The Fifth International Conference on Government Education Management and Tourism
(ICoGEMT 5)
Bandung, Indonesia, January, 24th, 2026

ANALYSIS OF LIGHT STEEL CANOPY FRAME STRUCTURE BASED ONSOLIDWORKS SIMULATION FOR FRAME DESIGN OPTIMIZATION

[1]**Rhovy Nurhazani**,[*2]**Jana Hafiza**,[3]**Muchlisinalahuddin**
[4]**Yuni Vadila**,[5]**Reyhan Stevano**

[1,2,3,5]*Mechanical Engineering, Faculty of Engineering, Muhammadiyah University of West Sumatra*
Padang, Indonesia
[4]*Mechanical Engineering, Faculty of Engineering, Padang State University*
Padang, Indonesia

Author's email:
[1]*rhovynurhazani2642@gmail.com* [*2]*Janahafizaumsb@gmail.com*
[3]*muchlisinalahuddin.umsumbar@gmail.com;* [4]*yunifadilabkt@gmail.com*
[5]*reyhanstevano55@gmail.com*

**Corresponding author: Janahafizaumsb@gmail.com*

Abstract.*The lightweight steel canopy frame serves as the main structural element that must be able to withstand static loads safely and efficiently. However, canopy frame design is often carried out without adequate structural analysis, potentially causing excessive deformation and the risk of structural failure. The main problem in this study is the low stiffness and safety factor in the initial canopy frame design. As a solution, geometric modifications were made to the frame to improve structural performance. This study used a SolidWorks-based Finite Element Analysis (FEA) simulation method with static loading conditions. The parameters analyzed included Von Mises stress, displacement, strain, and safety factor with a load of 700 N. The simulation results showed that the design modification was able to significantly reduce stress and displacement and increase the safety factor. Modification Model 2 produced the smallest displacement of 1.00 mm and a safety factor of 2, so it is recommended as the most optimal and safe canopy frame design for application.*

Keywords: *Canopy Frame; Finite Element Analysis; Light Steel; Safety Factor; Von Mises Stress.*

1. INTRODUCTION

The use of lightweight steel in building construction is growing rapidly due to its structural efficiency, economy, and ease of fabrication and installation. Lightweight steel has the advantages of low density, high tensile strength, and corrosion resistance, making it widely used in lightweight structures, including canopy frames. Canopy frames serve as exterior protection from the sun, rain, and wind, and therefore must be designed with safety and adequate structural performance in mind.(Manurung et al., 2021).

Although visually simple, a lightweight steel canopy frame must be able to withstand a combination of dead load, wind load, and rain load in accordance with the provisions of SNI 1727:2020. Improper design has the potential to cause excessive deformation, instability, and even structural failure that can endanger users.(Purwanto, 2017)However, field practice shows that canopy frame design is often carried out empirically based on experience, without adequate structural analysis. This approach generally does not consider stress distribution, deformation behavior, or safety factors quantitatively, resulting in suboptimal designs and a risk of long-term failure.(Alvian et al., 2020).

The development of structural engineering technology has encouraged the use of Finite Element Analysis (FEA) as an analysis method capable of modeling structural behavior in detail and accurately. FEA is a numerical method that divides a structure into small elements to calculate the structure's response to loading as a whole. Important parameters such as stress distribution, Von Mises stress, deformation, internal forces, and safety factors can be analyzed with a high degree of accuracy. One software that supports this method is SolidWorks with the SolidWorks Simulation module, which enables the integration of geometric modeling and structural analysis in a single platform.(Diinil Mustaqiem, 2020).

The application of FEA to light steel canopy frames allows the identification of critical areas, prediction of potential failures, and evaluation of structural performance under various loading conditions more realistically than manual calculations.(Trahair, 2018). In addition, FEA supports the design optimization process to make the structure safer and more efficient. However, research related to FEA-based analysis of lightweight steel canopy frames that refers to SNI standards and uses SolidWorks is still limited. Therefore, this study was conducted to assess the structural performance of lightweight steel canopy frames using the FEA approach as a basis for effective, efficient, and applicable design recommendations in the field.(Desmarita Leni, 2022).

