Research and Innovation in Technical and Vocational Education and Training Vol. 3 No. 1 (2023) 080-086
© Universiti Tun Hussein Onn Malaysia Publisher's Office

RITVET

Homepage: http://publisher.uthm.edu.my/periodicals/index.php/ritvet
e-ISSN : 2785-8138

Design and Simulate Robotic Arm Structures from Dissimilar Materials Between T6061 Aluminum and Cast-Iron Alloy

Muhammad Nor Aiman Abd Razak [1], Muhammad Mohamed Salleh[1]*, Mohd Hafiz Ghazali [1], Reyanhealme Rohanai [1], Nurhayati Baharudin [1]

[1]Faculty of Technical and Vocational Education,
University Tun Hussein Onn Malaysia, Batu Pahat, 86400, MALAYSIA

*Corresponding Author Designation

DOI: https://doi.org/10.30880/ritvet.2023.03.01.011
Received 16 May 2023; Accepted 14 June 2023; Available online 30 June 2023

Abstract: Robotic arms offer exceptional precision and versatility, making them indispensable for high-precision work, seamless welding applications, and multi-component assembly, thereby revolutionizing industries. To fully leverage the power of automation and robotics in heavy industries, we specialize in customizing the structural components of our robotic arms and performing comprehensive analyses. This study involves designing and analyzing a robotic arm that mimics the structure of a human arm. The robotic arm is subjected to various loading conditions, and the Finite Element Analysis (FEA) simulation is employed for analysis. The design of the robotic arm follows a minimalistic approach. Additionally, this paper describes the development of a Computer-Aided Design (CAD) model and analyzes static analysis for the robotic arm using CAD software. The study shows that the nature of the material is essential in reducing the weight of robotic arms. Although cast iron alloys have stronger characteristics than aluminum alloys, a 63% reduction in the weight of the robotic arm was achieved. Static analysis using the finite element method (FEM) provides a clear view of critical parts of robotic arm structures and contributes to material, time, and cost savings.

Keywords: Robotic Arm, Aluminum Alloy, Cast Iron, Design, Static Analysis

1. Introduction

The purpose of this study was to utilize Solidworks software to design and analysis of a robotic arm that is lightweight, ergonomic, and efficient. The study aimed to reduce the mass rate of the arm by changing the material from cast iron to aluminum alloy (specifically, alloy 6061) and to compare the two materials. The resulting lightweight arm would provide an advantage in tasks such as periodic

---

**Corresponding author: muhammads@uthm.edu.my* maintenance, cleaning, and mobility. The study also sought to increase the movement efficiency of the arm by minimizing mass inertia, thereby maximizing its functionality. This research contributes to the field of robotics by showcasing the impact of material selection and design on the performance of robotic arms.

Aluminum alloys have become more popular as structural materials in recent years due to their advantageous qualities such as a high strength-to-weight ratio, ease of fabrication, significant ductility, excellent thermal conductivity, high corrosion resistance, and attractive natural finish. As a result, the building industry currently consumes 25% of the world's aluminum production Georgantzia et al. (2021). Aluminum is utilized in various maritime applications, such as the hull and deck of fast boats, especially catamarans, as well as in the superstructure of ships. It is also used in box-girder bridges, walls and floors of offshore modules, and shipping containers. The use of aluminum in the maritime industry is driven by the need to reduce structural weight, increase cargo capacity, improve speed, and reduce fuel consumption Hosseinabadi et, al., (2021).

The researchers' objective was to achieve a compact and streamlined design for the robotic arm's servomotors at each joint, which would minimize their size while supporting the load of the robotic arm structure. Furthermore, they carried out static analysis through the application of the finite element technique (FEM) using Solidworks 2021 software. Purandhara Sai Santosh et al. (2021) conducted a study investigating the design and analysis of a robotic arm under different loading conditions using FEA simulation. The study utilized a simple approach in the design of the robotic arm, with the CAD model created using Creo Parametric software and exported to Ansys. The results showed that material properties have a significant impact on the performance of the robotic arm under various loading conditions. The study employed design optimization using Ansys Workbench to evaluate the effect of changing materials, with the Finite Element Method (FEM) providing a clear picture of design, optimization, and failure analysis. The findings of this study highlight the importance of material selection and FEM in the design and optimization of robotic arms.

