International Journal of
Innovation in Mechanical Engineering & Advanced Materials (IJIMEAM)
https://publikasi.mercubuana.ac.id/index.php/ijimeam

Vol.4 (No.3). 2022. pp. 66-73
Published online: December 1, 2022
ISSN: 2477-541X

IMPLEMENTATION OF THE FINITE ELEMENT METHOD IN SOLIDWORKS TO OPTIMIZE THE FRONT CAST WHEEL DESIGN FOR MOTORCYCLES

R. N. A. Kurniawan [1], D. Romahadi [1,2], M. Fitri [1], and M. R. Karim[2]*

[1] Universitas Mercu Buana, Jakarta 11650, INDONESIA
[2]School of Mechanical Engineering, Beijing Institute of Technology, Beijing 100811, CHINA

Abstract

Cast wheel rims often experience damage that causes damage to the lip of the rim, or the spokes rupture if it supports the excessive load. The safety aspect is very important to be considered in the automotive industry because it involves the lives of passengers. Structural optimization of various vehicle components has shown that component weight strongly influences vehicle performance. Based on these problems, this research aims to design a lightweight cast wheel design model that can withstand a load of 535 N. So, it is necessary to make an analysis using a comparison of design models and material variations and static simulations using SolidWorks 2018 software. The results sought are von Mises, displacement, strain, a factor of safety, and produce a lightweight design. The simulation results on the three models are still safe in holding a load of 535 N because the value of the factor of safety is not less than 1. The results of the design mass with material variations are lighter than the original wheels.

Keywords: Cast Wheel Rims, Finite Element Analysis, Von Mises Stress, Safety Factor

*Corresponding author: Tel. +62 21-5840815 Ext. 5200
E-mail address: dedik.romahadi@mercubuana.ac.id

DOI: 10.22441/ijimeam.v4i3.18794

1. Introduction

The development of the automotive industry in Indonesia is grown rapidly. These had been stated by researchers who have analyzed automotive products in Indonesia[1]. Many motorcycle components are produced by various manufacturers, one of which is wheel rim[2].

Wheel rim is a component at the bottom that supports the vehicle's weight. As a support, the wheel rim often gets a sudden shock load[3]. In general, cast wheels rim are widely used on vehicles in Indonesia because cast wheel wheels are stronger, lighter, have a sporty design and are easier to maintain. Unlike the case with spokes rim which are not easy to maintain and have the appearance of an old model[4]. Despite having the advantages of cast wheel rim, they still have drawbacks. Namely, they are often dented and broken due to damage caused by collisions during accidents or holding excess loads[5].

Things that need to be considered for rider comfort and safety include the mass of the wheels, the level of manufacturing process capability, and the performance of the wheels, which are the main points of optimizing the wheel design. The choice of material for motorcycle wheels is very influential on the level of strength. There are various motorcycle wheels, namely spoke and cast wheels[6].

The finite element method is a numerical method used to solve engineering problems, such as stress analysis in structures, personal frequencies and their shape modes, heat transfer, electromagnetics, and fluid flow. The essence of the Finite Element Method is to divide an object to be analyzed into several parts with a finite number. These parts are called elements, and a node connects each element. The process of dividing an object into several parts is called meshing[7]. The finite element method solves a problem using integral formula equations in linear and non-linear algebraic systems that have relatively accurate accuracy. The finite element method's advantage is that the formed elements will approach the actual element values[8]. Although it is an approach, this method is known to be quite effective in solving complex structures in the analysis of solid mechanics and heat transfer[9,10]. Besides that, this method is owned by one of the SolidWorks software[11].

SolidWorks is a design software used to do structural work[12]. The SolidWorks software is equipped with tools to calculate and analyze design outcomes such as stress, strain, and temperature effects[13]. Static simulation is a structural analysis model of a part that determines the limits of the ability of a specific material part to withstand static loads

Vol.4 (No.3). 2022. pp. 66-73
Published online: December 1, 2022
ISSN: 2477-541X

acting on it, including compressive, tensile, and axial loads and torsional weight. Static simulation is a type of simulation commonly used in CAD software, especially SolidWorks software. In SolidWorks, static simulation is a method of placing a load on an object which changes the objects shape and distributes the effects of the load on the workpiece[14,15].

Based on the background problems above, cast wheel rim can break due to holding excessive loads. This research will design a cast wheel design model with variations of the spoke model, which will be tested by static simulation in the SolidWorks 2018 software to determine the stress, deformation, and strain that occur and obtain the value of the design safety factor. The designed design will be made lighter by comparing the results of the selected material.

2. Experimental and Procedure

Variations of wheel designs using the SolidWorks 2018 software, an application for designing a machine part and simulation, can be carried out to find the design results. The design made in this study is a cast wheel front design model for motorcycle vehicles. Design simulation uses static simulation in SolidWorks software. In this simulation, the design model is in a fixed state or does not move. Tests were carried out to determine the strength of the wheel design on the spoke base plane and the inter-spoke plane in holding static loads.

