Strength Analysis of Light Vehicle Rim Design Using Finite Element Method: An Innovative Approach for Safety and Efficiency

Agus Dwi Putra[1,*], Diama Rizky Septiawan[1], Muhammad Arif Nur Huda[1], Dewi Izzatus Tsamroh[2], Bella Cornelia Tjiptady[3]

[1]Mechanical Engineering, State Polytechnic of Malang
Jl. Soekarno Hatta No. 9 Malang, 65141
[2] Faculty of Vocational Studies, Universitas Negeri Malang
Jl. Semarang No. 5 Malang, 65145
[3]Faculty of Saintek, Universitas Islam Raden Rahmat Malang
Jl. Mojosari No. 2 Kepanjen, Kab. Malang, 65163
Corresponding email: agus.dwi@polinema.ac.id

Abstrak

Velg mobil merupakan salah satu komponen vital yang berfungsi untuk menahan beban kendaraan dan memberikan kestabilan selama berkendara. Penelitian ini bertujuan untuk menganalisis kekuatan velg menggunakan metode elemen hingga (Finite Element Method/FEM) dengan perangkat lunak Solidworks 2020. Parameter yang dianalisis meliputi tegangan von mises, regangan ekuivalen, perpindahan (displacement), dan faktor keamanan (Factor of Safety). Hasil simulasi menunjukkan bahwa tegangan maksimum sebesar 1,978 × 10^{5} N/m² dan regangan maksimum sebesar 2,449 × 10^{-6} berada dalam batas aman material aluminium alloy. Perpindahan maksimum sebesar 5,687 × 10^{-4} mm menunjukkan kekakuan struktural velg yang memadai. Faktor keamanan minimum sebesar 1,087 × 10^{3} mengindikasikan toleransi tinggi terhadap kegagalan. Penelitian ini membuktikan bahwa desain velg aman, efisien, dan memenuhi standar keselamatan. Metode elemen hingga terbukti efektif dalam mengidentifikasi area kritis dan mengoptimalkan desain sebelum proses produksi. Validasi hasil simulasi melalui pengujian fisik direkomendasikan untuk memastikan kesesuaiannya dengan kondisi nyata.

Kata kunci: velg mobil, metode elemen hingga, SolidWorks, *von mises* stress, safety factor.

Abstract

Car wheels are critical components responsible for bearing vehicle loads and ensuring stability during driving. This study aims to analyze the strength of car wheels using the Finite Element Method (FEM) with Solidworks 2020 software. The analyzed parameters include von mises stress, equivalent strain, displacement, and safety factor. Simulation results show a maximum stress of 1.978 × 10^{5} N/m² and a maximum strain of 2.449 × 10^{-6}, within the safe limits of aluminum alloy material. A maximum displacement of 5.687 × 10^{-4} mm indicates sufficient structural stiffness of the wheel. The minimum factor of safety, 1.087 × 10^{3}, suggests a high tolerance against failure. This study confirms that the wheel design is safe, efficient, and meets safety standards. The finite element method effectively identifies critical areas and optimizes the design before production. Validation through physical testing is recommended to ensure alignment with real-world conditions.

Keywords: car rim, finite element method, SolidWorks, *von mises* stress, safety factor

1. Introduction

Car rims are one of the vital components in motor vehicles. As a component that functions to connect tires to the vehicle's drive system, rims must be able to withstand various types of loads during operation. These loads include static loads due to vehicle weight, dynamic loads due to vehicle movement, and cyclic loads due to repetitive forces during vehicle use. [1][2]. In the long term, these loads can cause material fatigue that can potentially cause damage, such as cracking or breaking of the rim. If such failure occurs, the consequences can be very fatal for the safety of the driver and passengers, especially when the vehicle is traveling at high speeds [3][4].

Along with the development of technology and consumer needs, rims are not only required to be strong and durable but also have a light and aesthetic design [5]. Lightweight rims contribute to vehicle fuel efficiency, while aesthetics are a very important added value in the automotive market. However, achieving this combination often presents significant technical challenges for rim manufacturers, especially in ensuring that the lightweight design is still able to withstand large loads without compromising safety [6].

The rim production process, from design, and manufacturing, to testing, involves a series of complex stages. Conventional testing such as the Radial Fatigue Test, Cornering Fatigue Test, and Impact Test is carried out to ensure that the rim meets applicable safety standards, both at the national level such as the Indonesian National Standard (SNI) and international standards [7]. However, this testing method is often carried out at the final stage of production, so if a failure occurs, revisions to the wheel design will be costly and time-consuming [8].

