International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 11 Issue: 04 | Apr 2024 www.irjet.net p-ISSN: 2395-0072

Structural Optimization of Wheel Rim by Using Finite Element Analysis

[1]Mr.Shubham Choudhari, [1]Mr.Ganesh Takmoge, [1]Miss.Chandani Raut, [1]Mr.Kiran Pingale, [1]Miss.Nisha Ghate

[1]B.E Students, Dept. of Mechanical Engineering, Jayawantrao Sawant College of Engineering, Hadapsar Pune, Maharashtra.

Abstract - This research explores the application of Finite Element Analysis (FEA) software in optimizing the structural design of a wheel rim for enhanced performance and durability. The study aims to address the need for light weight yet robust wheel rims to improve vehicle efficiency and safety. In this study the optimization is done by modifying the design and using different material which plays significant role in weight reduction. FEA is used for investigation the stress and deformations which occurs in the wheel rim. This comparative study helps us to analyze the optimal rim without affecting its strength.

***Key Words*: Finite Element Analysis, Optimization, Deformation, Efficiency & Safety.**

1. Introduction

The structural optimization of wheel rims is a critical endeavor in the automotive industry, aiming to enhance the performance, safety, and efficiency of vehicles. Finite Element Analysis (FEA) has emerged as an indispensable tool in this pursuit, enabling engineers to simulate and evaluate the mechanical behavior of wheel rims under various loading conditions.

This process not only facilitates the identification of weaknesses in existing designs but also empowers engineers to iteratively refine & optimize wheel rim structures to meet the demanding requirements of modern vehicles. In this study, we delve into the realm of structural optimization of wheel rims using FEA, exploring the methods, challenges, and benefits associated with this innovative approach in the pursuit of safer, more durable, & high-performance wheels for the automotive industry.

1.1 Literature Review

Nowadays optimization has become crucial due increase in global demand of mechanical components. While doing the optimization of components the economical factor must be satisfied. This Research revolves around optimization of wheel rim. This can be achieved by using modeling software.

Dr. B. Venkata Shiva, Y. Vara Prasad Ram [1] et.al have considered Al7068 material due to its strength to weight ratio, this study examines the wheel rim of Kia Carnival Prestige vehicles using CAD/CAE tools. This research identifies Al-7075 and Al-7068 materials as suitable for structural weight, good natural frequency range, and high safety factor values. This could lead to reduced fuel consumption and improved vehicle performance. The meshing process converts the wheel rim into small particles using elements and nodes, providing more accurate results for real-time applications. The vehicle's speed is 150kmp/h, and the 3members load is 24271N. The best material was selected for its ability to withstand high speed and maximum boundary load conditions, considering 3 to 5- and 7-members' loads.

Rajat Yadav, Kamal Sharma [2] et.al this research paper discusses the importance of wheels in vehicle performance, focusing on alloy wheels that reduce vehicle weight. The study uses CATIA software to design and optimize wheel hubs using finite element analysis on three composite materials: magnesium, aluminum, and steel. The research was conducted in two phases, starting with fatigue analysis and design and optimization analysis. The analysis also included stress strain, parameters, car rim geometry, and loading conditions. Cad models were created using CATIA v5 tools, and finite element analysis was used to find variations in complex regions. The results showed that Mg alloy has less stress compared to other materials, optimizing wheel design for vehicle load, increasing life expectancy, flexibility, and efficiency. Reducing rim weight also improved overall efficiency.

Karikalan Loganathan, S.K. Vijaya Siva Subramani [3] et.al this research aims to optimize the wheel rim of a vehicle by using aluminum alloy Al6061-T4 and magnesium alloy AZ80. The existing design is being studied and optimized to achieve a weight reduction of nearly 58.33% without compromising safety parameters. The cost of aluminum alloy is found to be lower than magnesium alloy. The research begins by identifying the need for the project and collecting data on future research scope. A preliminary design is created and analyzed using SolidWorks 2021 software. Aluminum 6061 is chosen for the wheel rim due to its lower cost and safety factor of 2.09 compared to magnesium alloy AZ80's high factor of 4.03. ANSYS 2021 R1 Academic is used for analysis, supporting the ongoing development of innovative technology and a process-centric approach to design.

1.2 Objectives of Project

The objectives of the Project are as follow:

- To improve the existing structure for better performance.
- Reduce weight of wheel rim by modifying its design.
- Comparison of wheel rim by using different alloy materials.