Kiran Bollineni et al. (2020) presented the design of a rover and a robotic arm with six degrees of freedom. Aluminum 6061 alloy was used for the rover and robotic arm design due to its optimal strength-to-weight ratio, machinability, and availability. The study used Ansys Workbench to perform stress and deformation analysis, determining the highest and minimum stresses to be 1.5 MPa and 5.3x10-13 MPa, respectively, with a total deformation fluctuating between 0.00168 mm and 1.28x10-5 mm. The chosen aluminum 6061 alloy was able to handle large stresses for a prolonged period (high fatigue strength). The factor of safety was calculated to be 9 for a payload of 10 kg.

Bugday and Karali (2019) conducted a study on the optimization of robotic arm design with the aim of reducing redundant weight, thereby making the robotic arm lighter. The study analyzed five major robotic arms from various brands, examining the payload distribution of the arm in terms of both region and quantity and compared various designs. To reduce unnecessary weight while maintaining positional precision, the researchers made changes to geometry and materials in alternative designs. The primary objective of this research was to utilize Solidworks software to determine the optimal locations for material reduction in the robot arm. In addition, the study carried out hole-drilling experiments and evaluated them to determine the ideal geometric shape for reducing material. The primary purpose of the optimization effort was to minimize motor loads by reducing the arm's mass and obtaining lower deformation and stress values compared to those estimated in static analysis. The results demonstrated that the robot arm was lighter under the same stress and that no further deformation occurred.

2. Methodology

The design and simulation of a robotic arm were conducted using Solidworks, wherein the behavior of the arm under static conditions, specifically its self-weight, was analyzed and studied through the utilization of Finite Element Analysis. Finite Element Analysis is a numerical method that proves useful in solving computational mechanics problems. By performing such an analysis, potential flaws were identified and addressed prior to the construction of the robot, minimizing the implementation of faulty ideas. The maximum load that the robotic arm could lift at both the bottom and top was applied, ensuring an exhaustive evaluation of the design Pupăză et, al (2014) and Sahu, S., & Choudhury, B. B. (2017).

2.1 Designing

Each part is designed to be light and strong. This not only helps in performance but also helps increase the lifespan of the robot arm. Another important factor to consider is the factor of safety (FOS). Even if it is just a design, it needs to be emphasized so that the injury or resilience of this built design is stronger and has its own limitations in terms of what it can achieve. For this reason, the critical parts of the upper arm and lower arm are designed considering the aspect of the safety factor. The study provides two different designs which will be shown in this section. Individual parts and their assemblies are shown starting with the first model. Individual parts and their assemblies are shown in Figure 1, Figure 2, Figure 3, Figure 4, Figure 5, Figure 6, and Figure 7.

