Terbit *online* pada laman web jurnal : https://jes-tm.org/index.php/jestm/index

Journal of Engineering Science and Technology Management

| ISSN (Online) 2828 -7886 |

JES-TM

Journal of Engineering Science & Technology Management

Article

Static Stress Simulation Analysis of Skid Frame of Portable Open Waste Container Using FEA Method on SolidWorks

Nurul Ulfah[1], Yosef Adicita[2*], Nugroho Pratomo Ariyanto[3], Roza Puspita[4] , Nurul Laili Arifin[5] , Ari Wibowo[6] and Rama Dani Eka Putra[7]

[1,2,3,4,5,6] Departement of Mechanical Engineering, Politeknik Negeri Batam, Batam,Indonesia
[7,] Departement of Mechanical Engineering, Universitas Bengkulu , Bengkulu,Indonesia

DOI: 10.31004/jestm.v5i2.315

E-mail: [1]nurululfah@polibatam.ac.id, [2*]yosef.adicita@polibatam.ac.id(Corresponding author)

ARTICLE INFORMATION

Volume 5 Issue 2
Received: 29 October 2025
Accepted: 28 September 2025
Publish *Online*: 30 September 2025
Online: at https:JESTM.org/

Keywords

Portable open waste container,
Skid frame,
Finite element analysis (FEA),
Stress simulation test,
Factor of safety (FOS) using
SolidWorks

A B S T R A C T

Portable open waste container is a traditional waste transport tool that is still widely used in various places. Skid frame is an important component in the construction of portable open waste container that serves to withstand static and dynamic loads during waste transport. Therefore, it is necessary to analyse the strength of the skid frame structure to ensure the design is able to withstand the given load. This research aims to analyse the stress distribution, deformation, and safety factor of the portable open waste container skid frame using the Finite Element Analysis method in Solidworks software. The analysis results show that the maximum stress that occurs in the skid frame is 249.921 MPa which almost reaches the yield stress of ASTM A36 steel material, which is 250 MPa, with a maximum deformation of 20.544 mm, and a minimum safety factor of 0.6. Based on these results, it can be concluded that the skid frame design needs to be optimised to improve the durability of the structure and safety during operation.

1. Introduction

This Waste management is a critical aspect in preserving the environment and maintaining sustainable development. One important element in the waste management system is the design and implementation of portable waste containers that are designed to facilitate the transportation and collection of waste from various locations (Adicita & Afifah, 2022). The skid frame of a portable waste container has a primary function as a load-supporting structure, which directly affects operational efficiency and safety. Thus, the structural stability and mechanical strength of the skid frame are parameters that cannot be ignored in the design and development process (Yingjun Guan et al., 2010).

The structural design of skid frames often faces various challenges, especially in ensuring that the operational loads received by the structure do not cause material failure. Previous research has shown that structural failures generally occur at connection areas or critical points that receive the highest stresses. For example, research highlights that conventional designs often fail to comprehensively consider stress distribution, making them prone to cracking and permanent deformation. This indicates an urgent need to use numerical simulation-based approaches to evaluate and optimise structural designs (Fitrianisa et al., 2023).

The Finite Element Analysis (FEA) method has proven to be an effective tool in analysing the mechanical behaviour of structural components (Abbas et al., 2020). Using finite element-based simulation, designers can model stress distribution, deformation and factors of safety under various loading conditions. Research by Patel and Parmar shows that the use of FEA is able to identify weak areas in the initial design as well as provide recommendations to strengthen the structure (Restu Pahlawan et al., 2021). In this context, SolidWorks software is one of the most widely used platforms due to its ability to integrate 3D design and numerical analysis in one working environment.

Static stress analysis is the main focus in this study, as static stress reflects the internal force distribution generated by a fixed load. This parameter is important for evaluating whether the material used is capable of withstanding the load without suffering permanent damage. Several previous studies have shown that static stress analysis is effective in evaluating the mechanical performance of structures, particularly in applications involving heavy and repetitive loads, such as waste transport systems (Adicita & Afifah, 2022).

The urgency of this research lies in the need to ensure the reliability of skid frame designs under increasingly challenging operational conditions. The waste management industry is constantly evolving, with an increasing need for portable waste containers capable of withstanding large loads and being used in a variety of environmental conditions. Without in-depth analysis, inadequate design can result in structural failure, which not only results in material loss but also safety risks to users (Putri Adnyani et al., 2019). Therefore, this research has not only academic value but also high practical relevance.

