WWW.IJRPR.COM

International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

A-Review Paper on Four Wheeler Wheel Rim

Mukesh Bamboriya[1], Dr. Satyendra Sharma[2]

[1] PG scholar, Department of Mechanical Engineering, SAGE University, Indore

[2] Associate Professor, Department of Mechanical Engineering, SAGE University, Indore

ABSTRACT

This work presents the design and analysis of an automotive wheel rim using software's. The wheel rim was designed using by different software's Catia, Pro-E and Solid work, and the model was imported into Ansys for further analysis. A test was carried out on the newly designed rim, and the results were used to identify areas of the rim that were under high stress. The design was then modified to improve the strength and durability of the rim. The results of the Ansys analysis were validated with experimental results.

The findings of this study show that most of the author use Ansys software used to effectively design and analyze automotive wheel rims. The software allows for a variety of analyses to be performed on the wheel rim model, including static analysis, fatigue analysis, and stress analysis. The results of analyses can be used to identify areas of the rim that are under high stress and make modifications to the design to improve its strength and durability.

Keywords: automotive wheel rim, Ansys software, design, analysis, fatigue test

1. Introduction

The wheel is a simple invention with a profound impact on human history. It has allowed for the transportation of goods and people over long distances, the development of new industries, and the advancement of civilization. The wheel is an essential part of modern transportation and is used in a variety of other applications. It is likely to continue to be an important part of our lives for many years to come. We can expect to see even more innovative designs for automotive wheels in the future. In the beloved section mainly focussed on past working methodology & result as well conclusion.

2. Literature Review

G. Ashokkumar [1] used CATIA to model a wheel rim and then imported the model into ANSYS for analysis. They conducted static analysis and modal analysis (a type of dynamic analysis) on the rim using two different materials: aluminum and forged steel. The results showed that forged steel was the better material for the wheel rim because it had a higher strength-to-weight ratio and was more resistant to fatigue than aluminum.

The use of ANSYS to analyze wheel rims was a valuable tool for engineers because it allowed them to test different designs and materials without having to build physical prototypes. This saved time and money, and it also helped to ensure that the final product was safe and reliable.

Yuwana Sanjaya [2] conducted a finite element analysis of a wheel rim to determine the effect of meshing size on the results. They used a vehicle wheel made of steel as their test case and ran the simulation using three different mesh sizes: 10 mm, 15 mm, and 20 mm.The results showed that the von Mises stress and displacement increased as the mesh size decreased. The error ratio of displacement was 0.011, 0.043, and 0.062 for the 10 mm, 15 mm, and 20 mm mesh sizes, respectively. The error ratio of stress was -0.174, 0.081, and 0.064 for the same mesh sizes. The authors concluded that the 10 mm mesh size was the most accurate, but it also required the most computational resources. The 15 mm mesh size was a good compromise between accuracy and computational cost.

S. Belodedenko [3] the reliability and durability of wheel rims were important for the energy efficiency of motor vehicles. One way to improve the reliability of wheel rims was to use optimized low-pearlite steels with increased impact strength. When determining the mechanical characteristics of these steels, it was important to consider mixed failure, which was done using a four-point asymmetric bending scheme. Conditions of mixed failure for an oblique crack led to a 25–45% decrease in the value of ΔKI relative to the stress intensity factor ΔKI obtained for the pure I mode. The same could be said about the II mode, when ΔKII determined during the growth of an oblique crack was 10% smaller than the value of ΔKII, which was calculated for the pure mode of failure. This indicated an increase in the fatigue crack growth rate for mixed failure compared to pure modes.

Yanchao Jianget [4] the authors of the paper studied the effect of die structure design on the forming of magnesium (Mg) alloy wheels by backward extrusion (BE). They found that the current design of Mg alloy wheels often led to upper rim cracking and coarse microstructure at the spokes, which reduced the yield. They proposed a new die structure design that included four parameters: increasing the spoke thickness, decreasing the spoke inclination angle, increasing the punch fillet radius, and increasing the inner fillet radius of the upper rim. They used numerical simulation to analyse the effect of these parameters on the forming process. They also conducted an experimental trial to verify the accuracy of their results. The results showed that the new die structure design was effective in improving the forming of Mg alloy wheels. The increased spoke thickness and decreased spoke inclination angle improved the metal flow and made the billet filling more smoothly. The increased punch fillet radius and increased inner fillet radius of the upper rim relieved the stress concentration at the window and made the filling at the upper rim complete. The macroscopic tearing phenomenon disappeared. The authors concluded that the new die structure design could effectively improve the forming of Mg alloy wheels and reduce the defects. This would lead to an increase in the yield of the wheels.

Yatong Zhual [5] the author of the paper studied the forming of magnesium (Mg) alloy wheels by single-step back extrusion. They found that the current process often led to coarse grains at the spokes, insufficient recrystallization, and easy cracking at the upper rim. They proposed a new billet design that could help to improve the forming process. The new billet design was a truncated cone billet. This billet design had a larger cross-section at the bottom and a smaller cross-section at the top. This design had helped to improve the flow of material during the forming process. It had also helped to increase the effective strain, dynamic recrystallize volume fraction, and refine grains at the spokes and the lower rim. The author conducted a numerical simulation to analyse the effect of the new billet design on the forming process. They also conducted an experimental trial to verify the accuracy of their results. The results had shown that the new billet design was effective in improving the forming of Mg alloy wheels. The truncated cone billet had helped to reduce the cracking tendency at the upper rim and improve the fluidity at the rim. This had made the rim filling easier and resulted in a more uniform microstructure. The author concluded that the truncated cone billet was of great significance to the back extrusion forming of Mg alloy wheels. It was a promising new design that had helped to improve the quality and performance of Mg alloy wheels.

