International Journal of Transport Development and Integration
Vol. 9, No. 2, June, 2025, pp. 441-448
Journal homepage: http://iieta.org/journals/ijtdi

Mathematical Models of a Car Wheel to Solve Its Failure Problems Under Impact Load

Mohammad Takey Elias Kassim[1], Zainab Mohamed Tahir Rashid[1], Rafal Khalid Jasim Sulaiman[1], Emad Toma Karash[1*], Ahmed Mohammed Mahmood[2], Ayad Dawood Sulaiman[1]

[1] Technical Institute of Mosul, Northern Technical University, Mosul 41000, Iraq
[2] Department of Optometry, Al-Noor University, Mosul 41012, Iraq

Corresponding Author Email: emadbane2007@ntu.edu.iq

https://doi.org/10.18280/ijtdi.090219

Received: 10 March 2025
Revised: 8 April 2025
Accepted: 17 April 2025
Available online: 30 June 2025

Keywords:
impact test, finite element, vehicle, ANSYS program, Von Mises stress

ABSTRACT

Vehicle tires are subjected to sudden and significant loads when driving at high speeds due to unexpected bumps in the road. To reduce the occurrence of these cracks, this research will address the occurrence of these cracks using various techniques. The Solid Works program will be used to design various wheel models and reinforce the areas where cracks occur. The models will then be loaded into the ANSYS program to determine the various deformations and stresses they experience, as well as the degree of improvement of the wheel models whose designs have been developed. The results demonstrated that the deformation models' values were substantially lower than those of the first model, with the third model showing the biggest percentage decrease (59.56%). The results showed that the Von Mises models' values and the maximum shear stress were considerably lower than those of the first model, with the third model showing the biggest percentage decline at (68.12 and 61.2%), respectively. The fact that these improved percentages are reached in the three models (64.74, 93.12, and 88.72%) indicates that the fatigue damage values of the three improved models in the design are significantly lower than the fatigue damage values of the first model. It is clear that the third model, with a safety factor increase of 93.12%, has the highest increase. This suggests that the third model, which has three collars reinforcing it in the region where cracks are developing, is the best.

1. INTRODUCTION

Wheels made of aluminum alloy have better heat conductivity. (>150 watts per kelvin per meter) compared to steel (45 watts per kelvin per meter), which is typically lighter and offers greater structural strength. It also improves handling by lowering unsprang mass, enabling the suspension to more closely follow the terrain and increase grip. Because alloy wheels do not rust internally as steel wheels do, they are more corrosion-resistant and last the lifetime of a vehicle [1-5]. For many years, automobiles have been the backbone of the global workforce. The roughest terrain, rocky roads, and other difficulties that the wheels are exposed to when driving may be traversed by them thanks to their size and strength. However, vehicles cannot move without functioning wheels. The wheel supports the static and dynamic loads experienced during a vehicle's operations and is a crucial structural part of the suspension system. Car rims must be carefully constructed because they are one of the most important parts of a vehicle. When building a mechanical structure, safety and economy are of vital importance to ensure that people can utilize it safely and affordably. The four main technical concerns involved in the design of a new wheel and its optimization are style, weight, manufacturability, and performance. Determining how a wheel will behave mechanically in actual service settings is crucial, but testing and inspecting wheels while they are being developed takes time and money. It's crucial to cut down on time spent during the new wheels' development and testing for financial reasons. To assess the mechanical performance of prototyping designs, Finite Element Analysis (FEA) utilizing the ANSYS program or other software is typically utilized in the design phase of product development.

Several studies have used FEA to examine how well wheel rims perform.

2. LITERATURE REVIEW

Several experimental studies [6-8] looked into the different wheel rim problems that could occur. The 13-degree impact test, which replicates the lateral impact of a wheel or other object on the side of a road during a collision, is the primary impact testing technique [9, 10]. Relying on the conventional design and development methodology lengthens the development cycle and prevents the product from being swiftly introduced to the market because it is difficult for new-style wheels to pass the impact test at one time. The use of finite element modeling in the design of aluminum alloy wheels has grown popular due to the quick advancement of computer simulation technology. Prior to now, the implicit finite element method-based simulation study for the wheel bending fatigue test and the radial fatigue test was very advanced [11-14]. The test duration and expense for developing a prototype wheel can be greatly decreased by using Computer-aided engineering (CAE) simulation. To simulate a wheel impact test, a variety of techniques and procedures can be applied [15-17]. The time and expense needed to complete a wheel design can be greatly decreased by computer modeling of wheel tests. The results of the rotating bending test and the radial fatigue test have demonstrated good agreement between simulation and experiment [18-21]. The main objective of this research is to reduce unscheduled brake failures caused by overheating [1]. Although not all alloy wheels are lighter than their steel counterparts, lighter wheels can increase hold by decreasing unsprang mass, allowing the suspension to monitor the ground more thoroughly and enhancing grip. A heavy load puts wheels at risk. Thus, a wheel must be observed for both strength and cyclic stress struggle. In addition, material thickness can be optimized to reduce material consumption [22]. This investigation used ANSYS15.0 to examine alloy wheel static stacking. This paper summarizes the finite element analysis that was done on an aluminum composite wheel. The effects of absolute dislocation, safety factor, and Von Mises stress were ascertained. Additionally, the results were favorable for the structure based on the computation of the obtained data [23]. The study's findings demonstrated that while carbon fiber experienced less deformation overall, the oval-shape AL6061 material was subjected to greater total deformation than the other materials. In comparison to the other materials, the oval-shaped AL6061 material experienced higher Von Mises stress and shear stress, whereas the carbon fiber experienced lower levels of both stressors. This study's comparison of all the data indicates that carbon fiber with a hexagonal shape is superior to the other materials and a good choice for making alloy wheels [24]. Three models for radial tests and finite element bending tests were created for this investigation. The stress and strain performance of each material can be determined using three different materials: steel, aluminum alloy, and magnesium alloy. We were able to get reasonable and better results for the magnesium alloy wheel by evaluating and analyzing the model in the radial test and bending test. The viability of the magnesium alloy wheel was validated by comparing the equivalent stress and deformation results. The structural design's dependability is predicted by this study, and some helpful references for the creation of magnesium alloy wheels are supplied [25]. Automobile wheel rims are extremely delicate because they support loads of varying intensities. Because it creates resistance to the vehicle's performance, the weight of the wheel should be considered when designing it. Thus, without sacrificing the wheel's safety, the wheel rim should be sufficiently light to enhance performance. Various test conditions that enhance vehicle safety have been used to assess the wheel rim's performance. This study discusses the loading conditions and test methodology of several test techniques in order to determine the true stresses and rim deformations. This study suggests using SAPH440 for heavy-duty vehicles like commercial trucks, etc., and AL356.2T6 to lower the vehicle's unsprung mass and increase performance [26]. The purpose of this study is to carry out additional analysis on static, modular, and harmonic structures. The equivalent Von Mises stress, total deformation, and ultimate principal stresses were discovered by the static structure study. AL7075, Inconel 718, and titanium alloy all performed well under the given load conditions and are excellent choices for alloy wheel design applications [27]. This study's researchers analyzed the aluminum wheel under radial load circumstances and found that its overall deformation was 0.2833 mm. These stresses, which resulted from different boundary conditions, were 163 MPa and 0.038 MPa less than the yield strength of aluminum alloy. Many studies in a variety of technical and engineering domains have used the finite element approach to examine different mathematical models developed using the ANSYS software. The stress and strain states of those models were ascertained by applying different loads to them in addition to assessing the degree of deformation. Several experimental studies [28-32] demonstrated results consistent with theoretical predictions and actual performance requirements.