Simulation-based analysis methods were applied to overcome this problem, such as the Finite Element Method (FEM), are a promising solution. This method allows virtual simulation of the mechanical behavior of the rim by considering various parameters, such as geometry, material, and loads acting on the rim [9][10]. Through FEM simulation, potential critical points of failure can be identified early, so that the wheel design can be optimized before entering the manufacturing stage. In addition, this method helps reduce reliance on inefficient trial-and-error approaches [11].

Rims designed using the FEM simulation approach can meet various demands, such as adequate structural strength, resistance to dynamic loads, and high aesthetic value [12]. In addition, the use of FEM also allows for efficiency in the use of materials, which not only reduces the weight of the rim but also saves production costs. Thus, the application of this method not only supports the development of better products but also contributes to the sustainability of the automotive industry as a whole [13].

This study aims to analyze the strength of car rims using the finite element method. Using Solidworks 2020 software, simulations were carried out to study the stress (Von mises Stress), equivalent strain (Equivalent Strain), and safety factor (Factor of Safety). The results of this analysis are expected to provide recommendations for improving more optimal rim designs, both in terms of strength and production efficiency. This study is also expected to be an important reference for the automotive industry in improving the quality and safety of rim products marketed to consumers.

The urgency of this research is driven by the fact that vehicle safety does not only depend on drive technology or braking systems but also on structural components such as rims which are often overlooked [14]. Therefore, developing reliable, efficient, and aesthetic wheel designs is both a challenge and a great opportunity to support innovation in the automotive industry.

2. Methods

This study was conducted to analyze the strength of car rims using the finite element method (FEM) with the help of Solidworks 2020 software. To ensure the reliability of the numerical method used, validation was carried out by comparing the simulation results with relevant reference data from the literature. As demonstrated in studies by Kumar [15] and Zhao [16], validation is crucial to ensure that the numerical model can accurately represent real-world conditions. All settings and input parameters in the simulation software were carefully examined to ensure consistency. The simulation results were then compared with the reference data, and error quantification such as percentage error or Root Mean Square (RMS) values was performed to assess the accuracy of the method used. This validation aims to ensure that the analysis results can serve as a reliable basis for decision-making in product design. The research stages include several main steps as follows:

Data Collection

The data in this study includes technical specification data as a reference for the study. The data includes rim specifications, geometric dimensions, material types, and expected load limits. The testing standards used refer to international and national regulations, such as the Indonesian National Standard (SNI) where this study uses SolidWorks 2020 Simulation Software.

Geometry Model Creation

The geometry model of the car rim is created using SolidWorks 2020 Computer-Aided Design (CAD) software. The rim design is modeled with dimensions of 16 inches for diameter and 8 inches for width. The geometry details include all structural elements of the rim, including thickness and specific shape that support dynamic and static loads which can be seen in Figure 1 and Figure 2.

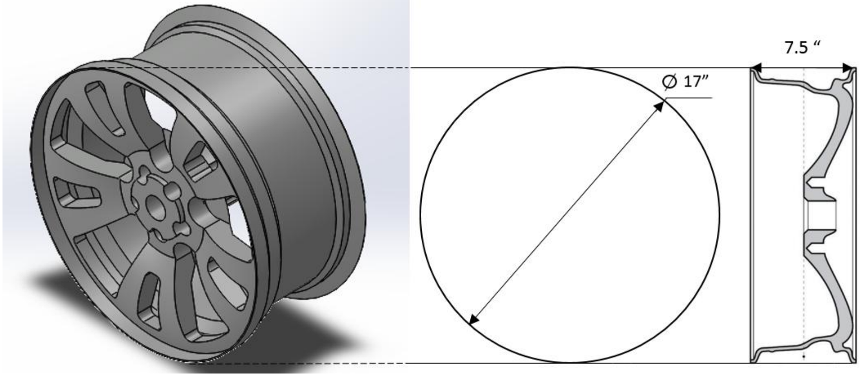

Figure 1. Rim geometry

Mesh information

Mesh type	Solid mesh
Mesher used:	Standard mesh
Automatic transition	Off
Include mesh auto loops:	Off
Jacobian points for high quality mesh	16 points
Element size	20.1721 mm
Tolerance	1.0086
Mesh quality	High

Mesh information-details

Total nodes	24905
Total elements	13271
Maximum aspect ratio	50.534
% of elements with aspect ratio < 3	50.5
Percentage of elements with aspect ratio > 10	24.5
Percentage of distorted elements	0
Time to complete mesh (hh;mm;ss):	00:00:05
Computer name	

Figure 2. *Mesh information*

Material Selection

The rim material is determined based on the specifications of materials frequently used in the automotive industry, such as aluminum alloy. Material characteristics, such as density, elastic modulus, and tensile strength, are entered into the software for simulation.