2. LITERATURE REVIEW

2.1 Light Steel

Light steel is a low-carbon steel-based material produced through the cold-formed steel (CFS) process, which is the formation of steel at room temperature without heating. This process results in increased strength and stiffness due to the strain hardening effect, so that light steel can still withstand structural loads even though it has a relatively thin thickness. To increase resistance to the environment, light steel is coated with zinc (galvanized) or a combination of zinc-aluminum (galvalume) which serves as protection against corrosion and extends the service life of the material, especially in outdoor applications such as canopy frames.(Purwanto, 2017).

In modern construction, lightweight steel is widely used because of its high strength-to-weight ratio. Its light weight reduces the dead load of the structure, while efficient profiles such as C-, U-, and Z-channels allow this material to perform optimally under bending and compression loads with more economical material usage. The tensile strength of lightweight steel is generally in the range of 550–650 MPa, depending on the quality and production process, so it can withstand significant structural forces even though its dimensions are thin. These characteristics make lightweight steel widely used in roof frames, canopy frames, ceilings, and other secondary structural elements.(Husnah et al., 2019).

The mechanical characteristics of lightweight steel are fundamental factors in structural behavior analysis. The main parameters used in the design and numerical analysis include an elastic modulus of 190–210 GPa, a maximum tensile strength of 550–650 MPa, a yield stress of approximately 550 MPa for the G550 type, and a Poisson's ratio of approximately 0.3. These parameters are used to model the stress–strain relationship, deflection, and safe loading limits of the structure in a Finite Element Analysis (FEA)-based analysis using SolidWorks Simulation. The combination of high stiffness and strength with a light mass makes lightweight steel highly suitable for canopy structures that receive wind and rain loads.(Stevens & Tedianto, 2018).

56Finite Element Analysis (FEA) is a numerical method that divides a structure into small elements connected by nodes, so that structural responses such as stress, deformation, and stability can be analyzed in detail. This method is particularly relevant for lightweight steel structures that have thin profiles and are susceptible to stress concentrations and local and global buckling. The FEA stages include geometric modeling, material definition, meshing, application of loads and boundary conditions, solving processes, and interpretation of results through postprocessing.(Muharni & Maulana, 2023).

In light steel structure analysis, several types of FEA analysis are used, including Linear Static Analysis to evaluate stress and deflection due to static loads, Buckling Analysis to determine critical buckling loads, Frequency (Modal) Analysis to avoid resonance, and Dynamic Loading Analysis to analyze the response to dynamic loads such as fluctuating winds. With the comprehensive application of FEA, the performance of light steel canopy frames can be thoroughly evaluated so that the resulting design is safer, more efficient, and more reliable before the construction stage.(Admin et al., 2022).

3. RESEARCH METHODS

This research falls into the engineering research category, focusing on structural analysis and design using a numerical simulation-based approach. This type of research was chosen because it aims to evaluate the structural performance of an engineering system and generate more optimal design recommendations based on the results of the technical analysis. The object of study in this research is a lightweight steel canopy frame, which is analyzed to determine the level of strength, stiffness, and safety of the structure in supporting the working load it receives. The research method used is a simulation-based analysis and design method using SolidWorks software, specifically the SolidWorks Simulation module. The stages to be carried out in this research are as shown in **Figure 1.**

Figure 1. Flow Chart

3. 1 Literature Study

The initial stage of the research began with a literature review related to canopy frame structures, the characteristics of lightweight steel materials, structural loading theory, and structural analysis methods using computer-based simulations. The literature used included scientific journals, mechanical and civil engineering textbooks, design standards, and relevant previous research. This literature review aimed to obtain a strong theoretical foundation and understand the appropriate analysis methods as a basis for the design and simulation process of canopy frame structures.

3.2 Identify the Problem

Following the literature review, issues related to the structural strength of lightweight steel canopy frames were identified. This identification included an analysis of potential structural weaknesses, material efficiency, and the likelihood of failure due to loading during operation. This stage is crucial for determining the research focus and formulating the analysis objectives to be achieved.