3. OBJECTIVE OF THE STUDY

As a result, this investigation will comprise identifying the deformations, stresses, and strains brought on by consistent loads in the market-standard Mercedes wheels. In addition, this study intends to treat wheels cracking brought on by sudden loads they are exposed to, employing various techniques to lessen the appearance of cracking while these wheels are exposed to significant unexpected loads, such as the existence of bumps in the road. In the current study, wheels are modeled using a mathematical model, and the model component is then analyzed using the ANSYS15.0 program. Typically, a static analysis is undertaken as part of the analysis.

4. RESEAECH NETHODOLGY

4.1 Design of wheel rim using solid works

The dimensions listed in Table 1 were used to design a wheel rim, and these same dimensions were used for analysis in this research.

Table 1. Technical details and dimensions of wheel

NO.	Item	Specification
1	Outer diameter (Wheel size)	431.8 mm
2	Bolt pattern (Pitch circle diameter)	112 mm
3	Rim diameter	431.8 mm
4	Item diameter	431.8 mm
5	Rim width	190.5 mm
6	Item weight	13.199 kg
7	Bolt pattern (Number of Holes)	5
8	Offset	49 mm
9	Material	Alloy Aluminum

Solid Works 2016 is also used to create the 3D model of the wheel rim. Figure 1 depicts the wheel rim's final look.

4.2 Materials used

The mechanically specific Aluminum alloy $[Density\ (\rho) = 2800\frac{kg}{m3},\ Young's\ Modulus\ (E) = 73.1\ GPa,\ Poisson's\ Ratio\ (\mu) = 0.33,\ Yield\ Stress = 505\ MPa]$ was chosen due to its superior stability under vibration loads, improved heat dissipation capabilities, improved braking performance, and extended fatigue life.

These factors contribute to its overall superiority over carbon steel wheel rims [33].

