BIS ENERGY AND ENGINEERING

Volume 2 2025 | DOI: 10.31603/biseeng.212
6th Borobudur International Symposium on Science and Technology (BIS-STE) 2024

UNIMMA PRESS
UNIVERSITAS MUHAMMADIYAH MAGELANG

Design and development of an innovative universal rotor holder automotive tool for operational efficiency of partner

Akbar Ela Heka[1], Muhammad Firdaus Jauhari[2, 3*]

[1] Department of Mechanical Engineering, Politeknik Negeri Banjarmasin, Banjarmasin, Indonesia
[2] Department of Automotive Engineering Technology, Politeknik Negeri Banjarmasin, Banjarmasin, Indonesia
[3] Center of Research and Community Services, Politeknik Negeri Banjarmasin, Banjarmasin, Indonesia
[*] Corresponding author email: firdaus.jauhari@poliban.ac.id

Abstract

This study presents the design and analysis of a universal rotor holder intended for automotive applications, focusing on its structural integrity under static loading conditions. The rotor holder was subjected to a static load at both fixed and sliding retainer, with a fixed geometry support for stability. Using SolidWorks, we conducted simulations to evaluate the Von Mises stress, displacement, and strain of the holder. The analysis revealed a maximum Von Mises stress of 16.390 MPa, which is well below the alloy steel's yield strength of 620.422 MPa, verifying the design's safety during operation. Furthermore, the measured displacement of 0.01214 mm and zero strain highlighted the holder's capability to preserve structural integrity without significant deformation. These results indicate that the universal rotor holder is both durable and dependable, making it ideal for accommodating various rotor sizes in automotive maintenance tasks. The implications of this research highlight the importance of using high-quality materials and precise engineering in tool design, ultimately enhancing safety and efficiency in automotive repair practices. This study contributes to the field by providing a validated design framework for future tool development in the automotive industry.

Published:
May 31, 2025

Selection and Peer-review under the responsibility of the 6th BIS-STE 2024 Committee

Keywords

Universal rotor holder, Structural analysis, Automotive tool development

Introduction

Currently, the automotive industry is facing increasingly complex challenges in vehicle maintenance and repair. One of the critical components in this process is the rotor holder, which serves as a support tool to maintain the position of the rotor during servicing. The rotor, being a rotating part of the vehicle, is subject to wear and damage over time. Therefore, the use of appropriate tools to secure the rotor is essential to ensure efficiency and safety in mechanical work [1].

This research aims to develop a design for a universal rotor holder, intended for use in the maintenance and repair of both cars and motorcycles. With the growing variety of vehicle types and brands, the demand for tools that can adapt to various dimensions and shapes of components has become increasingly urgent. Currently, many rotor holders available in the market have specific functions limited to certain brands, thus failing to meet the needs of workshops that service a wide range of vehicles.

Through collaboration with a small and medium-sized enterprise (SME) engaged in vehicle maintenance services, this study focuses on the design and development of a universal rotor holder. This tool is expected to meet the mechanical needs for maintaining the stability and safety of components during the maintenance process. With an innovative design, this tool will not only enhance work efficiency but also reduce the risk of damage to sensitive vehicle components. The design process utilizes SolidWorks software, allowing for the creation of complex 3D models and conducting simulations and analyses [2]. The design stages include user needs analysis, technical specifications, and conceptual design development. Once the design is approved, the next step is the fabrication of the tool, selecting appropriate materials to ensure reliability and optimal performance.

Similar tools have been designed, including a device to hold the rotor movement in motorcycle engines, which features adjustable arm openings and a simple design. This invention provides a practical alternative and improves time efficiency in machine maintenance [3]. Additionally, there are tools for valve spring removal on cylinder heads for larger engines, simplified in design and mechanism. This tool is designed for ease of use and assembly, particularly for tugboat engines [4]. Several references indicate that the design of various special tools can yield different functions and operational methods. One paper discusses a special tool for removing and installing rod cylinders on HD Dump Truck units, designed to lift and hold the rod cylinder to prevent elongation, thereby speeding up the process and reducing downtime, while also saving on labor costs and lead time [5]. Furthermore, the author designed a tool for removing caliper pistons during car brake maintenance using pneumatic power at 300 psi, which accelerated the process time to 7.6 seconds [6]. A valve spring compressor design for cylinder head assembly is also found in a patent, featuring a C-shaped tool that can be adjusted to define two opposing jaws [7].