Determining the design of the wheel rim required a determining factor that can be used as a reference. This factor is the wheel safety factor. In terms of the static load safety factor for motorcycle wheels, the minimum static load requirement is 1 to avoid design failure[16]. In addition, the success of the wheel rim design will be lighter, as explained by Joshi[17], who currently weighs 3.61 kg on a 150-cc motorcycle class.

2.1 Calculation of Load Distribution

In testing this design, static loading will be carried out from the amount of force exerted by the overall weight of the motorbike and its two passengers, including the driver[18]. The value of the cast wheel rim load uses a motorbike load of the Yamaha Vixion motorbike type with a vehicle load of 132 kg and two passengers (including the driver); the average adult weight is an assumption to be 63.2 kg for one person, so the total wheel load is 132 kg + (63.2 kg x 2) = 258.4 kg[19]. Given that the vehicle weighs 132 kg ≈ 1320 N and an adult weighs 63.2 kg ≈ 632 N, the following calculation can be performed.

$$\sum MB = 0$$

$$R_{va} \,.\, (1300) - 1320\,(700) - 632\,(320) = 0.$$

$$R_{va} = \frac{1320\,(700) - 632\,(320) - 632\,(40)}{1.300} = 535 \text{ N.}$$

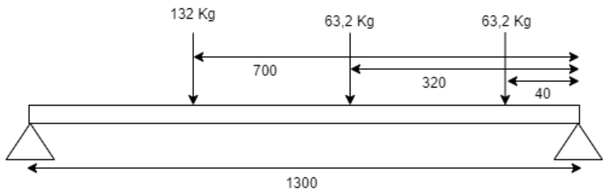


Fig. 1. Illustration of distance and load measurements on vehicles

2.2 Design and Simulation Process

The simulation uses static simulation in SolidWorks software. In this simulation, the design model is in a fixed state or does not move. Tests were carried out to determine the strength of the wheel design on the spoke base plane and the inter-spoke plane. The simulation stages are as follows:

1. Part design

In the first stage, the researchers made the essential test parts for the wheel design. Fig. 3 is the primary test shape made with a length of 60 mm, a width of 73.97 mm, and a height of 20 mm.

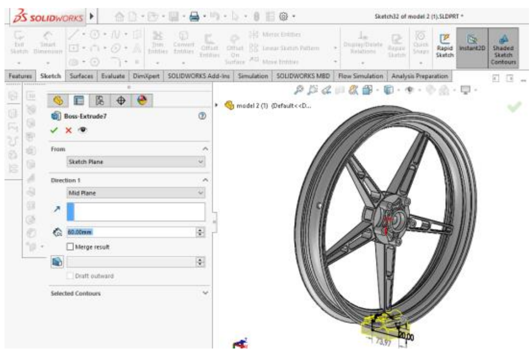


Fig. 2. Basic field of testing

2. Tools Apply Material

The next step is the Tools Apply Material menu for selecting the material to be applied to the design model. In these tools researchers can choose the material that has been provided in the software or can be custom material that they want to use.

3. Make Rigid

In the basic model of the test field, the test field is the basis that holds the design.

4. Fixed Geometry

Next is to make a fixed geometry or fix geometry on the wheel axle.

Vol.4 (No.3). 2022. pp. 66-73
Published online: December 1, 2022
ISSN: 2477-541X

5. Force
The force stage provides a large amount of force received on the wheel design. Giving force loads on the test base
6. Meshing
The meshing process is carried out, which is the process of dividing components into small particles.
7. Tools run
Then run the simulation by selecting Tools Run This Study.