Figure 1. Isometric view alloy wheel rims compatible

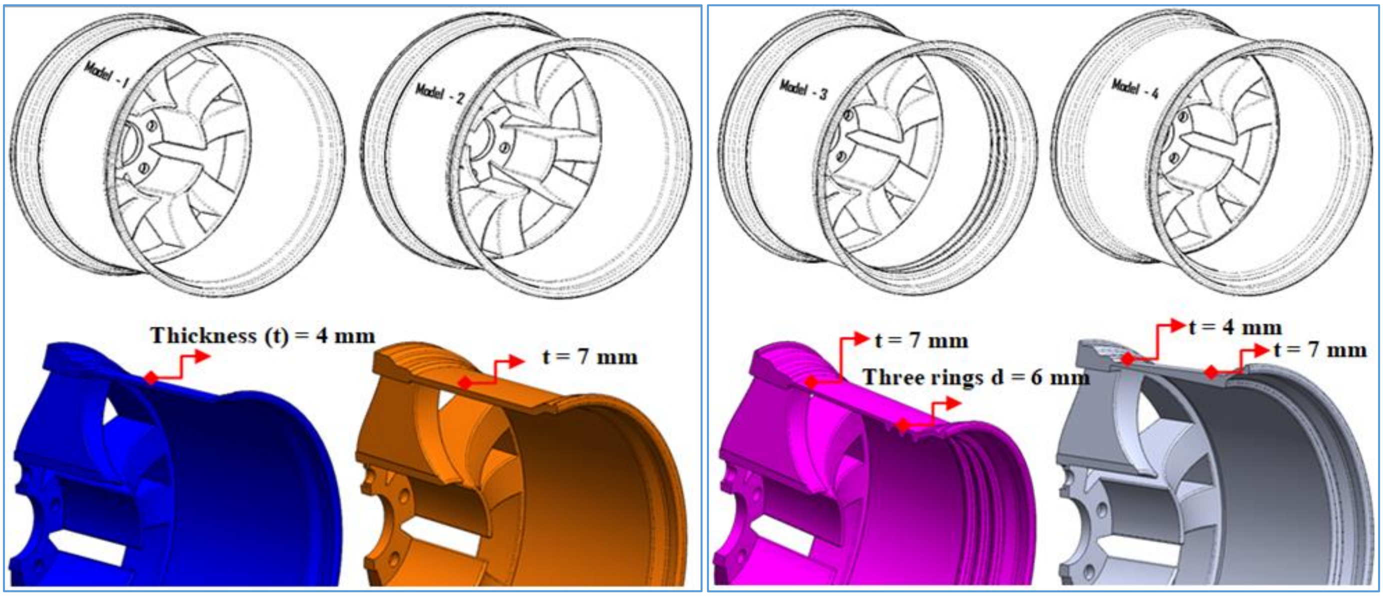

Figure 2. The dimensions and shapes of the four models

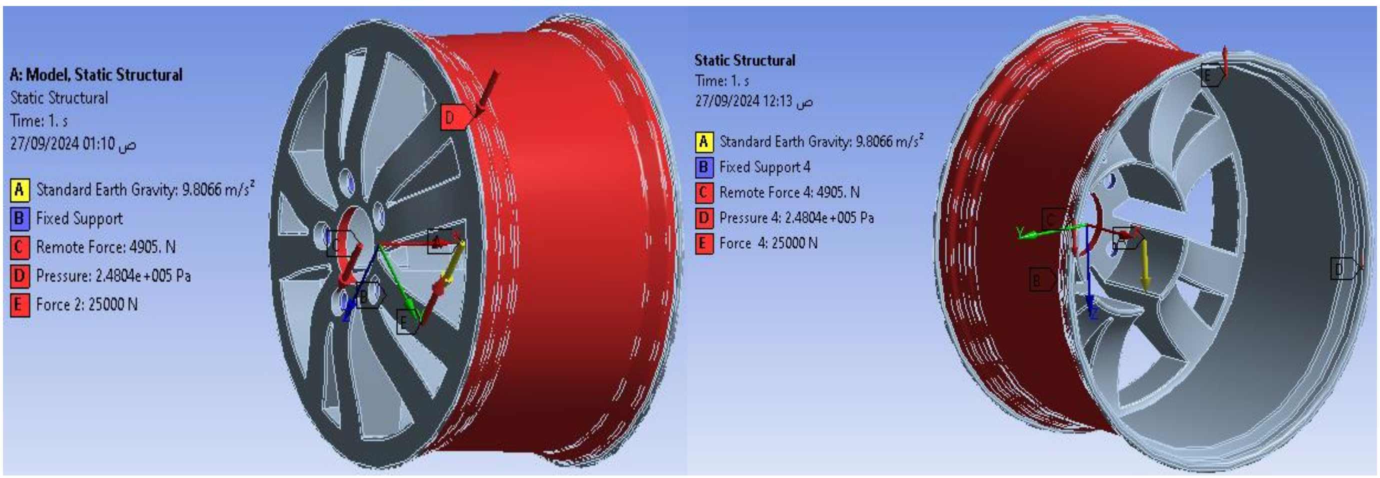

Figure 3. Type of force applied, its magnitude, and where it affects the wheel

4.3 Numerical modeling

In order to load the wheels with the loads they are exposed to and determine their resistance to those loads, four-wheel models were designed in Solid Works and then converted to an ANSYS program. The first model is an accurate representation of the wheel in use, while the other three have been altered from the original blueprint to reinforce the areas of weakness that are prone to breaking. Figure 2 depicts the dimensions and shapes of the four models that were employed in the present study.

4.4 Forces applied

Figure 3 illustrates the different kinds of forces acting on the wheel, along with their locations of influence and values across the four models. The identical loads were applied to each model in order to compare it with the original model, determine the degree of improvement, and determine the impact of the treatment type on the model's resistance to these loads in order to select the best model that can withstand them.

5. RESULTS AND DISCUSSION

Figure 4 shows that the maximum values of the three models are significantly lower than those of the first model, according to simulation results in the ANSYS program for model deformation. The resistance of the three models whose designs were improved over the first model was improved, as evidenced by the values of the models (9.3393, 4.6298, 3.777, and 4.8034 mm), with the third model having the lowest value.

The simulation results in ANSYS for the Von Mises stress models show that the maximum values of the three models are significantly lower than those of the first model (Figure 5). The model values (31778.1, 1987.6, 1013.1, and 1233 MPa) of the three models whose designs were optimized also demonstrated an improvement in strength over the first model; the third model had the lowest value (1013 MPa).

The maximum shear stresses models' ANSYS simulation results demonstrate that the maximum values of the three models are noticeably less than those of the first model (Figure 6). In the three models whose designs were optimized, the values (1816.5, 1043.1, 569.95, and 692.03 MPa) also showed an increase in strength over the first model; the third model had the lowest value (569.95 MPa).

A comparison of the four models' fatigue results for the safety factor is displayed in Figure 7. The safety factor values in the areas that were reinforced during the design of the three improved models are clearly improved, as the figure illustrates. In the regions where the safety factor was noticeably low, the figure also demonstrates an increase in the safety factor values. Their respective values were (0.027123, 0.043368, 0.085084, and 0.0691), and the third model out of all the models was the best one.

Figure 8 presents a comparative analysis of the fatigue damage outcomes from all four models. The three improved models' respective damage values (6.3183 × 107; 2.228 × 107; 4.3483 × 106; 7.1279 × 106) in the areas that were reinforced during design were significantly lower than the original model value, and the third model performed the best out of all the models.