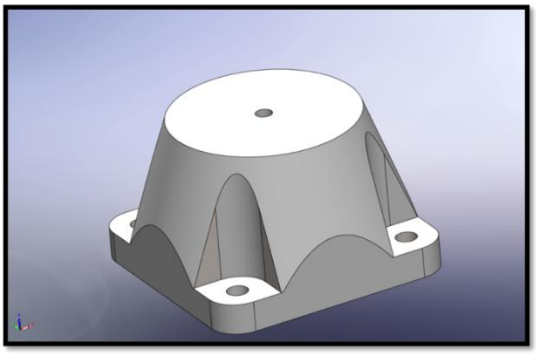


Figure 1: Base

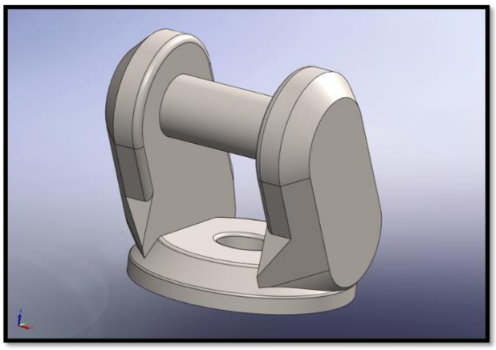


Figure 2: Waist

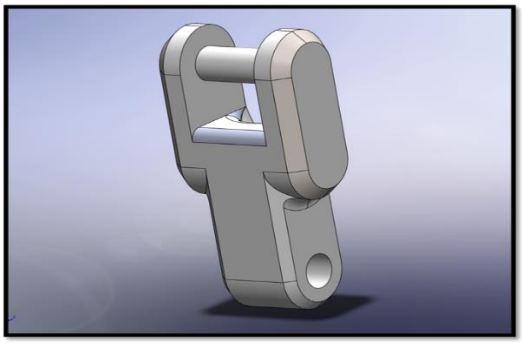


Figure 3: Shoulder

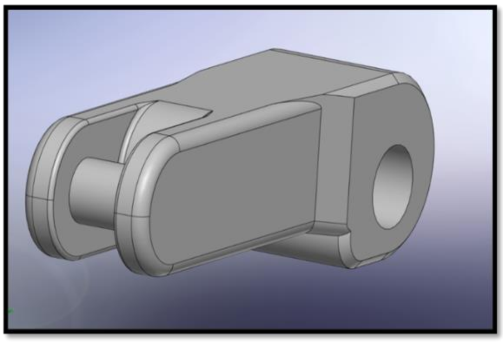
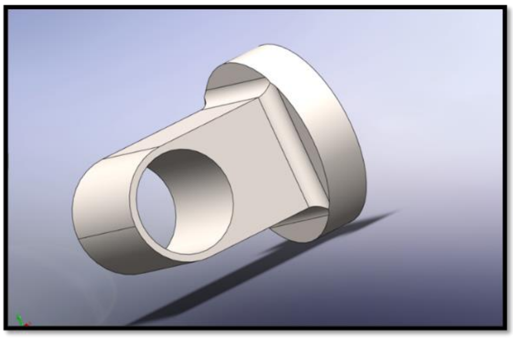


Figure 4: Forearm



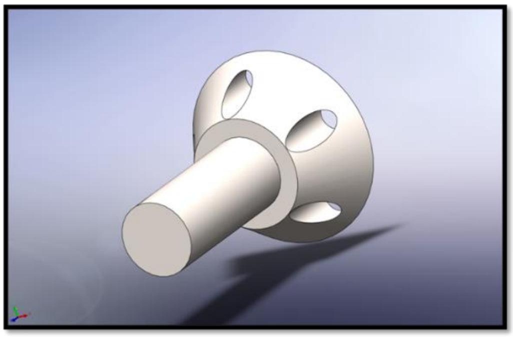


Figure 5: Wrist **Figure 6: End of effector**

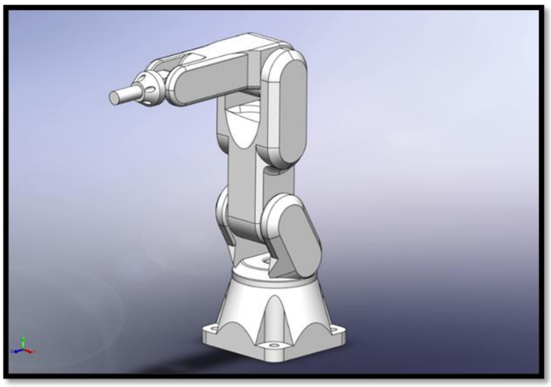


Figure 7: Robot Arm Assembly

2.2 Analysis

The lower and upper arms of the robotic arm were subjected to analysis using Solidwork software. To evaluate the behavior of the robotic arm under static conditions, including its self-weight, the study utilized finite element analysis, which is a commonly used numerical method for solving computational mechanics problems. The robotic arm was made up of two materials in this study, namely, aluminum alloy (6061 alloy) and cast iron (gray cast iron). The point of failure in the robotic arm could be identified easily by analyzing the von Mises stress, strain, and displacement.

