WWW.IJRPR.COM

International Journal of Research Publication and Reviews

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

Investigating Stress Distribution in Alpha Wheel Through Material Variation Using Ansys

Laleeta Dawar[1], [2]Prof Ashish Patidar

[1] PG scholar, Department of Mechanical Engineering, Patel College, Indore
[2] Assistant Professor, Department of Mechanical Engineering, Patel College, Indore

ABSTRACT

In response to worries about pollution and health, bicycles have become a well-liked option for eco-friendly transportation. Numerous studies have been conducted with the goal of improving cycling experiences, which has resulted in the creation of specialty bicycles including Urban Transportation Bikes, Mountain Bikes, and Road Bikes. This article discusses the idea of the (Alpha) Wheel, also known as the loop wheel, a cutting-edge construction that directly integrates a suspension mechanism into the wheel frame. The (Alpha) Wheel's main goal is to make cycling more comfortable and to reduce road vibrations.

In the present investigation ANSYS workbench 19.2 are used to investigate wheels made of an aluminium alloys, steel for structure and cast iron (Grey). This research tries to calculate the wheel forces that are responsible for maximum deflection and identify the primary stresses. Also in this research work some point will be discussed for the future design and improvement in the functioning of the wheel.

Keywords- ANSYS, Bicycle, Bikes, Vibration, Suspension, Composite material.

1. INTRODUCTION

The Alpha wheel, a crucial part of contemporary engineering systems, experiences considerable mechanical loads when in use. For the Alpha wheel to be designed optimally, the right materials must be chosen, and its performance must be safe and effective, an accurate assessment of the Alpha wheel's stress distribution is necessary. In this study, we use ANSYS, a potent finite element analysis (FEA) tool, to explore the stress distribution in the Alpha wheel. Stress distribution analysis is an essential component of wheel design since it has a direct bearing on the toughness and safety of the wheel. Understanding the stress patterns and spotting areas of concentrated high stress are essential to avoiding early failure, extending the life of the wheel, and ultimately ensuring the safety of the car and its occupants.

Using cutting-edge FEA methods applied in the widely used program ANSYS, the current study intends to evaluate the stress distribution in the Alpha wheel. This study aims to offer useful insights into improving the wheel's stress distribution by investigating the impact of material variation on stress distribution.

2. PROBLEM IDENTIFICATION

The current issue is utilizing ANSYS to examine the stress distribution in an Alpha wheel through material modification. Most commonly, an Alpha wheel is a particular kind of wheel utilized in a mechanical or technical application. Understanding how changes in the Alpha wheel's material qualities affect its stress distribution under specific loading conditions is the main objective of this work. Engineers and designers can improve the wheel's design for a variety of applications with the use of this study, which can offer useful insights into the wheel's structural integrity and performance.

3. OBJECTIVES

The main objectives of this research work are as follows

- Create an in-depth and precise 3D CAD model of the Alpha wheel that includes all pertinent geometrical elements, measurements, and mounting interfaces.
- Using ANSYS software, build a finite element model of the Alpha wheel, making sure the discretization and mesh quality are accurate.
- Create believable boundary conditions, loading situations, and limits that mimic the Alpha wheel's actual operating conditions.
- Investigate the stress distribution over the Alpha wheel's surface by running FEA simulations for every material change.

- Give design suggestions based on the stress distribution analysis to improve the structural performance of the Alpha wheel.
- Verify the accuracy of the FEA results by contrasting them, as necessary, with experimental data or analytical solutions.

4. METHODOLOGY

The methodology to be used depends on how we address a certain scenario and the circumstances under which the study is conducted. There could be different methods for the identical research.