S. Karuppusamy [6] the authors of the paper used finite element analysis (FEA) to improve the design of automotive wheel rims. They used SOLIDWORKS and ANSYS to analyse the static and fatigue performance of four different materials for wheel rims: steel alloy, aluminium alloy, magnesium alloy, and forged steel. The results of the analysis showed that steel alloy was the best material for wheel rims, as it had the highest critical stress and number of cycles to failure. The article also highlighted the importance of considering both static and fatigue loads when designing wheel rims. Static loads were the forces that were applied to the rim when the vehicle was stationary, while fatigue loads were the forces that were applied to the rim as the vehicle was moving. Fatigue loads could cause the rim to crack and eventually fail, so it was important to design the rim to be able to withstand these loads. The article provided valuable insights into the design of automotive wheel rims. The results of the analysis could be used to improve the design of wheel rims and make them more durable and reliable

M. Borecki et al [7] a new method for identifying the technical condition of wheel rims. The method used forced vibration spectra measurement and data processing with neural network use. The authors integrated the existing wheel balancer with a computer, shaft actuator, vibration exciter, and accelerometer head. The developed software automated the measurement procedure and results classification. The algorithm of data processing enabled the proper detection of the three states of rim: new, used, and fit for further use, and not usable. The article was limited in that it was only tested on a small number of wheel rims. The authors also did not provide any information on the accuracy of the method. Additionally, the method was not yet commercially available. Despite these limitations, the article was a valuable contribution to the field of automotive engineering. The proposed method had the potential to improve the safety and reliability of vehicles by helping to identify defective wheel rims.

Amarnath [8] the author focused on the design and analysis of alloy wheel rims using ANSYS. The project began by discussing the importance of wheels in the automobile industry. Wheels were classified as a safety critical component, and international codes and criteria were used to design them. The project then discussed the two main methods for structural analysis of wheels: strain gauge and finite element method. The project then went on to analyze different types of wheels and their functions. The project was well-written and well-organized. The authors did a good job of explaining the different concepts involved in the design and analysis of alloy wheel rims. The project also included a number of useful diagrams and illustrations.

3. Conclusion

a comprehensive overview of the current state of research on wheel rims. The papers highlight the importance of using high-quality materials for wheel rims, such as forged steel. They also emphasize the importance of considering both static and fatigue loads when designing wheel rims. Additionally, the papers discuss the use of finite element analysis (FEA) to improve the design of wheel rims. The authors also discuss a new method for identifying the technical condition of wheel rims. This method uses forced vibration spectra measurement and data processing with neural network use. The method was found to be able to correctly identify three states of rim: new, used, and fit for further use, and not usable. Overall, the papers provide valuable insights into the design, analysis, and testing of wheel rims. The findings of the papers could be used to improve the safety and reliability of wheel rims.

References

[1] G.Ashokkumar Design and Analysis of Wheel Rim by Using Catia & Ansys Volume: 03 Issue: 12 | Dec -2016 www.irjet.net

[2] Yuwana Sanjaya Design and Analysis of Mesh Size Subjected to Wheel Rim Convergence Using Finite Element Method IGF26 - 26th International Conference on Fracture and Structural Integrity 2452-3216 © 2021 The Authors. Published by Elsevier B.V

[3] S. Belodedenko Development of a methodology for mechanical testing of steel samples for predicting the durability of vehicle wheel rims Available online 24 April 2023 journal homepage: www.sciencedirect.com/journal/results-in-engineering

[4] Yanchao Jiang Die structure optimization study for magnesium alloy wheel formed by backward extrusion Available online at www.sciencedirect.com Available online 14 February 2023

[5] Yanchao Jiang, Yatong Zhu, Qichi Le, Qiyu Liao, Weiyang Zhou, Ping Wang, Tong Wang, Effect of truncated cone billet on single-step back extrusion forming process of magnesium alloy wheel, Journal of Materials Research and Technology, Volume 20,2022,

[6] Karuppusamy, Selvaraj & Karthikeyan, G. & Dinesh, s & Rajkumar, T. & Vijayan, Venkatraman & Basha, Kalil. (2016). Design and Analysis of Automotive Wheel Rim by using ANSYS and MSC Fatigue Software. Asian Journal of Research in Social Sciences and Humanities. 6. 196. 10.5958/2249-7315.2016.01007.8.

[7] M. Borecki, A. Rychlik, O. Vrublevskyi, A. Olejnik, M.L. Korwin-Pawlowski,Method of Non-Invasive determination of wheel rim technical condition using vibration measurement and artificial neural network,Measurement,Volume 185, 2021,

[8] Guojiang Dong, Shide Li, Shaozhong Ma, Dongsheng Zhang, Jiang Bi, Ji Wang, Mikhail Dmitrievich Starostenkov, Zuo Xu, Process optimization of A356 aluminum alloy wheel hub fabricated by low-pressure die casting with simulation and experimental coupling methods, Journal of Materials Research and Technology, Volume 24, 2023,

[9] Mr. M.A.Amarnath Design And Analysis Of Alloy Wheel Rim 2020 IJRAR March 2020, Volume 7, Issue 1

[10] Jitendra Shinde Review Paper on Design and Analysis of Automotive Wheel Rim Using Finite Element Analysis Volume: 04 Issue: 07 | July -2017 www.irjet.net

[11] Ahmad, M., Ismail, K.A., Mat, F., 2013. Convergence of finite element model for crushing of a conical thin-walled tube. Procedia Engineering 53, 586-593.