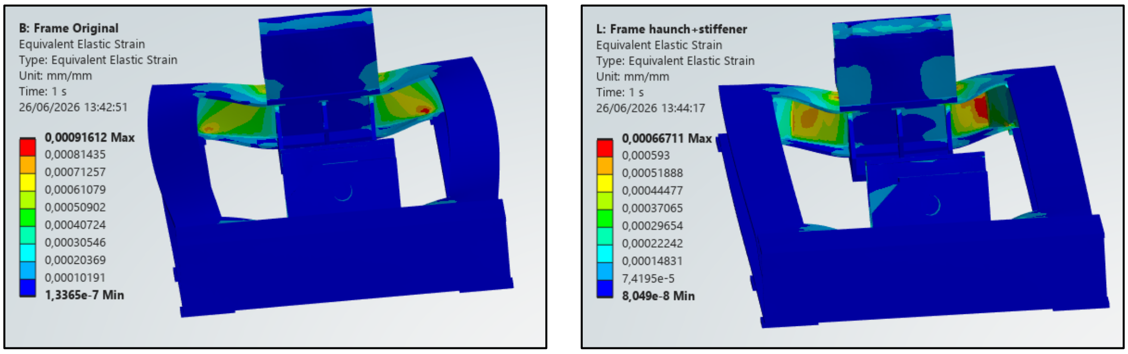


Fig. 12. Graph of the strain distribution results



(a) (b)

Fig. 13. Strain distribution results: (a) original frame and (b) frame combining a haunch plate and a stiffener plate

3.1.4. Deformation results

The deformation simulation results show that the original frame model experienced a maximum deformation of 1.4062 mm. After adding a stiffener plate. the maximum deformation decreased to 1.0293 mm. The model with a haunch plate resulted in a maximum deformation of 0.73291 mm. while the model combining a haunch plate and a stiffener plate resulted in a maximum deformation of 0.7801 mm. In general. all modification variations were able to reduce deformation compared to the original model. The deformation results are presented in Table 4.

Table 4. Deformation results

No.	Model variations	Maximum deformation (mm)	Minimum deformation (mm)
1.	Frame original	1.4062	0
2.	Frame with stiffener plate	1.0293	0
3.	Frame with haunch plate	0.73291	0
4.	Frame with haunch plate dan stiffener plate	0.7801	0

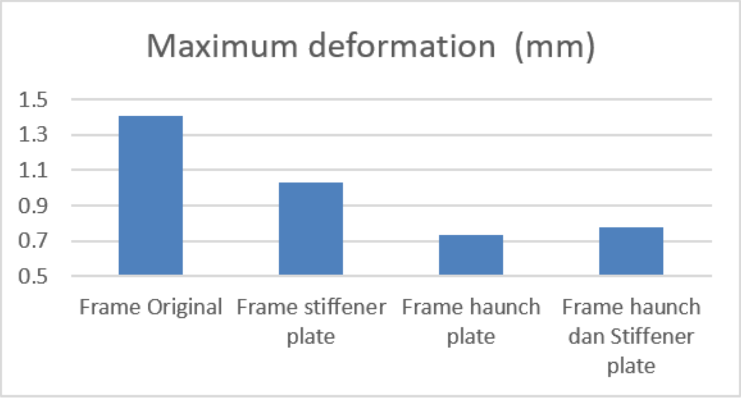
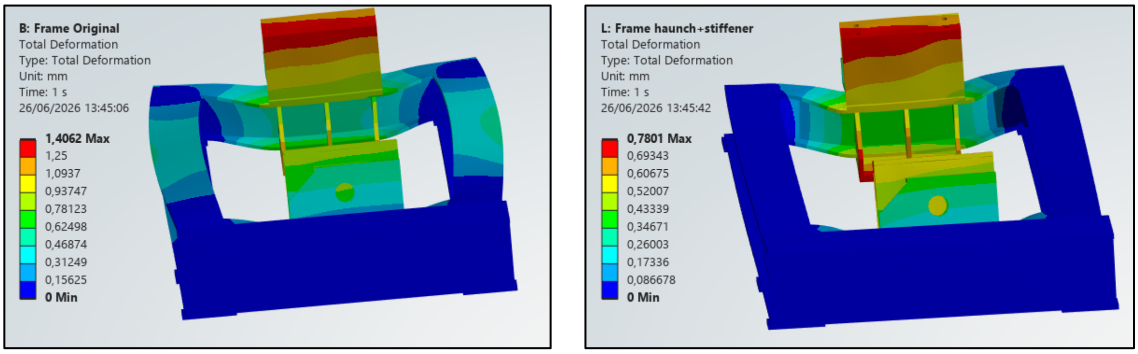


