FRAME STRENGTH ANALYSIS ON MOTORIZED RICKSHAWS IN BUKITTINGGI CITY, WEST SUMATRA USING SOLIDWORKS SIMULATION

[1]**Zalyah Syukri**,[*2]**Muchlisinalahuddin**,[3]**Rudi Kurniawan Arief**
[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]*zalyah1403@gmail.com; *[*2]*muchlisinalahuddin.umsumbar@gmail.com;* [3]*rudikarief@umsb.ac.id*
[4]*yunifadilabkt@gmail.com;*[5]*reyhanstevano55@gmail.com*

*Corresponding author: muchlisinalahuddin.umsumbar@gmail.com

Abstract.*This study analyzes the strength of the frame of a goods-carrying motorized rickshaw in Bukittinggi City, which is often manufactured without clear engineering regulations. This study tested the initial model and five modified design variations with static loads on ASTM A36 steel material. This was done using Finite Element Analysis (FEA) simulation in SolidWorks. The main objective was to find the best design by assessing the stress, deformation, and safety factor. The simulation results showed that the initial model had a Von Mises stress of 106 MPa and a deformation of 0.589 mm. However, the Modification 3 configuration was the best because it reduced the stress to 86 MPa and reduced the deformation to 0.301 mm, while increasing the safety factor to 2.9. The results of this study recommend the Modification 3 configuration as a good design solution.*

Keywords: *Finite Element Analysis; Frame Strength; Motorized Rickshaw, SolidWorks Simulation, Structural Analysis.*

1. INTRODUCTION

Motorized rickshaws are a traditional mode of transportation that is still widely used in various regions of Indonesia, including Bukittinggi City, West Sumatra.(Nugroho et al., 2024)In addition to being used as passenger vehicles, motorized rickshaws also function as freight vehicles due to their relatively compact dimensions and ability to navigate narrow, uneven, and winding roads. The geographic characteristics of Bukittinggi City mean that motorized rickshaws operate under a combination of static loads and quite demanding operational conditions. These conditions have the potential to cause significant stress and deformation on the vehicle frame if not designed with adequate strength.

The frame is the main component of a motorized rickshaw, which supports the entire weight of the vehicle, including the driver, cargo, and other components.(Mail, 2020)Frame strength and reliability significantly impact user safety and comfort. However, in practice, the process of constructing motorized rickshaw frames at the craftsman level is generally based on empirical experience without the support of measurable structural analysis. This lack of technical evaluation of frame strength can potentially lead to design errors, shorten the vehicle's lifespan, and increase the risk of structural failure.(Agustina & Soedarsono, 2020).

The development of computer-aided engineering (CAE) technology enables more efficient and accurate structural analysis through numerical simulation approaches.(Prabandono et al., 2024)SolidWorks is a widely used software program in mechanical engineering, which includes a Finite Element Analysis (FEA)-based simulation module. FEA can provide a virtual

representation of stress distribution, deformation, and safety factors in a frame structure, making it an alternative to more expensive and time-consuming physical testing.(Intec Precision Engineering Sdn. Bhd. et al., 2024).

Although the use of motorized rickshaws for goods is quite dominant in Bukittinggi City, technical studies related to the strength of motorized rickshaw frames based on FEA simulations are still limited. Therefore, this study was conducted to analyze the strength of motorized rickshaw frames for goods using Finite Element Analysis (FEA) simulations in SolidWorks software. This study aims to evaluate the stress distribution, deformation, and safety factor values of the frame based on operational load conditions, so that it can serve as a technical basis in assessing the feasibility of frame design and providing recommendations for improving the strength and reliability of the structure.(Arif Nurul Mustofa et al., 2025).

2. LITERATURE REVIEW

2.1 Motorized Goods Transport Rickshaw

A cargo-carrying motorized rickshaw is a motorcycle modified with a carriage or cargo bed, designed to increase its cargo-carrying capacity. These vehicles are widely used in urban areas with limited road infrastructure, such as narrow, hilly, and uneven roads. In Bukittinggi City, cargo-carrying motorized rickshaws play a vital role in supporting small-scale distribution of goods, particularly in traditional markets and densely populated residential areas.(Oliveira et al., 2017).