3.3 Determination of Initial Specifications and Dimensions

The next stage is determining the initial specifications and dimensions of the canopy frame. This determination is based on building requirements, literature studies, and applicable design standards. The specifications include the type of lightweight steel profile, frame element dimensions, bar spacing, and overall structural configuration. This stage aims to produce a realistic initial design that meets field conditions.

3.4 Making a Canopy Frame Model

Based on the predetermined specifications and dimensions, a three-dimensional model of the canopy frame was created using SolidWorks software. The model was created to closely mimic real-world conditions, including the construction form and framing system. This CAD model served as the primary basis for the structural simulation process.

3.5 Structure Simulation

After the frame model was completed, a structural simulation was performed using the simulation features in SolidWorks. At this stage, the model was assigned boundary conditions and loads consistent with the operating conditions of the canopy frame, such as self-weight, roof load, and environmental loads. The simulation was performed to obtain the structure's response to the applied loads.

3.6 Analysis of Simulation Results

The simulation results were analyzed to determine the stress distribution, deformation magnitude, and safety factor of the canopy frame. The simulation data was then compared against structural strength and safety criteria to assess the design's feasibility.

3.7 Design Optimization

If the analysis results indicate that the frame design does not meet safety or efficiency criteria, design modifications are made. This process is carried out iteratively until a more optimal design is achieved.

3.8 Final Design Determination

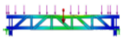
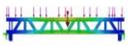
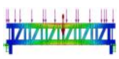
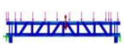
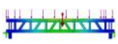
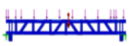
The final stage of the research is to determine the final design of the light steel canopy frame that meets the criteria for strength, safety and efficiency based on the results of simulations and analysis.

4. RESULTS AND DISCUSSION

4.1 Results

From the results of static simulations, several models from the existing initial model to the modified model, obtained the results in **Table 1**.

Table 1. FEA Simulation Results for Canopy Frame

Model Type	**VMS (Mpa)**		**Displacement (mm)**		**Strain (Nm)**		**FOS**	
Original Model	137		3.90		0.01		1.3	
Modification Model 1	92		1.10		0		2	
Modification Model 2	92		1.00		0		2	
Modification Model 3	135		6.30		0.01		1.3	

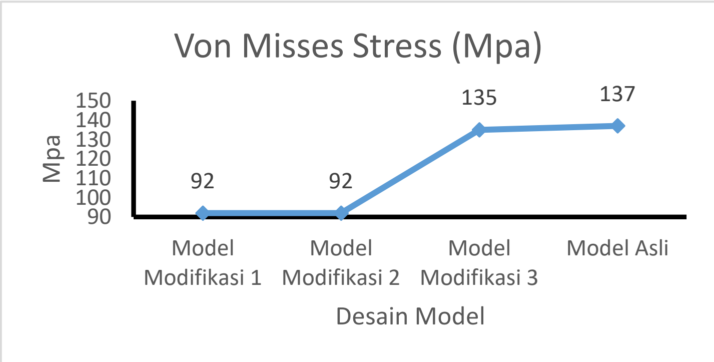
From table 1. The initial model of the canopy frame, the von misses stress results were 137 Mpa, with a maximum displacement of 3.9 mm, a strain value of 0.01 Nm, and a safety factor of

4.1 This Value Is The Value of The Existing Model Design

In modified model 1, the stress value obtained decreased by 92 MPa, 32.85% compared to the initial design model. From the displacement results, the results were obtained at 1.1 mm which resulted in a significant decrease of 71.79%. From the strain results obtained at 0 which was not too significant compared to the initial model. The Safety Factor value obtained was 2 which had a change of 53%.