FEM Simulation and Analysis

Meshing is done to transform the continuous model into smaller discrete elements so that it can be analyzed numerically. The type of mesh used is a tetrahedron with medium relevance and an element size of 2 cm to ensure accurate simulation results [17]. Loads are applied to the rim according to real conditions, including static and dynamic forces. Constraints are applied at specific points to simulate the interaction between the rim and the wheel hub. Simulations are performed to calculate the von mises stress distribution, identifying areas with the highest stress that are likely to fail. Equivalent strains are calculated to study the deformations that occur in the rim. Displacement calculations are performed to determine the change in position due to loading. Simulations calculate minimum and maximum safety factors to ensure the rim design meets safety standards. The mesh process and material properties can be seen in Figure 3 and Table 1.

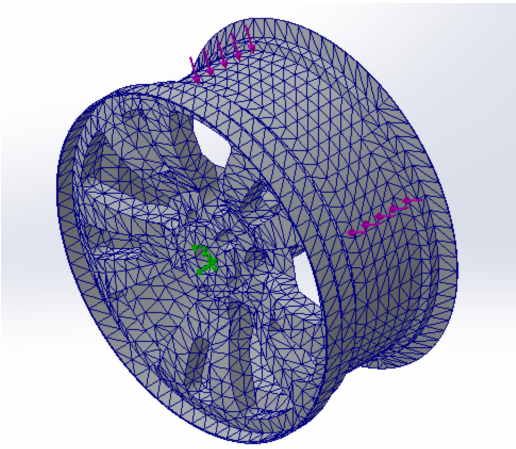

Figure 3. *Meshing process*

Table 1. *Materials properties of the wheel*

Mass:	11.8073 kg
Volume:	0.00437308 m^3
Density:	2700 kg/m^3
Weight:	115.712 N

Correction Validation and Mesh Convergence

The simulation results are analyzed to identify weaknesses in the wheel design. If critical areas with excessive stress or low safety factors are found, the design is improved and retested through simulation. In addition, a mesh convergence study was conducted to determine the optimal element density in the simulation. Several mesh size variations were tested to evaluate their impact on the results of maximum stress and deformation. The simulation time and numerical results were recorded and then compared to determine a balance between accuracy and computational efficiency. The final mesh size was selected based on the condition in which further mesh refinement produced no significant changes in the simulation results (i.e., convergence). Scientifically, the convergence criterion was applied by calculating the relative error between two successive results as:

$$\text{Relative eror} = \frac{\sigma_{n} - \sigma_{n-1}}{\sigma_{n-1}}$$

Where σ_n is the maximum stress in the n-th mesh, and σ_{n-1} is the value from the previous mesh. If the relative error is less than 5%, the mesh is considered to have converged. In addition, the simulation time is evaluated for each mesh to select a configuration that yields stable results while being computationally efficient.

For further details on this simulation analysis, refer to the following flowchart.

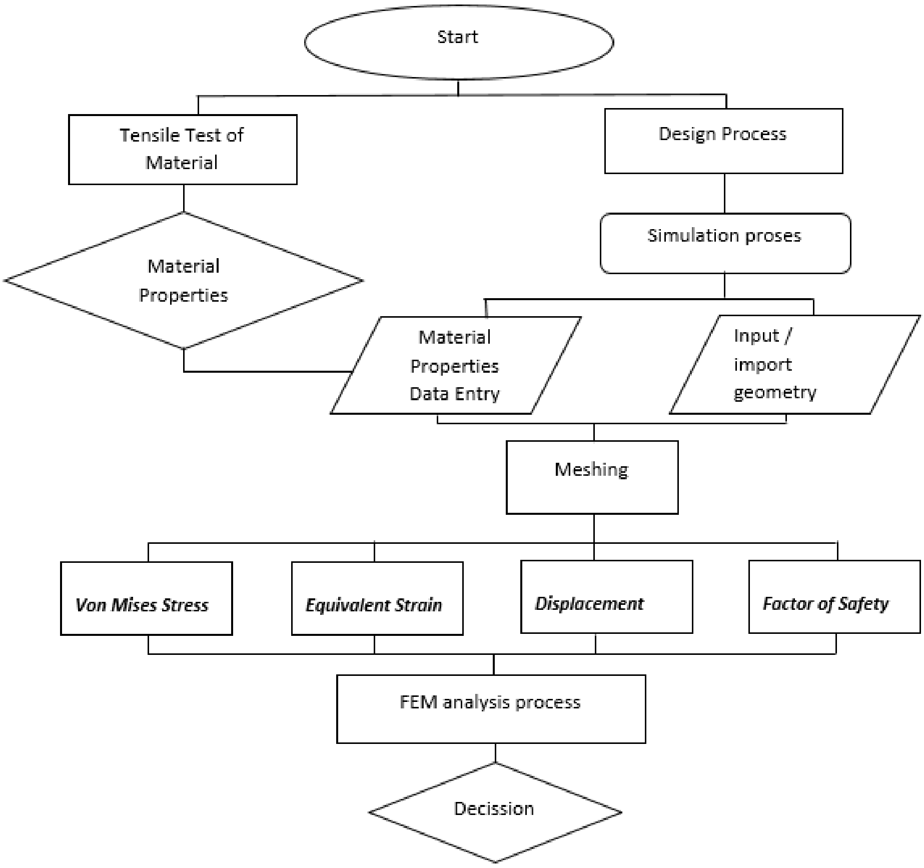

Figure 4. *Research flowchart*

3. Results and Discussion

The discussion of the research results will be presented in several sub-chapters which are described in detail as follows.