Structurally, motorized rickshaws for transporting goods use an additional metal frame as the main structure of the carriage. This frame is designed to support a wide range of cargo loads and is equipped with support wheels to maintain vehicle stability.(Ardiyansyah et al., 2023)However, modifying conventional frames without proper technical calculations has the potential to cause safety problems due to overloading or uneven load distribution.

2.2 Vehicle Structure and Frame

The vehicle's frame structure serves as the primary element supporting all vehicle components and distributing loads during operation. In motorized tricycles used for transporting goods, the frame must be able to withstand the weight of the driver, the load, and additional forces caused by uneven road conditions.(Blatnický et al., 2022; Wu et al., 2025).

The motorized rickshaw frame configuration combines a motorcycle frame and an additional frame for a freight train. This combination results in complex load distribution and can potentially lead to stress concentrations at specific points, particularly at frame joints. Therefore, structural analysis is necessary to understand the frame's behavior and ensure the design's feasibility.

2.3 Loads Acting on the Frame

The frame of a motorized rickshaw transporting goods experiences various types of loads during operation. Static loads result from the weight of the frame, the driver, and the cargo being transported, which directly affect the basic deformation of the frame.(Hussin et al., 2020; Nabil & Faizin, 2025).

In addition, the frame is also subjected to dynamic loads due to road shocks, acceleration, deceleration, and vehicle maneuvers. Impact loads can occur when a vehicle passes over potholes or uneven road surfaces, while engine vibrations produce repetitive loads that can accelerate material fatigue. The combination of these loads must be considered in a structural strength analysis.(Bucharest Civil Engineering University Calea Plevnei 59, Bucharest, Romania & Cristian Ștefan, 2024).

2.4 Properties and Characteristics of Materials

Frame material is a critical factor in determining a structure's load-bearing capacity. Metal materials such as carbon steel are widely used in vehicle frames due to their good mechanical strength, adequate ductility, and ease of shaping.(Blatnický et al., 2020; Kulkarni & Sawant, 2024; Trzepieciński & Najm, 2024).

Material characteristics are not only determined by mechanical properties such as elastic modulus and yield strength, but are also influenced by manufacturing processes and environmental conditions, such as corrosion. Therefore, proper material property selection and modeling are crucial in simulation-based structural analysis.(Bean et al., 2022).

2.5 Stress and Deformation

Stress and deformation are key parameters in evaluating structural strength. Stress arises as a material's internal response to external loads, while deformation indicates the change in shape or size of a structure due to those loads.

To ensure structural safety, a safety factor is used, which compares the working stress to the material strength. The safety factor accounts for loading uncertainty, variations in material properties, and potential design or manufacturing errors. This concept forms the basis for assessing the suitability of a motorized rickshaw frame for transporting goods.(Cochrane et al., 2019; Poutanen et al., 2020).

2.6 SolidWorks Simulation

SolidWorks Simulation is a structural analysis module integrated into SolidWorks software. This module allows users to perform numerical frame strength analysis directly on previously created three-dimensional models.(Ferroudji et al., 2017).

SolidWorks Simulation's advantages lie in its integration of CAD modeling and structural analysis, the availability of material libraries, and the ease of visualizing analysis results. This software allows for efficient evaluation of motorized rickshaw frames prior to physical testing.(Hao et al., 2023; Institute of Solid Mechanics of the Romanian Academy et al., 2022).

2.7 Finite Element Analysis (FEA)

Finite Element Analysis(FEA) is a numerical method used to analyze the behavior of structures by dividing the model into small, interconnected elements. Each element is represented by a mathematical equation that describes the relationship between forces, stresses, and deformations.(Güler & Karagülle, 2016; Partheniadis et al., 2022).

The FEA method allows the analysis of structures with complex geometries and loading conditions, making it widely used in mechanical engineering. In the analysis of the frame of a motorized tricycle for transporting goods, FEA is used to predict stress distribution, deformation, and structural safety factors, thereby identifying critical areas prone to failure.