This research also seeks to bridge the gap that still exists between structural design theory and its real-world application. For example, although various design standards have been published for mechanical components, many skid frame designs are designed based on conservative assumptions without considering in-depth numerical analyses (Rizki Amelia & Prasetyo, 2019). This study aims to integrate numerical simulation-based approaches with structural design principles to produce more reliable and efficient designs.

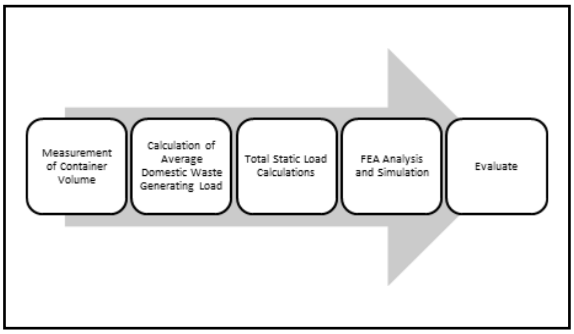

Figure 1. Study workflow

In this study, the material used for analysis is ASTM A36 structural steel, which is one of the most common materials in structural applications. This material was chosen because it has adequate mechanical strength as well as wide availability, making it suitable for application in skid frames. Previous studies have shown that ASTM A36 steel has ideal characteristics for structural applications, especially under static loading conditions (Sajid & Kiran, 2018).

Overall, this research offers a comprehensive approach in evaluating and improving the skid frame design of portable waste containers. By utilising FEA in SolidWorks, this research not only provides an in-depth overview of the stress distribution and deformation in the structure, but also generates design recommendations that can be implemented in the industry. The contribution of this research is expected to provide a solid foundation for further development in numerical simulation-based structural design.

2. Literature Riview

2.1 FEA on Steel Frame Structures

Finite Element Analysis, or FEA, has become one of the most practical ways to understand how steel frame structures behave under different loads. It allows engineers to visualize stress distribution, identify weak areas, and make design adjustments before any physical prototype is built. This approach saves both time and material, while giving designers more confidence that the frame can handle real working conditions. In many engineering studies, FEA is also used as a learning tool to compare how different frame geometries respond to the same load, which helps improve both accuracy and safety in structural design.

This method has been widely applied to diverse structural components, including boat trailer sliders and steel module frames, to evaluate performance under various loading conditions, and to predict buckling loads in modular steel structures. Furthermore, FEMs have been successfully employed to investigate the compressive behaviors of corner-supported modular steel sway frames, demonstrating high accuracy in predicting compressive resistance and stiffness (Khalid et al., 2024).

2.2 Static Analysis and Deformation

Static analysis plays a key role in evaluating whether a structure can remain stable when it is exposed to fixed or constant loads. Through this analysis, it becomes possible to estimate how much stress and deformation occur in each part of a frame. In practice, most deformation tends to appear at connection points or where the geometry changes, since those spots often carry higher stress. A well-designed frame should keep the deformation within elastic limits, ensuring that the material can return to its original shape when the load is removed. When deformation values are high, it usually means the frame needs reinforcement or modification of its cross-section. This process helps prevent long-term failure and ensures that the structure can perform safely during operation.

However, a significant challenge in structural analysis, particularly for modular steel frames, is that conventional steel design codes often overestimate column strength, highlighting the inadequacy of isolated member-based design equations for complex assemblies (Yang et al., 2025). Previous research has often focused on simplified mechanical analysis, with finite element method analyses rarely utilizing solid modeling, resulting in less detailed structural optimization (Yang et al., 2025). Moreover, the reliability of finite element models is highly dependent on the accuracy of their input parameters and boundary conditions, with simplifications in material modeling, support conditions, and geometric imperfections often leading to significant scattering in predictive outcomes (Khan et al., 2023).

2.3 Use of ASTM A36

ASTM A36 is one of the most widely used types of mild steel for construction and mechanical applications. It is favored because it combines moderate strength with good ductility and ease of fabrication. The material is easy to weld and form, which makes it suitable for many structural projects. Although its yield strength is not extremely high, its predictable behavior makes it ideal for testing and simulation. In static loading conditions, A36 tends to show consistent stress–strain patterns, allowing designers to rely on simulation data for accurate predictions. When used properly, this steel can provide both strength and flexibility, balancing safety and cost efficiency.