Fig. 14. Deformation results graph



(a) (b)

Fig. 15. Deformation results: (a) original frame and (b) frame combining a haunch plate and a stiffener plate

3.1.5. Safety factor results

A safety factor analysis was conducted to evaluate the structural safety level under an operational load of 130 metric tons. The original frame model yielded a minimum safety factor of 1.2874. After adding a stiffener plate. the safety factor increased to 1.6324. The model with a haunch plate yielded a safety factor of 1.8178. while the model combining a haunch plate and a stiffener plate yielded the highest safety factor of 1.8461. These results indicate that all structural modifications improved the safety level compared to the original model. The safety factor results are presented in Table 5.

Table 5. Safety factor results

No.	Model variations	Minimum safety factor	Percentage increase (%)
1.	Frame original	1.2874	-
2.	Frame with stiffener plate	1.6324	26.80
3.	Frame with haunch plate	1.8178	41.20
4.	Frame with haunch plate dan stiffener plate	1.8461	43.39

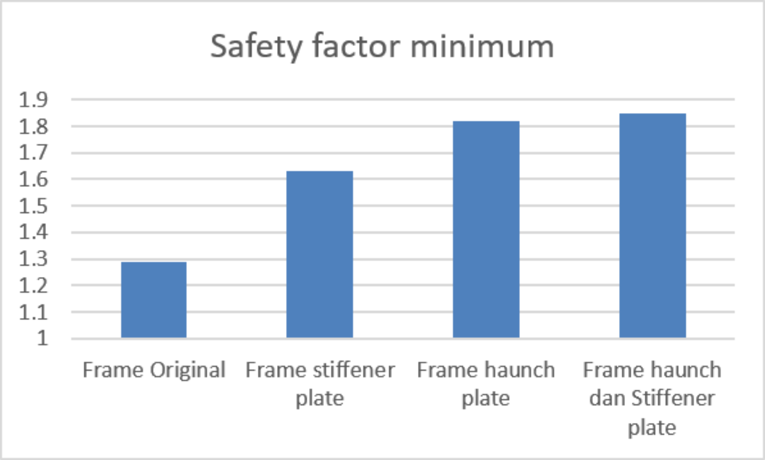
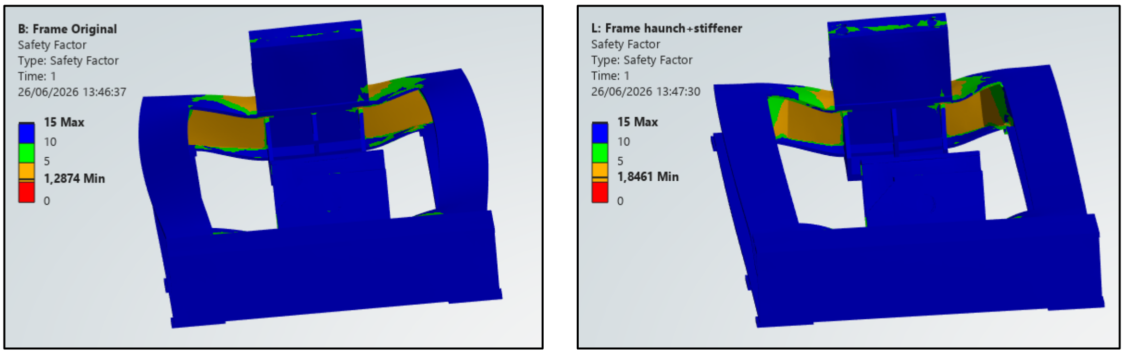


Fig. 16. Graph of safety factor results



(a) (b)

Fig. 17. Safety factor results: (a) original frame and (b) frame combining a haunch plate and a stiffener plate

3.2. Discussion

The mesh convergence results show that refining the mesh size has different effects on the maximum stress and maximum strain. The maximum stress still fluctuates because this parameter is sensitive to local stress concentrations at the joint area. haunch plate. and stiffener plate. Meanwhile. the maximum deformation shows relatively small changes. indicating that the structure’s global response has reached a stable state. Based on these results. a face size of 25 mm was selected because it produces relatively convergent stress and deformation values while maintaining good computational efficiency. An evaluation of mesh quality using the skewness parameter also indicates that the mesh falls within the “acceptable” category for finite element analysis.