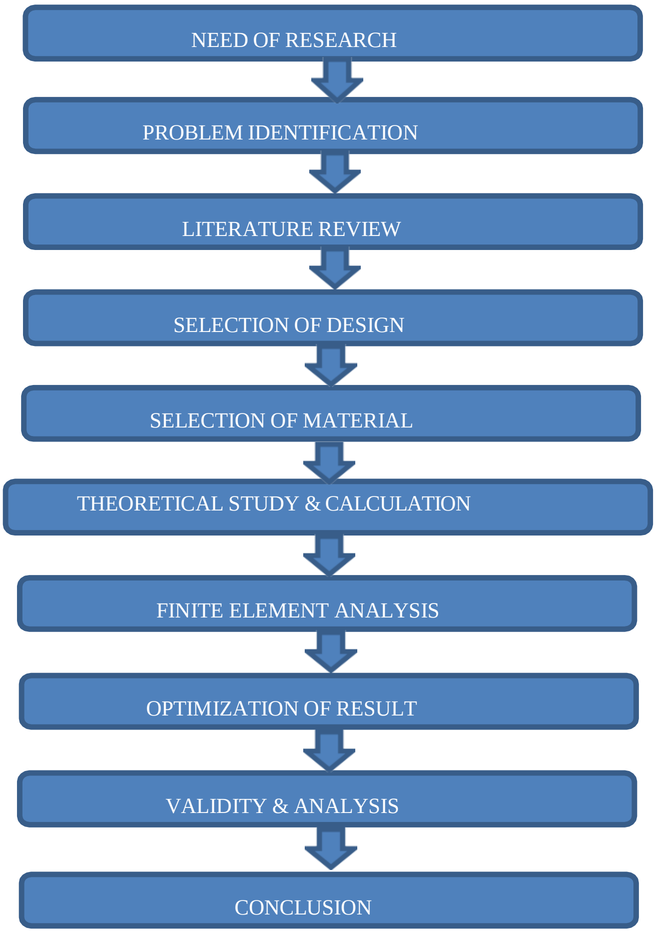

Figure 4.1 Methodology

5. RESULTS

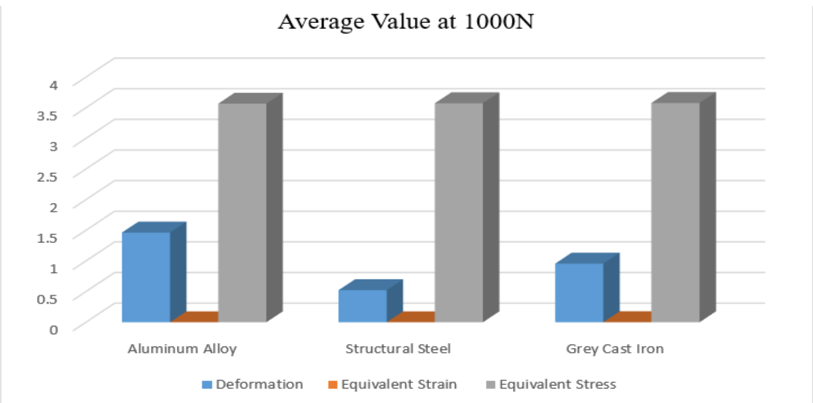

Figure 5.1 Values of deformation, Equivalent strain, stress Comparison at 1000 N

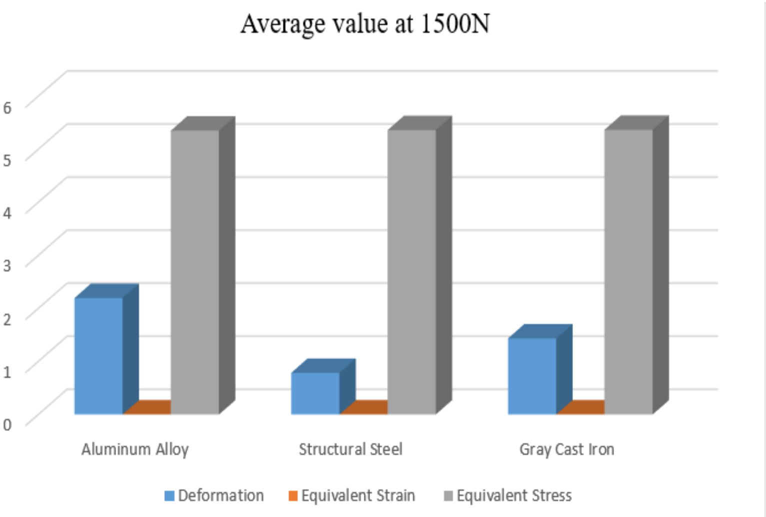

Figure 5.2 Values of deformation, Equivalent strain, stress Comparison at 1500 N

6. CONCLUSION

Based on the analysis, the following conclusions can be drawn:

Aluminum Alloy: The aluminum alloy suspension wheel has advantageous qualities such a high strength-to-weight ratio, low weight, and superior corrosion resistance. Under the applied loads, it demonstrated adequate rigidity and durability. To prevent excessive stress concentrations in crucial locations, caution must be exercised.