For instance, its application in cold-formed steel components, such as I-sections, C-sections, and L-sections, demonstrates its versatility in both non-structural and load-bearing built-up sections, which can be further optimized through advanced finite element analysis (Thangavel et al., 2025). The adaptability of ASTM A36 also extends to its use in composite cold-formed steel beams, offering a lighter alternative to hot-rolled steel beams and enabling various structural system applications (Nawar et al., 2025). This broad applicability, however, necessitates careful consideration of connection integrity, as modular steel structures rely heavily on robust load transfer mechanisms between members and modules (Morido-García & De Santos-Berbel, 2024; Yang et al., 2025).

2.4 Relevance to Skid Frames and Containers

Skid frames and waste container structures are good examples of how these analytical approaches come together in real engineering practice. These frames must support large, uneven loads and face repeated handling during daily operations. Because of this, designers often focus on how stress spreads along the longitudinal beams and at welded joints. When those areas are not well reinforced, cracks or excessive bending can occur over time. By applying FEA and static analysis, engineers can identify where to place stiffeners, adjust beam thickness, or modify the overall geometry. Even small changes in design can significantly improve load distribution and reduce deformation. This makes the combination of A36 material and simulation-based optimization a practical choice for developing safe, durable, and efficient skid frame structures.

Previous research on steel structures has explored various aspects, including the cyclic-plastic response of mild steels, which are extensively used due to their ductility and work-hardening potential, especially in applications demanding low-cycle fatigue assessment (Nath et al., 2021). Further studies have also focused on the computational modeling of residual stress in welded high-strength steel box sections, where finite element method models are validated using experimental data to explore the impact of various parameters on residual stress distributions (Tu et al., 2025).

3. Research Methodology

This study was designed to analyse the static stresses in the skid frame of a portable waste container using the Finite Element Analysis (FEA) method performed on SolidWorks software. The method used includes the following steps: measurement of container volume, calculation of average domestic waste load, calculation of total container static load, and FEA simulation based on the parameters obtained from the previous steps. Each step is described in detail with real measurement results to provide an evaluation of the approach taken.

3.1 Measurement of Container Volume

Volume measurement using standard containers from waste transport vehicles. In this study using a sample of the most common vehicle used by the Environmental Service with the type of ArmRoll vehicle with Portable Container. This high efficiency container bin has the advantage of being able to stay at the waste disposal site, then the full container bin will be transported on the next travel route.

Figure 2. Specification of truck (Spesifikasi Teknis pengadaan Arm Roll Kontainer, 2012)

Portable Container Bin specifications consist of a 3 mm SPHC floor and UNP 100 main frame with ASTM A36 Casbon Stell Material (Widiastuti et al., 2019). The container volume is calculated based on the outer dimensions specified in the technical specifications. The actual data shows that the container has outer dimensions of 3310 mm length, 1890 mm width and 1350 mm height. The volume is calculated using the basic geometry formula.

$$V = P \times L \times T \quad (1)$$

According to the calculation, the maximum volume capacity was found to be 8.45 M3. However, the effective capacity was 6m3 to account for the unfilled empty space.

3.2 Calculation of Average Generated Load of Domestic Waste

The waste load is calculated based on the standard density ρ of waste and the effective volume of the container. Based on SNI 19-3964-1994, the density of domestic waste is in the range of 350-400 Kg / m3 (Adicita & Afifah, 2022). With ρwaste = 375 Kg/M3 and filled volume (V) = 6 m3, the waste period is calculated as follows:

$$M = \rho waste \times filled\ volume \quad (2)$$

According to the calculation, the waste load was 2250 kg.

3.3 Total Static Load Calculation of Container

The total static load is the sum of the empty weight of the container and the weight of the transported waste. Based on the material specifications, the empty weight of the container is estimated to be 600 Kg. The total static load is calculated using the formula:

$$W\ Total = W\ empty + (M \times g) \quad (3)$$

According to the calculation, the W total was 22.815 N.