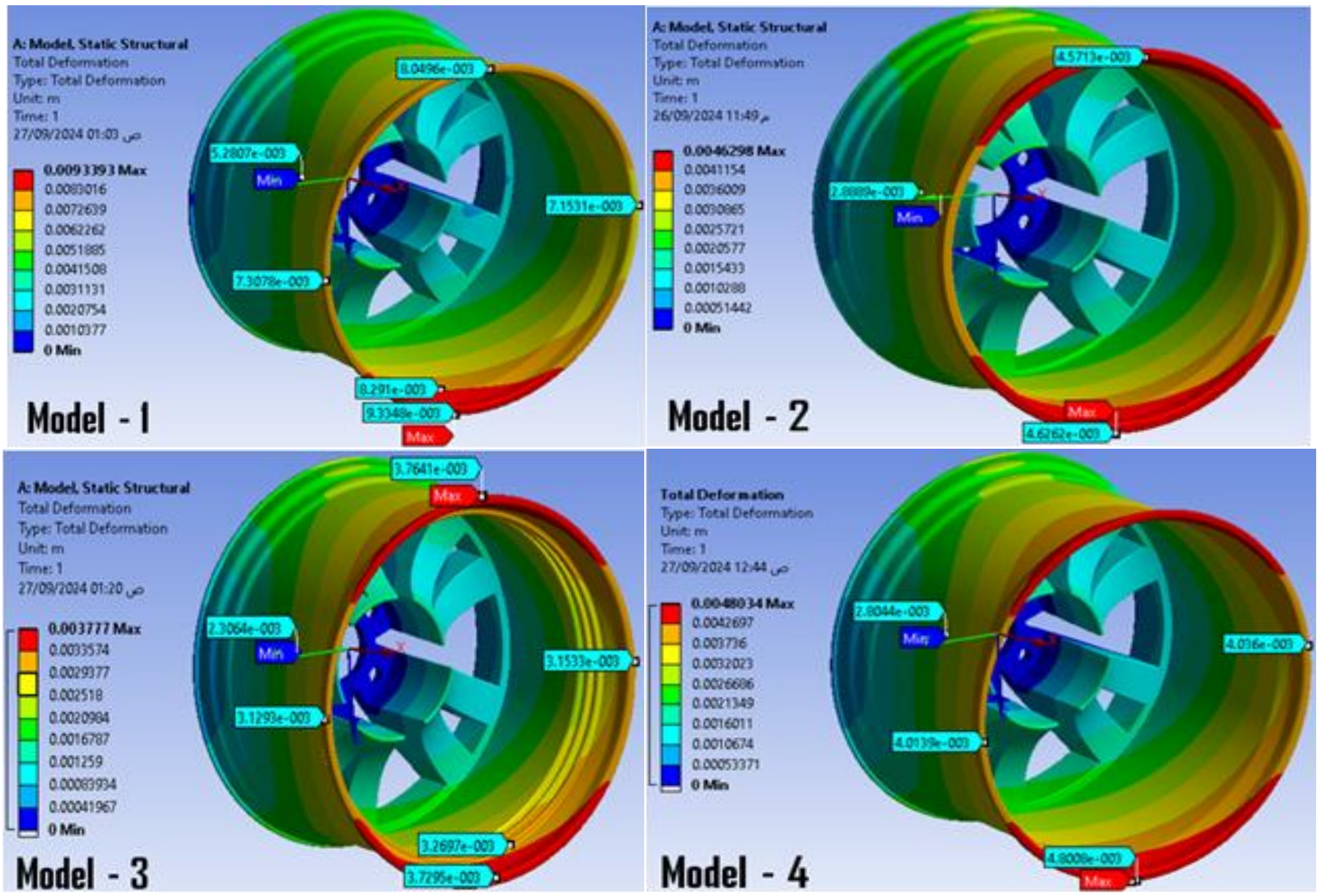

Figure 4. Compares the simulation deformation results of the four models in the ANSYS program

