INTERNATIONAL JOURNAL OF INTEGRATED ENGINEERING
ISSN: 2229-838X e-ISSN: 2600-7916

IJIE

Vol. 16 No. 2 (2024) 259-269
https://publisher.uthm.edu.my/ojs/index.php/ijie

# Finite Element Analysis of Stress Distribution in AL6061 Frame Structures Under Varying Applied Loads

**Muhammad Amin Sabtu[1,2], Aidid Ezmi Admi[2], Muhammad Mohamed Salleh[3], Saliza Azlina Osman[2], Shahrul Azmir Osman[2*]**

[1] *Intec Precision Engineering Sdn. Bhd. No. 20 Jalan Mega 1, Kawasan Perindustrian Nusa Gemilang, 79200 Gelang Patah, Johor, MALAYSIA*
[2] *Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia 86400, Parit Raja, Johor, MALAYSIA*
[3] *Faculty of Technical and Vocational Education, Universiti Tun Hussein Onn Malaysia 86400, Parit Raja, Johor, MALAYSIA*

*Corresponding Author: shahrula@uthm.edu.my
DOI: https://doi.org/10.30880/ijie.2024.16.02.027

**Article Info**

Received: 27 February 2024
Accepted: 11 June 2024
Available online: 3 August 2024

**Keywords**

Finite element analysis, SolidWorks, stress, strain, displacement, safety factor, AL6061

**Abstract**

The demand for lightweight and high-performance structures has driven the popularity of commercially available aluminum alloys such as AL6061, particularly in the transporter industry. However, predicting stress distribution accurately within AL6061 frame structures under diverse loading conditions remains a significant challenge for clients due to the complex nature of their behavior. To address this challenge, a study has been conducted that utilizes the advanced 3D simulation software SolidWorks 2023 to model a custom dimension frame structure and evaluate their behavior under varied loads of up to 200% of the structure's weight, which is a significant departure from the standard loading conditions. The goal is to offer a thorough and accurate assessment of the frame structure's capabilities. This investigation analyzed the stress, strain, displacement, and safety factors of a specifically designed frame structure, ensuring that each value met or exceeded acceptable standards. This study also provides valuable insights into the structure's performance under various loading conditions, including the handling of heavy items and specimens that require the use of a transporter in industrial engineering applications. Additionally, a detailed assessment of the material properties of the aluminum alloy used in constructing the frame structure provides useful information for designing and optimizing high-performing structures in the industry. Based on the results, it appears that the critical von Mises stress, when exposed to maximum varying loads (76.54 MPa), remains below the structure's yield strength of 275 MPa. The equivalent strain reaches a maximum value of 0.5607E-03, and the structure's deformation at maximum loads is just 0.6705 mm. The safety factors of the structure, tested at three different varying loads, are above 3. These values conclusively demonstrate that the AL6061 frame structure is safe and reliable for industrial applications and can withstand loads up to 200% wt.

## 1. Introduction

Aluminium alloys are extensively utilized in the manufacturing industry because of their exceptional electrical and thermal conductivity, impressive strength-to-weight ratio, good hardness, corrosion resistance, and flexibility and resilience [1-3]. Among these materials is AL6061, an aluminium alloy with high adaptability, workability, and ease of joining. Owing to its remarkable strength and corrosion-resistant properties, this material is particularly advantageous in the aerospace, marine, and military sectors [4-5].

Technologists are driven to develop a reliable and effective method for joining aluminium alloys due to the growing use of aluminium in various industrial sectors. The objective of these advancements is to avoid the negative impacts of welding on the desired mechanical, chemical, and metallurgical properties of aluminium alloys, ultimately leading to a longer lifespan [6]. There is an increasing demand for lightweight and high-performance constructions in many industries, especially industrial transporter, leading to a rise in demand for commercial aluminium alloys such as AL6061 [7]. However, despite advancements in structural analysis, accurately predicting stress distribution in frame structures made from AL6061 under various loading conditions still poses a significant challenge. This is because of the complex behavior exhibited by these structures, which involves intricate geometries and material properties. The ability to accurately predict stress distribution is crucial for ensuring the structural integrity and safety of these frame structures, particularly in a wide variety of industrial engineering.

In this present paper, the primary objective of this research is to evaluate the structural behavior of an AL6061 custom-dimension frame structure under various loading conditions. This evaluation will be carried out through finite element analysis (FEA) techniques, leveraging computer-aided engineering (CAE) tools. Specifically, the research will employ 3D modeling software (SolidWorks) and its integrated FEA capabilities to analyze critical factors such as stress, strain, displacement, and factors of safety within the frame structure. By utilizing these computational tools and methodologies, the study aims to gain insights into the structural performance, reliability, and safety considerations of the frame design under different loading scenarios. The findings from this research can contribute to advancements in structural engineering by providing a framework for optimizing frame designs and ensuring their structural integrity under various loading conditions as a case study for the manufacturer.

## 2. Literature Review

To analyze the impact of different loads on the stress distribution of AL6061 frame structures, a Finite Element Analysis (FEA) will be conducted. This advanced computational technique accurately models how complex structures respond to various loading scenarios. FEA is widely used in product development and is becoming increasingly popular in industrial processes as it improves efficiency and product quality and provides insights into the impact of multiple process variables. The Finite Element Method (FEM) has been a staple in product development for some time and is now gaining momentum as a cutting-edge computational mechanics application in the industrial sector [8]. The most important reason for this development is the industrial need to improve product productivity and quality and better understand the influence of different process parameters [9].

