

A Review on Stress Analysis of Alloy Wheel

Mr. Sujoyot Rasal¹, Mr. Harshal Pimple², Mr. Sachin Muralidharan³, Mr. Rahul Yadav⁴

¹Department of Mechanical Engineering, Mumbai Univesity, MUMBAI
Email : sujyotrasal@gmail.com

²Department of Mechanical Engineering, Mumbai Univesity, MUMBAI
Email : hpimple12@gmail.com

³Department of Mechanical Engineering, Mumbai Univesity, MUMBAI
Email : sachin060997@gmail.com

⁴Department of Mechanical Engineering, Mumbai Univesity, MUMBAI
Email : ry1815877@gmail.com

Abstract—The research paper focuses on to review the determination of best suitable wheel rim material by performing FEA analysis on them. Wheel is the main mechanical part of the vehicle suspension system that has to withstand various loads encountered during vehicle action. The alloy wheel rim should be strong enough to withstand these loads. The material used for alloy wheel should be selected very carefully to withstand different stresses and deformations. Accordingly, materials like aluminium, magnesium and zinc alloy are studied by applying loads. The model is designed on Solidworks software and then the analysis is done using Abaqus software. The static condition is used for the analysis of the alloy wheel. At last the results of total deformation and equivalent stresses are obtained for different wheel rim materials and compared with each other. Thus, the best material can be selected for the manufacturing of the alloy wheel.

Keywords—FEA, Alloy wheel, ABAQUS

I. INTRODUCTION

The significance of wheel and tires in the vehicle cannot be challenged. Without engine, the car may pull, but without the wheels, it is not possible. The wheel thru tires takes complete load and decreases friction, and delivers cushioning effect to the passenger by absorbing vibration due to road surface unevenness and support in steering control. The wheel is a device that enables efficient movement of an object across a surface. Early wheels were simple wooden discs with a hole for the axle which further seemed to be inefficient. The spoke wheel was invented more recently, and allowed the construction of lighter and swifter vehicles. Alloy wheels are automobile wheels which are made from alloys like aluminium and magnesium. Alloy wheels are better heat conductors than steel wheels and also lighter in weight than them. Automotive manufacturers have been developing safe, fuel efficient and lightweight vehicular components to meet government regulations. In the real service conditions, the determination of mechanical behaviour of the wheel is important, but the testing and the inspection of the wheels during their development process is actually time consuming and costly. For economic reasons, it is important to reduce the time spent during the development and testing phase of a new wheel. Since it is difficult to estimate the stresses by 3-D stress analysis using elementary mechanical approximations, Finite Element Analysis(FEA) is used for the purpose.

- 1) Wheel: Wheel is generally composed of rim and disc.
- 2) Rim: This is a part where the tire is installed.
- 3) Disc: This is a part of the rim where it is fixed to the axle hub.
- 4) Offset: This is a distance between wheel mounting surface where it is bolted to hub and the centerline of rim.
- 5) Flange: The flange is a part of rim, which holds the both beads of the tire.
- 6) Bead Seat: Bead seat comes in contact with the bead face and is a part of rim, which holds the tire in a radial direction.
- 7) Hump: It is bump what was put on the bead seat for the bead to prevent the tire from sliding off the rim while the vehicle is moving.

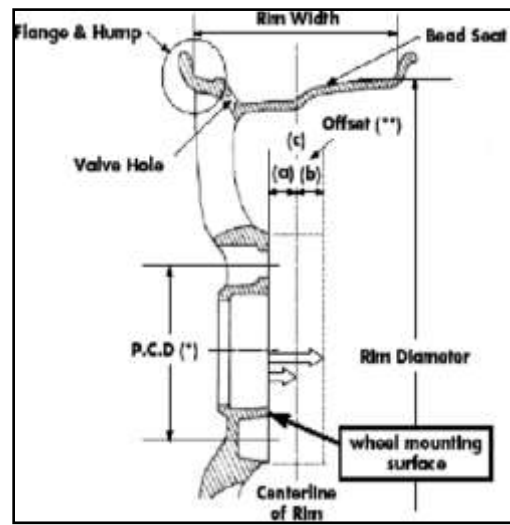


FIGURE 1: Wheel Rim Nomenclature

II. OBJECTIVE

- 1..To optimize the design of alloy wheel .
2. To determine the stresses on the alloy wheel under operational loading.
3. To achieve reduction of time by reducing the number of product development cycle.
4. To reduce cost by simulating the testing of model on computer instead of expensive test.