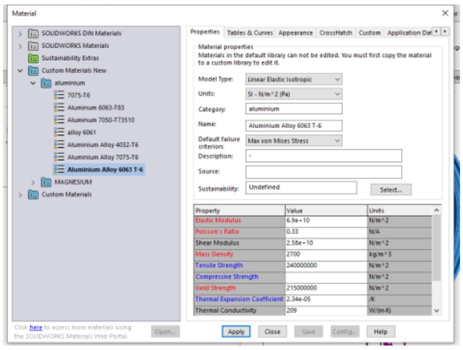


Fig. 3. Basic field of testing

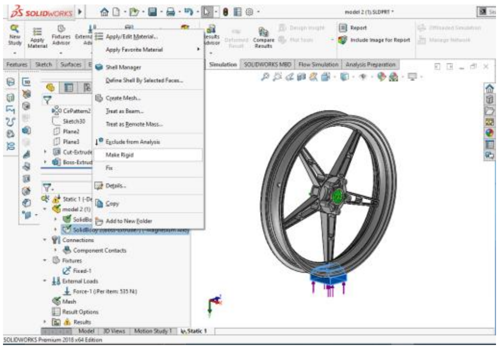


Fig. 4. Make Rigid

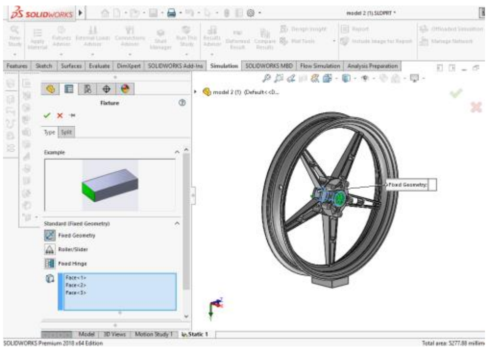


Fig. 5. Fix geometry shaft wheel rim

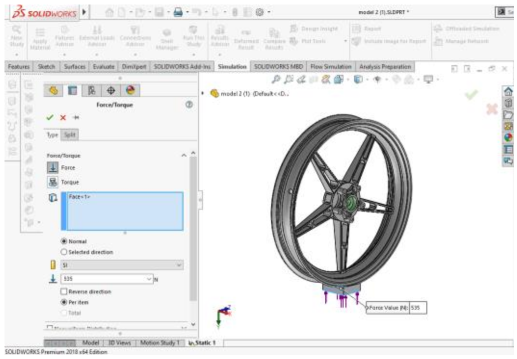


Fig. 6. Force strength

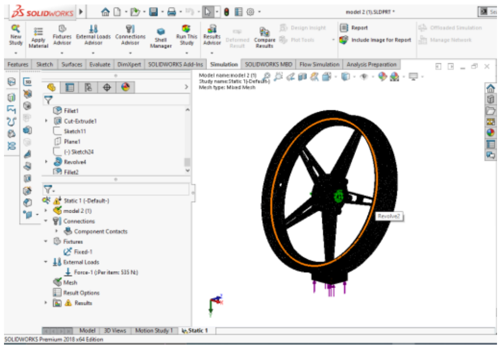


Fig. 7. Meshing

3. Results and Discussion

The simulation investigates the effect of the number of elements, or what can be called a meshing independence study and look at the distribution of von Mises stress, displacement, and strain. The analysis uses a static simulation with a static load of 535 N. Design testing is carried out on the spoke base plane and the inter-spoke plane to determine the safety level and design mass.

3.1 Results of Meshing Independence Study

The mesh analysis used is the tetrahedral elements. The meshing independence study is carried out by determining the least number of elements to the most significant number. The stress results obtained will be graphed to determine the effect of the number of elements. Fig. 8 shows the results of the von Mises stress graph that occurs in the design model. This research uses number of elements from 80,000 to 600,000. The results obtained here use 600,000 number of elements because the more elements used, the more accurate will be the results.

Vol.4 (No.3). 2022. pp. 66-73
Published online: December 1, 2022
ISSN: 2477-541X

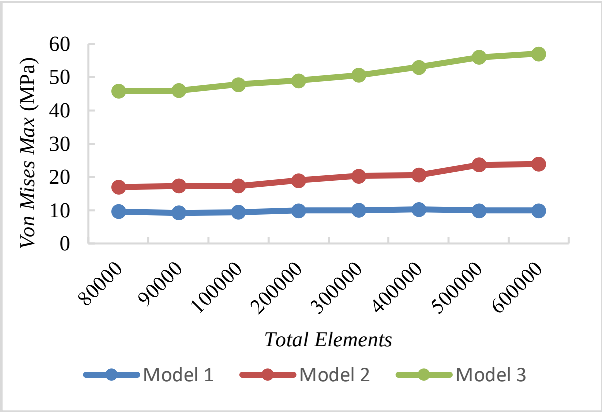


Fig. 8. Von Mises stress max graph on the three models

3.2 Simulation Design Results of Aluminum Alloy 6061-T6 Wheel Rim

1. Model A

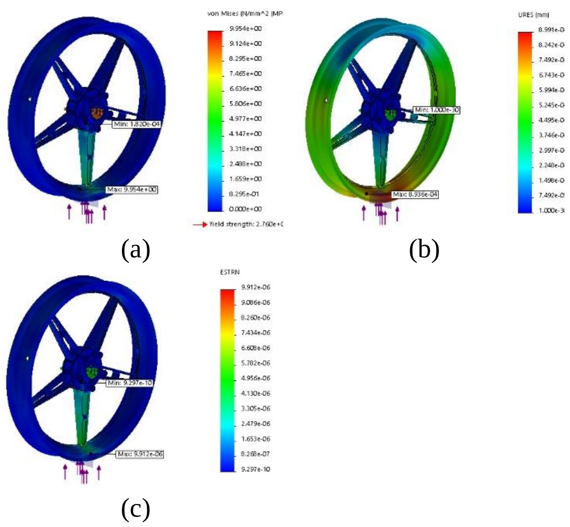


Fig. 9. Spoke base of model A aluminum alloy 6061-T6 wheel rim (a) von Mises stress, (b) displacement, and (c) strain

Fig. 9 (a) is the result of a static simulation of the wheel design on the spoke base plane, which shows the result of the maximum von Mises stress of 9.954 MPa. Fig. 9 (b) shows a maximum displacement value of 0.0008936 mm. Fig. 9 (c) shows the results of the maximum strain value of 0.000009912.

Fig. 10 (a) is the result of a static simulation of the wheel design in the inter-spoke plane, which shows the result of the maximum von Mises stress of 47.26 MPa. Fig. 10 (b) shows a maximum displacement value of 0.005614 mm. Fig. 10 (c) shows the results of the maximum strain value of 0.00004077.

