IJRAME

INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING
WWW.IJRAME.COM
ISSN (ONLINE): 2321-3051

Vol.9 Issue 08,
AUGUST 2021
Pg: -01-06

OPTIMIZATION OF CAR RIM USING DESIGNING AND ANALYSIS SOFTWARES

B. ABHIMAN[1], M. YASHWANTH KUMAR[2]

[1]Assistant Professor, abhi4060.rj@gmail.com
[2] Assistant Professor, yashwanth.megnit@gmail.com
Author Correspondence: [1]Swami Vivekananda Institute of Technology, Mahbub College Campus, Secunderabad 500003
[2]G. Narayanamma Institute of Technology & Science for Women, Shaikpet, Hyderabad 500104

Abstract

The essence of car wheel rim provides a firm base on which to fit the tire. Its dimensions, shape should be suitable to adequately accommodate the particular tire required for the vehicle. In this project a tire of car wheel rim belonging to the disc wheel category is considered. The modern vehicle is also seen today a fashion item to complement people's individual requirements. Motor vehicles are produced according to very strict rules to ensure the safety of the passengers. Every component is therefore designed according to the criticality of the component. Wheels are classified as a safety critical component and international cods and criteria are used or design a wheel. The purpose of the car wheel rim provides a firm base on which to fit the tire. Its dimensions, shape should be suitable to adequately accommodate the particular tire required for the vehicle. In this study a tire of car wheel rim belonging to the disc wheel category is considered. The wheel rim is modelled by using modelling software CATIA V5. By using this software the time spent in producing the complex 3- D models and the risk involved in the design and manufacturing process can be easily minimized. So the modelling of the wheel rim is made by using CATIA. Later this CATIA modal is imported to ANSYS for analysis work. ANSYS is the latest software used for simulating the different forces, pressure acting on the component and also calculating and viewing the results. By using ANSYS software reduces the time compared with the method of mathematical calculations by a human. ANSYS structural analysis work is carried out by considered two different materials namely aluminium forged steel and their relative performances have been observed respectively. The main objective of this paper is to conduct structural analysis on the car wheel rim for different materials with modified dimensions. In addition to this the wheel rim is subjected modal analysis, a part of dynamic analysis is carried out its performance is also analysed

Keywords*:* Catia, Ansys, Car rim, Stresses, static stresses and Von-misses stresses.

1. Introduction

Automotive wheels have evolved over the decades from early spoke designs of wood and steel, carryovers from wagon and bicycle technology, to flat steel discs and finally to the stamped metal configurations and modern cast and forged aluminium alloys rims of today's modern vehicles. Historically, successful designs arrived after years of experience and extensive field testing. Since the 1970's several innovative methods of testing well aided with experimental stress measurements have been initiated. In recent years, the procedures have been improved by a variety of experimental and analytical methods for structural analysis (strain gauge and finite element methods). Materials to produce these wheels have become has sophisticated as a design and materials can range from steel to nonferrous alloys like magnesium and aluminium. Automotive wheels have

IJRAME

INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING
WWW.IJRAME.COM
ISSN (ONLINE): 2321-3051

Vol.9 Issue 08,
AUGUST 2021
Pg: -01-06

evolved over the decades from early spoke designs of wood and steel. Carry over's from wagon and bicycle technology, to flat steel discs and finally to the stamped metal configurations and modern cast and forged aluminium alloys rims of today's modern vehicles historically successful designs arrived after years of experience and extensive field testing. Wheel rims affect the braking performance of a vehicle as result of the following for parameters: size, weight, design or ventilation, materials. The size of the wheel rim governs how much space there is between the rim and the brake rotor. By moving up to a higher diameter wheel rim there will be more scope for air flow around the brakes and therefore better cooling. The weight of the wheel rim is an obvious issue. The mass is not only important in terms of the overall weight of the wheel, the rotational inertia of the wheel goes up with more weight as well, causing even more work for the brakes. The development of wheel is traced from a material viewpoint beginning with wood, the first documented wheel material and ending with new materials under development such as composites and titanium. While it is impossible to imagine what civilization would like without a wheel, many early civilizations has numerous other tools but did not possess wheels. In this paper a model is designed and analysis is carried out to optimize the dimensions of the car rim that can be improvised with effective modifications. Aluminium alloy wheels will not fail during service. The strength of the rim and fatigue life is critical. In order to reduce the cost, design for light weight and limited life is increasingly used for all vehicle components. In the actual product development rotary fatigues test is being used to detect the strength life of the wheel. In the present paper reliable design and a proper test procedure is conducted through simulation to guarantee the service strength under operational conditions and functioning of the wheel.