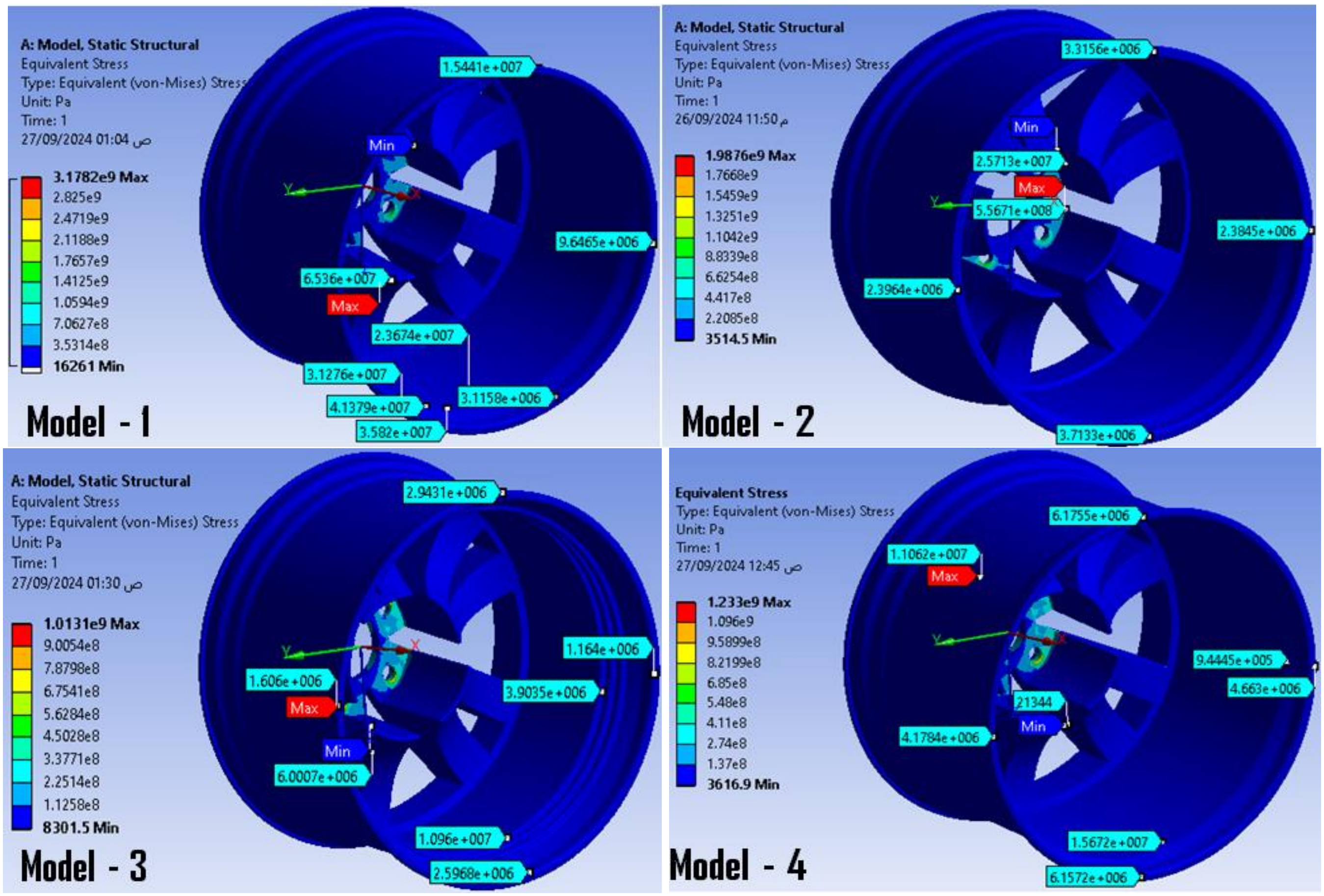

Figure 5. Comparison of the simulation Von Mises stress results of the four models in the ANSYS program

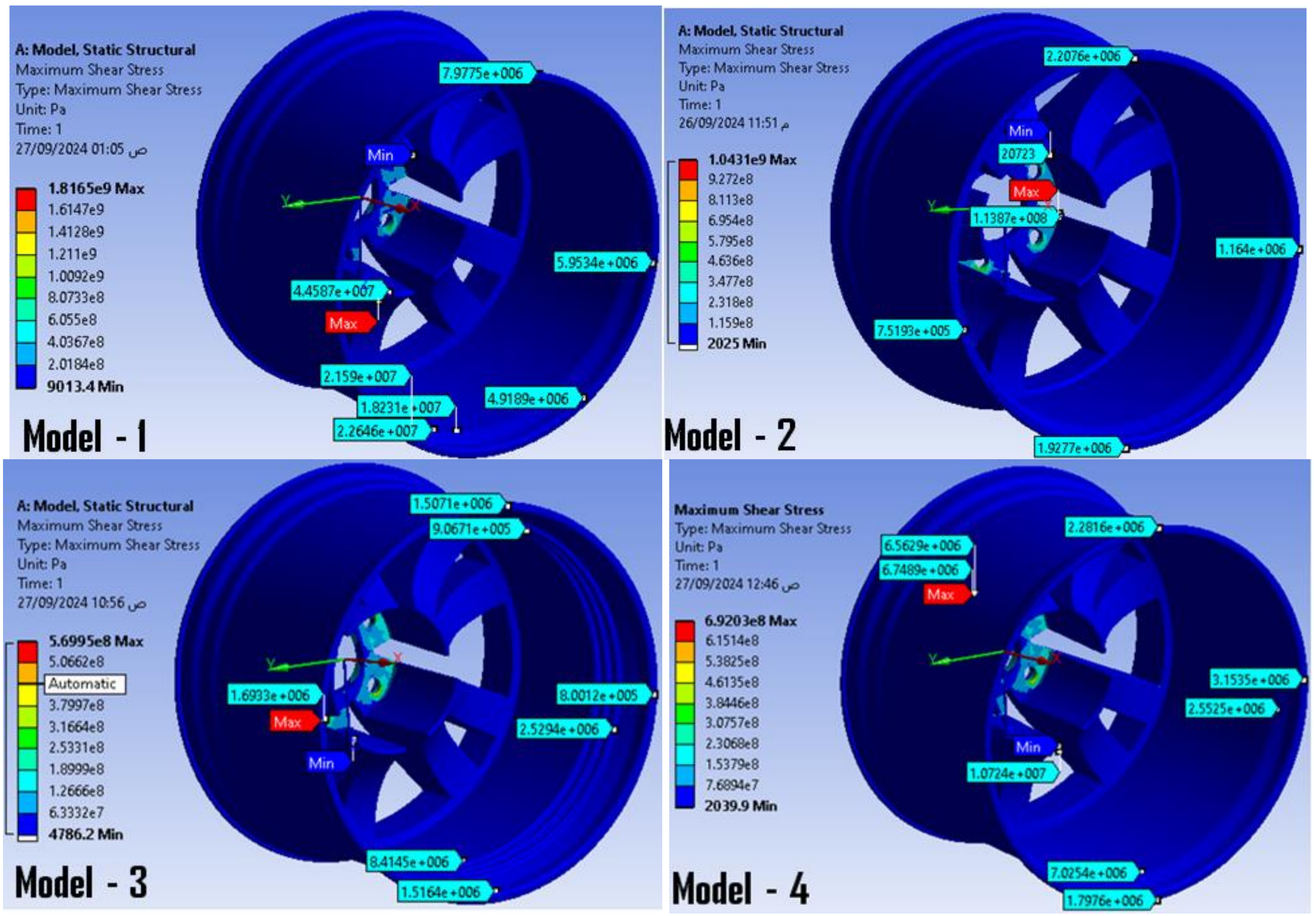

Figure 6. Comparison of the simulation maximum shear stress results of the four models in the ANSYS program