3. Result and Discussion

In this robotic arm design, the lower arm (shoulder) using cast iron as a structure can support a maximum load of 7,000,000N, while the upper arm can accommodate a maximum load of 2,500,000N, with a factor of safety (FOS) value of 2 or greater. The analysis of aluminum alloys will yield similar load results. The objective is to evaluate the variation in the capacity of the two materials, cast iron and aluminum alloys, to support the forces in the robotic arm structures.

The findings for the lower arm (Fig. 8 and 9) indicate that the maximum von Mises stress value for cast iron is 191086064 N/m2, while for aluminum alloy, it is 189829200 N/m2, indicating a marginal decrease of 0.66%. Additionally, the maximum displacement value for cast iron is 0.117477 mm, and for aluminum alloy, it is 0.147368 mm, showing a percentage increase of 25.4%. The strain value was determined to be 0.002346 for cast iron and 0.002359 for aluminum alloy. The variation in strain values for both materials was insignificant. The factor of safety (FOS) value for cast iron was 2.2, whereas, for aluminum alloy, it was 0.29, indicating a substantial decline of 86.8%. To ensure that the safety factor is at a value of 2 or greater, aluminum alloy could not support the load of 7,000,000N used for the lower arm of the initial model.

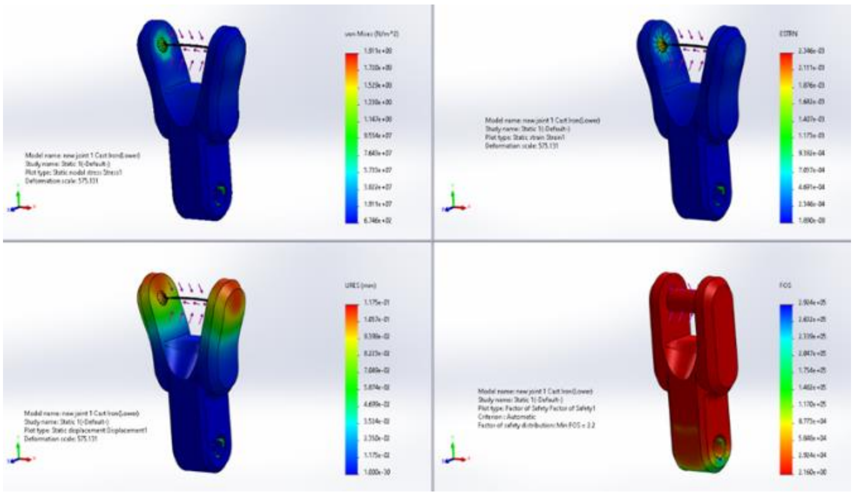


Figure 8: Result Lower Arm for Cast Iron

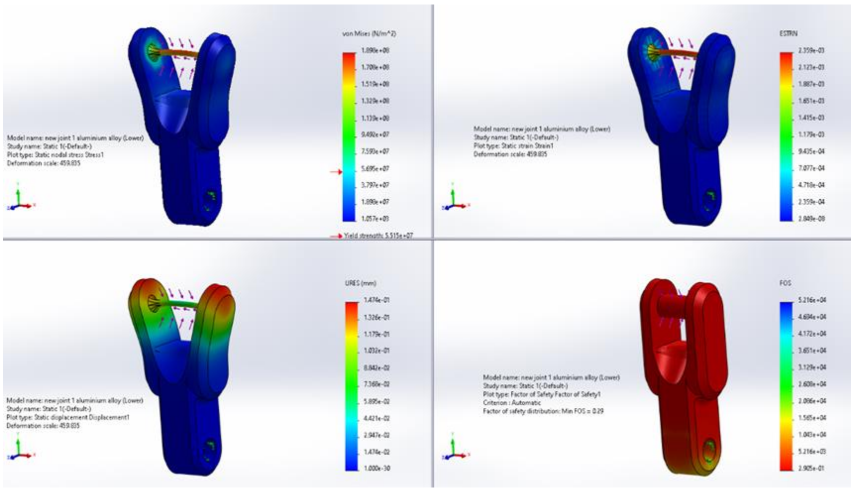


Figure 9: Result Lower Arm for Aluminum Alloy

Subsequently, the upper arm results were examined (refer to Fig. 10 and 11). The maximum von Mises stress value for cast iron was 205450576 N/m2, while for aluminum alloy, it was 203351216 N/m2, indicating a decline of only 1.02%. Moreover, the maximum displacement value for cast iron was 0.065467 mm, and for aluminum alloy, it was 0.066521 mm, reflecting a percentage increase of 1.6%. The strain values were found to be 0.002577 for cast iron and 0.002555 for aluminum alloy.

The variation in strain values for both materials was minimal and not significant. The subsequent factor of safety (FOS) values was determined, and cast iron had a FOS value of 2.1, while aluminum alloy had a value of 0.27, indicating a significant reduction of 87.1%. Consequently, the load of 2,500,000N used for the upper arm of the initial model could not be accommodated by the aluminum alloy to ensure the safety factor remained at a value of 2 or higher.