Numerous research studies have utilized Finite Element Analysis in their case studies. Mahakul et al. [10] analyzed different materials used in making spur gears, which used SolidWorks simulation to compare the performance of gears made from stainless steel, Ti-3Al-8 V-6Cr-4Mo-4Zr composite, and Zinc AC41A alloy. The results showed that the lighter composite material, Ti-3Al-8 V-6Cr-4Mo-4Zr, was the best choice for the gears. This was due to its moderate stress distributions, lower strain values, and minimal deformation under load. Dabit et al. [11] analyzed the strength of an Autonomous Unmanned Surface Vehicle Feeder Boat. Using SolidWorks software, they used Finite Element Analysis to evaluate the boat's durability under static loads, which focuses on assessing the stress, strain, and displacement values of the boat hull made of balsa wood. The prototype test proved that the boat could carry 8 kg of fish feed without breaking, demonstrating the effectiveness of the modifications to enhance its strength and rigidity. Singh et al. [10] utilized computer analysis to examine a tape-sealing machine. Their findings revealed that the machine's pin experienced the most significant stress due to its small size. Additionally, they observed that the machine's components exhibited less movement over time, indicating elasticity in the material. The part of the machine that made contact with the boxes experienced the most movement, likely due to the box's motion.

Dzulfiqar et al. [12] an Automatic Thickness Checking Machine for the braking pad industry. Their design was tested thoroughly using Finite Element Analysis to ensure its structural integrity and reliability, which evaluated the frame by testing it with different static loads on each axis and considering factors such as safety, von Misses stress, and displacement due to static loading. They showed that the values were within an acceptable range. Wibawa [13] conducted a comprehensive analysis of the impact of fillet radius size on the strength and durability of Unmanned Aerial Vehicle (UAV) landing gear through Ansys Workbench. The research revealed that larger fillet radiuses resulted in increased stress but reduced deformation. Despite the stress levels being below the yield strength of the material, the safety factor was deemed satisfactory. This study highlights the significance of fillet size in determining the performance of UAV landing gear, particularly in terms of stress and deformation under impact loads. Vardaan & Kumar [14] utilized SolidWorks Simulation to optimize the design of the thresher machine flywheel, comparing two variations made of S-glass fiber and grey cast iron with different geometries. The study revealed that the flywheel's material and geometry can impact its energy storage capacity. While cast iron flywheels had higher stress values, they exhibited lower deformation. On the other hand, S-glass fiber flywheels showed promise as an alternative, if deformation can be compensated for. Mistry et al. [15] investigated various joint types utilized in step lap joints through Finite Element Analysis (FEA) in Ansys Workbench. The study compared the load-carrying capacity, stress distribution, and deformation of joints with different step lengths, materials, and mechanical fasteners. The results yielded valuable insights into joint configurations' stress and deformation parameters under varying conditions.

Rao et al. [16] utilized Finite Element Analysis to examine freight railway wagons. They employed SolidWorks to create a wagon model and Ansys software to analyze it. Through static structural analysis, they evaluated four different materials to assess stress and deformation, while modal analysis was conducted to gain insight into the wagons' vibration behavior. Ultimately, their findings revealed that AISI 1018 low-carbon steel was the optimal material for open-top wagons, as it demonstrated superior stress, deformation, and vibration behavior performance. Pandey et al. [17] assess the strength and durability of bolts created through 3D printing technology using two different materials, Polylactic acid (PLA) and Acrylonitrile butadiene styrene (ABS). The study utilized SolidWorks software to design standard M16 bolts and subjected them to tests measuring tangential stress and distortion. Ultimately, the findings indicated that bolts fashioned from PLA exhibited superior quality compared to those made from ABS. Dhevesh Kannan et al. [18] examined the resistance capabilities of an aluminium alloy target plate against a 7.62 Armour Piercing (AP) projectile. Employing Ansys 19.2 explicit dynamics software simulations, they studied the plate's ballistic resistance with and without ceramic reinforcements. The findings indicated a significant enhancement in the plate's resistance with the inclusion of ceramic reinforcements, resulting in reduced deformation and residual velocity of the projectile.

## 3. Methodology

### 3.1 3D Modelling

Initially, the SolidWorks 2023 Student Edition software, which is a registered Computer-Aided Design (CAD) tool, is used to create a 3D model of the design frame structure. In this case study, the frame structure of a custom design from an industrial transporter is fabricated using the standard dimensions specified in Table 1. The resulting model of the frame structure is presented in Fig. 1.

**Table 1** *Dimensional parameter of frame structure*

| Parameter | Description | Value |
|---|---|---|
| L | Frame structure length | 1028.7 mm |
| W | Frame structure width | 1016.0 mm |
| H | Frame structure height | 50.8 mm |

### 3.2 Finite Element Analysis

The Finite Element Method (FEM) is widely recognized as a valuable tool for design optimization. FEM, or Finite Element Analysis, is a powerful numerical method that enables engineers to simulate physical phenomena and predict the behavior of parts or assemblies under specific conditions. This advanced technology helps identify weak points, high voltage areas, and design flaws in a safe and efficient manner. The simulation results are easily interpreted using a color scale that displays the pressure distribution of the object. The Finite Element Method is considered an extremely reliable computer algorithm for investigating complex engineering problems that involve intricate geometries, limitations, and loading conditions. Not only is it a dependable structural analysis tool, but it also saves valuable time and resources. When using FEM, the body or structure is discretized into nodes and elements, which are then summed to represent the entire body. The solver assigns each node a function, and once solutions are obtained for all nodes and elements, they are integrated to form the complete solution for the entire body [20-23].

In this proposed case study, SolidWorks software utilizes FEA to conduct static structural on proposed frame structure models. To initiate the procedure, it is necessary to import the 3D configuration of the frame structure model into the SolidWorks simulation. In the material selection tab, SolidWorks presents an assortment of pre-established materials, and it's crucial to know the precise material property value for optimal analysis efficiency.

Additional information regarding the meshing process and boundary conditions is provided, and it's imperative to apply boundary conditions and loads within SolidWorks diligently.