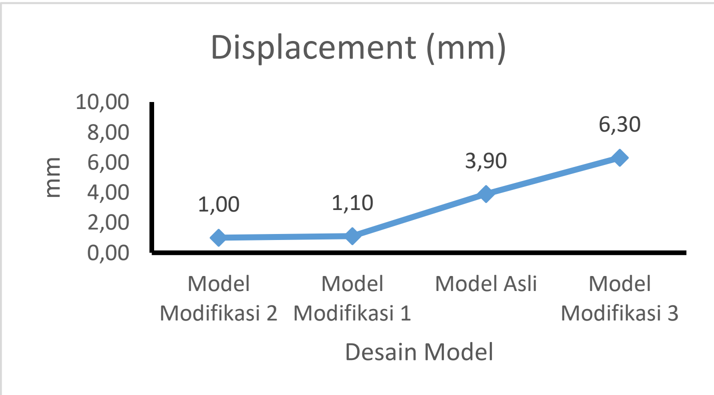
Modification model 2, The results obtained the same stress value as the first modification model of 92 Mpa, Then for the displacement obtained the result of 1 mm where when compared with the initial model and the first modification model, the results of the second modification are the best results. For the strain value, the results are the same as the strain value in the first modification model along with the value of its factor of safety.

Modification Model 3, Gets a result of 135 MPa where the result is not too far from the initial design model, from the results when compared with the first and second modification models of 47%. Then the displacement section becomes not the best result with a large displacement of 6.3 mm, which when compared with the best model in the second modification model makes a change of 530%. Then in the strain section gets a result of 0.01 Nm where the result is the same as the result in the initial design model. Same with the safety factor, Has the same result as the initial design model. From the results obtained, a comparison graph of the entire model was made



Graph 1. Von Misses Stress (VMS)

The graph above shows that the lowest Von Misses Stress value was obtained for the modified model 1 and modified model 2, which was 92 MPa, while the modified model 3 obtained 135 MPa and the initial model 137 MPa. This trend graph indicates that the shape of the support connection affects the results of the various models.



Graph 2. Displacement

Graph 2 This figure shows a comparison of the maximum displacement values on the canopy frame for four design variations. Modification Model 2 produced the smallest displacement of 1.00 mm, followed by Modification Model 1 at 1.10 mm, which indicates an increase in structural stiffness compared to the original model. The Original Model experienced a displacement of

3.90 mm, while Modification Model 3 showed the largest displacement of 6.30 mm, which indicates a decrease in structural stiffness. These results indicate that the geometric modifications in Modification Models 1 and 2 are more effective in reducing deformation, while Modification Model 3 is less optimal in resisting static loads.

4.2 Von Mises Stress

Von Mises stress is used as the primary parameter to evaluate a structure's ability to withstand static loads. Simulation results show that Modified Models 1 and 2 provide the most significant stress reduction, from 137 MPa in the original model to 92 MPa, or a reduction of approximately 32.8%.

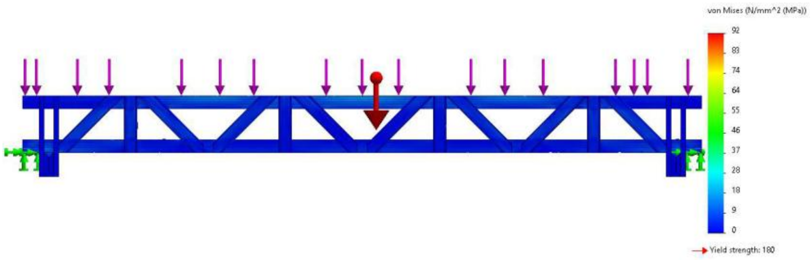


Figure 2. Von Mises Stress Modification 2

The stress distribution in both models appears more even, particularly in the main members and frame joints, reducing the potential for local stress concentrations. Conversely, Modified Model 3 shows an increase in stress approaching the original model's values, indicating a suboptimal configuration of the frame elements in distributing the load.

4.3 Displacement

Displacement analysis was used to assess the stiffness of the canopy frame structure. The original model experienced a maximum displacement of 3.90 mm, indicating a relatively large degree of deformation.