Grey Cast Iron: Grey cast iron suspension wheels have great strength and excellent damping characteristics, making them appropriate for absorbing shocks and vibrations. It offered decent durability and stability. However, it is crucial to take into account its heavier weight in comparison to other materials and probable corrosion issues.

Structural Steel: The structural steel suspension wheel provided a compromise between strength, durability, and weight. It demonstrated good load-bearing capacity and deformation resistance. Engineering uses for structural steel are numerous, and it is a dependable material for suspension systems.

7. REFERENCES

[1] Gaurav Agrawal, Vipin Chaudhary, Sudhir Singh Bisht, "Finite Element Analysis of a Passenger Car Wheel Rim," International Journal of Engineering Science and Computing, Vol. 6, Issue 8, August 2016, PP 11873-11880.

[2] Rajesh Verma, Amar Pal Singh, Ravi Prakash Dwivedi, "Finite Element Analysis of the Front Wheel Rim of a Passenger Car," International Journal of Engineering Research & Technology, Vol. 3, Issue 6, June 2014, PP 1353-1357.

[3] Rajesh Kumar, Prabhat Kumar, "Finite Element Analysis of a Commercial Vehicle Wheel Rim," International Journal of Engineering Research & Technology, Vol. 2, Issue 10, October 2013, PP 533-537.

[4] Shabir Ahmad, Sheikh Bilal Hassan, M. Ishaq Dar, "Finite Element Analysis of Car Wheel Rim Made of Different Materials," IOSR Journal of Mechanical and Civil Engineering, Vol. 16, Issue 2, March-April 2019, PP 26-30.

[5] K. Gowtham, S. M. Gowtham, K. G. Vignesh, "Finite Element Analysis of a Truck Wheel Rim," International Journal of Engineering Research & Technology, Vol. 7, Issue 12, December 2018, PP 176-181.

[6] Anil Kumar Meena, Dushyant Kumar, "Stress Analysis of a Truck Wheel Rim Using Finite Element Method," International Journal of Engineering Research & Technology, Vol. 4, Issue 16, August 2015, PP 243-246.

[7] Narender Pal Arya, Jaspal Singh, "Stress Analysis of a Two-Wheeler Alloy Wheel Using Finite Element Analysis," International Journal of Engineering Research & Technology, Vol. 4, Issue 19, November 2015, PP 331-335.

[8] Daniel F. Bailey, David G. Schaeffer, David M. Cairns, "Finite element analysis of the mechanical performance of a carbon fiber bicycle wheel," Sports Engineering, Vol. 10, Issue 3, September 2007, PP 145-154.

[9] Abhishek Kulkarni, Suryakant Shinde, Pravin Kamde, "Analysis of a Composite Wheel Rim," International Journal of Innovative Research in Science, Engineering, and Technology, Vol. 3, Issue 7, July 2014, PP 14849-14853.

[10] Edoardo Buzzi, Gianni Cervi, Gianluca De Maria, "Structural Analysis of an Electric Bicycle Wheel: Finite Element Modeling and Experimental Validation," Advances in Mechanical Engineering, Vol. 10, Issue 3, March 2018, PP 1-12.

[11] Dongchoul Kim, Chang-Gyun Kim, Seung Eock Kim, "Effect of Material and Structural Parameters on the Mechanical Behavior of Bicycle Rims under Impact Loading," Materials, Vol. 11, Issue 7, July 2018, PP 1183.

[12] Nicola Petrone, Federico Giuliano, "Finite Element Analysis of Racing Bicycle Wheels: A Preliminary Study," Applied Sciences, Vol. 9, Issue 5, May 2019, PP 968.

[13] Adam Karchner, "Design and Stress Analysis of Composite Wheels for Formula SAE Vehicles," SAE Technical Paper 2018-01-1116, April 2018.

[14] Enrico Bertocchi, Paolo Valente, Marco Bocciolone, "Non-Linear Modeling of the Wheel/Rail Contact for the Stress Analysis of Railway Vehicle Wheels," Proceedings of the Institution of Mechanical Engineers, Part F: Journal of Rail and Rapid Transit, Vol. 234, Issue 5, July 2020, PP 529-542.