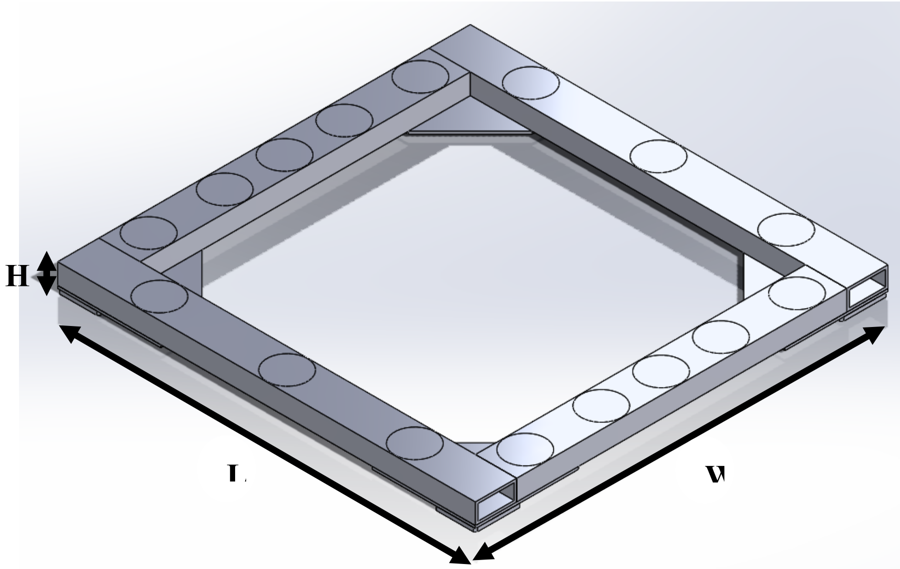


**Fig. 1** *Model of design frame structure*

## 3.3 Material Properties

For the finite element analysis, the selected material is aluminium alloy 6061-T6, a pre-existing option in SolidWorks' materials database. This alloy comprises magnesium and silicon, which give it desirable welding properties and impressive mechanical characteristics [24-25]. Table 2 [26] illustrates the composition of aluminium alloy 6061-T6, and one of its key advantages is its capacity to endure heat treatment. Table 3 [26] presents the numerical values for the material's mechanical properties.

## 3.4 Meshing

After the materials were assigned, the mesh was created using a form of grid or density between each material that forms the material. The smaller the mesh value, the better the shape of the content on the part to be simulated. In mathematics, this grid or mesh is called the discretization process [27]. In the 3D mechanical design application, a meshing method is used to enter the mesh density parameter at a 'fine' meshing value. The grid is formed by equilateral triangles with a global size of 16.3 mm and a tolerance value of 5.43328 mm, as seen in Fig. 2.

For the simulation, a highly efficient Blended Curvature-Based Mesh type mesher is utilized. This mesh type offers superior performance compared to other alternatives, due to its optimized code architecture, multithreading capabilities, and parallel multicore processing. Furthermore, this mesher can also achieve significantly faster meshing of parts and large assemblies [28-29]. The Solid Mesh type is employed, and the corresponding values can be found in Table 4. However, before analyzing the data, a mesh convergence test is essential to ensure the appropriate grid size.

### 3.4.1 Mesh Convergence

The structural frame undergoes a mesh convergence test to assess whether the numerical analysis is free from any dependence on the discrete element size. The test involves deploying smaller and finer elements to enhance the accuracy of simulation results. However, it also increases the number of elements, leading to longer operation times. As such, it becomes crucial to identify the element size that does not affect the simulation results or the quality of the elements, ensuring the mesh's independence [30].

To maintain the integrity of the current model, a mesh independence analysis was carried out using element sizes ranging from 21 to 16.3 mm. Subsequent analysis revealed that there was negligible variation in the increase of nodes and elements beyond the 16.3 mm threshold. Consequently, an element size of 16.3 mm was chosen for the model. Of these sizes, the 16.3mm option (with 71,117 nodes and 34,877 elements) produced a fine mesh and generated stable results in terms of deformation and stress. It is worth noting that this mesh size generated high-quality results, with an Aspect ratio of less than 5 for over 90% of its elements [31]. Fig. 2 shows a visual representation of the applied grid on the relevant parts of the model frame during the computational work and Table 5 gives the values for different element sizes and their number of nodes and elements.

Int. Journal of Integrated Engineering Vol. 16 No. 2 (2024) p. 259-269

**Table 2** *Composition of aluminium alloy AL6061 T6*

| Chemical Elements | Percentage (%) |
|---|---|
| Si | 0.4-0.8 |
| Fe | 0.7 |
| Cu | 0.15-0.4 |
| Mn | 0.15 |
| Mg | 0.8-0.12 |
| Cr | 0.04-0.35 |
| Ti | 0.15 |
| Zn | 0.25 |
| Al | 96-97.36 |

**Table 3** *Mechanical properties of aluminium alloy*

| Properties | Values |
|---|---|
| Density | 2700 kg/m$^3$ |
| Poisson's Ratio | 0.33 |
| Elastic Modulus | 6.9E+10 MPa |
| Shear Modulus | 2.6E+10 MPa |
| Tensile Strength | 3.1E+08 MPa |
| Yield Strength | 2.75E+08 MPa |