III. LITERATURE REVIEW

Karthik A.S. et. al, 2016 [1]used Finite Element Techniques to find out stress and displacement distribution in vehicle wheels subjected to increase pressure and radial load. The model was made using "CATIA V5" and the analysis was done through "ANSYS Workbench".Jaspreet Singh et. al, 2015 [2] considered anwheel for the analysis. During the part of thesisproject aluminium alloy was carried out for the FEA analysis. The static condition was chosen for the analysis and software used was ANSYS 15.0 .The pressure was applied on the outer rim of the aluminium alloy wheel.P. Meghashyam et. al, 2013 [3]created a model of the wheel rim with the help of CATIA software. Later this CATIA model was imported to ANSYS for the analysis work. With the help of ANSYS software, the different forces, pressure acting on the component were analysed. ANSYS static analysis was done by taking into consideration two different materials aluminium and forged steel.N. Satyanarayanaet. al, 2012 [4]performed a static analysis on aluminium alloy wheel A356 by using FEA package. The 3D model was designed by using CATIA and imported into ANSYS using iegs format. The analysis was performed in a static condition. The pressure was applied on the rim. FEA was carried out by simulating the test conditions to analyze stress distribution, fatigue life, safety and damage of alloy wheel.Kalpesh R. S. et. al, 2017 [5]analysed stress and displacement distribution in automobile wheels subjected to increased pressure and radial load , and took essential efforts to discover the Finite element techniques. Alloy wheel was

designed using Creo software and static structural analysis was done with different materials, load and boundary conditions using Ansys software. The maximum total deformation and equivalent stresses obtained were lowest for ZA21. Hence it is the best suitable material for alloy wheel rim.

IV. MATERIAL

TABLE 1
MATERIAL USED (5)

Properties	Aluminium6061	Magnesium AZ91D	Titanium 6A-4V
Density(g/cm ³)	2.70	1.8	4.6
Young's Modulus(GPa)	71	45	96
Poisson's Ratio	0.33	0.35	0.36

V. METHODOLOGY

In this paper reviewed the concept of radial fatigue testing for estimating the fatigue life of aluminium alloy wheel by conducting the manual testing and comparing the same with finite element analysis using ANSYS software. Fatigue life prediction using the stress approach is mostly based on local stress. In radial fatigue test the recommended minimum external diameter is 1.2m for wheels with rolling radius of less than 0.5m and 2.4m for larger wheels. The finite element analysis method is an effective approach for predicting the failure mode of an automotive component of wheel during wheel design stage [6]. FEA simulation of the wheel can significantly reduce time cost required to finalize the wheel design. As in the case of an automobile wheel maximum load is applied on the alloy wheel. Among aluminium, magnesium and titanium, maximum deformation noted was for Magnesium and Titanium has less equivalent stress and deformation but it is costly as compared to other [1]. The CATIA file was imported into ANSYS15.0. After application of boundary conditions and loads it was found that maximum deformation for aluminium alloy was at the circumference of the wheel and minimum at the bolt portion [2]. ANSYS static analysis is carried out using two different materials namely aluminium and forged steel and found that aluminium wheel rim is subjected to more stress compared to forged steel [3]. The analysis was executed in a standing state on aluminium alloy wheel A356.2 using FEA package. After applying pressure on the rim, using ANSYS found that the total deformation of wheel maximum is 0.2833 and minimum is 0.031478 [4]. A static FEA was performed on the alloy wheel to determine their critical areas. The mesh size in the analysis should be considered because it will affect the accuracy of the result. The accuracy of result increased with the decrease in the number of element size. The analysis indicated that the most critical part of an automobile alloy wheel is located at the circumferential area of the rim. In particular, titanium alloy is the most suitable material but is very expensive [5]