2. Methodology

There are number of cars which are still using solid wheel rim. It has high weight in their respective rim segment. High weight means high mass due to which the cost requires to manufacture rim increases; hence a typical solid wheel rim of Hyundai Santro Car has taken for Analysis.

Solid rim CAD Model was prepared in Creo Parametric 4.0 and the analysis is done by giving suitable constrains and boundary conditions. Comparison between various metal alloys is done in ANSYS Workbench Software. This comparison helped to obtain suitable Alloy. The project work included following steps,

1) Study of literature Review related to Finite Element Analysis.

2) The actual dimensions of wheel rim were measured and cross-verified from web.

3) CAD model of wheel rim is created with the help of Sketch, Mirror, Revolve, Extrude, Pattern, and Chamfer command etc. From Creo Parametric 4.0

4) The .igs file is exported from Creo Parametric 4.0

5) For FE Analysis, the rim .igs file is imported in ANSYS Workbench Software & following boundary and loading conditions are applied on rim geometry.

i. the fixed support is applied at nut-bolt holes.
ii. remote location force is applied over the circumference of rim.
iii. rotational velocity is given at central drive shaft.

6) Results are obtained by running model design in Solver environment.

7) Comparison of various Alloys are done to get optimum alloy for wheel.

2.1 Assumptions & Velocity, Loading Calculations

Following assumptions are made for the analysis:

Car Weight – 935 Kg

Passenger Weight – 400 Kg (assuming 4 adults, child)

Luggage – 65 Kg

Car Accessories – 100 Kg

Total Weight – 1500 Kg

Wheel rims will experience 1500 Kg of total weight, which equivalent to 14709.97N.

Each wheel of car will bear 3677.49N of Load.

Assuming speed – 60 Km/Hr.

Linear velocity – 16.66 m/s

Angular velocity,

W = V/R =16.66/0.19 = 87.68 Rad/s

2.2 Rim Nomenclature

2D Diagram of old & new rim,

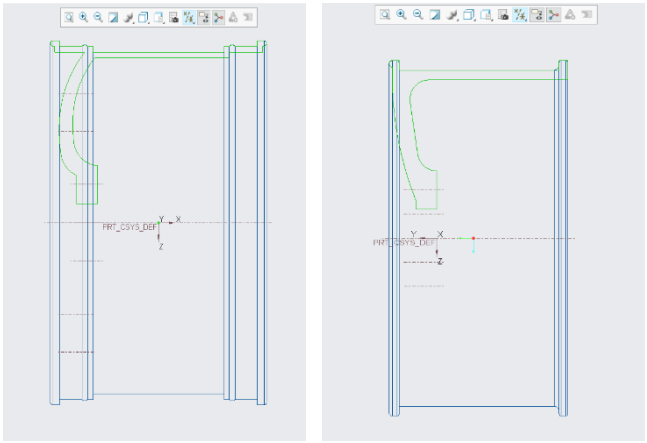

Fig.1 CAD Model of Wheel Rim

Table 1. Dimensions of Wheel Rim

Sr.No	Specification	Old Rim	New Rim
1	Rim Width	174	170
2	Wheel Diameter	370	370
3	Rim thickness	6	8
4	Bolt diameter	20	20
5	Number of bolt hole	4	6

2.3 Material Selection

For comparative study following materials were taken. Properties of materials are mentioned below.

Table 2. Properties of Material

Sr. No	Material Name	Density (Kg/m3)	Youngs Modulas (Gpa)	Poisson's Ratio
1	Al Alloy	2770	69	0.31
2	Mg Alloy	1800	44.8	0.35
3	Titanium Alloy	4400	114	0.32

3. Finite Element Analysis

The meshing is done after importing the .igs file. The type of meshed element used is tetrahedron and coarse mesh is generated in the critical regions.

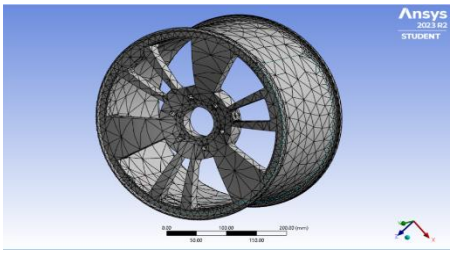

Fig. 2 Meshed Model of optimized Rim

As The bolts holes and shaft hole will be directly connected rotating shaft, they are considered for fixed support as shown in image. Fixed support is applied at six holes for old rim and four holes for new rim.