The simulation results show that variations in structural geometry significantly affect stress distribution and safety factors. The original model produced the highest maximum stress because the load was concentrated at the joint area and the H-beam web. resulting in greater stress concentrations. The addition of a stiffener plate increased local stiffness and reduced maximum stress. while the use of a haunch plate expanded the force transfer area. leading to a more even load distribution. The combined model with haunch plates and stiffener plates demonstrated the best performance based on stress and safety factor indicators. with a maximum stress of 127.29 MPa—a 30.27% reduction compared to the original model. This model also produces a maximum deformation of 0.7801 mm and the highest safety factor of 1.8461—a 43.39% increase compared to the original model. Although the lowest deformation was obtained in the haunch plate model. the difference in deformation relative to the combined model is relatively small; therefore. the combined model was selected because it provides the best balance between stress reduction. increased stiffness. and an improved safety factor.

Overall. the combination of a haunch plate and a stiffener plate is the most optimal configuration for improving the performance of a 130-metric-ton hoist frame under static loading conditions. However. this study still has limitations because the analysis was conducted using a static structural approach and therefore did not consider the effects of dynamic loads. vibrations. and cyclic loading during lifting operations. Furthermore. the use of bonded contact and fixed supports assumes ideal joint conditions. so the flexibility of actual joints is not fully represented in the simulation. Further research is recommended to evaluate the effects of dynamic loads. fatigue. variations in joint conditions. and experimental validation on actual structures.

4. Conclusions

A structural simulation of a 130-metric-ton hoist frame using the FEA method in ANSYS Mechanical was successfully conducted on four model variations: the original frame. a frame with a stiffener plate. a frame with a haunch plate. and a frame with a combination of a haunch plate and a stiffener plate. The simulation results show that the addition of reinforcing elements reduces maximum stress and increases the safety factor compared to the original model.

The highest stress. strain. and deformation distributions occurred in the H-beam web area and the hoist mount. which bear direct loading. The model combining haunch plates and stiffener plates produced a maximum stress of 127.29 MPa. a maximum strain of 0.00066711 mm/mm. and a maximum deformation of 0.7801 mm. The 30.27% reduction in maximum stress indicates that the combination of stiffening elements is capable of increasing structural stiffness and improving load distribution in critical areas.

The combined haunch plate and stiffener plate model yields the highest safety factor of 1.8461. representing a 43.39% increase compared to the original model. This improvement demonstrates that the addition of reinforcing elements can increase the structural safety margin. reduce working stress. and improve load distribution in the critical areas of the hoist frame. Therefore. the combined configuration of haunch plates and stiffener plates is recommended as the optimal design for static loading conditions. with the caveat that dynamic and experimental validation remain necessary in future research.

Conflict of interest

The author declares that there are no conflicts of interest in this study.

Financing

This study did not receive any external funding.

Acknowledgement

The author would like to express gratitude to the Applied Bachelor’s Degree Program in Mechanical Engineering. Faculty of Vocational Studies. Yogyakarta State University. as well as the companies that provided support and the necessary data during the course of this research.

References

[1] X. Chen and Y. He. “Finite element analysis of stress distribution on portal crane structure under loading conditions.” Journal of Mechanical Engineering Science. vol. 236. no. 8. pp. 4215–4226. 2022.

[2] M. Abdullah. A. Rahman. and D. Putra. “Effect of structural dimension variation on stress distribution in crane structures using finite element analysis.” International Journal of Mechanical Engineering and Applications. vol. 11. no. 3. pp. 145–152. 2023.

[3] L. Cheng. Z. Wang. and H. Liu. “Structural optimization of crane components using stiffener and reinforcement methods based on finite element analysis.” Engineering Structures. vol. 275. p. 115248. 2023.

[4] Y. Huang. Q. Zhang. and X. Li. “Finite element analysis and structural optimization of crane systems under dynamic loading.” Applied Sciences. vol. 11. no. 14. p. 6543. 2021.

[5] A. A. Kharisma and D. M. Yanuar. “Simulasi kekuatan rangka gantry crane single girder menggunakan metode elemen hingga (finite element analysis).” Jurnal Ilmiah Teknologi dan Rekayasa. 2023.

[6] S. Maruyama. “Effect of the haunch angle and stiffener types on column-beam connection behaviour under static loading.” Civil Engineering Beyond Limits. vol. 3. no. 4. pp. 1–13. 2022.

[7] T. Prahasto. M. T. S. Utomo. O. Kurdi. and P. D. Nababan. “Analysis of load variations and operational conditions on the standardization of spreader bar geometric design.” Civil Engineering Journal and Sustainable Research. 2025.