The results of this research are expected to be implemented as an innovative support tool in vehicle maintenance and repair work, contributing to the automotive industry and enhancing the quality of vehicle maintenance services in an increasingly competitive market. The increasing demand for versatile, safe, and efficient tools in the automotive industry highlights the necessity of innovative solutions to support diverse vehicle maintenance needs. This study aims to design and analyze a universal rotor holder with a primary focus on its structural integrity under static loading conditions. Using SolidWorks simulations, the analysis revealed that the maximum Von Mises stress was significantly below the material's yield strength, with no evidence of significant deformation. These findings confirm the holder's robustness, reliability, and adaptability to various rotor sizes. This research contributes to the advancement of tool design in the automotive sector by offering a validated framework, ultimately enhancing safety and efficiency in vehicle maintenance practices.

Method

The methodology applied in this research involves a systematic and detailed series of steps. First, an in-depth survey was conducted regarding vehicle maintenance practices related to rotor components. Subsequently, a literature review was performed to understand the existing designs and functions of rotor holders, which served as a foundation for designing a universal rotor holder in the context of vehicle maintenance. The next stage involved designing a prototype of the tool using SolidWorks for three-dimensional modeling, ensuring the accuracy and effectiveness of the design. Testing of the universal rotor holder design was carried out in SolidWorks by applying static loads to the retaining elements (fixed retainer and sliding retainer). The selection of materials for this specialized tool was aligned with the static analysis applied to the designed structure, resulting in values that are safe for the users of the tool. A flowchart for the design of the universal rotor holder is presented in Figure 1.

The first stage of the design and analysis in this research involves creating a model design for the universal rotor holder using the selected material to achieve a good safety factor, specifically Alloy Steel (Figure 2). Alloy steel exhibits a range of remarkable material properties, making it suitable for various engineering applications. With an elastic modulus of 2,141,391 N/mm^2 and a Poisson's ratio of 0.28, it demonstrates high stiffness and predictable deformation behavior under load. Its shear modulus of 805,570.9 N/mm^2 further indicates its resistance to shear forces. The material's mass density of 0.0077 kg/m^3 ensures an optimal balance between strength and weight, while its tensile strength of 7,380.92 N/mm^2 and yield strength of 6,326.51 N/mm^2 highlight its exceptional resistance to deformation and failure under tensile stress. These properties underscore the robustness and versatility of alloy steel in various engineering and industrial applications. Once the design is completed, a load analysis is conducted by establishing relevant parameters, including the application of loads and constraints. With the specified load conditions, meshing is performed for the prototype. Subsequently, simulations are carried out to evaluate the performance of the prototype. The simulation results are then analyzed for key metrics such as Von Mises stress, displacement, strain, and factor of safety (FOS), ensuring that the outcomes meet the established criteria. This structured approach ensures a comprehensive evaluation of the design and functionality of the model, thereby supporting informed decision-making in its manufacturing and development processes.