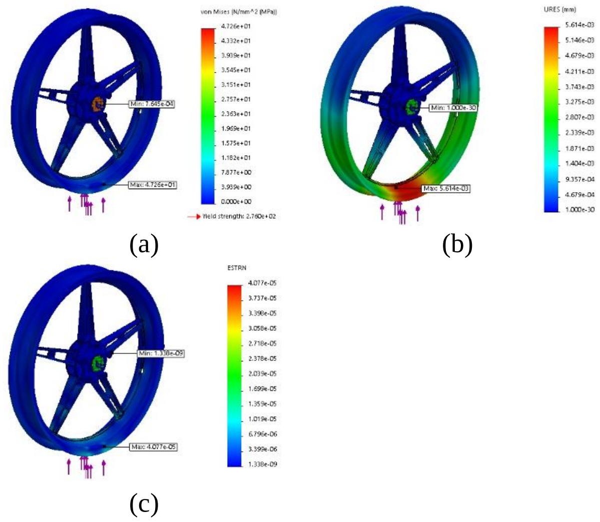


Fig. 10. Field between spoke of model A aluminum alloy 6061-T6 wheel rim (a) von Mises stress, (b) displacement, and (c) strain

2. Model B

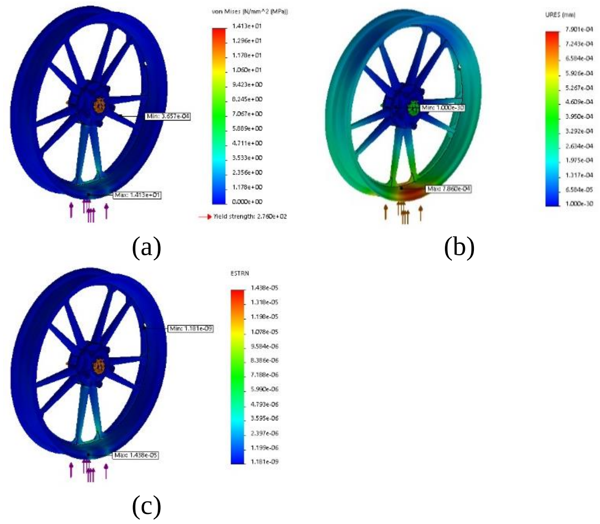
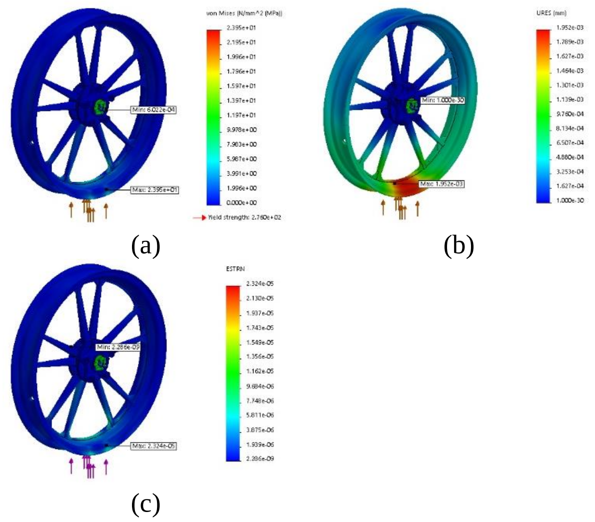


Fig. 11. Spoke base of model B aluminum alloy 6061-T6 wheel rim (a) von Mises stress (b) displacement, and (c) strain (part 2)

Fig. 11 (a) is the result of a static simulation of the wheel design on the spoke base plane, which shows the result of the maximum von Mises stress of 14.93 MPa. Fig. 11 (b) shows a maximum displacement value of 0.0007860 mm. Fig. 11 (c) shows the results of the maximum strain value of 0.00001438.

Fig. 12 (a) is the result of a static simulation of the wheel design in the inter-spoke plane, which shows the result of the maximum von Mises stress of 23.95 MPa. Fig. 12 (b) shows a maximum displacement value of 0.001952 mm. Fig. 12 (c) shows the results of the maximum strain value of

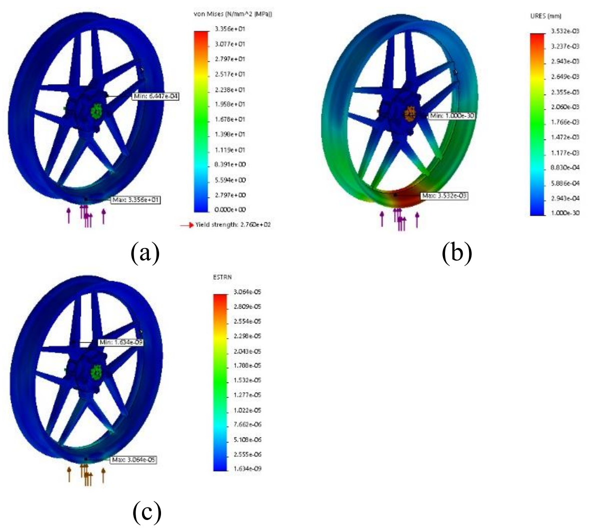
0.00002324.