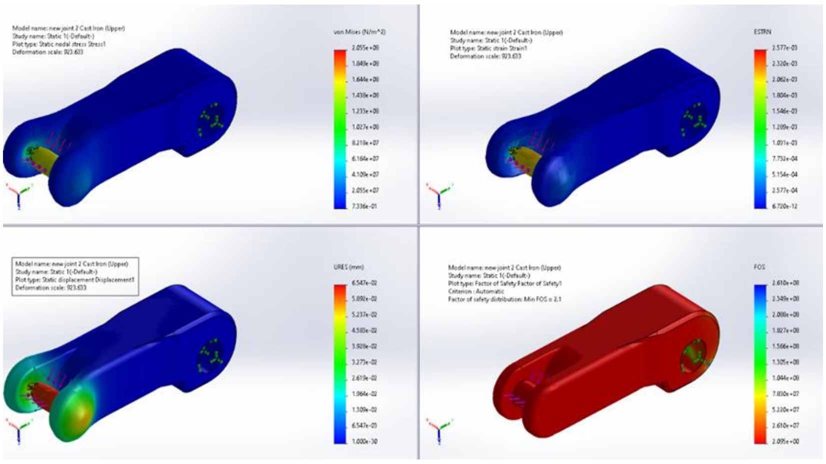


Figure 10: Result Upper Arm for Cast Iron

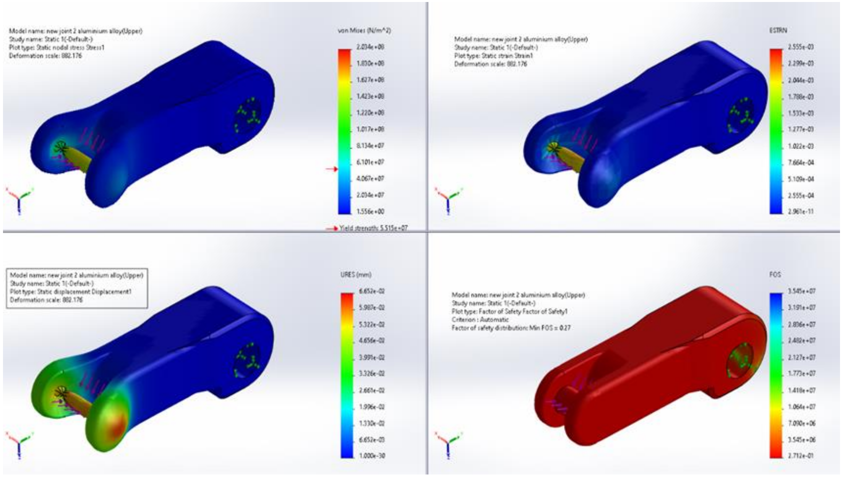


Figure 11: Result Upper Arm for Aluminum Alloy

3.1 Discussions

The researcher used Solidworks software to choose aluminum alloy as the structural material for the robotic arm. In order to determine if using aluminum alloy could decrease the weight of the robotic arm structure, the researcher compared it to a commonly used material in the market, gray cast iron. The study revealed that using aluminum alloy resulted in a weight reduction of 63% compared to cast iron, which is more than half of the original weight. According to several scholars, aluminum alloy has the potential to reduce the weight of structures, lower material costs, and save processing time Gutiérrez, et al (2017), Ghosh, et, al (2018) and Kruthika, K., Kumar, and Lakshminarayanan, (2016). Additionally, the static analysis results indicated that the robotic arm structure made with aluminum alloy material could withstand loads up to 102 tons with a safety factor of 2. According to Akin, (2010) some designers refer to it as the factor of ignorance. Remember that a FoS of unity means that failure is imminent; it does not mean that a part or assembly is safe. Many authors argue that the factor of safety should be calculated as the product of terms greater than one but less than eight, depending on materials and structure.

In heavy industries, the load capacity of robotic arms is generally limited, and it is considered large if a load of 100 tons can be successfully accommodated. This is evident from the fact that even the FANUC M-2000IA model robotic arm, which has a maximum load capacity of only 2.3 tons, as stated on Fanuc's official website, is not used for lifting such heavy loads. Therefore, it can be concluded that using aluminum alloy (6061) as the structural material for robotic arms can significantly reduce their weight and enhance their load capacity.

4. Conclusion

In conclusion, the study successfully utilized the finite element method (FEM) static analysis to design and analyze the robotic arm, and Solidworks software was found to be a cost-effective and efficient tool with a range of features that make it a preferred choice among engineers for preliminary design work. The study effectively achieved its objectives and satisfactorily answered the research questions. Future research is expected to focus on the development of a robotic arm that utilizes an aluminum structure.