3. RESEARCH METHODS

This study applies a quantitative descriptive method with an engineering analysis approach to evaluate the structural strength of motorized rickshaw frames in Bukittinggi City through computer-based simulations. The descriptive nature of this study aims to describe the technical conditions in depth, while the quantitative aspect is sourced from data processing from field measurements and Finite Element Analysis (FEA) calculations using SolidWorks software.(Shkut, 2023)As a simulation-based study, the analysis was conducted by modeling the frame in 3D to test the structural response to operational loads, allowing researchers to accurately identify critical points, deformations, and safety factors. Overall, this study is applicable because it provides a concrete engineering solution regarding the feasibility and safety of local transportation structures through a measurable, scientific approach.

Figure 1. Flow chart

Several stages of research are listed in the flowchart above as follows:

1. *Identification of problems*

   This phase was conducted to identify key issues related to the strength and reliability of motorized rickshaw frames operating in Bukittinggi City. Field observations indicated that commonly used frames did not meet safety standards and faced varying road conditions. Based on these conditions, the researchers formulated the core problem: evaluating frame strength using an analytical approach using SolidWorks Simulation.

2. *Literary Studies*

   At this stage, various scientific references were collected, such as journals, textbooks, previous research, and technical standards related to frame analysis, motorized tricycle frame design, and the FEA (Finite Element Analysis) method. This literature review aims to strengthen the theoretical basis, understand the development of previous research, and determine relevant variables in the study.

3. *Design*

   The frame design was created in 3D using SolidWorks software based on collected field data. A detailed CAD model was prepared to accurately represent the shape and configuration of the motorized rickshaw frame for transporting goods. This model included profile dimensions, structural connections, and supporting frame elements that affect the structure's strength. The resulting CAD design was then used as the primary object in Finite Element Analysis.

4. *FEA Simulation*

   The 3D model was tested numerically by applying boundary conditions, such as determining the type of material, which in this study used ASTM A36 steel material, determining the fulcrum (fixture), and applying static loads that represent the load of the goods.

5. *Analysis*

   The simulation results are evaluated by reviewing the distribution of Von Mises Stress, Displacement, Strain, and Factor of Safety (FoS) to identify critical points that have the potential to experience structural failure.
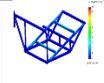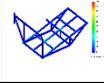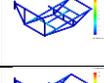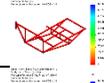
6. *Conclusion*

   This study uses a systematic procedure based on field observations and Finite Element Analysis (FEA) simulations to evaluate structural strength, accurately identify critical points, and provide recommendations for a more optimal and safe motorized rickshaw frame design.

4. RESULTS AND DISCUSSION

4.1 Results

Table 1. FEA Simulation Results

Typography Design	VMS (MPa)	FOS	STRAIN (Nm)	URES (mm)

Early Model	106		2.4		0.0003		0.589	
Modification 1(2nd Design)	109		2.3		0.0004		0.474	
Modification 2(3rd Design)	89		2.8		0.0003		0.380	
Modification 3(4th Design)	86		2.9		0.0002		0.301	
Modification 4(5th Design)	98		2.6		0.0003		0.533	
Modification 5(6th Design)	101		2.5		0.0003		0.457	
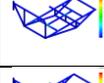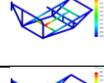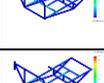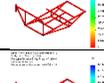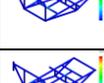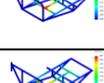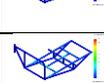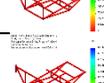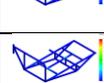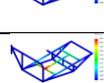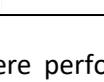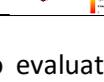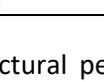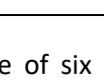
Static simulations were performed to evaluate the structural performance of six design variations under identical loading conditions. The applied loads represent the working loads acting on the structure during operation, while fixed supports were applied at designated support regions to simulate actual installation constraints. Evaluation parameters included von Mises stress (VMS), displacement, strain, and factor of safety (FOS).

The initial model yielded a von Mises stress of 106 MPa, a factor of safety of 2.4, a strain of 0.0003, and a maximum displacement of 0.589 mm. These values served as a basis for comparison with the modified design.