(a) (b) (c)

Fig. 12. Field between spoke of model B aluminum alloy 6061-T6 wheel rim (a) von Mises stress, (b) displacement, and (c) strain

3. Model C

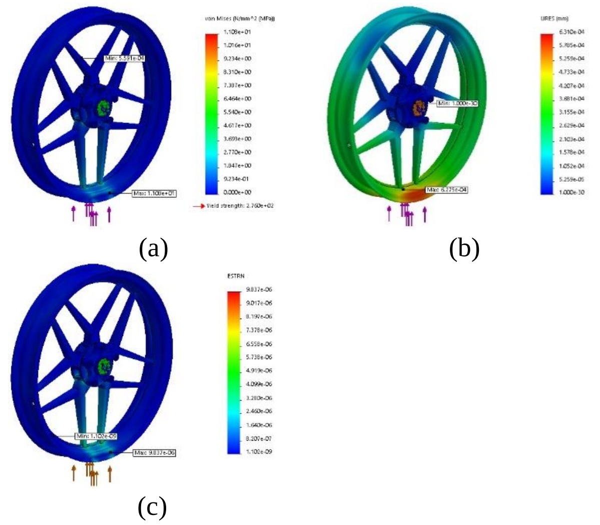


(a) (b) (c)

Fig. 13. Spoke base of model C aluminum alloy 6061-T6 wheel rim (a) von Mises stress (b) displacement (c) strain

Fig. 13 (a) is the result of a static simulation of the wheel design on the spoke base plane, which shows the result of the maximum von Mises stress of 33.56 MPa. Fig. 13 (b) shows a maximum displacement value of 0.003532 mm. Fig. 13 (c) shows the results of the maximum strain value of 0.00003064.

Fig. 14 (a) is the result of a static simulation of the wheel design in the inter-spoke plane, which shows the result of the maximum von Mises stress of 11.08 MPa. Fig. 14 (b) shows a maximum displacement value of 0.0006275 mm. Fig. 14 (c) shows the results of the maximum strain value of 0.000009837.

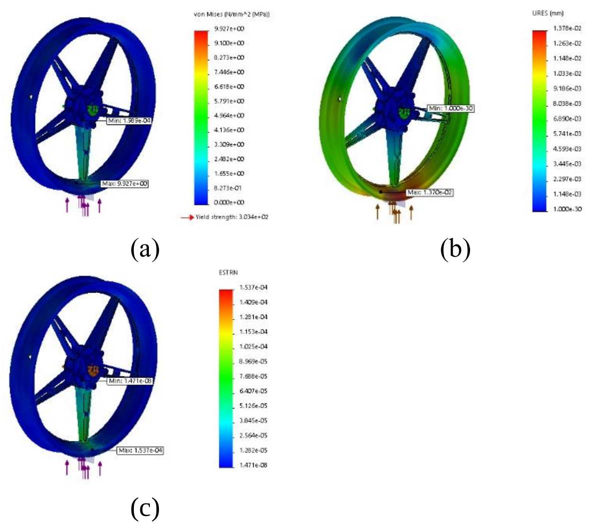


(a) (b) (c)

Fig. 14. Field between spoke of model B aluminum alloy 6061-T6 wheel rim (a) von Mises stress, (b) displacement, and (c) strain

3.3 Simulation Design Results of ZK60A Magnesium Alloy Wheel Rim

1. Model A



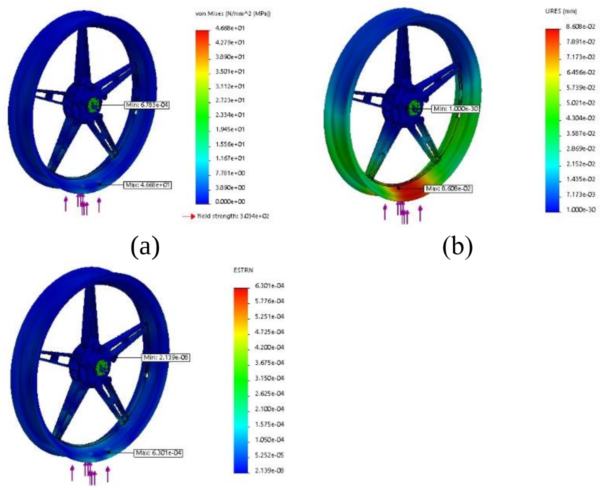
(a) (b) (c)

Fig. 15. Spoke base of model A magnesium alloy ZK60A wheel rim (a) von Mises stress (b) displacement, and (c) strain

Fig. 15 (a) is the result of a static simulation of the wheel design on the spoke base plane, which shows the result of the maximum von Mises stress of 9.927 MPa. Fig. 15 (b) shows a maximum displacement value of 0.01370 mm. Fig. 15 (c) shows the results of the maximum strain value of 0.0001537.