Acknowledgement

The authors would also like to thank the Faculty of Technical and Vocational Education, Universiti Tun Hussein Onn Malaysia for its support.

References

Georgantzia, E., Gkantou, M., & Kamaris, G. S. (2021). Aluminium alloys as structural material: A review of research. *Engineering Structures, 227*, 111372.

Hosseinabadi, O. F., & Khedmati, M. R. (2021). A review on ultimate strength of aluminium structural elements and systems for marine applications. *Ocean Engineering*, *232*, 109153.

Santosh, L. P. S., Mishra, N., Mahanta, S. S. A., Dharmarajan, V., Varma, S. K., & Shoor, S. (2022). Design and analysis of a robotic arm under different loading conditions using FEA simulation. *Materials Today: Proceedings, 50*, 759-765.

Kiran Bollineni, R., Menon, S. S., & Udupa, G. (2020). Design of Rover and Robotic Arm. *Materials Today: Proceedings*, *24*, 1340-1347.

Bugday, M., & Karali, M. (2019). Design optimization of industrial robot arm to minimize redundant weight. *Engineering Science and Technology, an International Journal*, *22*(1), 346-352.

Pupăză, C., Constantin, G., & NEGRILĂ, Ș. (2014). Computer aided engineering of industrial robots. *Proceedings in Manufacturing Systems*, *9*(2), 87-92.

Sahu, S., & Choudhury, B. B. (2017). Static analysis of a 6-axis industrial robot using finite element analysis. *International Journal of Mechanical Engineering and Technology*, *8*(3), 49-55.

Akin, J. E. (2010). *Finite element analysis concepts: via SolidWorks*. World Scientific.

Gutiérrez, S. C., Zotovic, R., Navarro, M. D., & Meseguer, M. D. (2017). Design and manufacturing of a prototype of a lightweight robot arm. *Procedia Manufacturing*, *13*, 283-290.

Ghosh, T., Roy, A., Mishra, R., & Kamlesh, S. S. (2018). Structural optimization of a CT guided robotic arm based on static analysis. *Materials Today: Proceedings*, *5*(9), 19586-19593.

Kruthika, K., Kumar, B. K., & Lakshminarayanan, S. (2016, October). Design and development of a robotic arm. In *2016 International Conference on Circuits, Controls, Communications and Computing (I4C)* (pp. 1-4). IEEE.