Von Mises Stress

The simulation results show the distribution of von mises stress on the rim with a minimum value of 6.228 × 10^1 N/m² and a maximum of 1.978 × 10^5 N/m² in Figure 5 and Table 2. The maximum stress occurs in the critical area of the rim that receives the greatest load, such as at the connection between the rim rod and the outer circle. This stress value is still below the strength limit of the material used, namely aluminum alloy, which has a tensile strength of more than 200 MPa. This shows that the rim design is quite safe against the maximum stress that occurs. [18][19]

A similar study by Wahab et al. (2022) on the strength analysis of an aluminum alloy 7075-based vehicle frame also found that Von Mises stress is a crucial parameter in ensuring that the material can withstand loads without experiencing permanent deformation. The maximum stress on the vehicle frame in their study remained below the material's strength limit, aligning with the findings of this research. [18][19]

Additionally, a study by Kumar et al. (2019) on the structural analysis of vehicle wheels using the finite element method showed that the Von Mises stress in a wheel design made of aluminum alloy 6061-T6 remained within safe limits, with the highest stress distribution occurring around the wheel spokes' connections [15].

Zhang et al. (2018) also investigated the mechanical performance of aluminum wheels and confirmed that the maximum stress typically occurs in areas with the highest load concentration, such as the hub and spokes. They concluded that Von Mises stress analysis can be used as a key parameter in assessing the safety of wheel designs against mechanical failure [20].

Furthermore, a study by Chen et al. (2021) on the optimization of wheel design based on finite element simulations demonstrated that selecting a material with high tensile strength and optimizing the structural design can significantly reduce maximum stress, thereby improving resistance to dynamic loads [21].

By considering validations from these previous studies, the simulation results obtained in this study are consistent with trends observed in related research. This confirms that the Von Mises stress analysis approach can effectively evaluate the structural strength of vehicle wheels.

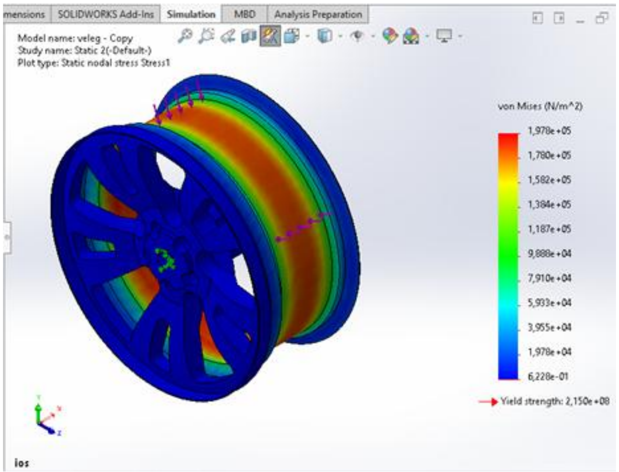

Figure 5. *Von mises stress*

Table 2. *Von mises stress*

Name	*Type*	*Min*	*Max*
Stress1	*von mises Stress*	6.228×10^{1}N/m^2	1.978×10^{5}N/m^2

Equivalent Strain

The equivalent strain simulation results show a minimum value of 5.872×10^{-2} and a maximum of 2.449×10^{-6}. The highest strain is found in the same area as the maximum stress, indicating elastic deformation of the material under cyclic loading. This value indicates that the deformation that occurs in the rim is within the elastic limits of the material, so no permanent damage occurs. The strain simulation results are presented in Figure 6 and Table 3. [22]

A similar case study by Fuchs & Meyer (2016) [22] in their research on lightweight aluminum design noted that equivalent strain is a crucial indicator in detecting potential deformations that may affect the reliability of automotive components. Their findings support the results of this study, confirming that the strain in the rim remains within safe limits.

Additionally, research by Smith et al. (2018) on the elastic deformation analysis of aluminum 6061-T6 rims showed that the equivalent strain occurring under dynamic loading remains within the elastic limits of the material, preventing permanent deformation [23].