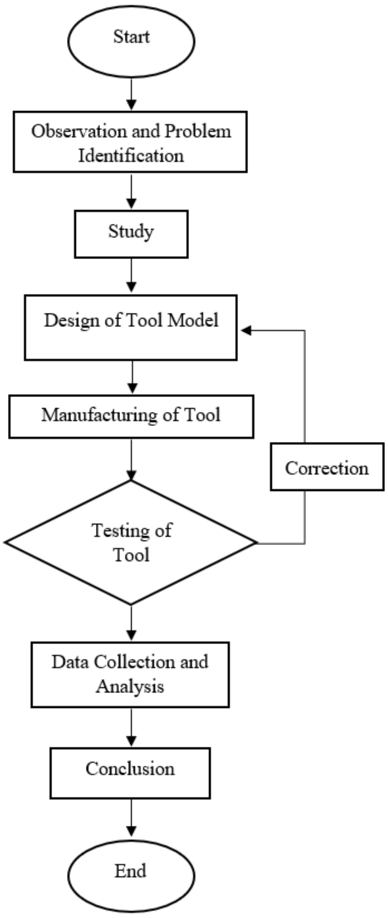

Figure 1. Flowchart of the universal rotor holder design

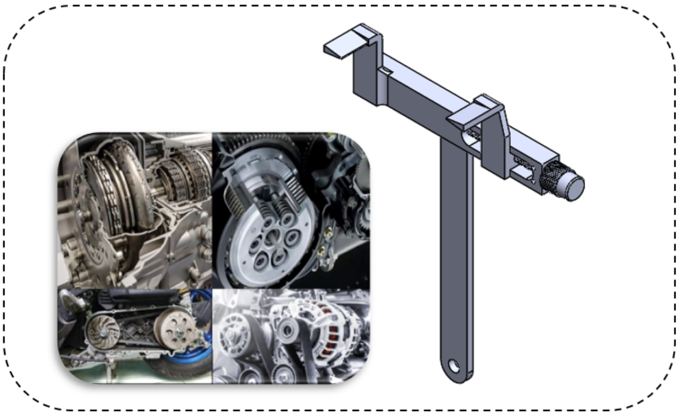

Figure 2. Rotor components and the universal rotor holder model

a. The Von Mises Stress method is a technique used to assess the stress experienced by a material when subjected to a load. In load analysis using SolidWorks, this method serves to predict the location and magnitude of stress in specific parts of the model or structural design [8]. The fundamental principle of the Von Mises method is that a material will fail if its equivalent stress exceeds the yield strength of that material. In this context, the calculated equivalent stress is used to compare the stresses occurring at various points within the material.

b. Strain in load analysis using SolidWorks refers to the relative change in size or shape of an object due to the applied load. This parameter is crucial for evaluating the material's response to the forces or pressures exerted. Strain analysis includes calculating the changes in dimensions or deformation in the parts of the model that are subjected to loads, allowing users to understand the extent of deformation occurring in different sections of the structure [9].

c. Displacement in load analysis using SolidWorks refers to the change in position or shift of a point or element in the model as a result of the applied load or force. When analyzing a model or structure under load, this software provides information regarding the displacement of various parts of the model from their original positions due to the load. Displacement can also be represented as a vector indicating the direction and magnitude of the movement of points within the model.

d. The Factor of Safety (FoS) is the ratio between the maximum load capacity that a structure or component can withstand and the actual load applied to that structure [10]. In load analysis using SolidWorks, the FoS serves to evaluate how safe or strong a design is against the applied loads. A higher FoS value indicates a greater level of safety offered by the design.

Load analysis is crucial in engineering and design as it ensures the safety, performance, and efficiency of the materials used in a structure or component. Through this analysis, it can be confirmed that the design can withstand loads without failure, thereby protecting user safety. Additionally, understanding the effects of loads allows for the optimization of performance and efficient material usage, reducing waste and production costs. Load analysis also aids in planning and developing better designs while meeting industry safety and performance standards. The results of the analysis provide insights for design improvements, making load analysis a critical step in ensuring the optimal performance of the models we develop.

Results and Discussion

The Finite Element Method (FEM) is an essential technique in structural modeling and analysis used in software such as SolidWorks. FEM allows for the representation of physical objects as a collection of small elements, facilitating detailed analysis of structural responses. In SolidWorks, users can assign material properties, boundary conditions, and loads, and apply FEM to calculate the structural response. The application of FEM enables simulations that closely approximate real-world conditions, predicting deformation and stress, and evaluating design performance before mass production, thereby reducing the risk of errors and production costs [11], [12].

In the design of the universal rotor holder, the researcher utilized SolidWorks 2020 to design and perform static load analysis. The material chosen for this application is Alloy Steel, based on discussions with technicians from Denta Motor and the results of the simulations conducted. Alloy steel possesses highly attractive and beneficial mechanical properties, making it a primary choice in various industrial applications. These properties are influenced by different alloy compositions, which include elements such as chromium, nickel, molybdenum, and vanadium. Alloy steel exhibits high tensile strength, allowing the material to withstand heavy loads without experiencing permanent deformation. Additionally, alloy steel demonstrates good resistance to wear and corrosion, thanks to the alloying elements that enhance the material's durability in harsh environments. Its high toughness is also a significant characteristic, enabling alloy steel to absorb energy without fracturing, making it ideal for applications requiring impact resistance. Furthermore, alloy steel can be modified through heat treatment processes to improve its mechanical properties, such as strength and hardness, making it highly versatile for various engineering needs. With this combination of properties, alloy steel is an excellent choice for structural components, tools, and machinery that require high performance and durability.