3.4 FEA Analysis & Simulation in SolidWorks

The simulation process uses the updated dimensions and is performed in the following stages:

1. Geometry Modelling: A 3D model was created based on the new sizes, including length 3310 mm, width 1890 mm, and height 1350mm, using SolidWorks software.
2. Material Definition: ASTM A36 is a standard carbon steel widely used for its strength, ductility, and economical cost. With a main chemical composition such as carbon (0.25-0.29%), manganese (1.03%), and phosphorus (0.04%), this material offers a balance between strength and ease of fabrication. The yield stress of 250 MPa and maximum tensile stress of 400-550 MPa make it ideal for structural applications that require high resistance to static loads (Abbas et al., 2020). The modulus of elasticity of 200 GPa indicates adequate stiffness, while the elongation at break of 20% indicates good deformation capability before fracture, providing additional safety under repeated load conditions. ASTM A36 is also easily welded and formed, making it suitable for a wide range of applications such as building frames, bridges, and mechanial components. With these properties, ASTM A36 supports applications such as skid frames, where material strength and load distribution are critical (Buranapunviwat & Sojiphan, 2021).
3. Application of Loads and Boundary Conditions: In this simulation, the application of loads and boundary conditions is designed to reflect the real operational conditions of the skid frame as the main structural component of the portable waste container. This step is critical to ensure that the analyses reflect stress and deformation distributions that match the actual loads that the structure will receive in a working environment (Farida et al., 2024). The load is statically applied as a vertical normal force (Z-direction) on the bottom longitudinal element, which acts as the main support of the structure. The magnitude of the load was calculated based on a combination of the empty weight of the container and the full waste mass, with a total applied load of 22,815 N. This loading was evenly distributed at key contact points to simulate the waste mass distribution conditions inside the container. This approach is designed to provide a realistic simulation of force distribution, where the largest loads are concentrated on structural elements that directly receive gravitational forces (Farida et al., 2024). Boundary conditions are set at key support points at the base of the skid frame. These points are fixed to represent direct contact between the skid frame and the ground surface or transport vehicle. The establishment of these boundary conditions ensures that the reaction forces generated during loading can be realistically transferred to the structural elements. These conditions simulate the structural interactions that occur in waste transport in real environments.

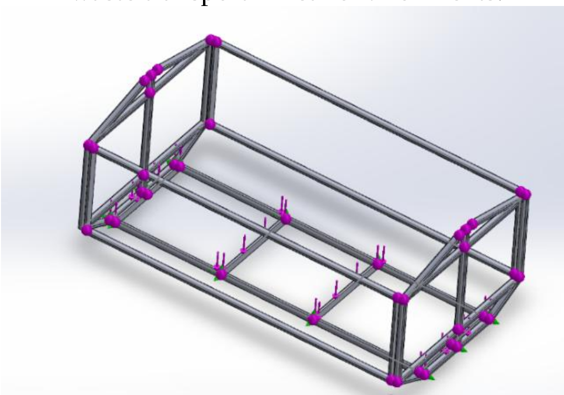

Figure 3. Joint force with load 22,815 N

4. Simulation Analysis: Simulation analysis shows that the current design is close to the

material tolerance limit, with high stress concentrations, significant deformation, and a low factor of safety. Design modifications are required to increase the safety margin and ensure long-term structural reliability. These measures will not only improve mechanical performance but also ensure that the structure is capable of meeting more stringent operational requirements in the future.

4. Results and Discussion

4.1 Static Stress Test Simulation Results and Design Analysis

The static stress simulation results performed on the skid frame showed a maximum stress concentration of 249.921 MPa, which almost reached the yield stress of ASTM A36 steel material, which is 250 MPa. The maximum stress was identified at the connection area of the lower longitudinal element, which serves as the main force channeling point. This phenomenon can be explained by the structural design and load distribution, where non-optimal frame geometry results in high stress concentrations in certain areas. The influence of the shape of the structural elements as well as the limited placement of the reinforcing elements contributed to this result.

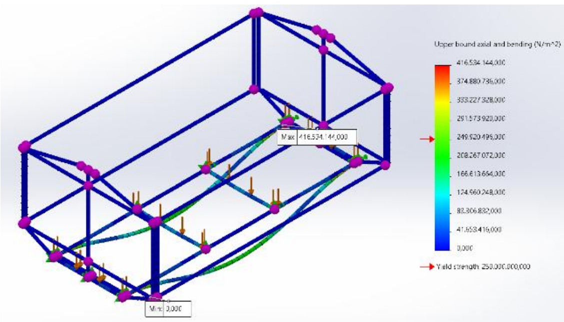

Figure 4. Static stress simulation

This Skid frame designs with a rectangular configuration and a single longitudinal element provide efficient performance for resisting vertical loads. However, this design has inherent weaknesses, especially at the joints of the horizontal and vertical elements that receive the greatest concentration of forces. This uneven stress distribution indicates that the main structural elements are working at maximum capacity, while some other elements are underutilised. These results are consistent with the findings of previous research by Mustasyar, who found that conventional designs on skid frames often exhibit high stress concentrations at joint areas (Perkasa et al., 2021).