In Modification 1 (Design 2), the von Mises stress increased to 109 MPa, which is equivalent to a 2.83% increase compared to the initial model. The safety factor decreased to 2.3 (−4.17%), while the displacement decreased to 0.474 mm, indicating a reduction of 19.52%. However, the strain increased to 0.0004, indicating higher local deformation.

Modification 2 (Design 3) resulted in a significant reduction in the von Mises stress to 89 MPa, which is equivalent to a 16.04% reduction from the initial design. The safety factor increased to 2.8 (+16.67%), and the displacement decreased to 0.380 mm, which represents a 35.48% reduction. The strain value remained at 0.0003, indicating stable elastic behavior.

The best performance was obtained in Modification 3 (Design 4). This design produced the lowest von Mises stress of 86 MPa, which represents an 18.87% reduction compared to the initial model. The safety factor increased to 2.9 (+20.83%), the strain decreased to 0.0002 (−33.33%), and the displacement was minimized to 0.301 mm, indicating a significant reduction of 48.90%.

For Modification 4 (Design 5), the von Mises stress decreased moderately to 98 MPa (−7.55%), with a safety factor of 2.6 and a displacement of 0.533 mm (−9.51%). Meanwhile, Modification 5 (Design 6) produced a von Mises stress of 101 MPa (−4.72%) and a displacement of 0.457 mm (−22.41%), with a safety factor of 2.5.
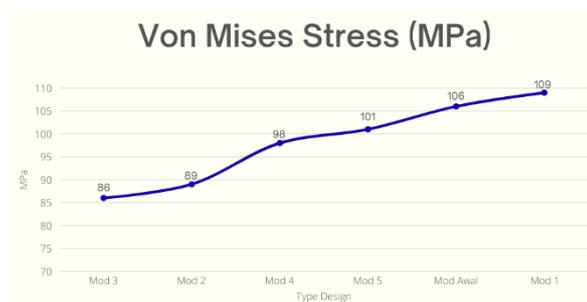
Graph 1. Von Mises Stress (VMS)

The von Mises stress (VMS) graph shows the variation in stress values for each design configuration. The highest stress occurred in Modification 1 with a value of 109 MPa, while the lowest stress was observed in Modification 3 at 86 MPa. This trend indicates that geometric modifications significantly affect stress distribution, with better design configurations able to reduce stress concentration.

Graph 2. Safety Factor

The von Mises stress (VMS) graph shows the variation in stress values for each design configuration. The highest stress occurred in Modification 1 with a value of 109 MPa, while the lowest stress was observed in Modification 3 at 86 MPa. This trend indicates that geometric modifications significantly affect stress distribution, with better design configurations able to reduce stress concentration.

4.2 Stres

Von Mises stress analysis was used as the primary indicator to evaluate the structural strength of each design under static loading conditions. The results showed that geometric modifications significantly affected the stress distribution within the structure. Designs with better geometry exhibited lower stress concentrations, as indicated by lower VMS values.
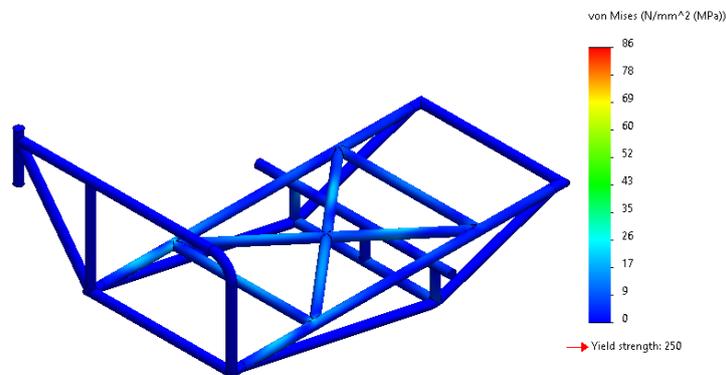

Figure 2. Modified Von Mises Stress 3

The lowest stress value of 86 MPa observed in Modification 3 (Figure 3) indicates that this configuration provides a more uniform load distribution compared to the initial design. Conversely, the increased stress observed in Modification 1 indicates that the applied geometric changes resulted in stress concentrations, which could negatively impact structural durability.