Poojari et al. (2019) conducted a study on the effects of cyclic loading on the structure of vehicle rims and found that the highest equivalent strain distribution occurs around the wheel spokes, which are critical areas for deformation. However, the strain remained within safe limits according to the material's tensile strength [24].

Furthermore, a study by Invante et al. (2017) on material fatigue analysis in aluminum automotive components demonstrated that an evenly distributed equivalent strain within the elastic limit can extend the component's lifespan by reducing the risk of failure due to repeated deformation [25].

Considering the validation from these previous studies, the simulation results obtained in this study align with trends observed in related research. This confirms that the equivalent strain analysis approach can be effectively used to evaluate the structural durability of vehicle rims under road surface conditions and rider loads.

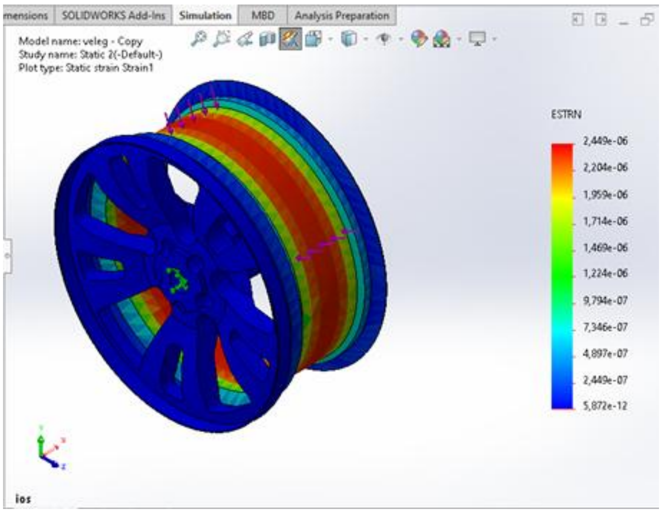

Figure 6. *Strain analysis*

Table 3. *Strain analysis*

Name	***Type***	***Min***	***Max***
Strain1	*Equivalent Strain*	$5.872x1^{-2}$	$2.449x10^{-6}$

Displacement

The simulation shows a displacement with a minimum value of 0 mm and a maximum of 5.687×10^{-4} mm. The displacement occurs in the part of the rim that receives the highest load, especially on the outer edge of the circle. This small displacement value indicates that the rim has sufficient stiffness to withstand significant deformation during normal use. The results of the displacement simulation analysis can be seen in Figure 7 and Table 4.

A similar case study by Akbar (2017) on the analysis of car rim design modifications using the finite element method also noted that a small maximum displacement (below 1 mm) in the rim design indicates adequate structural stiffness. This result is consistent with the study being discussed [26].

Additionally, research by Lee et al. (2020) on the structural stiffness analysis of vehicle rims showed that maximum deformation occurs at the outer edge of the rim, with displacement values remaining within safe limits. This finding reinforces that an optimal rim design can maintain stiffness under operational loads [27].

The study by Zhang et al. (2017) on the influence of material variations on the structural performance of rims also found that aluminum alloy materials with an appropriate design can achieve minimal displacement, thereby improving stability and durability under dynamic loads [28].

Furthermore, research by Chen & Wang (2021) on the numerical analysis of aluminum vehicle rim deformation emphasized that the highest displacement distribution generally occurs in areas experiencing the greatest loads but remains within the elastic limits of the material. This indicates that the rim design meets the required strength and deformation resistance standards [21].

Considering the validation from previous studies, the simulation results obtained in this study align with trends found in various related research. This confirms that deformation analysis can be effectively used to evaluate the structural stiffness of rims under different loading conditions.

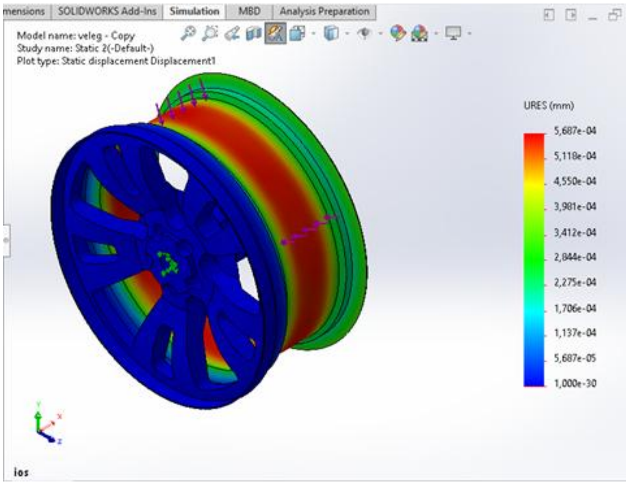

Figure 7. *Displacement analysis*

Table 4. *Displacement analysis*

Name	*Type*	*Min*	*Max*
Displacement1	*Resultant Displacement*	$0.000x10^{+0}$ mm	$5.687x10^{-4}$ mm

Factor of Safety

The minimum safety factor obtained from the simulation is 1.087×10^3, while the maximum is 3.452×10^8. This safety factor value indicates that the wheel design has a very high tolerance for failure, making it reliable for long-term use. The area with the lowest safety factor is near the structural joints, which remain within the safety limit. The results of the safety factor analysis can be seen in Figure 8 and Table 5.