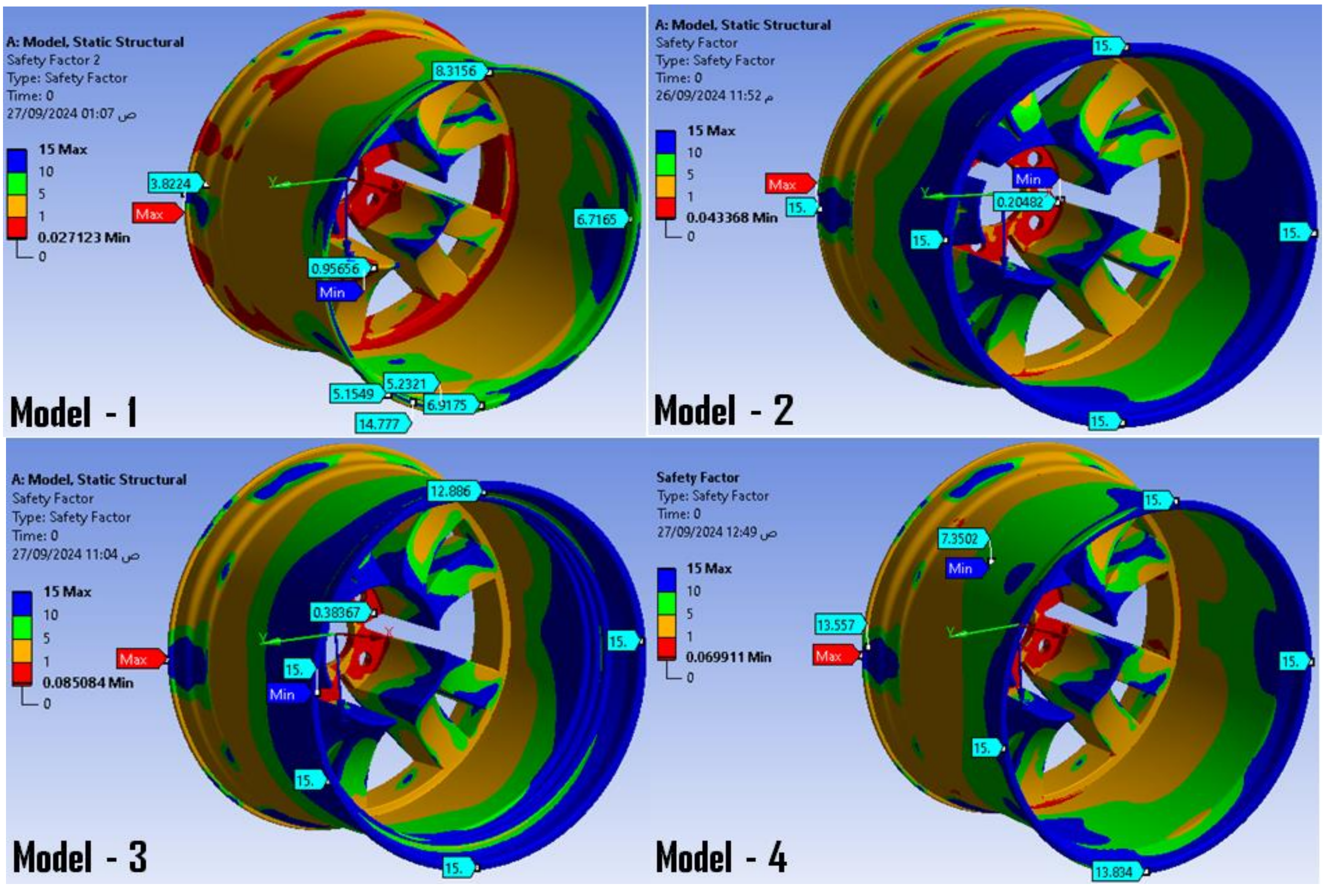

Figure 7. Comparison of the simulation safety factor results of the four models in the ANSYS program