VI RESULTS

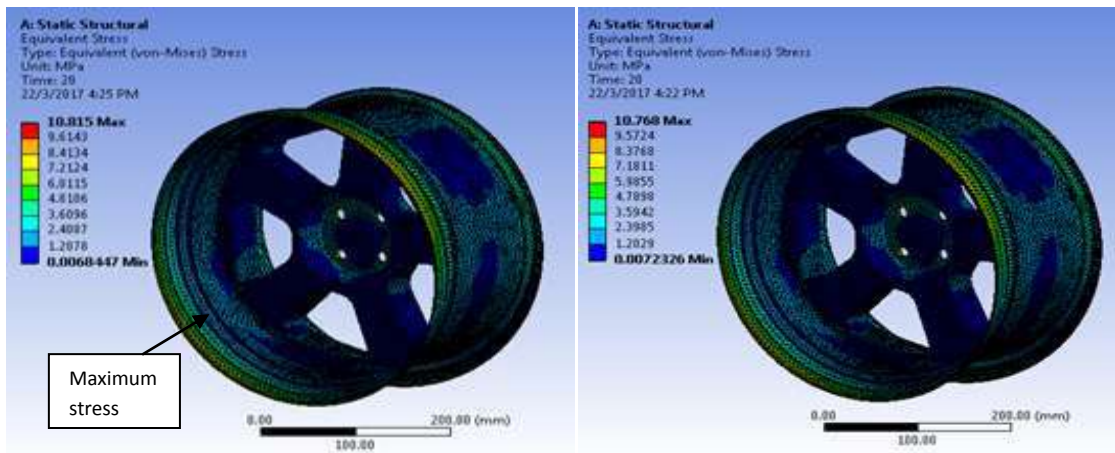


FIGURE 2: Stress distribution (Aluminium)[5]FIGURE 3: Stress distribution (Magnesium) [5]

VII CONCLUSION

A wide literature survey is carried out in the research area of wheel rim materials to study the performance of the alloy wheel rim using established experimental methods. Commonly used materials are AL alloy, Mg alloy, etc. The analysis indicated that the most critical part of an alloy wheel is located at the circumferential area of the rim for all materials. Also from the analysis, it was found that stresses are more at the bolt holes than the remaining area of the wheel which agreed from the literature review. The simulation for the three types revealed that titanium alloy demonstrates the highest fatigue life and to be the most suitable material but is more expensive. Further the work can be extended by the dynamic analysis.

VIII REFERENCES

- [1] Karthik A.S. , Praveen S. Ullagaddi, "Static Analysis of Alloy Wheel using FEA", IJRST, Vol 2, Issue 12,Page 397-401, 2016
- [2] Jaspreet Singh, Siddhartha Saha, "Static Analysis of alloy wheel using Ansys 15.0",IJRET,Vol 04,Issue 07,Page 217-220, 2015
- [3] P. Meghashyam , S . Girivardhan Naidu, "Design and Analysis of Wheel rim using Catia and Ansys", IJAIEM, Vol 2, Issue 8 ,Page 14-20, 2013
- [4] N. Satyanarayana ,Ch.Sambaiah, "Fatigue Analysis of Aluminium alloy under Radial load" , IJMIE, Vol2,Issue 1,Page 1-6, 2012
- [5] K.A. Zakaria, "Fatigue Life Simulation of an Alloy Wheel Design" ,IJET,Vol 7(4.40) 81-84
- [6]V.Dharanikumar, S.Mahalingam, "Review on Fatigue Analysis of Aluminium Alloy Wheel under Radial Load for Passenger Car",IJEDR,Vol 3,Issue 1,Page 121-125, 2014