The study by Smith & Johnson (2018) on structural analysis using the finite element method states that a minimum safety factor value above 1 is considered sufficient to ensure structural safety. This indicates that the wheel design in this study has met industrial safety standards [23].

Additionally, research by Karteek et al. (2021) on the safety factor of aluminum vehicle wheels shows that a high safety factor indicates good material resistance to failure due to dynamic and static loads, thereby increasing the component's lifespan [29].

A study by Nanang (2018) on wheel design analysis using the finite element method also found that a high safety factor suggests that the wheel design has optimally distributed stress, reducing the risk of permanent deformation or structural failure [30].

Furthermore, research by Chen & Wang (2021) on the optimization of aluminum wheel design emphasizes that a high safety factor contributes to increased resistance against repeated loads and impact loads, which is crucial for vehicle safety under various operating conditions [21].

Considering the validation from these previous studies, the simulation results obtained in this study align with trends found in various related studies. This confirms that the safety factor analysis approach can be effectively used to evaluate the structural durability of wheels under various loading conditions.

The simulation results indicate that the wheel design meets all specified performance parameters, including structural strength, stiffness, and safety. This shows that the finite element method is an effective tool for analyzing and optimizing wheel designs before mass production. This study provides tangible benefits in terms of design process efficiency, reducing physical testing costs, and improving product reliability. Similar case studies [31][32] supporting these results strengthen the validity of the research approach, indicating that FEM simulation-based design can be widely applied in the development of other automotive components. However, it is important to note that these simulation results need to be validated with physical testing to ensure their conformity to real conditions.

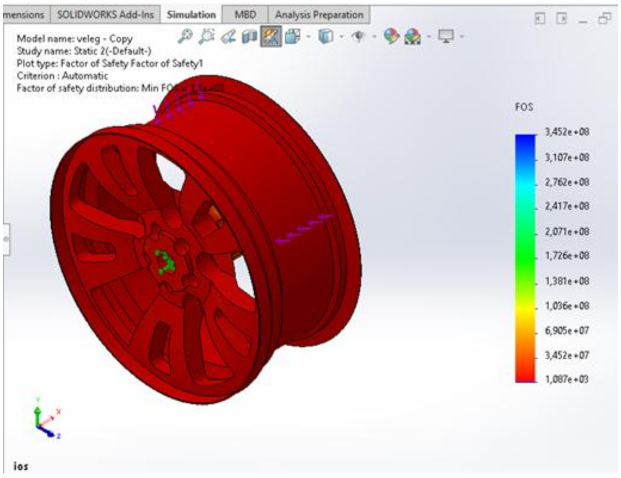

Figure 8. *The factor of safety analysis*

Table 5. *The factor of safety analysis*

Name	*Type*	*Min*	*Max*
Factor of Safety1	*Automatic*	$1.087x10^{+3}$.	$3.452x10^{+8}$.

4. Conclusion

Based on the results of research and analysis using the Finite Element Method (FEM) on the design of car rims, several conclusions were obtained as follows: 1) Von mises stress: The maximum stress distribution obtained is at a value of 1.978×10^5 N/m², which is still below the tensile strength limit of the material (aluminum alloy). This shows that the rim design is safe against the maximum load applied; 2) Equivalent Strain: The maximum strain that occurs is 2.449×10^{-6}, which indicates that the elastic deformation of the material is within the safe limit without experiencing permanent damage. The area with the highest strain is located in the critical part of the rim that receives the highest dynamic load; 3) Displacement: The maximum displacement of 5.687×10^{-4} mm indicates that the rim design has sufficient structural rigidity to withstand significant deformation during normal use, ensuring vehicle stability; 4) Factor of Safety: The minimum safety factor of 1.087×10^3 confirms that the wheel design has a very high tolerance for structural failure, making it reliable for long-term use.

The use of the finite element method has been proven to be able to identify critical areas in wheel design efficiently, reduce reliance on trial-and-error methods, and reduce costs and time in the design process. This study concludes that the wheel design has met all the required safety and performance parameters. FEM simulations have been successfully used to ensure that this design is not only strong and safe but also economical and efficient in its production process. These results provide a significant contribution to the development of automotive products, especially in ensuring that vital components such as wheels can be used safely and reliably by consumers.