2. Literature Review

P. Meghashyam et al [1] conducted static and buckling analysis comparatively on steel and aluminium wheel rim. It is found that the deflections are more in aluminium than forged steel rim. It is recommended that forged steel is preferable than aluminium wheel rim. **Kalpesh R. Salunkhe [2]** analysed on alloy wheel by applying the three different materials namely aluminium (AL 6061), zinc (ZA 21) and Magnesium (Mg), the maximum total deformation and equivalent stresses are obtained lowest for zinc (ZA 21) compared to aluminium and magnesium. **N. Satyanarayana & Ch.Sambaiah[3]** conducted a finite element analysis on Aluminium alloy wheel to analyze stress distribution and fatigue life, safety and damage of alloy wheel and determined the safety factors for fatigue life and radial load. **R.Vijayan et al [4]** analysed to prevent cracks, and to improve aesthetics by reducing the weight, a new design is assimilated and conducted test on design using ansys software to optimize the stresses and deformation on the wheel rim**. B. Venkat Vinay Kumar & K. Devaki Devi [5]** reviewed to identify the existing gaps and proposed work by filling those gaps using CREO software and ANSYS used to analyze the design and calculate the stresses, deflections, bending moments and their relations. Static structural analysis has been carried out and the performance of the rim has been checked in dynamic analysis. **K. Srinivasa Rao [6]** determined the product life, the damage factor and safety factors through fatigue analysis done on three different materials of different designs**. Rahul K. Jape and S. G. Jadhav [7]** concluded that load acting on the alloy wheel rim is calculated as per Japanese industrial standard given by the company. Finite element analysis is performed on both wheel rim i.e. base wheel rim and optimized wheel rim the stresses and fatigue life of the wheel is calculated which is satisfying company criteria. Optimization techniques helped largely in reducing the mass of solid components which results in overall body weight reduction and thus lesser cost. Lesser weight in turn gives better performance and better fuel efficiency. **XiaofengWanga et al [8]** described a practical and comprehensive method for simulating the dynamic cornering fatigue test of

IJRAME

INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING
WWW.IJRAME.COM
ISSN (ONLINE): 2321-3051

Vol.9 Issue 08,
AUGUST 2021
Pg: -01-06

the automotive wheels. The test of a steel passenger car wheel is simulated by combined use of the linear transient dynamic finite element analysis and the local strain approach. . The strain history of the element whose local stress–strain characteristic keeps linear and closest to the critical element is applied to predict the fatigue life of the critical element with Neuber's rule and local strain approach, which is quite close to the test results.

3. Design and Analysis

The use of technological tools to assist in the creation, modification, analysis, or optimization of a design is termed as computer-aided design (CAD). CAD software is used to increase the designer's productivity, improve the quality of the design, improve communication through documentation, and create a database for manufacturing. CAD output is usually in the form of computer files it can be used for printing, machining, or other manufacturing processes. CAD output is usually in the form of computer files it can be used for printing, machining, or other manufacturing processes. The CAD model represents a traditional wheel rim with nave plate at the centre which is modelled over Catia software.

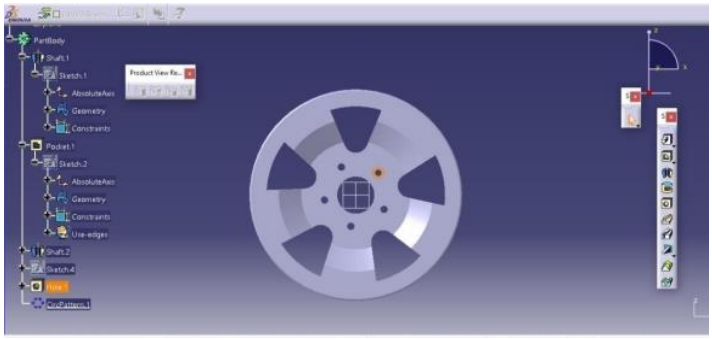

Figure 1: Design of Rim model

The designed model is imported into Ansys software for analysis by applying load conditions. The analysis is test is performed on the aluminium and forged steel material. The displacement, Von misses stresses and stress intensities of aluminium and forged steel materials were analysed