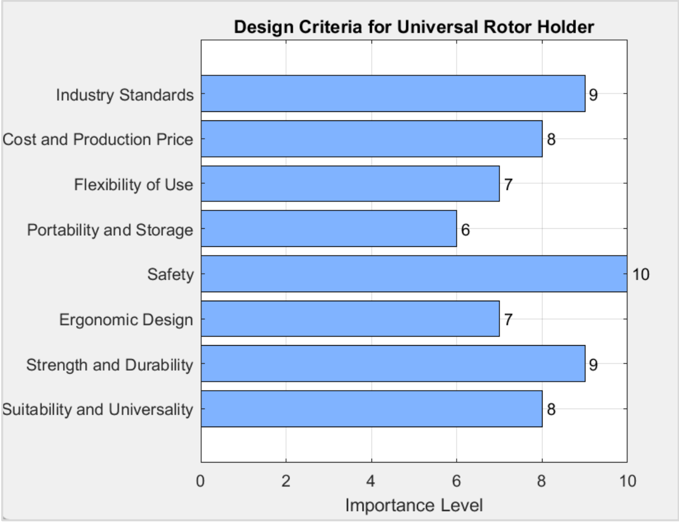

Figure 3. Design criteria

In designing the universal rotor holder, a 3D model was developed using SolidWorks, with an emphasis on critical criteria related to functionality and safety as outlined in the criteria shown in Figure 3. The holder must be adjustable to accommodate various rotor sizes for motorcycles and cars, ensuring compatibility with different brands and models of vehicles. Strength and durability are prioritized, utilizing high-quality materials such as steel for a robust construction that can withstand stress during the installation and removal of the rotor. An ergonomic design is also essential, featuring a comfortable grip to reduce hand fatigue and an intuitive layout for ease of use. Safety is addressed with an effective locking mechanism to prevent the rotor from rotating during maintenance and smooth edges to avoid user injuries. Additionally, portability and storage considerations are taken into account, with a compact size for easy transportation and efficient packaging for secure access. The tool should also be versatile, capable of being used on other vehicle components and equipped with additional accessories for various applications. Finally, cost-effectiveness is considered to ensure a competitive price without compromising quality, alongside a balanced selection of materials that weigh cost against performance. All these criteria aim to meet industry standards related to safety and performance, including durability testing for repeated use.

In this static load analysis, the universal rotor holder is subjected to a static load of 25 kg, which translates to a force of 250 N applied at both of its supports: the fixed retainer and the sliding retainer. The design incorporates a fixed geometry support on the side of the holder for the fixed retainer, while a roller support is utilized for the sliding support. This configuration allows for effective load distribution and stability during operation. Additionally, the remaining portion of the holder is assumed to be rigid and immovable, as it is secured in place by the force exerted by the user's hand, ensuring that the holder remains stable under load.

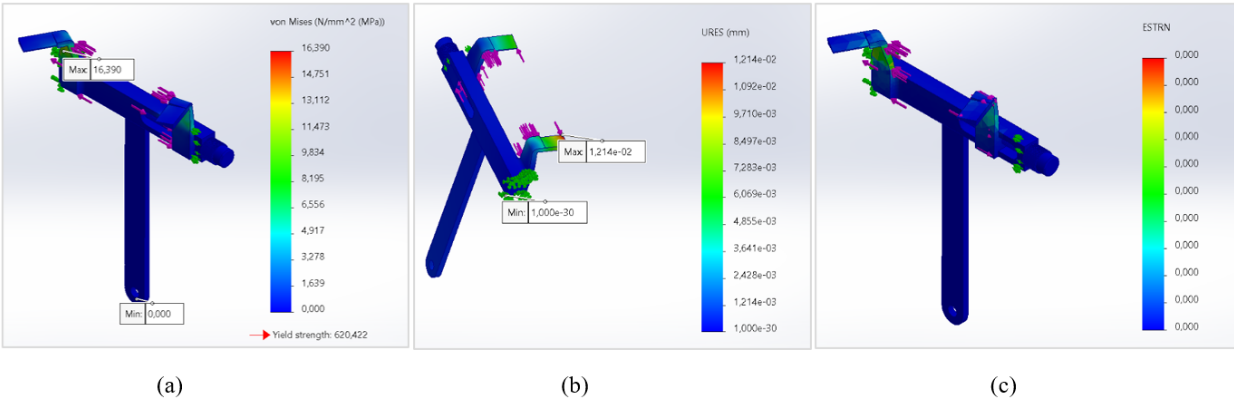

Figure 4. Load simulation results

Von Mises Stress Analysis (Figure 4. a)