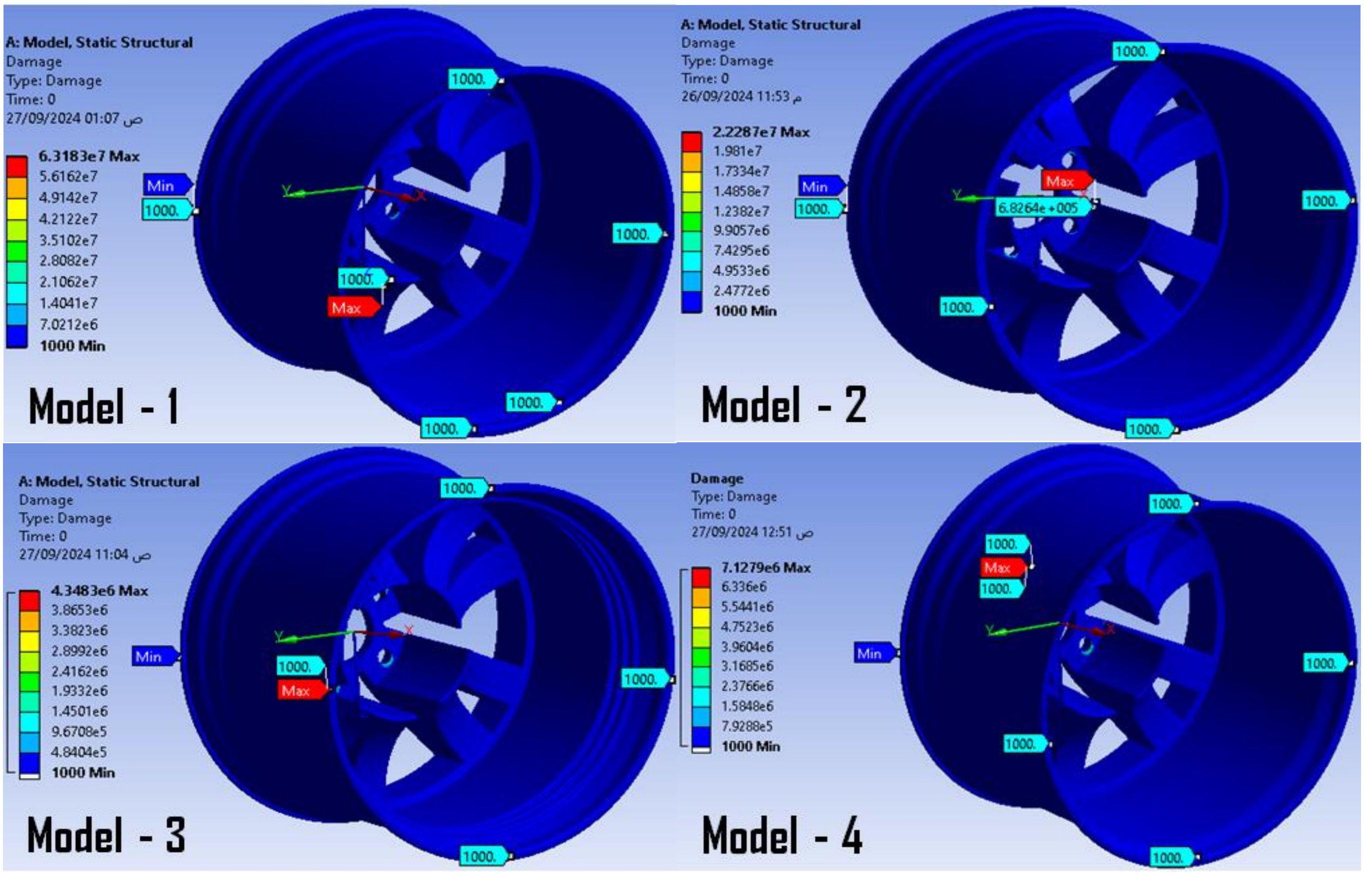

Figure 8. Comparison of the simulation damage results of the four models in the ANSYS program

6. CONCLUSION AND RECOMMENDATIONS

6.1 Conclusion

Following the creation of the four Solid Works models and refinement of the second, third, and fourth models. In order to apply various loads to the four models, Solid Works models were imported into ANSYS. Subsequently, simulation was performed to determine the various deformations and stresses that the models experience under load. We can draw the following conclusions from this study:

(1) The results show that the deformation models' values are substantially lower than the first model's. Accordingly, these percentages were attained (50.43, 59.56, and 48.57%), with the third model showing the biggest percentage drop (59.56%). This is due to the fact that sections that are exposed to strong, unexpected loads are reinforced while operating.

(2) The findings demonstrate that compared to the first model, the Von Mises stress models have noticeably lower values. The corresponding percentages were 37.46, 68.12, and 61.20%; the largest percentage drop (68.12%) was observed in the third model.

(3) The results show that the Maximum Shear stress models have values that are significantly lower than those of the first model. The corresponding percentages reached were 37.46, 68.12, and 61.20%, and the third model showed the biggest percentage decrease (61.20%).

(4) The fact that these percentages were reached (37.46, 68.12, and 60.75%), it can be concluded that the safety factor values in the three improved models in the design are significantly higher than the safety factor values in the first model. Because the areas subject to sudden heavy loads were strengthened during design, the third model had the highest percentage increase in safety factor, coming in at 60.75%.

(5) The fatigue damage values in the three improved models in the design are substantially lower than the fatigue damage values in the first model, as evidenced by the fact that these percentages (64.74, 93.12, 88.72%) were reached. The third model, with a safety factor increase of 93.12%, had the lowest percentage increase of any model because the areas susceptible to sudden heavy loads were strengthened during design.

6.2 Recommendations

(1) Suggest exploring the use of other materials or testing the models under different environmental conditions.

(2) Manufacture models according to the designed models and compare the experimental results of these models with the results obtained in this study.

(3) Comparatively analyze the theoretical and practical results after doing a practical investigation of the outcomes.

REFERENCES

[1] Prabhu, R., Natrayan, L., Kumar, M.S. (2017). Modeling and cyclic stress analysis of automotive alloy wheel rim. International Journal of Research, 4(17): 86-91.

[2] Adigio, E.M., Nangi, E.O. (2014). Computer aided design and simulation of radial fatigue test of automobile rim using ANSYS. IOSR Journal of Mechanical and Civil Engineering, 11(1): 68-73.

[3] Apelian. D. (2009). Aluminum Cast Alloys: Enabling Tool and Improved Performance. North American Die Casting Association, Wheeling, Illinois. https://aluminium-guide.com/wp-content/uploads/2019/05/WWR_AluminumCastAlloys.pdf.

[4] Srikanta, B.A., Veeraraju, P. (2014). Geometrical and material optimization of alloy wheel for four wheeler. International Journal of Research and Innovation, 1: 1401-1402.

[5] Natrayan, L., Santhakumar, P., Dinesh, P., Mohan, K.R., Scholar, U., Mohandass, R. (2016). Design and comparative analysis of old & new model car wheel rims with various materials. Journal for Research, 2(2): 69-73.

[6] Cheng, L., Nianzu, R., Yang, Q.S., Chao, C., Cheng, T., Liu, Z.G. (2020). Stress state measurement and result analysis of car wheels. IOP Conference Series Materials Science and Engineering, 784(1): 012021. https://doi.org/10.1088/1757-899X/784/1/012021

[7] Mahajan, P., Ishar, R., Prajapati, D.R. (2017). Static analysis of truck wheel rim using ANSYS software. International Journal of Management, IT and Engineering, 7(1): 104-121.

[8] Zheng, Y.Q., Li, B., Wang, Z.L. (2011). Dynamic simulation on impact test of vehicle wheel. In Advanced Engineering Forum. Trans Tech Publications, Ltd., pp. 890-893. https://doi.org/10.4028/www.scientific.net/AEF.2-3.890

[9] Zang, M.Y., Qin, T. (2012). Simulation analysis of car a-alloy wheel 13 impact test. Journal of Mechanical Engineering, 46(2): 83-87.

[10] Yang, L., Yang, H.B., Tan, H.J., Hu, P., Cheng, X.W., Lu, J.L. (2018). Impact test simulation and structural optimization of aluminum alloy A356.2 wheel hub. Journal of Physics: Conference Series, 1074(1): 012068. https://doi.org/10.1088/1742-6596/1074/1/012068

[11] Raju, P.R., Satyanarayana, B., Ramji, K., Babu, K.S. (2007). Evaluation of fatigue life of aluminum alloy wheels under radial loads. Engineering Failure Analysis, 14(5): 791-800. https://doi.org/10.1016/j.engfailanal.2006.11.028

[12] Ati, M.K. (2025). Managing smart cities by designing a communications system using LoRa technology. The Nexus of Sustainability and Energy Technology Journal, 1(1): 1-6.