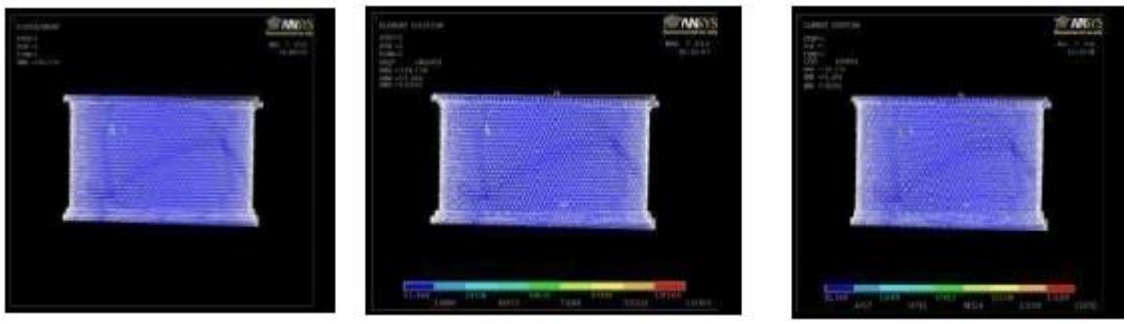

Figure 2: Displacement, Von-misses Stresses, and Stress intensities of Aluminium wheel rim

IJRAME

INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING
WWW.IJRAME.COM
ISSN (ONLINE): 2321-3051

Vol.9 Issue 08,
AUGUST 2021
Pg: -01-06

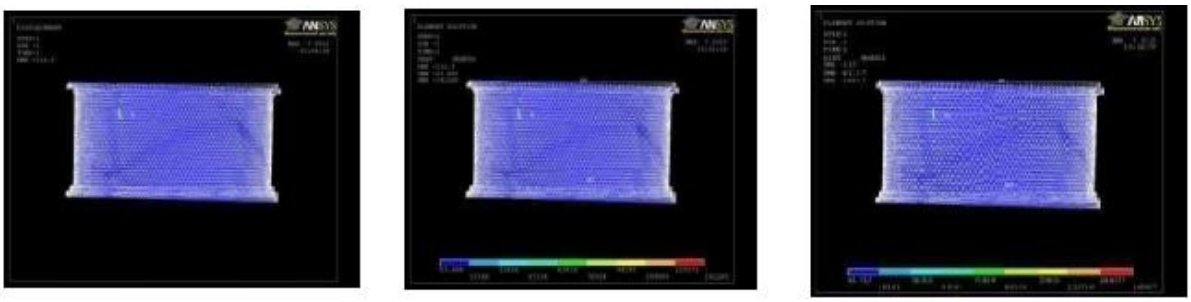

Figure 3: Displacement, Von-misses Stresses, and Stress intensities of Forged steel rim

The analysis performed on the different materials was studied with stresses induced in the car wheel with increases in number of rotations.

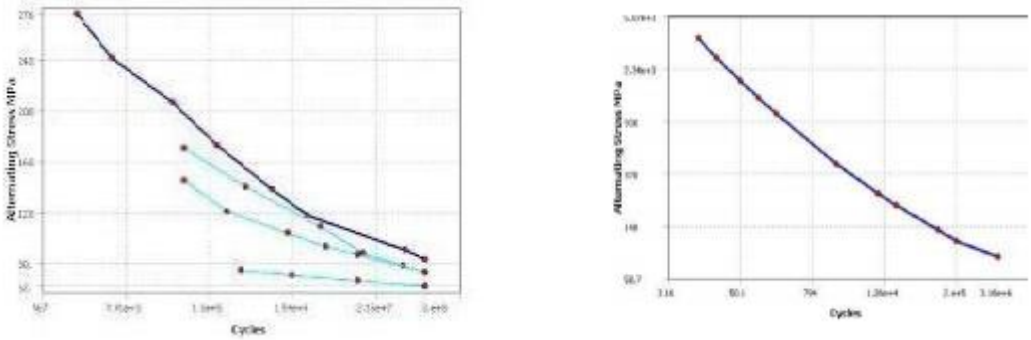

Figure 4: Graph of statistical stresses of forged steel and Aluminium car rim

The results of stresses and structural analysis of aluminium and forged steel with comparison is sown the following tabular column.