4.3 Displacement

The analysis displacement represents the stiffness of the structure under applied loads. The initial model experienced the largest displacement of 0.589 mm, indicating the lowest stiffness among all designs. A gradual reduction in displacement was observed in subsequent modifications, with the smallest displacement recorded in Modification 3 at 0.301 mm.
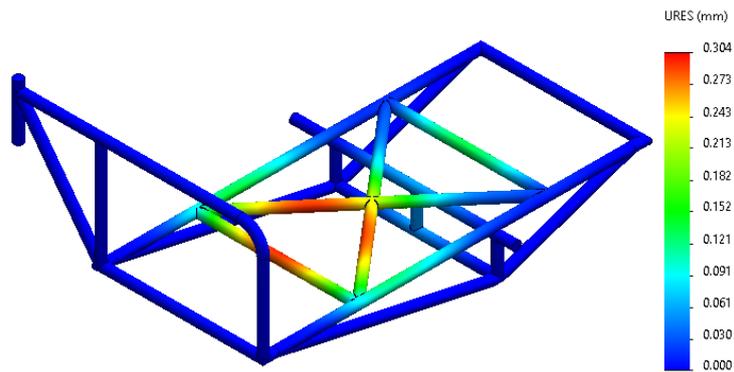
Figure 3. Displacement Modification 3

The nearly 50% reduction in displacement achieved by Modification 3 confirms a significant increase in structural stiffness. Lower displacement values indicate better resistance to deformation, which is crucial for maintaining dimensional stability during operation.

4.4 Safety Factors

Factor of Safety (FOS) analysis is defined as the ratio between the allowable material strength and the stress induced. A design is generally considered safe if the FOS value exceeds 2 for static loading conditions.
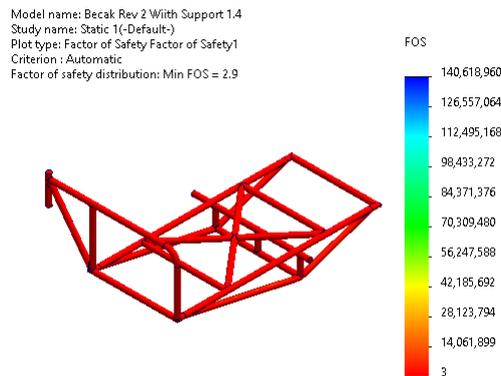

Figure 4. Modified Safety Factor 3

All design variations produced FOS values greater than 2, indicating a safe structural condition. However, Modification 3 achieved the highest FOS value of 2.9, reflecting the largest safety margin. Conversely, Modification 1 produced the lowest FOS value (2.3), making it the least favorable design from a safety perspective.

4.5 Discussion

Simulation results indicate that design modifications play a significant role in improving structural performance. While some modifications reduce displacements, they can simultaneously increase stress levels, as observed in Modification 1. This highlights the importance of balancing stiffness and stress distribution in structural optimization. Modification 3 (Design 4) provides the most optimal configuration by achieving the lowest stress, highest safety factor, minimum strain, and smallest displacement. These results demonstrate that optimal design does not necessarily require maximum material usage, but rather an efficient geometric configuration that balances strength, stiffness, and safety.

CONCLUSION

Based on the results of static finite element analysis conducted on six design variations, it can be concluded that geometric modifications have a significant impact on structural performance. The parameters evaluated include von Mises stress, safety factor, strain, and displacement.

Simulation results show that Modification 3 (Design 4) is the most optimal configuration among all designs. This design produces the lowest von Mises stress of 86 MPa, the highest safety factor of 2.9, the minimum strain value of 0.0002, and the smallest displacement of 0.301 mm. Compared to the initial model, this configuration reduces the stress by 18.87% and the displacement by almost 50%, while increasing the structural safety margin.

All design variations exhibit safety factor values greater than 2, indicating that all configurations are structurally safe under static loading conditions. However, certain modifications exhibit increased stiffness without corresponding stress reduction, highlighting the importance of balancing stress and deformation distribution in structural optimization.