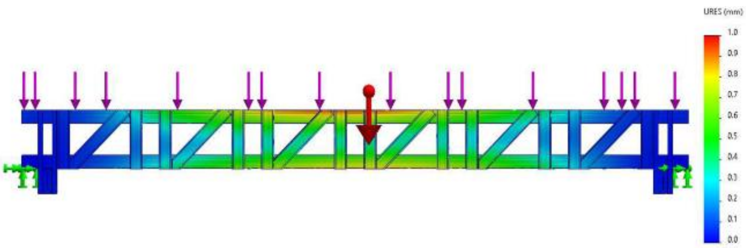


Figure 3. Displacement Modification 2

Modification Models 1 and 2 showed a significant reduction in displacement of 1.10 mm and 1.00 mm, respectively, or a reduction of more than 70% compared to the original model. This indicates that both modified models have better structural stiffness.

In contrast, Modified Model 3 experienced an increase in displacement of up to 6.30 mm, indicating that the structure became more flexible and less able to withstand deformation due to working loads.

4.4 Strain

The strain values in Modification Models 1 and 2 are close to 0, indicating that the deformation is still within the elastic range of the material and is relatively small. This indicates that both models operate safely under the applied load.

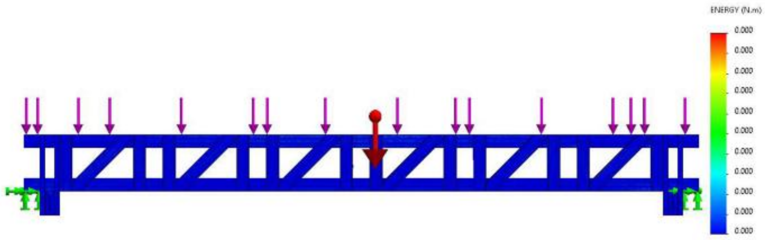


Figure 4. Modified Strain 2

In the Original and Modified Model 3, the strain recorded was 0.01, which indicates greater deformation and the potential for increased risk of material fatigue if the structure is used for a long period of time.

4.5 Safety Factor

The safety factor is used to assess a structure's level of safety against failure. The original model and Modified Model 3 have a FOS value of 1.3, which is relatively low and close to the minimum safety limit for lightweight steel structures.

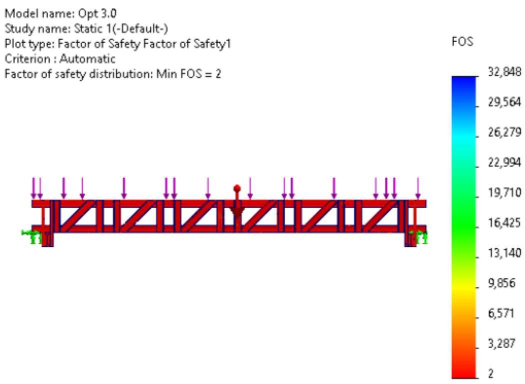


Figure 5. FOS Modification 2

In contrast, Modified Model 1 and Modified Model 2 show an increase in the safety factor to 2, indicating that the structure has a better margin of safety against the working load. This value meets the safe design criteria for static structures and allows for tolerance to load variations and manufacturing imperfections.

4.6 Discussion

Based on the overall simulation results, it can be concluded that modifying the frame geometry significantly impacts the structural performance of the canopy frame. Modification Models 1 and 2 have been shown to significantly reduce stress, minimize displacement, suppress strain, and increase the safety factor.

In contrast, Modified Model 3 showed no improvement in structural performance and even resulted in higher deformations and stresses. This confirms that adding or changing structural elements without proper geometric planning does not always improve structural strength.

From these results, Modified Model 2 can be recommended as the most optimal design because it produces a combination of low stress, minimum displacement, and adequate safety factors, making it safer and more efficient to be applied to canopy frames.

CONCLUSION

Based on the results of the finite element analysis (FEA) conducted on a light steel canopy frame with several variations of the frame structure design, it can be concluded that geometric modifications have a significant influence on the structural performance of the frame. The parameters evaluated include Von Mises stress, displacement, strain, and factor of safety (FOS) with a load of 700 N.

Simulation results show that the original model produces relatively high stresses and displacements with a safety factor of 1.3, indicating that the structure is operating close to the material's safety limits. Modified Model 3 also exhibits suboptimal performance, producing the largest displacements and a relatively low safety factor.