STRESSES RESULTS			**STRUCTURAL RESULTS**		
TYPE OF RESULT	**ALUMINIUM**	**FORGED STEEL**	**TYPE OF RESULT**	**ALUMINIUM**	**FORGED STEEL**
STATISTICAL DISPLACEMENT	4.1156×10^{-5}	4.046×10^{-6}	**DISPLACEMENT (DMX VALUE)**	336.167	122.3
VON-MISES STRESS	9.748×10^{7}	3.2348×10^{6}	**VON-MISES STRESS**	138103	141467
STRESS INTENSITY	1.0533×10^{6}	3.3459×10^{6}	**ULTIMATE STRESS VALUE**	152091	162955
DYNAMIC DISPLACEMENT	0.71735	0.43427			

IJRAME

INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING
WWW.IJRAME.COM
ISSN (ONLINE): 2321-3051

Vol.9 Issue 08,
AUGUST 2021
Pg: -01-06

4. Conclusions

CAD model of the wheel rim is generated in CATIA and this model is imported to ANSYS for processing work. An amount of pressure is applied along the circumference of the wheel rims made of both ALUMINIUM & FORGEDSTEEL and bolt circle of wheel rims is fixed. The following conclusions obtained from the result are

1. Aluminium wheel rim is subjected to more stress compared to Forged Steel.
2. In both the cases Von-misses stresses are less than Ultimate strength.
3. Deflections in Aluminium are more when compared to Forged Steel.
4. Since in both the cases Von-misses stresses is less than the Ultimate strength, taking deflections into account , Forged steel is preferred as best material for designed wheel rim.

References

1. Meghashyam, P., S. Girivardhan Naidu, and N. Sayed Baba, "Design and Analysis of Wheel Rim using CATIA & ANSYS." *International Journal of Application or Innovation in Engineering & Management (IJAIEM)*, Volume 2, Issue 8, August 2013, ISSN 2319 - 4847, pp 14-20.
2. Ganesh, S and Dr.P. Periyasamy, "Design and analysis of spiral wheel rim for four wheeler." *The International Journal of Engineering and Science*, Volume 3, Issue 4, 2014, ISSN (e): 2319 – 1813, ISSN (p): 2319 – 1805, pp. 29-37.
3. N. Satyanarayana & Ch.Sambaiah, "Fatigue Analysis of Aluminium Alloy Wheel Under Radial Load", *International Journal of Mechanical and Industrial Engineering (IJMIE)*, Vol-2, Issue-1, 2012, ISSN No. 2231 –6477,pp 1-6.
4. R.Vijayan, T.Thanka Geetha, V.Jagan Saai, M.A.S.MuthuKumara Swamy, M.Saravanan. "Design and Finite Element Analysis of Tractor Wheel Rim", *International Journal of Pure and Applied Mathematics*, Volume 115, Issue No. 7, 2017, ISSN: 1311-8080 (printed version); ISSN: 1314-3395 (on-line version), pp 257- 261.
5. B. Venkat Vinay Kumar & K. Devaki Devi, "Design Analysis and Optimization of a Wheel Rim", *The International journal of analytical and experimental modal analysis*, Volume XI, Issue IX, September/2019, ISSN NO: 0886-9367, pp 1879-1886.
6. K. Srinivasa Rao, M .Rajesh ,G.Sreedhara Babu, "Design and Analysis of Alloy Wheels", *International Research Journal of Engineering and Technology (IRJET)*, Volume: 04 Issue: 06 June -2017, ISSN: 2395 -0056, pp 2036-2042.
7. Rahul K. Jape and S. G. Jadhav, "CAD Modeling and FEA Analysis of Wheel Rim for Weight Reduction" *International Journal of Engineering Science and Computing(IJESC)*, Volume 6 Issue No. 6, June 2016DOI 10.4010/2016.1756 ISSN 2321 3361, pp 7404-7411
8. XiaofengWanga, XiaogeZhang,"Simulation of dynamic cornering fatigue test of a steel Passenger car wheel", *International Journal of Fatigue* 32 (2010).

IJRAME

INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING
WWW.IJRAME.COM
ISSN (ONLINE): 2321-3051

Vol.9 Issue 08,
AUGUST 2021
Pg: -01-06

A Brief Author Biography

B. Abhiman – Assistant Professor in Mechanical Department, Swami Vivekananda Institute of Technology. Graduated in Master of Engineering from Osmania University, Hyderabad. His area areas of interest are Materials science, CAD, Production Engineering. He has 10 years of teaching experience and guided many projects for B.tech Students.

M .Yashwanth Kumar – Assistant Professor in Mechanical Department, G. Naryanamma Institute of Technology & Science for Women. Graduated in Master of Technology from Jawaharlal Nehru Technological University, Hyderabad. His area areas of interest are Materials science, Metallurgy, Tribology. He has 6 years of teaching experience and published research papers in various international conferences and journals.