However, compared to the study conducted by Ghea Rizki Amelia, who analysed skid frames for horizontal pressure vessels, the maximum stress identified in this study was higher (Rizki Amelia & Prasetyo, 2019). In that study, the maximum stress was recorded at 150,795 MPa, well below the yield stress limit of S275JR material of 275 MPa. This difference could be due to the different load distribution, where pressure vessels tend to have a more centralised load distribution at certain points, whereas waste containers have a more even load along the frame surface.

4.2 Deformation Simulation Results (Displacement test)

The simulation results show a maximum deformation of 20.544 mm at the bottom longitudinal area of the skid frame. This deformation occurs at elements with high stress concentrations due to uneven load distribution. This indicates that although the current design is able to withstand the total static load without direct failure, potential deformation problems need to be considered, especially in the structural elements that receive the largest loads. Further in-depth analysis shows that the existing design form significantly affects the deformation results, especially in terms of the overall stability of the structure. develop the calculation analysis.

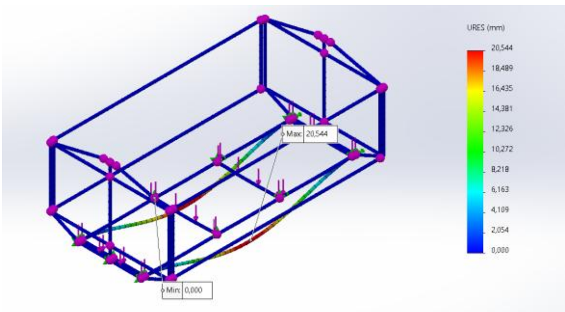

Figure 5. Displacement simulation results

The skid frame structure in this study was designed using frame elements in the form of rectangular beams, which generally provide better stress distribution compared to simpler profiles such as pipes or round bars (Restu Pahlawan et al., 2021). However, this design also creates force concentration points at the joints between elements, especially in areas that receive direct loads from the container. The high stress at the joint area triggers localised deformation, which then affects the total deformation of the structure.

The geometrically symmetrical shape of the skid frame helps to distribute the load throughout the structure. However, imbalances in loading, such as concentrated loads in certain areas, enlarge local deformations in the lower longitudinal elements. This situation is in line with the findings of simulation results of previous research on automotive frames, which showed that uneven load distribution in skid frame structures can cause excessive deformation in areas with lower stiffness (Abbas et al., 2020).

4.3 Factor of Safety Result

The geometrically symmetrical shape of the skid frame helps to distribute the load throughout the structure. However, imbalances in loading, such as concentrated loads in certain areas, enlarge local deformations in the lower longitudinal elements. This situation is in line with the findings of simulation results of previous research on automotive frames, which showed that uneven load distribution in skid frame structures can cause excessive deformation in areas with lower stiffness[4].

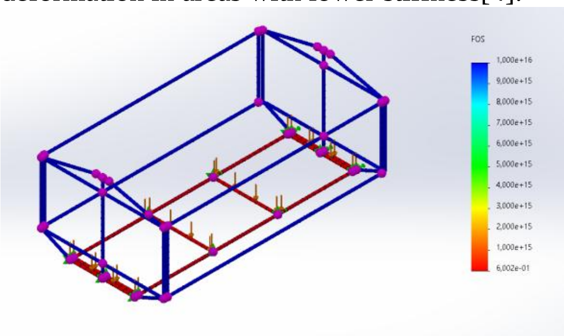

Figure 6. Factor of Safety Simulation Results

Skid frame designs consisting of main support elements, such as steel profiles with joints at critical points, tend to create stress concentrations in the joint areas and lower longitudinal elements. This configuration, although efficient in terms of materials, has limitations in distributing loads evenly throughout the structure. As shown in the simulation results, the maximum stress occurs at the longitudinal joint area, which receives the largest load from the static force distribution.