Vol.4 (No.3). 2022. pp. 66-73
Published online: December 1, 2022
ISSN: 2477-541X

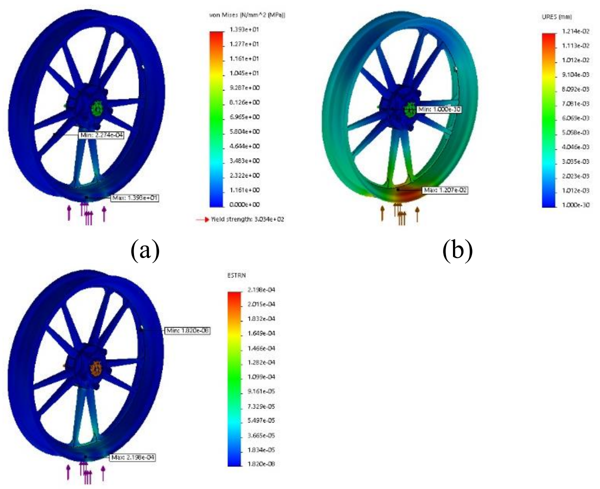
Fig. 16 (a) is the result of a static simulation of the wheel design in the inter-spoke plane, which shows the result of the maximum von Mises stress of 46.68 MPa. Fig. 16 (b) shows a maximum displacement value of 0.08608 mm. Fig. 16 (c) shows the results of the maximum strain value of 0.0006301.



(a) (b) (c)

Fig. 16. Field between spoke of model A magnesium alloy ZK60A wheel rim (a) von Mises stress (b) displacement, and (c) strain

2. Model B

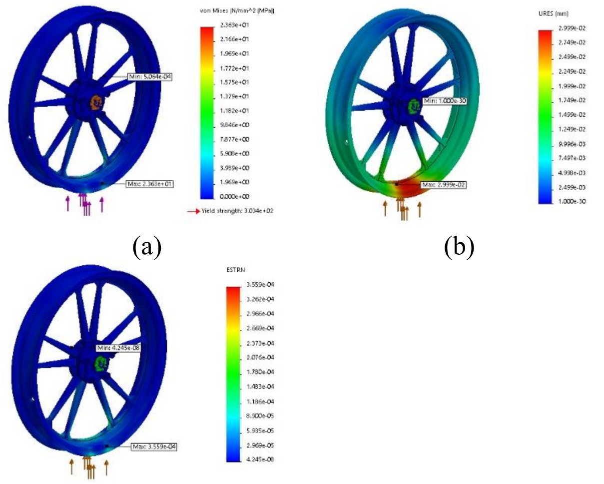


(a) (b) (c)

Fig. 17. Spoke base of model B magnesium alloy ZK60A wheel rim (a) von Mises stress (b) displacement, and (c) strain

Fig. 17 (a) is the result of a static simulation of the wheel design on the spoke base plane, which shows the result of the maximum von Mises stress of 13.93 MPa. Fig. 17 (b) shows a maximum displacement value of 0.01207 mm. Fig. 17 (c) shows the results of the maximum strain value of 0.0002198.

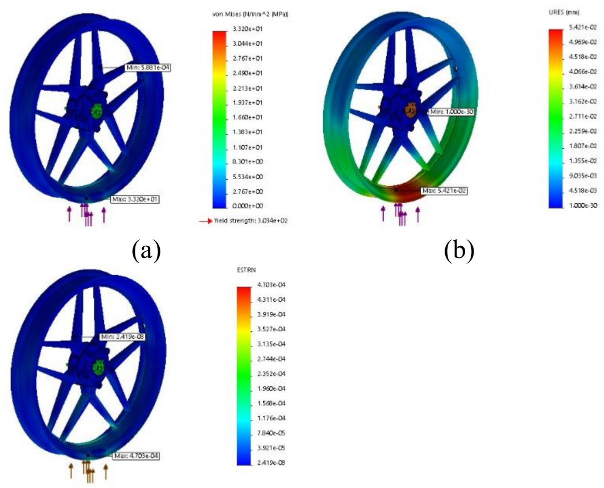
Fig. 18 (a) is the result of a static simulation of the wheel design in the inter-spoke plane, which shows the result of the maximum von Mises stress of 23.63 MPa. Fig. 18 (b) shows a maximum displacement value of 0.02999 mm. Fig. 18 (c) shows the results of the maximum strain value of 0.0003559.



(a) (b) (c)

Fig. 18. Field between spoke of model B magnesium alloy ZK60A wheel rim (a) von Mises stress (b) displacement, and (c) strain

3. Model C



(a) (b) (c)

Fig. 19. Spoke base of model C magnesium alloy ZK60A wheel rim (a) von Mises stress (b) displacement, and (c) strain.

Fig. 19 (a) is the result of a static simulation of the wheel design on the spoke base plane, which shows the result of the maximum von Mises stress of 33.2 MPa. Fig. 19 (b) shows a maximum

Vol.4 (No.3). 2022. pp. 66-73
Published online: December 1, 2022
ISSN: 2477-541X

displacement value of 0.05421 mm. Fig. 19 (c) shows the results of the maximum strain value of 0.0004703.

Fig. 20 (a) is the result of a static simulation of the wheel design in the inter-spoke plane, which shows the result of the maximum von Mises stress of 10.96 MPa. Fig. 20 (b) shows a maximum displacement value of 0.009630 mm. Fig. 20 (c) shows the results of the maximum strain value of 0.0001511.