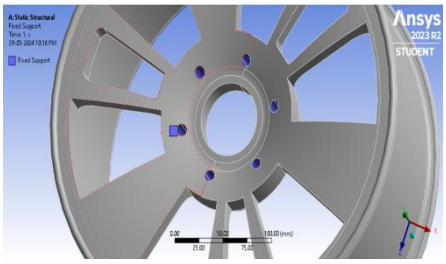

Fig. 3 Boundary Condition (Fixed support)

The rotational velocity is given at the central axis about which the wheel is going to rotate. The rotating action causes centrifugal force in the wheel structure.

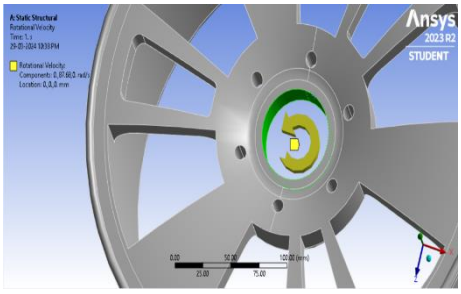

Fig. 4 Boundary Condition (Rotational Velocity)

The remote force will be applied on external circumference of rim as shown in image. The results are evaluated by combing all forces together.

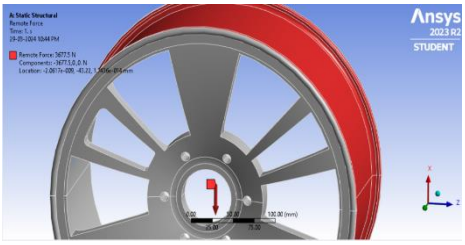

Fig. 5 Boundary Condition (Remote Foce)

3.1 Total Deformation and Stress Results

The stress and deformation for each wheel rim is calculated. Figure mentioned below shows the results of aluminum alloy for both old and new rim.

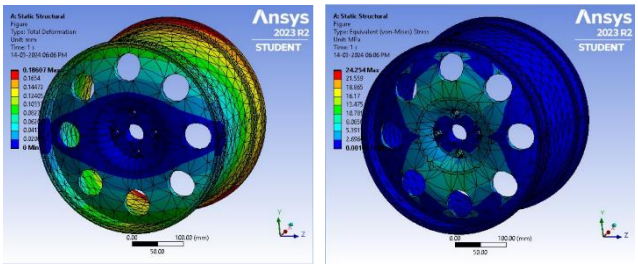

Fig. 6 Analysis of Al Alloy (Old rim)

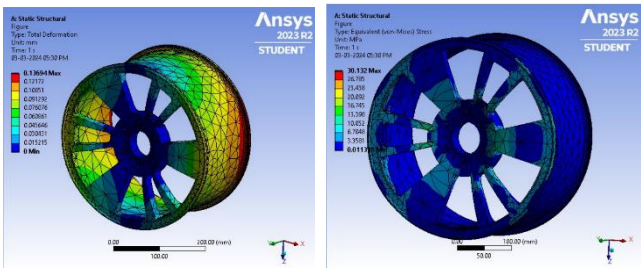

Fig. 6 Analysis of Al Alloy (New rim)

In similar manner the results of Mg and Titanium alloy are evaluated.

4. Result Table

The mass calculations of wheel rim are done based on the volume and density of each material.

Table 3. Properties of Material

Property	Old Rim	New Rim
Mass (Kg)		
Al Alloy	15.03	8.17
Mg Alloy	5.85	5.31
Titanium Alloy	15.03	1.36

Table 4. shows the stress and deformation values evaluated from the analysis.

Table 4. Result of Stress and Deformation

Property	Old Rim	New Rim
Stress (MPa)		
Al Alloy	24.25	30.13
Mg Alloy	24.08	30.10
Titanium Alloy	24.01	30.04
Deformation(mm)		
Al Alloy	0.1860	0.1369
Mg Alloy	0.2910	0.2158
Titanium Alloy	0.1358	0.1011

Here it observed that stresses in old and new rim are nearly same. But when we compare the masses of both rims, the optimized rim gives better results.

4.1 Comparison of results by using chart.

The figure below shows the comparison of masses of old rim and optimized rim.