Therefore, it can be concluded that optimal design does not necessarily require maximum material usage, but rather an efficient geometric configuration that balances strength, stiffness, and safety. The findings of this study can serve as a reference for further design development and future experimental validation.

REFERENCES

Agustina, S., & Soedarsono, S. (2020). Analysis of Retaining Walls for Water Pump Houses. Review in Civil Engineering, 4(1). https://doi.org/10.31002/rice.v4i1.1987

Ardiyansyah, M., Yunaldi, W., & Zulfiko, R. (2023). ENFORCEMENT OF THE CRIMINAL ACTION OF REPLACING MOTOR VEHICLE ENGINES BASED ON THE ROAD TRAFFIC AND TRANSPORTATION LAW. PALAR | PAKUAN LAW REVIEW, 9(3), 90–100. https://doi.org/10.33751/palar.v8i3

Arif Nurul Mustofa, M., Bahatmaka, A., Dwi Handiko, Y., Dwi Rinonggo, M., & Ichlas Imran, A. (2025). Analysis of Frame Strength on Vessel Deck Structures of Ships and Buses Using Finite Element Method. Enthalpy: Scientific Journal of Mechanical Engineering Students, 10(2), 75–87. https://doi.org/10.55679/enthalpy.v10i2.22

Bean, P., Lopez-Anido, R.A., & Vel, S. (2022). Integration of Material Characterization, Thermoforming Simulation, and Post-Forming Structure Analysis for Thermoplastic Composites. Polymer, 14(9), 1877. https://doi.org/10.3390/polym14091877

Blatnický, M., Dižo, J., Molnár, D., & Suchánek, A. (2022). Comprehensive Analysis of Tricycle Structures with Steering Systems for Improved Driving Properties When Turning. Materials, 15(24), 8974. https://doi.org/10.3390/ma15248974

Blatnický, M., Sága, M., Dižo, J., & Bruna, M. (2020). Application of EN AW 6063 Light Metal Alloy in Vehicle Frame Construction with Innovative Steering Mechanism. Materials, 13(4), 817. https://doi.org/10.3390/ma13040817

Cochrane, C., Muneer, T., & Fraser, B. (2019). Design of an Electric-Powered Rickshaw, for Use in India. Energies, 12(17), 3346. https://doi.org/10.3390/en12173346

Ferroudji, F., Khelifi, C., Meguellati, F., & Koussa, K. (2017). Design and Static Structural Analysis of a 2.5 kW Darrieus-Savonius Combined Wind Turbine. International Journal of Engineering Research in Africa, 30, 94–99. https://doi.org/10.4028/www.scientific.net/JERA.30.94

Güler, S., & Karagülle, H. (2016). Finite Element Analysis of Structures with Extruded Aluminum Profiles Having Complex Cross-Sections. Latin American Journal of Solids and Structures, 13(8), 1499–1514. https://doi.org/10.1590/1679-78252755

Hao, J., Song, Y., Liu, H., Zhang, P., Chen, L., Zhang, N., Jia, S., & Liu, Y. (2023). Optimal Design Model for a New Type of Scraper and Research on Its Material Properties. Lubricants, 11(4), 171. https://doi.org/10.3390/lubricants11040171

Hussin, MSF, Jumaidin, R., Mohd Ali, N., Fauzi, MA, Saat, S., Sadun, AS, Mohd Yassim, H., & Ahmad, EZ (2020). Conceptual Design and Structural Analysis of the Savonius Vertical Axis Wind Turbine at Low Wind Speeds. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, 71(2), 178–187. https://doi.org/10.37934/arfmts.71.2.178187

Institute of Solid Mechanics of the Romanian Academy, Dragne, C., Radu Frenţ, C., Institute of Solid Mechanics of the Romanian Academy, Iliescu, M., & Institute of Solid Mechanics of the Romanian Academy. (2022). MECHANICAL ENGINEERING OF ROBOTIC SYSTEMS WITH SOLIDWORKS. International Journal of Modern Manufacturing Technology, 14(2), 61–68. https://doi.org/10.54684/ijmmt.2022.14.2.61