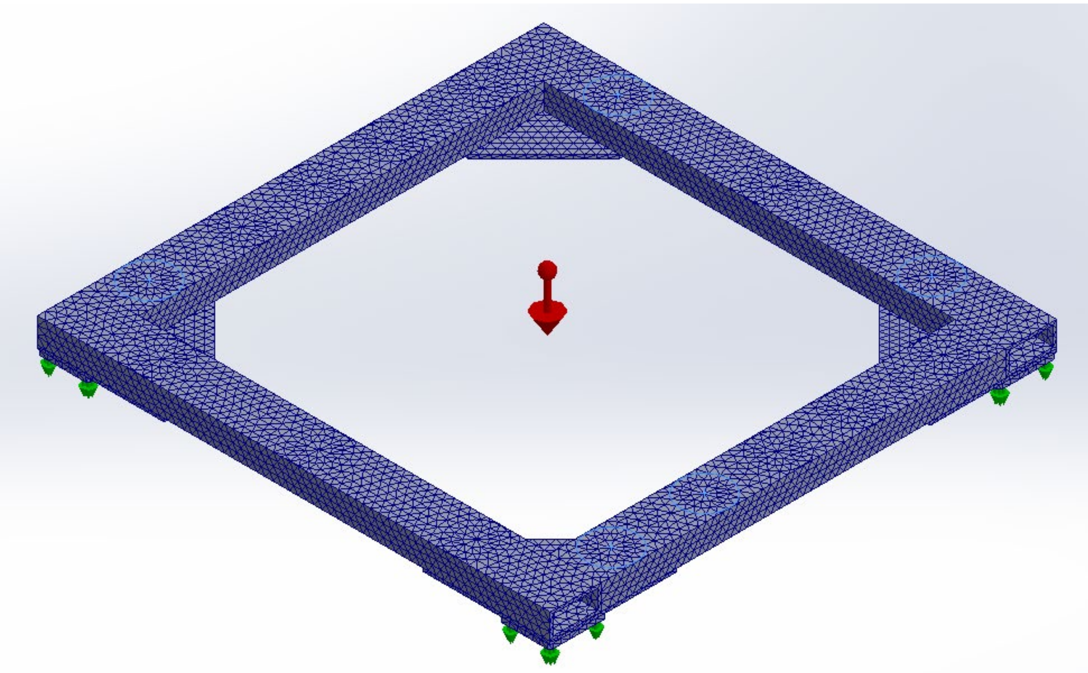

**Fig. 2** *Applied meshing on frame structure*

**Table 4** *Meshing data from the AL6061 frame structure*

| Mesh type | Solid Mesh |
|---|---|
| Mesher used | Blended curvature-based Mesh |
| Jacobian points | 29 points |
| Element size | 16.3 mm |
| Tolerance | 5.43328 mm |
| Mesh quality | High |
| Total nodes | 71117 |
| Total elements | 34877 |
| Maximum Aspect Ratio | 10.498 |
| Percentage of elements with Aspect Ratio < 3 | 97.4 |
| Percentage of elements with Aspect Ratio > 10 | 0.0172 |
| Percentage of distorted elements | 0 |

**Table 5** *Meshing data of different element sizes*

| Property of mesh | Values | Values | Values | Values |
|---|---|---|---|---|
| Element Size | 16 mm | 21 mm | 18 mm | 16.3 mm |
| Nodes | 72715 | 43211 | 55982 | 71117 |
| Elements | 35675 | 21069 | 27352 | 34877 |

## 3.5 Boundary Condition

In the process of structural analysis, applying boundary conditions after Meshing is crucial. The complete frame structure consists of interconnected substructures modeled as elastic bodies to reflect their structural dynamics. To simulate the external load, an external load advisor is used to select distributed mass, which is then evenly applied to specific areas of the structure. The applied weights, in English unit, are 1179.34 kg (100%wt), 1769.01kg (150%wt), and 2358.68kg (200%wt). To restrict the Degree of Freedom (DOF) of all nodes on the surface, the fixture advisor tab is used to apply roller/slider support to specific areas of the frame structure. Following these steps during analysis is crucial. Fig. 3 overviews the designed frame structure with the applied boundary conditions.

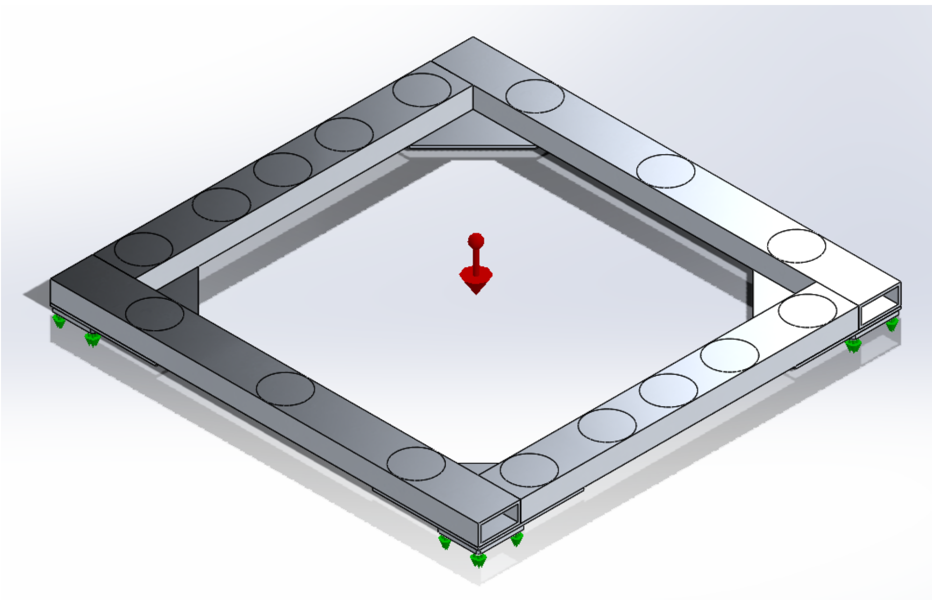

**Fig. 3** *Boundary conditions on frame structure*

Int. Journal of Integrated Engineering Vol. 16 No. 2 (2024) p. 259-269

## 4. Results and Discussion

The simulation conducted to design the AL6061 frame structure provides stress, strain, displacement, and safety factor values. Analyzing the results obtained from this study can determine the frame structure's performance and reliability.

### 4.1 Stress Simulation

Based on the Stress Simulation analysis, it has been determined that applying a distributed static loading of 2358.68 kg to specific parts of the frame structure results in a maximum stress value of 7.654E+07 N/m$^2$. It should be noted that this value is well within safe limits as it is lower than the yield strength of the material, which is 2.75E+08 N/m$^2$. It means that the frame structure will not fail at maximum weight variation due to the impact load. A visual representation of the Stress Simulation results can be seen in Fig. 4.