The material properties for the alloy steel used in the construction of the rotor holder indicate a yield strength of 620.422 MPa. During the static load simulation, which was conducted with the specified forces and support conditions, the analysis revealed a maximum Von Mises stress of 16.390 MPa occurring at the upper section of the holder component. This stress value is significantly lower than the yield strength of the material, confirming that the rotor holder operates within a safe limit. The results indicate that the design can withstand the applied load without experiencing permanent deformation, thus ensuring the reliability and safety of the tool during its intended use. This is particularly relevant in automotive applications, where safety and performance are critical, as the rotor holder must reliably support various rotor sizes without failure.

Displacement Analysis (Figure 4.b)

The tensile strength of the alloy steel is measured at 738.1 MPa, representing the maximum stress that the material can endure when subjected to tension before failure occurs. The static load simulation yielded a displacement of 0.01214 mm at the upper end of the holder component. This minimal displacement suggests that the holder maintains its structural integrity under load, as it is well below the tensile strength threshold of the material. The negligible displacement further reinforces the conclusion that the rotor holder is designed to function safely and effectively, even under the specified static load conditions. This aspect is crucial for users who require precision and stability when working on vehicle maintenance, as excessive displacement could lead to misalignment and potential damage.

Strain Analysis (Figure 4.c)

The elastic modulus of the alloy steel is determined to be 214.1391 MPa, which quantifies the material's resistance to elastic deformation when subjected to an applied force. The elastic modulus is defined as the slope of the stress-strain curve within the elastic deformation region, indicating how much the material will deform elastically under stress. The static load simulation results in a strain measurement of 0 mm along the body of the tool. This outcome indicates that there is no significant strain experienced by the holder, which is well within the elastic limits of the material. Consequently, this finding supports the assertion that the rotor holder operates safely and effectively without compromising its structural integrity. The ability to maintain low strain levels is particularly relevant in high-stress environments, such as automotive workshops, where tools are frequently subjected to varying loads. Testing the tool process under real condition can be seen in Figure 5.

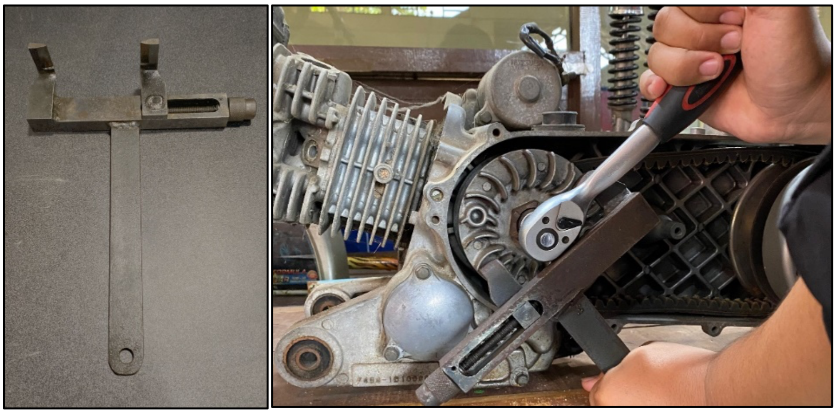

Figure 5. Testing the tool under real condition

In summary, the comprehensive analysis of the universal rotor holder under static loading conditions demonstrates that the design is robust and capable of withstanding the applied forces without exceeding the material limits. The results from the Von Mises stress, displacement, and strain analyses collectively affirm that the rotor holder is structurally sound, ensuring its safety and functionality during operation [13–15]. This thorough evaluation not only highlights the effectiveness of the design but also provides confidence in its performance in real-world applications. The relevance of this analysis extends to ensuring that the rotor holder meets industry standards for safety and performance, thereby enhancing user trust and satisfaction in automotive maintenance tasks. By confirming the tool's reliability, this analysis supports its adoption in both professional and DIY automotive settings, where safety and efficiency are paramount.