Intec Presisi Engineering Sdn. Bhd., Saturday, MA, Admi, AE, Universiti Tun Hussein Onn Malaysia, Mohamed Salleh, M., Universiti Tun Hussein Onn Malaysia, Osman, SA, Universiti Tun Hussein Onn Malaysia, Osman, SA, & Universiti Tun Hussein Onn Malaysia. (2024). Finite Element Analysis of Stress Distribution in AL6061 Frame Structure Under Various Applied Loads. International Journal of Integrated Engineering, 16(2). https://doi.org/10.30880/ijie.2024.16.02.027

Kulkarni, KS, & Sawant, SH (2024). Mechanical Analysis of Various Composites Used in Automotive Chassis Design. Journal of Mining, Metals and Fuels, 1337–1346. https://doi.org/10.18311/jmmf/2024/46210

Surat, A. (2020). ERGONOMIC STUDY BASED ON BIOMECHANICAL AND PHYSIOLOGICAL ASPECTS IN THE DESIGN OF MOTORIZED TRICYCLES (BENTOR) IN MAKASSAR CITY. Journal of Industrial Engineering Management, 5(1), 25–35. https://doi.org/10.33536/jiem.v5i1.549

Nabil, M., & Faizin, A. (2025). Analysis of the Effect of Dimensions and Thickness of Hollow ST 37 on the Strength of Electric Motorcycle Frames Using the Finite Element Method (FEM). Journal of Mechanical Engineering, 2(3), 11. https://doi.org/10.47134/jme.v2i3.4794

Nugroho, H., Sudarwo, F., & Artosa, OA (2024). Grassroots free-for-all neo-liberalism: Motorized pedicab (bentor) drivers as homo saker in downtown Yogyakarta, Indonesia. Journal of Urban Cultural Studies, 11(2), 241–260. https://doi.org/10.1386/jucs_00090_1

Oliveira, C., Albergaria De Mello Bandeira, R., Vasconcelos Goes, G., Schmitz Gonçalves, D., & D'Agosto, M. (2017). Sustainable Vehicle-Based Alternatives in Last-Mile Distribution of Urban Freight Transport: A Systematic Literature Review. Sustainability, 9(8), 1324. https://doi.org/10.3390/su9081324

Partheniadis, I., Terzi, V., & Nikolakakis, I. (2022). Finite Element Analysis and Modeling in Pharmaceutical Tablet Manufacturing. Pharmaceutics, 14(3), 673. https://doi.org/10.3390/pharmaceutics14030673

Poutanen, T., Länsivaara, T., Pursiainen, S., Mäkinen, J., & Asp, O. (2020). Eurocodes Safety Factor Calculation. Applied Science, 11(1), 208. https://doi.org/10.3390/app11010208

Prabandono, B., Putra, RBSP, Pradana, AC, Kurniawan, A., Nugraha, A., & Yusuf, M. (2024). Strength Analysis in Shelf Design Development on Base Cabinet with Finite Element Analysis using Solidworks CAE Software. Quantum Teknika: Journal of Applied Mechanical Engineering, 6(1), 12–17. https://doi.org/10.18196/jqt.v6i1.19979

Shkut, A. (2023). Methodological approach to assessing the durability of welded screen structures using SolidWorks Simulation software. Naukovyi Visnyk Natsionalnoho Hirnychoho Universytetu, 6, 79–85. https://doi.org/10.33271/nvngu/2023-6/079

University of Civil Engineering of Bucharest, Calea Plevnei 59, Bucharest, Romania, & Cristian Ştefan, D. (2024). SolidWorks Simulation Approach for Consolidation of Concrete Pillars with CFRP Wrapping. Romanian Journal of Physics, 69(7–8), 908–908. https://doi.org/10.59277/RomJPhys.2024.69.908

Trzepieciński, T., & Najm, S.M. (2024). Current Trends in Metal Materials for Body Panels and Structural Components Used in the Automotive Industry. Materials, 17(3), 590. https://doi.org/10.3390/ma17030590

Wu, Z., Pan, W., Niu, R., & Liu, L. (2025). Design and improvement of a small electric vehicle frame. Mechanical Science, 16(1), 263–272. https://doi.org/10.5194/ms-16-263-2025