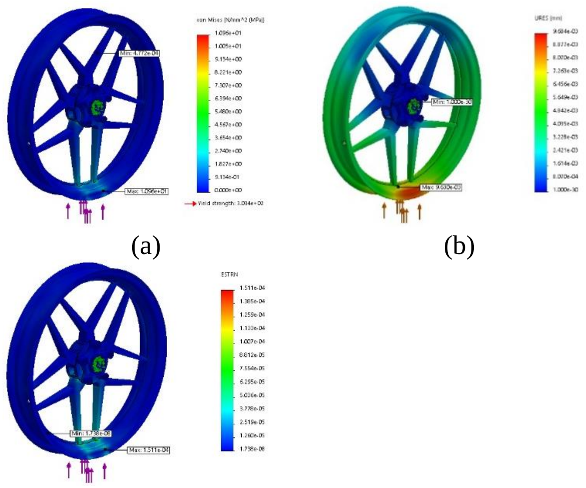


Fig. 20. Field between spoke of model C magnesium alloy ZK60A wheel rim (a) von Mises stress (b) displacement, and (c) strain

3.4 *Simulation Analysis*

The safety factors of the wheel designs based on the von Mises stress are shown Based in Fig. 21 and Fig. 22. The design results and materials used are included in the safe category for holding a load of 535 N. This is because the results of the safety factor are not less than 1 (minimum requirement of static load).

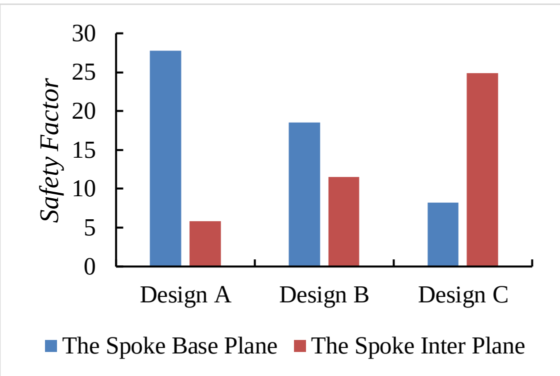


Fig. 21. Graph of safety factor for aluminum alloy 6061-T6 material

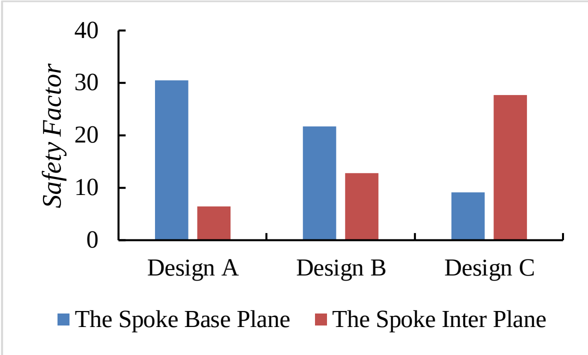


Fig. 22. Graph of safety factor for magnesium alloy ZK60A material

Fig. 23 shows that the modified wheel design has a lighter mass than the original wheel. Model A has a lighter mass because the number of spokes is less than Model B and Model C. A large number of spokes affects the mass of the design. Compared to these two materials, the magnesium alloy ZK60A material has a very light yield because it has a mass density of 1830 kg/m^3, while the aluminum alloy 6061-T6 material has a mass density of 2700 kg/m^3.

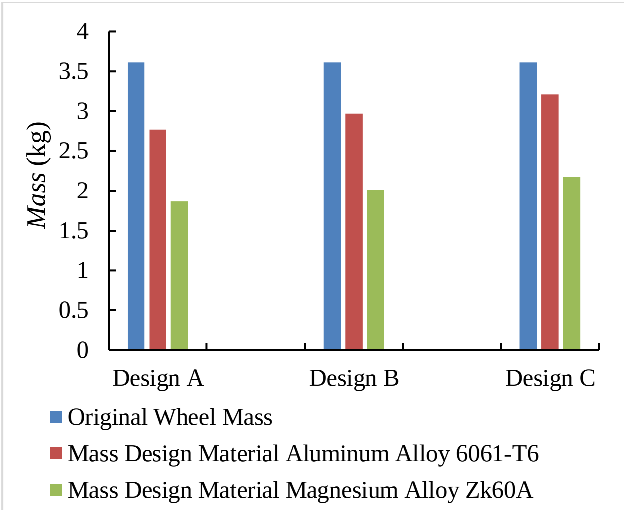


Fig. 23. Comparison of original wheel mass with design mass

4. Conclusions

The simulation of a motorcycle cast front wheel rim design results different safety factors. Seeing from the results of the safety factor in the three models and the variety of materials used, they are still categorized as safe because the results of the safety factor calculation have met the minimum requirements for static load, which is not less 1. For the results, the highest safety factor is in the magnesium alloy ZK60A material because this material has a yield strength of 303 MPa, and the aluminum alloy 6061-T6 material has a yield strength of 276 MPa. The result shows that the modified wheel design mass is lighter than the original wheel. For the lightest design, the ZK60A magnesium alloy material has a mass density of 1830 kg/m^3, and aluminum alloy 6061-T6 has a mass density of 2700 kg/m^3.