In contrast, Modification Model 1 and Modification Model 2 on the lightweight steel canopy frame structure showed significant improvements in structural performance. Both models were able to substantially reduce Von Mises stress, drastically reduce displacement, and increase the safety factor value to reach 2. This indicates that the geometric configuration in both models is more effective in distributing loads and increasing structural stiffness.

Of all the design variations analyzed, the Modified Model 2 frame structure can be recommended as the most optimal design because it produces the smallest displacement, lower stress compared to the original model, and a safety factor value that meets the static structural safety criteria. This finding confirms that improving structural performance is not always achieved through the addition of materials, but through the design of a more efficient and appropriate frame geometry.

The results of this study are expected to be a reference in the design and development of lighter steel canopy frames that are safer, more efficient, and more reliable for similar structural applications in the future.

REFERENCES

Admin, A., Permana, I., Arif Pratama, R., Bayu Setiajit, S., & Sriyanto, S. (2022). DESIGN AND ANALYSIS OF ALUMINUM DRONE STRUCTURE FOR MONITORING VIA AIR LINES WITH DEFLECTION CRITERIA. Teknika STTKD: Jurnal Teknik, Elektronik, Engine, 8(1), 44–51. Https://Doi.Org/10.56521/Teknika.V8i1.594

Alvian, L., Hanifi, R., & Fitri, M. (2020). ANALYSIS OF THE EFFECT OF SCREW LOCATIONS ON THE STRENGTH OF LIGHT STEEL CONNECTIONS USING ANALYTICAL AND TENSILE TEST METHOD. AME (Application of Mechanics and Energy): Scientific Journal of Mechanical Engineering, 6(2), 69. Https://Doi.Org/10.32832/Ame.V6i2.3307

Desmarita Leni, M. (2022). Comparison of Machine Learning Algorithms for Predicting Mechanical Properties of Low Alloy Steel. : https://Doi.Org/10.30596/Rmme.V5i2.11407

Dinil Mustaqiem, A. (2020). COMPARATIVE ANALYSIS OF SCOOTER FRAME SAFETY FACTORS USING SOLIDWORKS 2015 SOFTWARE. Journal of Mechanical Engineering, 9(3), 164. Https://Doi.Org/10.22441/Jtm.V9i3.9567

Husnah, H., Darfia, NE, & Hidayat, F. (2019). ANALYSIS OF LIGHT STEEL AND HEAVY STEEL (WF) FRAME STRUCTURES USING BRICSCAD AND FINITE ELEMENT METHOD. Cycle: Journal of Civil Engineering, 5(2), 87–96. https://Doi.Org/10.31849/Siklus.V5i2.3232

Manurung, SS, Violeta, I., & Maulina, SM (2021). ANALYSIS OF THE EFFECTIVENESS OF VERTICAL BAR SEGMENTATION LENGTH IN CANTILEVER STEEL CANOPY FRAME. Jurnal ENGINEERING-SIPIL, 21(2), 139. Https://Doi.Org/10.26418/Jtsft.V21i2.50663

Muharni, R., & Maulana, R. (2023). ANALYSIS OF THE EFFECT OF LOAD ON DRUM BRAKING SYSTEM TEST EQUIPMENT THROUGH BRAKE PRESSURE PARAMETERS USING THE TYE ROBOT GX 160 MACHINE. 19.

Purwanto, H. (2017). Efficiency Analysis of Light Steel Roof Frame Construction. Deformation Journal, 2(1), 26–36. https://Doi.Org/10.31851/Deformasi.V2i1.2817

Stevens, D., & Tedianto, LS (2018). ANALYSIS OF THE EFFECT OF ELEMENT LENGTH ON THE COMPRESSIVE STRENGTH OF LIGHT STEEL CHANNEL PROFILE USING THE FINITE ELEMENT METHOD. JMTS: Jurnal Mitra Teknik Sipil, 1(1), 159. Https://Doi.Org/10.24912/Jmts.V1i1.2253

Trahair, N.S. (2018). Non-Linear Biaxial Bending Of Steel Z-Beams. Thin-Walled Structures, 129, 317–326. https://Doi.Org/10.1016/J.Tws.2018.04.012