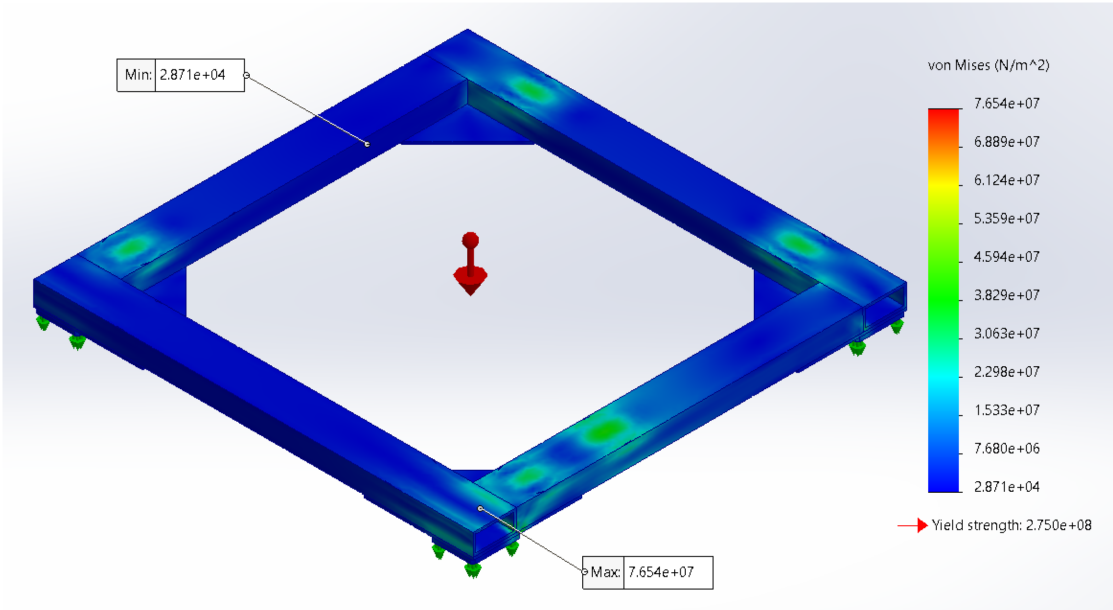


**Fig. 4** *Behaviour of frame structure under stress simulation*

### 4.2 Strain Simulation

A simulation was carried out on the underside of the frame structure of AL6061 to calculate the strain stress value under a distributed static loading of 2358.68 kg. The maximum equivalent strain value obtained was 0.5607E-03, which indicates elastic behavior, meaning the material is deforming within its elastic limit and will likely return to its original shape after load removal. The results of the strain simulation can be viewed in Fig. 5.

### 4.3 Displacement Simulation

Deformation is the term used to describe the alteration of a component's shape or distortion caused by a load's impact. During displacement simulations, a distributed static load is applied to determine the maximum resultant displacement value experienced by the frame structure. The findings reveal that when subjected to a distributed static load of 2358.68 kg on specific structure areas, the maximum deformation value is relatively minor, measuring only 0.6705 mm. This is clearly depicted in Fig. 6.

### 4.4 Safety Factor Simulation

When assessing the safety of a component or structure, the safety factor plays a crucial role, even when working with minimal dimensions. SolidWorks determines this factor by dividing the material's yield strength by the maximum von Mises stress. If this value falls below 1, it indicates that the design has permanently failed. Regarding the AL6061 frame structure, it can safely withstand a distributed static load of 2358.68 kg in specific areas of the structure, with a minimum safety factor value of 3.593, as depicted in Fig. 7. This value exceeds the standard requirement for a component capable of withstanding dynamic loads, ensuring a safe and reliable design.