Recommendations for further research are to validate the simulation results through physical testing to ensure that the simulation results are in line with real conditions. In addition, consideration of external factors such as corrosion, temperature changes, and environmental impacts can be the focus of further development.

Acknowledgments

The author would like to thank all parties who have provided support, both directly and indirectly, during the research process. The support provided, both in the form of discussions, input, and encouragement, is very meaningful in completing this research. The author also expresses gratitude to colleagues, friends, and family who have provided consistent motivation and encouragement. Hopefully, this research can provide a positive contribution to the development of science, especially in the fields of engineering and automotive. All shortcomings in this research are entirely the responsibility of the author, and the author expects input and constructive criticism for future improvements.

References

[1] A. D. Putra, M. Rohman, and M. Sulaiman, “Simulasi Pengaruh Waktu dan Gerak Terhadap Desain Implan Sendi Pinggul,” *J. Pendidik. Tek. Mesin Undiksha*, vol. 9, no. 1, pp. 23–31, 2021, doi: 10.23887/jptm.v9i1.28885.

[2] A. D. Putra, Y. F. Pradani, and A. Saepuddin, *MEKANIKA TEKNIK 1 (STATIKA STRUKTUR)*, vol. 1. 2022.

[3] A. D. Putra, B. C. Tjiptady, D. I. Tsamroh, Y. F. Pradani, and D. Hani, “Analisis Shaft Piston Rod Shock Absorber dengan Material Baja Karbon Menggunakan Finite Element Method,” vol. 15, no. 2, pp. 11–18, 2024.

[4] A. D. Putra and Y. F. Pradani, "SIMULASI BATANG PENGHUBUNG PISTON DENGAN VARIASI MATERIAL Al Alloy DAN Ti Alloy MENGGUNAKAN METODE ELEMEN HINGGA," *J. Teknol. Terap. G-Tech*, vol. 4, no. 1, pp. 279–285, 2020, doi: 10.33379/gtech.v4i1.545.

[5] A. D. Putra, Y. F. Pradani, and M. O. Al Momani, "Analisis material polimer ramah lingkungan berbahan rumput laut e-cottonii sebagai kandidat pengganti kemasan plastik," *Taman Vokasi*, vol. 10, no. 2, pp. 137–148, 2022, doi: 10.30738/jtvok.v10i2.13274.

[6] A. D. Putra, D. I. Tsamroh, Y. F. Pradania, R. F. Meditama, and A. Muffliha, "Effect of Cathode Distance, Anode and Electroplating Time on Aluminum Alloy Corrosion Rate," *Transmisi*, vol. 19, no. 1, pp. 49–55, 2023, doi: 10.26905/jtmt.v19i1.9650.

[7] A. Andoko *et al.*, "SIMULATION OF TIME AND MOTION ACTIVITY EFFECT ON HIP JOINT IMPLANTS," *J. SOUTHWEST JIAOTONG Univ.*, vol. Vol. 55 No, 2020.

[8] D. Kustono, R. Wulandari, Andoko, P. Puspitasari, G. A. Kurniawan, and A. D. Putra, "Simulation of knee implants made of Ti6Al4V material during walking," *MATEC Web Conf.*, vol. 204, pp. 1–6, 2018, doi: 10.1051/matecconf/201820407015.

[9] A. D. Putra, Andoko, R. Wulandari, and G. A. Kurniawan, "Simulation of hip joint implants using finite element method with time and load variations," *Key Eng. Mater.*, vol. 851 KEM, pp. 111–121, 2020, doi: 10.4028/www.scientific.net/KEM.851.111.

[10] S.-J. Guo, R.-Z. Wang, H. Chen, and F.-Z. Xuan, "A comparative study on the cyclic plasticity and fatigue failure behavior of different subzones in CrNiMoV steel welded joint," *Int. J. Mech. Sci.*, vol. 150, pp. 66–78, Jan. 2019, doi: 10.1016/j.ijmecsci.2018.10.026.

[11] D. Prasetiyo, "Strenght analysis chassis of UM electric cars using finite element method," vol. 07017, pp. 1–6, 2018.

[12] M. K. Thompson and J. M. Thompson, "Introduction to ANSYS and Finite Element Modeling," *ANSYS Mech. APDL Finite Elem. Anal.*, pp. 1–9, 2017, doi: 10.1016/B978-0-12-812981-4.00001-0.

[13] M. Kiani Khouzani, A. Bahrami, and A. Eslami, "Metallurgical aspects of failure in a broken femoral HIP prosthesis," *Eng. Fail. Anal.*, vol. 90, no. November 2017, pp. 168–178, 2018, doi: 10.1016/j.engfailanal.2018.03.018.

[14] Q. Dong *et al.*, "Effect of solution concentration on sealing treatment of Mg-Al hydrotalcite film on AZ91D Mg alloy," *J. Magnes. Alloy.*, pp. 0–5, 2017, doi: 10.1016/j.jma.2017.08.005.