References

[1] Sorongan, F. A. (2019). Pengaruh return on assets, return on equity, earning per share terhadap harga saham pada perusahaan otomotif yang terdaftar BEI. *JMBI UNSRAT (Jurnal Ilmiah Manajemen Bisnis dan Inovasi Universitas Sam Ratulangi)*, *6*(2), 106–114.

[2] Iskandar, A., Sulaeman, A., Rofiroh, R., & Rizky, P. (2022). analisis performance velg vrossi honda scoopy dengan Finite Element Analysis Method (FEAM) menggunakan Solidworks. *Motor Bakar: Jurnal Teknik Mesin*, *5*(1), 1-8.

[3] Widyantoro, M., Solihin, S., & Rifda, I. R. (2020). Peningkatan efisiensi pada lini proses machining velg motor dengan metode line balancing PT. XYZ Murwan. *Jurnal Penelitian dan Aplikasi Sistem & Teknik Industri (PASTI)*, *14*(01), 54-64.

[4] Zuli, M. (2016). Analisis sifat mekanik dan struktur mikro velg sepeda motor spoke wheel dan cast wheel. *Barometer*, *1*(1), 18-20.

[5] Dahlan, A., & Rusiyanto, R. (2021). Pengaruh penambahan unsur aluminium murni pada bahan aluminium scrap terhadap ketangguhan impak dan struktur mikro hasil pengecoran velg motor Honda. *Jurnal Dinamika Vokasional Teknik Mesin*, *6*(1), 58-68.

[6] Trimulya, C. T., Cholis, N., Pradana, S., & Wahyuni, F. (2020). Analisis produk pelek motor tipe cast wheel berbahan paduan alumunium. *Bina Tek.*, *16*(2), 79–84.

[7] Santoso, E. T. F., Anwar, C., & Heryana, G. (2020). Analisis kegagalan produk presered concrete girder I akibat deformasi cetakan dengan menggunakan metode elemen hingga. *Jurnal Teknologika*, *10*(2), 27-36.

[8] Warsito, A., Bukit, M., & Johannes, A. Z. (2022). Analisis sifat mekanik modifikasi kemiringan mercu spillway tipe ogee dengan studi kasus bendungan Raknamo menggunakan metode elemen hingga software Elmer. *Jurnal Fisika: Fisika Sains dan Aplikasinya*, *7*(1), 69-74.

[9] Lubis, S. (2019). Analisis pengaruh besar gesekan terhadap tegangan thermal pada sepatu rem mobil ketebalan 8 mm menggunakan perangkat lunak Msc. Nastran V. 9. *Jurnal Rekayasa Material, Manufaktur dan Energi*, *2*(1), 44-53.

[10] Romahadi, D., Ruhyat, N., & Dorion, L. D. (2020). Condensor design analysis with Kays and London surface dimensions. *Sinergi*, *24*(2), 81-86.

[11] Rasyid, M. K. (2018). Analisis bentuk penampang batang engkol menggunakan metode elemen hingga. *J. Tek. Mesin ITI*, *2*(1), 12–15.

[12] Anjasmoro, G., Robbi, N., & Choirotin, I. (2022). Analisis kekuatan pembebanan rangka pada perancangan mesin pencetak pelet menggunakan simulasi Solidworks. *J. Sains dan Teknol. Tek. Mesin Unisma*, *18*(2), 195-201.

[13] Adi, I. N. A., Dantes, K. R., & Nugraha, I. N. P. (2018). Analisis tegangan statik pada rancangan frame mobil listrik Ganesha Sakti (Gaski) menggunakan software Solidworks 2014. *Jurnal Pendidikan Teknik Mesin Undiksha*, *6*(2), 113-120.

[14] Randis, Maulana, N., & Hermansyah, H. (2021). Perancangan dan simulasi struktur rangka overhaul stand untuk penggunaan assembly dan diassembly hydraulic cylinder menggunakan software Solidwork, *Jurnal Teknik Mesin, 10*(1), 36.

[15] Romahadi, D., Feriyanto, D., Suprihatiningsih, W., & Setiawan, W. N. (2022). Perancangan sistem diagnosis getaran motor menggunakan jaringan saraf tiruan propagasi mundur. *Jurnal Rekayasa Mesin*, *13*(1), 37-46.

[16] Syaefudin, E. A., & Basori, I. (2013). Perancangan desain velg sepeda motor hybrid dan pengujian distribusi beban dengan software Autodesk Inventor. *Jurnal Konversi Energi dan Manufaktur*, *1*(1), 22-31.

[17] Joshi, H., Powar, A., Khuley, S., & Yesane, D. P. (2016). Analysis and topological optimization of motorcycle front wheel. *Int. J. Curr. Eng. Technol.*, *6*(6), 265–269.

[18] Yamaha Motor Co. (2022). *Yamaha Vixion R spesifikasi.*

[19] Syahputri, A., & Karjadidjaja, I. (2022). Hubungan pola aktivitas fisik dengan Indeks Massa Tubuh (IMT) dan lingkar pinggang pengunjung usia dewasa di Puskesmas Kecamatan Kalideres Jakarta Barat. *Tarumanagara Medical Journal*, *4*(1), 1-7.