In addition, the symmetrical and rectangular geometry of the skid frame also affects the factor of safety. According to Dewi Tumewu, non-optimal geometry shapes can create stress concentrations in the structure and lower the factor of safety (Tumewu et al., 2019). To increase the factor of safety, design modifications are required by considering aspects of load distribution, structural element configuration, and frame geometry optimisation. Thus, skid frame structures can be designed more reliably and safely to support hazardous waste container applications. (Abbas et al., 2020; Evtushenko S.I. et al., 2017; Restu Pahlawan et al., 2021)

5. Conclusion

The simulation results show a clear relationship between the structural design of the skid frame and the stress distribution, maximum deformation, and factor of safety (FOS). The maximum stress value of 416,534 MPa, far exceeding the ASTM A36 material yield stress limit of 250 MPa, highlights the major weaknesses of the current design, especially at the main joints and lower longitudinal elements. These locations became stress concentration points due to the suboptimal design in distributing static loads. The maximum deformation of 20.544 mm indicates that the structure underwent significant deformation, although it is still within the tolerance limits for certain applications. However, deformations of this magnitude indicate that the structure may suffer long-term damage, especially under repeated load conditions. The minimum factor of safety of 0.6, which is well below the recommended standard (≥1.2), emphasises the potential risk of structural failure.

These results are in line with previous studies, such as the one conducted by Abidin, which highlighted the importance of adding stiffening elements and geometry optimisation to reduce stress concentrations. The study showed that redesigning by adding transverse stiffeners and increasing the dimensions of the frame elements can reduce the maximum stress by up to 25% and increase the FOS to more than 1.2. Amelia and Prasetyo's research also confirmed that high stresses in the main joints are a common problem in skid frame design, especially without additional reinforcement (Rizki Amelia & Prasetyo, 2019). This linkage shows that the weaknesses of the current design can be overcome by approaches that have been proven in previous studies, such as load distribution optimisation, stiffening element addition, and geometry modification. With the implementation of these measures, stress distribution can be improved, deformation can be minimised, and FOS can be increased, making the structure safer and more reliable in long-term applications.

References

Abbas, H., Juma, D., Ma'ruf, M., & Jahuddin, R. (2020). ILTEK : Jurnal Teknologi Penerapan Metode Elemen Hingga Untuk Desain Dan Analisis Pembebanan Rangka Chassis Mobil Model Tubular Space Frame. *ILTEK: Jurnal Teknologi*, *15*(No. 2).

Adicita, Y., & Afifah, A. S. (2022). Analisis Sistem Pemilihan dan Daur Ulang Sampah Rumah Tangga di Daerah Perkotaan Menggunakan Pendekatan Life Cycle Assessment (LCA). *Jurnal Ilmu Lingkungan*, *20*(2), 406–413. https://doi.org/10.14710/jil.20.2.406-413

Buranapunviwat, K., & Sojiphan, K. (2021). Destructive testing and hardness measurement of resistance stud welded joints of ASTM A36 steel. *Materials Today: Proceedings*, *47*, 3565–3569. https://doi.org/10.1016/j.matpr.2021.03.562

Evtushenko S.I., Petrov I.A., Alekseev S.A., & Alekseeva A.S. (2017, May). *2017 International Conference on Industrial Engineering, Applications and Manufacturing (ICIEAM) : proceedings : South Ural State University (national research university), Chelyabinsk, Russia, May 16-19, 2017*.

Farida, I., Prasetyo, K., Furqon, M. M., & Dika, P. (2024). Analisis Kekuatan Desain Skid Frame Dengan Simulasi Linear Statik Berbasis Inventor-Nastran. *Jurnal Patriot Biru TNI AU*, *3*(3).

Fitrianisa, D., Luqyana, D., & Milah Muhamad, A. (2023). Numerical simulation analysis of structural strength of portable skid for storage tank with 50.000 liters capacity. *Disseminating Information on the Research of Mechanical Engineering-Jurnal Polimesin*, *21*(2). http://e-jurnal.pnl.ac.id/polimesin

Khalid, N., Hussin, N. H. F., Saat, A. M., Khalid, P. Z. M., & Rahamaddulla, S. R. (2024). Design and Strength Analysis of Boat Trailer Sliders Using the Finite Element Method. In *Advanced Structured Materials* (Vol. 208, pp. 321–334). Springer. https://doi.org/10.1007/978-3-031-56844-2_28

Khan, K., Chen, Z., Liu, J., & Tsavdaridis, K. D. (2023). Compressive behaviors of corner-supported modular steel sway frames with rotary inter-modular connections. *Thin-Walled Structures*, *193*. https://doi.org/10.1016/j.tws.2023.111245