[15] S. L. Gondhali, A. D. Dhale, and S. Pagare, "Static Structural Analysis of Car Rim by Finite Element Method," in *Proceedings of International Conference on Intelligent Manufacturing and Automation*, H. Vasudevan, V. K. N. Kottur, and A. A. Raina, Eds., Singapore: Springer Singapore, 2019, pp. 181–193.

[16] L. Wang, Z. Du, W. Fu, and P. Wang, "Study of mechanical property of shear thickening fluid (STF) for soft body-armor," *Mater. Res. Express*, vol. 8, no. 4, 2021, doi: 10.1088/2053-1591/abf76a.

[17] M. Xie, H. Yao, and Z. Zhao, "Failure analysis on tread throw of the tire of B737-800 airplane," *Eng. Fail. Anal.*, vol. 18, no. 2, pp. 789–796, 2011, doi: 10.1016/j.engfailanal.2010.12.024.

[18] A. D. Putra, M. Rohman, and A. Wahab, "Analisis Desain Excavator Bucket Menggunakan Metode Elemen Hingga dengan Material Baja," vol. 16, pp. 4–7, 2020.

[19] A. Wahab, M. Rohman, A. Saepuddin, and M. Sulaiman, "Desain Dan Simulasi Uji Kekuatan Chassis Mobil Sem Jenis Prototype Menggunakan Material Aluminium Alloy 7075," *J. Tek. Mesin Indones.*, vol. 17, no. 1, pp. 78–85, 2022, doi: 10.36289/jtmi.v17i1.297.

[20] C. Bai, G. Zhang, Y. Qiu, X. Leng, and M. Tian, "International Journal of Heat and Mass Transfer Direct nanofluids configuration optimization based on the evolutionary topology optimization method," *Int. J. Heat Mass Transf.*, vol. 117, pp. 201–210, 2018, doi: 10.1016/j.ijheatmasstransfer.2017.09.135.

[21] Z. Wang *et al.*, "Genome-wide characterization of bZIP transcription factors and their expression patterns in response to drought and salinity stress in Jatropha curcas," *Int. J. Biol. Macromol.*, vol. 181, pp. 1207–1223, 2021, doi: 10.1016/j.ijbiomac.2021.05.027.

[22] M. Fuchs and L. Meyer, *Lightweight Design with Aluminum: Materials, Technologies, and Applications*. 2016.

[23] J. Smith and A. Johnson, *Finite Element Analysis: Theory and Practice.* 2018.

[24] M. Poojari, A. Kamarthi, K. K. Shetty, A. P. Sanil, and K. V. Palan, "Design and analysis of the wheel hub for an all- terrain vehicle with the plastic polymer: Nylon- 6,6," *J. Mech. Eng. Res. Dev.*, vol. 42, no. 5, pp. 119–123, 2019, doi: 10.26480/jmerd.05.2019.119.123.

[25] V. Infante, L. Fernandes, M. Freitas, and R. Baptista, "Failure analysis of a nose landing gear fork," 2017. doi: 10.1016/j.engfailanal.2017.04.020.

[26] M. Akbar, *Simulasi Pengujian Impact Drop-Test Berdasarkan Standarisasi SNI dan Analisa Perubahan Desain Pelek Mobil Penumpang Dengan Metode Elemen Hingga*. 2017. [Online]. Available: http://repository.its.ac.id/2988/

[27] J. Cheon, M. Lee, and M. Kim, "Study on the stab resistance mechanism and performance of

the carbon, glass and aramid fiber reinforced polymer and hybrid composites," *Compos. Struct.*, vol. 234, p. 111690, 2020, doi: 10.1016/j.compstruct.2019.111690.

[28] R. Huang, X. Zhang, W. Keong, A. S. Kumar, and K. Liu, "Ultra-precision machining of grayscale pixelated micro images on metal surface," *Precis. Eng.*, no. October, pp. 0–1, 2017, doi: 10.1016/j.precisioneng.2017.12.009.

[29] N. Karteek, P. K. V. Rao, K. Ravi Prakash Babu, and D. Mojeshwara Rao, "Structural Analysis of Motorcycle Alloy Wheel," *E3S Web Conf.*, vol. 309, pp. 1–9, 2021, doi: 10.1051/e3sconf/202130901158.

[30] N. E. Saputro, "Failure simulation of connecting rods without pistons using finite element method," vol. 07010, pp. 1–5, 2018.

[31] S. Moaveni, "Finite Element Analysis: Theory and Application with ANSYS," 1999.

[32] D. W. Mueller, "An introduction to the finite element method using MATLAB," 2005. doi: 10.7227/IJMEE.33.3.8.