[13] Aziz, M., Ramasamy, D., Kumaresan, R., Subramaniam, S. (2025). Investigation of heat transfer rate in automotive radiator with graphene nanofluid with simulation. The Nexus of Sustainability and Energy Technology Journal, 1(1): 26-39.

[14] Shehab, H.W., Mohammed, M.S., Abdullah, M.N. (2025). Design, modeling, and comprehensive analysis for the wing of an aircraft. NTU Journal of Engineering and Technology, 4(1): 60-70. https://doi.org/10.56286/txv7rh29

[15] Al-Hamdany, A., Mohammed, H.A. (2025). Seismic guardians: Redefining shear walls for unshakable residential structures. NTU Journal of Engineering and Technology, 4(1): 81-86. https://doi.org/10.56286/7vn1h352

[16] Zainuddin, H., Ali, M.B., Muhammad Said, N.M., Zakaria, K.A. (2016). Correlation between vertical wheel impact energy with lateral wheel impact energy: A finite element analysis approach. International Journal of Automotive and Mechanical Engineering, 13(3): 3574-

3583. https://doi.org/10.15282/ijame.13.3.2016.4.0294
[17] Chang, C.L., Yang, S.H. (2009). Simulation of wheel impact test using finite element method. Engineering Failure Analysis, 16(5): 1711-1719. https://doi.org/10.1016/j.engfailanal.2008.12.010
[18] Chang, C.L., Yang, S.H. (2008), Finite element simulation of wheel impact test. Journal of Achievements in Materials and Manufacturing Engineering, 28(2): 167-170.
[19] Qian, W.X., Wu, N.X., Li, H., Yin, X.W., Xie, L.Y. (2023). Simulation analysis and experimental study of the strength of aluminum alloy suspension structure. Materials, 16(1): 128. https://doi.org/10.3390/ma16010128
[20] Lian, Z.H., Li, M.J., Lu, W.C. (2022). Fatigue life prediction of aluminum alloy via knowledge-based machine learning. International Journal of Fatigue, 157: 106716. https://doi.org/10.1016/j.ijfatigue.2021.106716
[21] Shang, R., Altenhof, W., Hu, H., Li, N. (2005). Kinetic energy compensation of tire absence in numerical modeling of wheel impact testing. SAE Technical Paper No. 2005-01-1825. https://doi.org/10.4271/2005-01-1825
[22] Meshram, P., Sinha, A. (2018). Design & optimization of alloy wheel rim using ANSYS. Smart Moves Journal IJO-Science, 4(6): 5. https://doi.org/10.24113/ijoscience.v4i6.147
[23] Shekhar, M.C., Rao, B.L., Krishna, M.V.R. (2018). Design and structural analysis of car alloy wheel using with various materials. International Journal, 5(7): 114-120.
[24] Jiang, X., Lyu, R., Fukushima, Y., Otake, M., Ju, D.Y. (2018). Lightweight design and analysis of automobile wheel based on bending and radial loads. IOP Conference Series: Materials Science and Engineering, 372(1): 012048. https://doi.org/10.1088/1757-899X/372/1/012048
[25] Pandey, V.K., Choubey, A., Shrivastava, A.K. (2018). A review on study of design and analysis of car wheel rim using fem. International Journal of Advance Engineering and Research Development, 3(4): 128-131.
[26] Hebbal, M.S., Dabair, M. (2019). Static structural, modal and harmonic analysis of alloy car wheel rim using ANSYS workbench. International Journal of Engineering Research & Technology (IJERT), 8(7): 599-611.
[27] Satyanarayana, N., Sambaiah, C. (2012). Fatigue analysis of aluminum alloy wheel under radial load. International Journal of Mechanical and Industrial Engineering (IJMIE), 2(1): 1-6.
[28] Karash, E.T., Ali, H.M., Kassim, M.T.E. (2024). Designing cantilever models from various materials and comparing them when they are under constant load and have holes. Revue des Composites et des Materiaux Avances, 34(3): 363-377. https://doi.org/10.18280/rcma.340312
[29] Kassim, M.T.E., Karash, E.T., Mahmood, A.M., Sultan, J.N. (2024). Mechanical property enhancement of stainless steel 12X18H10T through nc-TiN coating: A simulation study. Annales de Chimie Science des Matériaux, 48(2): 163-176. https://doi.org/10.18280/acsm.480203
[30] Agarwal, A., Rimal, A., Ilunga, M. (2024). Non-linear contact analysis and response surface optimization of railway wheel using ANSYS. Mathematical Modelling of Engineering Problems, 11(8): 2039-2047. https://doi.org/10.18280/mmep.110805
[31] Ali, H.M., Najem, M.K., Karash, E.T., Sultan, J.N. (2023). Stress distribution in cantilever beams with different hole shapes: A numerical analysis. International Journal of Computational Methods and Experimental Measurements, 11(4): 205-219. https://doi.org/10.18280/ijcmem.110402
[32] Sediqer, T.A.A., Karash, E.T., Sultan, J.N., Najem, M.K. (2023). Finding the weight difference of a rectangular structure with a parabolic arc for mathematical models made of steel and other composite materials. WSEAS Transactions on Applied and Theoretical Mechanics, 18: 178-194. https://doi.org/10.37394/232011.2023.18.17
[33] Khawaskar, M. (2023). Comparative analysis of steel wheel rim & alloy wheel rim of a car with respective materials. International Journal for Research in Applied Science & Engineering Technology (IJRASET), 11(VI): 1959-1969. https://doi.org/10.22214/ijraset.2023.53992