Morido-García, G., & De Santos-Berbel, C. (2024). Structural Performance of Bolted Lateral Connections in Steel Beams under Bending Using the Component-Based Finite Element Method. *Applied Sciences (Switzerland)*, *14*(9). https://doi.org/10.3390/app14093900

Nath, A., Barai, S. V., & Ray, K. K. (2021). Studies on the experimental and simulated cyclic-plastic response of structural mild steels. *Journal of Constructional Steel Research*, *182*. https://doi.org/10.1016/j.jcsr.2021.106652

Nawar, M. T., El-Zohairy, A., Eisa, A. S., Mohammed, A., & Gomaa, S. (2025). FE Parametric Study of Composite Cold-Formed Steel Beams Under Positive and Negative Loadings. *Journal of Composites Science*, *9*(5). https://doi.org/10.3390/jcs9050209

Perkasa, M., BesarTeknologi Kekuatan Struktur BRIN, B., & Selatan, T. (2021). Analisis Kekuatan Struktur Beam Skid Base Dengan Metode Elemen Hingga (FEM). *Jurnal Teknik Mesin: CAKRAM*, *4*(2).

Putri Adnyani, L., Aisyah, N., Gusriana Harahap, R., Dianiswara, A., Studi Teknik Kelautan, P., & Teknologi Kalimantan, I. (2019). Analisa Kekuatan Deck BARGE NANIA 10070 DWT Dengan Dua Metode Pembebanan. In *KAPAL* (Vol. 16, Issue 2). http://ejournal.undip.ac.id/index.php/kapal

Restu Pahlawan, A., Hanifi, R., & Santosa, A. (2021). Analisis Perancangan Frame Gokart dari Pengaruh Pembebanan dengan Menggunakan CAD Solidworks 2016. *Jurnal METTEK*, *7*(1), 1. https://doi.org/10.24843/mettek.2021.v07.i01.p01

Rizki Amelia, G., & Prasetyo, N. A. (2019). UJI

KEKUATAN SKID-FRAME PADA HORIZONTAL PRESSURE VESSEL BERBASIS SIMULASI NUMERIK. *Jurnal Teknologi Dan Riset Terapan (JATRA)*, *1*(1). http://jurnal.polibatam.ac.id/index.php/JATRA,https://jurnal.polibatam.ac.id/index.php/JATRA

Sajid, H. U., & Kiran, R. (2018). Influence of high stress triaxiality on mechanical strength of ASTM A36, ASTM A572 and ASTM A992 steels. *Construction and Building Materials*, *176*, 129–134. https://doi.org/10.1016/j.conbuildmat.2018.05.018

Spesifikasi Teknis Pengadaan Arm Roll Kontainer. (2012). *SJA 001-ARM ROLL + KONTAINER TERTUTUP*. www.karoserisenangjaya.com,

Thangavel, P., Palanisamy, M., Kumar, D. R., Wipulanusat, W., Sunkpho, J., & Roy, K. (2025). Investigation of cold formed steel angle compression through high throughput design FEA and machine learning. *Scientific Reports*, *15*(1). https://doi.org/10.1038/s41598-025-03991-9

Tu, Y., Liu, Z., Tu, Y., & Sheng, M. (2025). Computational modeling of residual stress in welded high-strength steel box sections. *PLOS ONE*, *20*(9 September). https://doi.org/10.1371/journal.pone.0332445

Tumewu, D., Saerang, E. J., & Harahap, J. T. (2019). Studi Perbandingan Metode Perhitungan Struktur Statis Tak Tentu pada Portal Bangunan Gedung. *Jurnal Teknik Sipil Terapan*, *1*(2). http://Jurnal.polimdo.ac.id/

Widiastuti, H., Arifin, N. L., & Mutiarani. (2019). Waste sorting campaign toward green campus: Case study in Politeknik Negeri Batam. *AIP Conference Proceedings*, *2085*. https://doi.org/10.1063/1.5094985

Yang, K., Khan, K., Yang, Y., Jiang, L., & Chen, Z. (2025). Impact of Inter-Modular Connections on Progressive Compressive Behavior of Prefabricated Column-Supported Volumetric Modular Steel Frames. *Crystals*, *15*(5). https://doi.org/10.3390/cryst15050413

Yingjun Guan, Yang Zhao, Deqiang Mu, & Yingzhi Guan. (2010, October). *2010 International Conference on E-Product E-Service and E-Entertainment*.