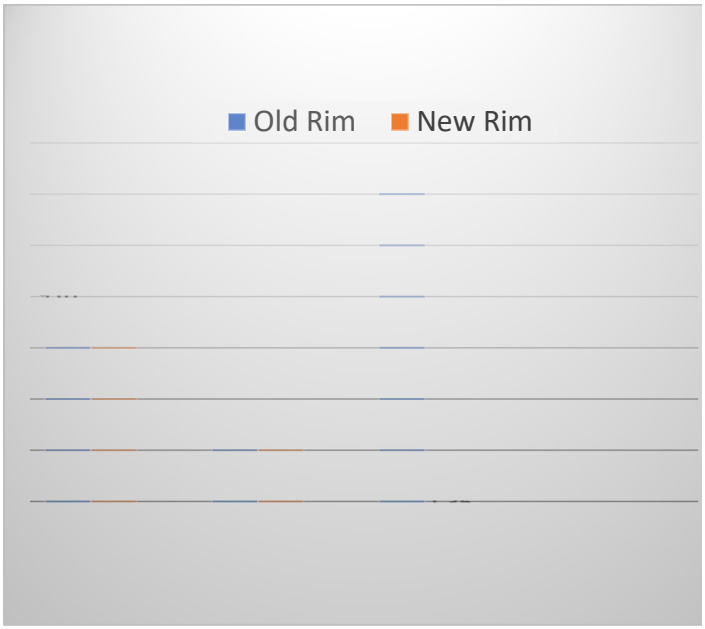

Fig. 7 Comparison of masses

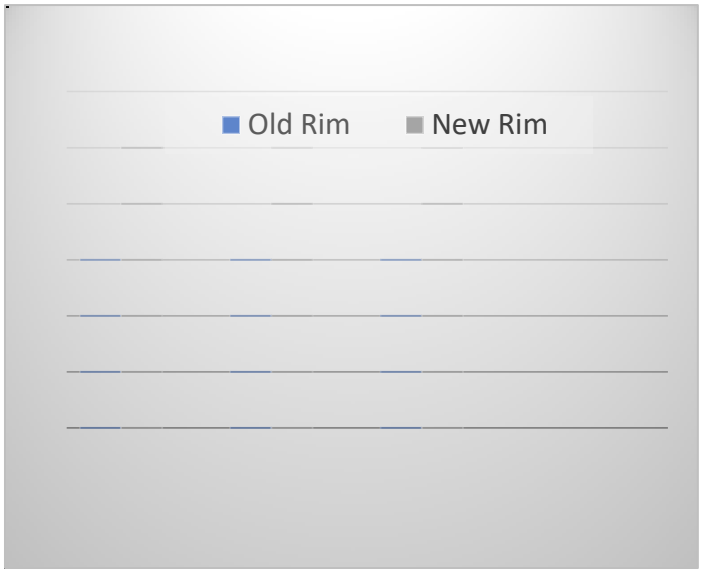

Fig. 8 Comparison of stresses and deformation

5. Conclusion

It is possible to reduce not only mass but also manufacturing cost of the wheel without harming its strength (Load Bearing Capacity). With the help of FEA analysis we can predict and understand how wheel rim might behave under various physical conditions.

1) The mass of old rim reduced after topology optimization in new rim.

2) Due to less amount of material as compared to old rim the stress increases in all materials by 4 to 5 MPa. But these stresses developed are within permissible range and hence new rim with aluminium alloy can gives us the optimum results.

Future Work

Further work can be done on manufacturing of this wheel rim that is the mould design and effect of temperature on the mould designed.

References

1. Aman Sharma, Rajat Yadav, Kunal Sharma, “Optimization & Investigation Of rim For High Performance of Vehicle”, Elsevier, Materials today: Proceeding(45), 2021, pp. 3601- 3604.

2. B. Abhiman, M. Yashwant Kumar and M.Vankanta Reddy “Optimization Car Rim Using Design & Analysis of Software”, International Journal of Research In Aeronautical And Mechanical Engineering (URAME), Vol. 9, Issue 8, 2021, pp. 1-6.

3. Y. Vara Prasad Ram and Dr. Vankanta Shiva "Modelling & Analysis of Rim Using CAE/ CAD Tools", International Journal of Mechanical Engineering, Vol. 7, 2022, pp. 1118-1124.

4. K. Venkateswara Rao and Dr. T. Dharmaraja "Analysis of Wheel Rim Using Finite Element Method", International Journal of Engineering Research & Technology (IJERT), Vol. 03, 2014, pp. 1259-1263.

5. Mr. Sushant. K. Bawane and Prof. Y. L. Yenarkar "Optimization of Car Rim", International Journal of Engineering Research and Applications (IJERA), ISSN: 2248-9622, Vol. 5, Issue 10, Part 2, 2015, pp. 01-08.

6. Mr. Chintapalli Shekhar et.al "Design and Structural Analysis of Car Alloy Wheel Using with Various Materials", International Journal Of Advance Scientific Research And Engineering Trends (IJASRET), ISSN: 2456-0774, Vol. 5, Issue 07, 2020, pp. 114-120.