Conclusion

A comprehensive analysis of the universal rotor holder demonstrates that the design of this tool not only meets the criteria for functionality and safety but also exhibits excellent performance under static loading conditions. The simulation results obtained using SolidWorks indicate that the maximum Von Mises stress generated is well below the yield strength of the material, which is 620.422 MPa, with a measured maximum value of 16.390 MPa. Furthermore, the measured displacement of 0.01214 mm and the negligible strain indicate that the tool is capable of maintaining its structural integrity without experiencing excessive deformation. Thus, this rotor holder is designed to operate safely and effectively in various automotive applications, meeting industry standards related to safety and performance. This research also emphasizes the importance of using alloy steel, which possesses superior mechanical properties, such as resistance to wear and corrosion, making it an ideal choice for applications requiring high durability. With these promising results, the universal rotor holder is expected to be widely adopted in vehicle maintenance practices, both in professional workshops and for personal use, thereby enhancing efficiency and safety in automotive maintenance processes.

Acknowledgments

We would like to express our appreciation and gratitude for the assistance of all our colleagues in this research, as well as to P3M Politeknik Negeri Banjarmasin for their support in the publication of our article.

References

1. Crolla, D.A. Automotive engineering: powertrain, chasis system and vehicle body.
2. Mario H. Castro-Cedeño, P. *Introduction to SolidsWorks*; 2015;
3. Jauhari, M.F.; Saberani, S. Alat Bantu Penahan Rotor Untuk Mesin Sepeda Motor Yang Disempurnakan 2023.
4. Jauhari, M.F.; Admaja, N.Y.; Arawi, R.R. Alat Bantu Pelepas Pegas Katup Pada Kepala Silinder 2023.
5. Purwono, H. Perancangan Special Tool Remove and Install Hoist Cylinder Pada Unit Dump Truck HD 1500-7. **2018**, 1–10.
6. Sunu, P.W.; Gede, S.; Bangse, K. A Design of a Tool of Car Brake Piston Caliper Remover with Pneumatic Power. **2023**, 446–450, doi:10.5220/0010947300003260.
7. K Line Industries, I. Valve Spring Compressor - Patent CA 2312123. *Paten* 2001, *CA 2312123*, CA 2312123.
8. Eko Siswono, M.M. Static Analysis of Frame Structure of Post-Stroke Patients Tricycle Design with Material Type Variations. *R.E.M. (Rekayasa Energi Manufaktur)* **2019**, *4*.
9. Sandy Suryady, E.A.N. Simulasi Faktor Keamanan dan Pembebanan Statik Rangka Pada Turbin Angin Savonius. *Jurnal Ilmiah Multidisiplin* **2022**, *1*, 42–48.
10. Kahfi, M.W.; Mufarida, N.A.; Kosjoko, K. Desain dan analisis chassis tipe tubular space frame. *ARMATUR : Artikel Teknik Mesin & Manufaktur* **2023**, *4*, 18–30, doi:10.24127/armatur.v4i1.3266.
11. Akin, J.E. *Finite Element Analysis Concepts via SolidWorks*; World Scientific: Rice University, Houston, Texas, 2009;
12. Shih, R.H. *Introduction to Finite Element Analysis Using SolidWorks Simulation 2010*; Schroff Development Corporation, 2010;
13. Widiyanti; Puspitasari, P.; Suyetno, A. The development of instructional materials mechanics of materials using solidworks simulation software. *AIP Conference Proceedings* **2016**, *1778*, doi:10.1063/1.4965792.

14. Wu, D.; Zhang, Z.; Wang, Z. Application research of solidworks in modeling of straw carbonization preparation plant. *Journal of Physics: Conference Series* **2019**, *1303*, doi:10.1088/1742-6596/1303/1/012048.
15. Shinde, J.; Kharade, S.; Mulani, A.; Kokare, K.; Tapase, C.; Terani, A. Design and analysis of front axle using Solidworks simulation. *International Research Journal of Engineering and Technology* **2022**.