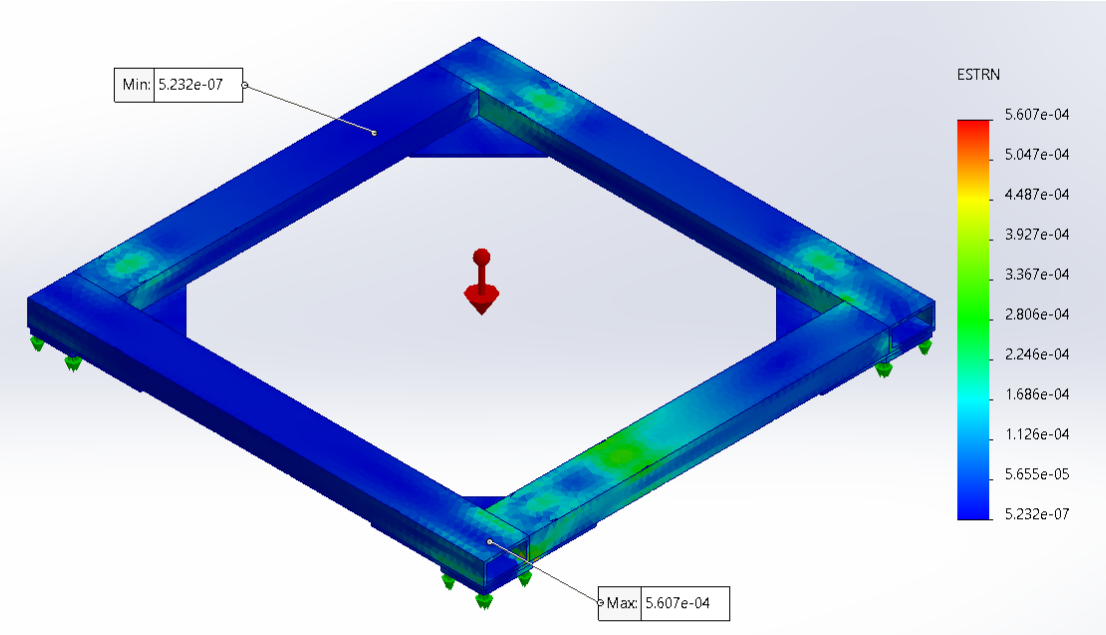


**Fig. 5** *Behaviour of frame structure under strain simulation*

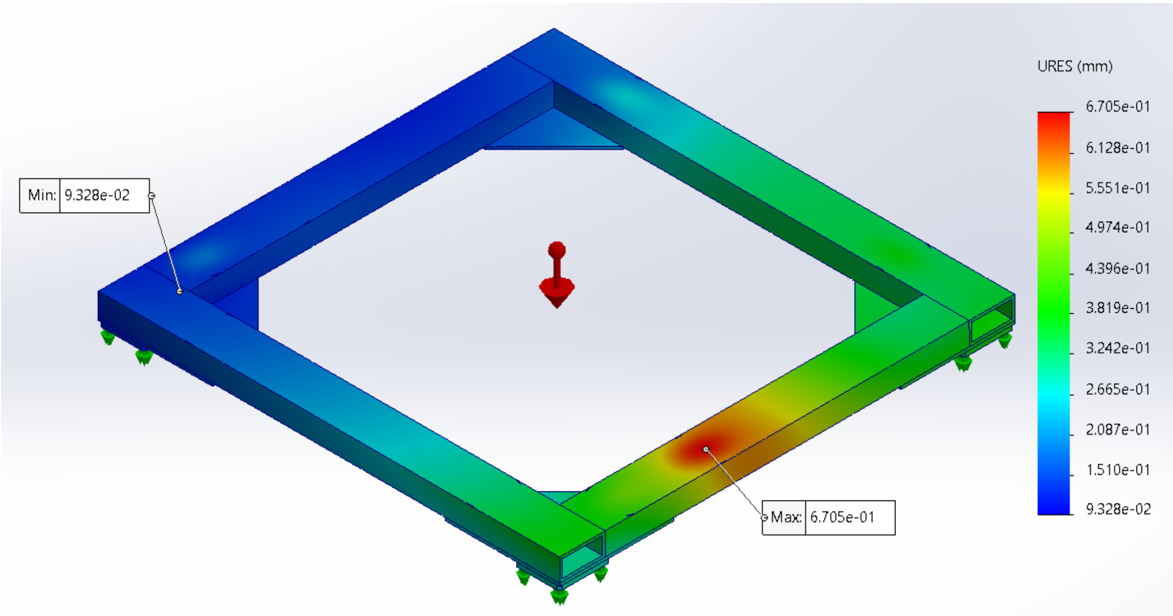


**Fig. 6** *Behaviour of frame structure under displacement simulation*

Table 6 displays the findings of the static stress simulation conducted on the AL6061 frame structure with applied weight variations. The analysis reveals that as the load weight increases, the von Mises stress and deformation of the frame structure also increase. Conversely, the safety factor of the frame structure decreases with increasing load weight. Nevertheless, even with the highest applied weight, the safety factor remains above 2, indicating that the structure can endure the loads.

**Table 6** *Static stress simulation results of the AL6061 frame structure with applied weight variations*

| Weight of Load (kg) | Von Mises Stress (MPa) | Equivalent Strain ($\times 10^{-3}$) | Resultant Displacement (mm) | Safety Factor |
|---|---|---|---|---|
| 2358.68 | 76.54 | 0.5607 | 0.6705 | 3.593 |
| 1769.01 | 57.50 | 0.4213 | 0.5026 | 4.783 |
| 1179.34 | 38.45 | 0.2819 | 0.3347 | 7.152 |

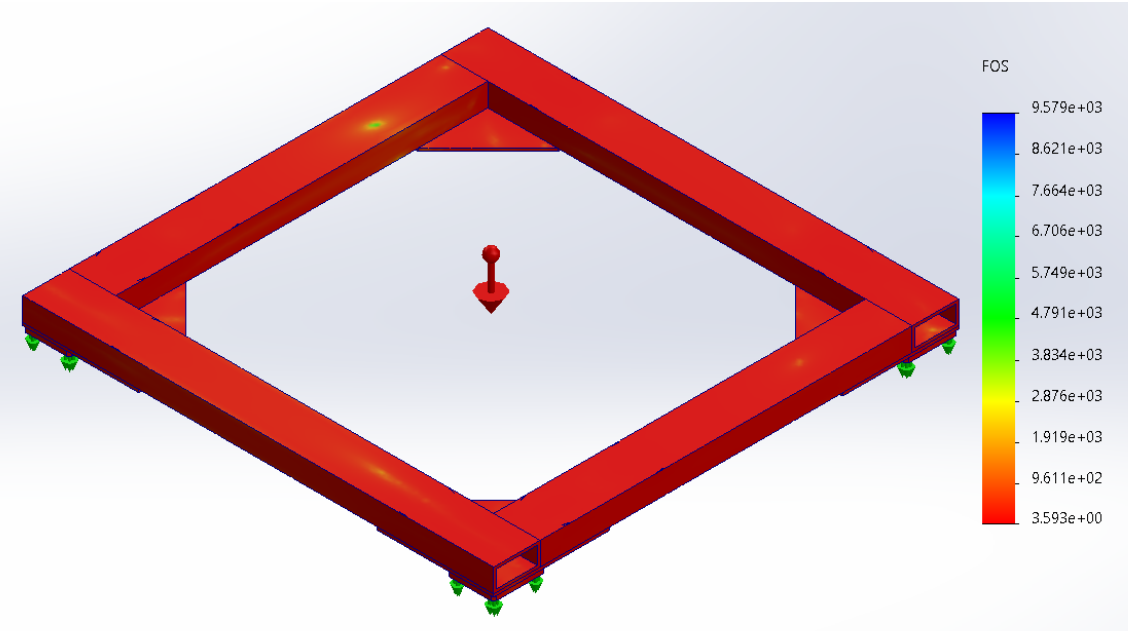


**Fig. 7** *Behaviour of frame structure under safety factor simulation*

## 5. Conclusion

In this study, the following conclusions were reached:

1. After undergoing thorough testing with varying loads, it has been determined that the structure is safe to use. The highest amount of stress observed during testing was 76.54 MPa, which is well below the structure's yield strength of 275 MPa at maximum weight variation. This suggests that the structure is sturdy enough to endure the applied loads without any lasting deformation. Nonetheless, it is worth noting that there was a 24.88% increase in Von Mises stress value when the weight of the load was increased from 1769.01 kg to 2358.68 kg.
2. At an applied load of 2358.68 kg, the structure underwent a maximum strain of 0.6705E-03. The maximum strain increased proportionally as the applied load was gradually raised from 1179.34 kg to 1769.01 kg, reaching 0.2819E-03 and 0.4213E-03 respectively. The linear correlation between the applied load and maximum strain signifies that the material behavior within the analyzed load range can be approximated as elastic.
3. Upon subjecting the structure to loads of 1179.34 kg and 1769.01 kg, a significant increase in resultant displacement is observed, with a positive trend of 33.41%. These findings indicate that the material demonstrates elastic behavior within the tested load range, whereby the resultant displacement is recoverable after the load is removed. Furthermore, the maximum resultant displacements are relatively minor, indicating that the structure can withstand the applied loads without experiencing significant bending. This suggests that the structure possesses commendable stiffness.
4. Based on the load-carrying capacity testing, it has been determined that the structure is proficient in accommodating loads within the tested ranges and maintaining a safety factor above 3. This safety factor ensures that the structure can handle loads beyond the tested values without surpassing its material limits. However, the safety factor shows a consistent decrease as the load increases, indicating that the design may be less efficient at higher loads. This suggests that the structure may not utilize its material capacity as effectively as the load increases, resulting in a decreasing safety margin.

According to the proposed study, it was found that the frame structure experiences greater strain and deformation as the weight of a load increases. This is due to the increase in von Mises stress. Additionally, as the load weight increases, the safety factor of the structure decreases. In simpler terms, the structure becomes less safe as the load gets heavier. However, it's important to note that even when subjected to the heaviest weight, the safety factor of the structure remains above 3. This indicates that the frame structure is practical and capable of withstanding heavy loads while still maintaining a safe level of use.

## Acknowledgement

This research was supported by the Matching Grant [Q274] and Industrial Grant [M116]. The author would like to thank to Faculty of Mechanical Engineering and Manufacturing, Universiti Tun Hussein Onn Malaysia, and Intec Precision Engineering Sdn. Bhd. For providing necessary research facilities for this study.

## Conflict of Interest

Authors declare that there is no conflict of interest regarding the publication of the paper.

## Author Contribution

*The authors confirm contribution to the paper as follows:* ***study conception and design:*** *Muhammad Amin Sabtu, Aidid Ezmi Admi, Shahrul Azmir Osman;* ***data collection:*** *Muhammad Amin Sabtu, Aidid Ezmi Admi;* ***analysis and interpretation of results*** *Muhammad Amin Sabtu, Aidid Ezmi Admi;* ***draft manuscript preparation:*** *Muhammad Amin Sabtu, Aidid Ezmi Admi, Muhammad Mohamed Salleh, Saliza Azlina Osman, Shahrul Azmir Osman. All authors reviewed the results and approved the final version of the manuscript.*

## References


[1] N. Hoffman, "Finite Element Analysis of Extruded AA6061-T6 Mechanical Connection in Low-Cycle Fatigue," Mississippi State Univerity, Mississippi, 2020.

[2] A. Khoshroyan and A. R. Darvazi, "Effects of Welding Parameters and Welding Sequence on Residual Stress and Distortion in Al6061-T6 Aluminum Alloy for T-Shaped Welded Joint," *Transactions of Nonferrous Metals Society of China (English Edition)*, vol. 30, no. 1, pp. 76–89, Jan. 2020, doi: 10.1016/S1003-6326(19)65181-2.

[3] P. Ganesh and V. S. Senthil Kumar, "Finite Element Simulation in Superplastic Forming of Friction Stir Welded Aluminium Alloy 6061-T6," *International Journal of Integrated Engineering*, vol. 3, no. 1, pp. 9–16, 2011.

[4] T. Akbari, A. Ansari, and M. R. Pishbijari, "Influence of Aluminum Alloys on Protection Performance of Metal Matrix Composite Armor Reinforced with Ceramic Particles Under Ballistic Impact," in *Ceram. Int*, Jul. 2023.

[5] C. S. Ho, M. K. M. Nor, and M. S. A. Samad, "Modelling Temperature and Strain-Rate Dependence of Recycled Aluminium Alloy AA6061," *International Journal of Integrated Engineering*, vol. 14, no. 6, pp. 135–145, 2022, doi: 10.30880/ijie.2022.14.06.012.

[6] R. Rajan, P. Kah, B. Mvola, and J. Martikainen, "Trends in Aluminium Alloy Development and Their Joining Methods," 2016. [Online]. Available: https://api.semanticscholar.org/CorpusID:150378819

[7] A. Sabarinivas and D. R. Subramanian, "Comparative Study of Al6061 and AlSi10Mg Produced by Selective Laser Melting Process," *IOP Conf Ser Mater Sci Eng*, vol. 1059, no. 1, p. 012052, Feb. 2021, doi: 10.1088/1757-899X/1059/1/012052.

[8] Imaginationeering Design Solutions, "How You Can Improve the Quality of Product with FEA," Imaginationeering Design Solutions. Accessed: Dec. 26, 2023. [Online]. Available: https://imaginationeering.com/how-you-can-improve-the-quality-of-product-with-fea/

[9] Raymont Osman Product Design, "Finite Element Analysis (FEA): Simulating your Pre-Manufactured Design," Raymont Osman Product Design. Accessed: Dec. 26, 2023. [Online]. Available: https://www.raymont-osman.com/what-is-fea-finite-element-analysis/

[10] R. Mahakul, D. Nath Thatoi, S. Choudhury, and P. Patnaik, "Design and Numerical Analysis of Spur Gear using SolidWorks Simulation Technique," in *Materials Today: Proceedings*, Elsevier Ltd, 2019, pp. 340–346. doi: 10.1016/j.matpr.2020.09.554.

[11] A. S. Dabit, A. E. Lianto, S. A. Branta, F. B. Laksono, A. R. Prabowo, and N. Muhayat, "Finite Element Analysis (FEA) on Autonomous Unmanned Surface Vehicle Feeder Boat Subjected to Static Loads," in *Procedia Structural Integrity*, Elsevier B.V., 2020, pp. 163–170. doi: 10.1016/j.prostr.2020.07.022.

[12] A. R. Singh, A. Dixit, and J. Dhupal, "Finite Element Analysis of Case Box Tape Sealer Mechanism," in *Materials Today: Proceedings*, Elsevier Ltd, 2019, pp. 570–576. doi: 10.1016/j.matpr.2019.07.104.

[13] M. F. Dzulfiqar, A. R. Prabowo, R. Ridwan, and H. Nubli, "Assessment on the Designed Structural Frame of the Automatic Thickness Checking Machine - Numerical Validation in FE Method," in *Procedia Structural Integrity*, Elsevier B.V., 2021, pp. 59–66. doi: 10.1016/j.prostr.2021.10.009.

[14] L. A. N. Wibawa, "Effect of Fillet Radius of UAV Main Landing Gear on Static Stress and Fatigue Life using Finite Element Method," in *Journal of Physics: Conference Series*, IOP Publishing Ltd, 2021. doi: 10.1088/1742-6596/1811/1/012082.

[15] K. Vardaan and P. Kumar, "Design, Analysis, and Optimization of Thresher Machine Flywheel using Solidworks Simulation," *Mater Today Proc*, vol. 56, pp. 3651–3655, Jan. 2022, doi: 10.1016/j.matpr.2021.12.348.

[16] S. Mistry, P. Joshi, R. Dhandhukiya, S. Gandhi, N. Bhanushali, and C. Desai, "Finite Element Studies of Bolted, Riveted, Bonded and Hybrid Step-Lap Joints of Thick Plate," in *Materials Today: Proceedings*, Elsevier Ltd, 2021, pp. 1080–1087. doi: 10.1016/j.matpr.2021.07.467.

[17] P. K. V. Rao, G. Rama Prudhvi Varma, and K. Sri Vivek, "Structural Dynamic Analysis of Freight Railway Wagon using Finite Element Analysis," *Mater Today Proc*, vol. 66, pp. 967–974, Jan. 2022, doi: 10.1016/j.matpr.2022.04.770.

[18] S. Pandey, S. Vidya, and P. Suresh, "Comparative Finite Element Analysis of ISO Standard Bolt Using ABS and PLA Material," *Mater Today Proc*, vol. 72, pp. 3063–3067, Jan. 2023, doi: 10.1016/j.matpr.2022.09.063.
[19] M. Dhevesh Kannan, S. Chand Kundurti, B. Ranta Sunil, and A. Sharma, "Material Design for Ballistic Applications: A Numerical Analysis of Surface Reinforced AA6061-T6 Metal Matrix Composite," *Mater Today Proc*, Oct. 2023, doi: 10.1016/j.matpr.2023.10.113.
[20] E. Menacho-Mendoza, R. Cedamanos-Cuenca, and A. Díaz-Suyo, "Stress Analysis and Factor of Safety in Three Dental Implant Systems by Finite Element Analysis," *Saudi Dental Journal*, vol. 34, no. 7, pp. 579–584, Nov. 2022, doi: 10.1016/j.sdentj.2022.08.006.
[21] T.-S. Lan, H.-W. Zhang, J.-S. Gao, and X.-J. Dai, "Design and Optimizated Simulation of Bicycle Frame Structure," in *2nd IEEE Eurasia Conference on IOT, Communication and Engineering 2020*, 2020.
[22] H. Samekto, "Design Modifications of a Thin Wall Part from Aluminium to Magnesium," *International Journal of Integrated Engineering*.
[23] N. N. Nasrudin, N. F. Ariffin, A. Alias, A. M. Hasim, and M. N. S. Zaimi, "Experimental Validation of Reinforced Concrete Beam Incorporating Coal Fly Ash and Coal Bottom Ash Using Numerical Analysis," *International Journal of Integrated Engineering*, vol. 15, no. 2, pp. 228–236, 2023, doi: 10.30880/IJIE.2023.15.02.022.
[24] "All About 6061 Aluminum Alloy | Xometry." Accessed: Dec. 30, 2023. [Online]. Available: https://www.xometry.com/resources/materials/6061-aluminum-alloy/
[25] "All About 6061 Aluminum (Properties, Strength and Uses)." Accessed: Dec. 30, 2023. [Online]. Available: https://www.thomasnet.com/articles/metals-metal-products/6061-aluminum/
[26] I. Das Chowdhury *et al.*, "Investigation of Mechanical Properties of Al6061-T6 Alloy by EAFSW with Straight and Reverse Polarity of Direct Current," in *Materials Today: Proceedings*, Elsevier Ltd, 2020, pp. 4024–4030. doi: 10.1016/j.matpr.2020.10.216.
[27] M. S. Ul Abrar, K. F. Nadim Ezaz, M. J. Hasan, R. I. Pranto, T. A. Alvy, and M. Z. Hossain, "Speed-dependent Impact Analysis on A Car Bumper Structure using Various Materials," *Results in Engineering*, p. 101927, Mar. 2024, doi: 10.1016/j.rineng.2024.101927.
[28] "Blended Curvature-Based Mesher - 2022 - What's New in SOLIDWORKS." Accessed: Apr. 18, 2024. [Online]. Available: https://help.solidworks.com/2022/English/WhatsNew/c_wn2022_simulation_bcb_mesher.htm
[29] "Enhanced Blended Curvature-Based Mesher - 2021 - What's New in SOLIDWORKS." Accessed: Apr. 18, 2024. [Online]. Available: https://help.solidworks.com/2021/English/WhatsNew/c_wn2021_simulation_blended_curvature_mesher.htm
[30] A. Abid and A. A. Bhuiyan, "A Finite Element Analysis Approach to Design and Optimize The Static Structural Impact on A Skateboard," *Mater Today Proc*, vol. 60, pp. 2171–2187, Jan. 2022, doi: 10.1016/j.matpr.2022.02.424.
[31] Dassault Systèmes, "Mesh Quality Checks," 2023. [Online]. Available: https://help.solidworks.com/2023/english/SolidWorks/cworks/c_mesh_quality_checks.htm?format=P&value=1