REVIEW PAPER ON DESIGN AND ANALYSIS OF AUTOMOTIVE WHEEL RIM USING FINITE ELEMENT ANALYSIS

Nikhil Jamadade

P.G. Student, Department of Mechanical Engineering, TKIET, Warnanagar, Kolhapur, India

Jitendra Shinde

P.G. Student, Department of Mechanical Engineering, BV Engineering College, R. K. Nagar Kolhapur, India

Abstract

This work involves the application of radial load fatigue test to study the fatigue life of the rim with weight reduction. In this work we are going to carry out the radial load fatigue test on newly designed rim with material and design optimization. For finite element analysis we are going to use software like ANSYS. Results obtained by these two methods will be compared to each other for validation of work.

Introduction

A wheel rim is a highly stressed component in an automobile that is subjected to bending and torsional loads. Because of the long life and high stresses, as well as the need for weight reduction, material and manufacturing process selection is important in rim design. There are competitions among materials and manufacturing processes, due to cost performance, and weight. This is a direct result of industry demand for components that are lighter, to increase efficiency, and cheaper to produce, while at the same time maintaining fatigue strength and other functional requirements.

In the fatigue life evaluation of aluminum wheel design, the commonly accepted procedure for passenger car wheel manufacturing is to pass two durability tests, namely the radial fatigue test and cornering fatigue test .Since alloy wheels are designed for variation in style and have more complex shapes than regular steel wheels, it is difficult to assess fatigue life by using analytical methods. The newly designed wheel is tested in laboratory for its life through an accelerated fatigue test. Based on these test results the wheel design is further modified for high strength and less weight. But stress analysis will not yield the optimum wheel design described a probability based model for prediction of fatigue failure of automotive wheel rim. Aluminum alloys, magnesium alloys and structural steel are the three important materials by which rim of wheel is casted and then manufactured. Fatigue life of all of these three components gets varied. It is necessary to know the fatigue life of each of these materials. By conducting design of experiments best parametric design can be done. Also thickness of the rim can be varied for each of these materials. It is stated that by varying the thickness level we can get varied fatigue strength for different materials. It is recommended to use optimum thickness which gives higher strength without compromising the fatigue life of the rim. Weight reduction can cause high level of cost saving.

Problem Statement

The failure of rim wheel is due to crack initiated near the ventilation hole which further gets propagated throughout the rim which leads to fatigue failure. In order to improve the Fatigue life of rim, material optimization and design optimization is necessary for which best material has to be selected by conducting design of experiments to find parametric design which gives higher fatigue life.

Objective

1. To design the proposed automotive wheel rim.
2. To carry out stress and fatigue analysis of wheel by using finite element method.
3. To study the effect of various materials on fatigue life of the rim.
4. To optimize thickness of rim to reduce the material consumption and improve life of component by using advanced fatigue strain life approach.
5. To develop the experimental setup for proposed automotive wheel rim.
6. To compare experimental and finite element analysis results

Proposed Work

Methodology

In this dissertation work it is proposed to study the effect of various materials on fatigue life of the automotive wheel rim by using finite element analysis and radial load testing.

Phase I - Literature Survey.

In this phase literature survey of fatigue life analysis, Design optimization etc. will be carried Out by referring journals like ASME journal, Springer link, Elsevier, US patents, etc.

Phase II - One of the method to find fatigue life of rim is by conducting radial load test. This Phase includes fatigue analysis of newly designed rim by finite element analysis and by radial Load test.

1. Experimental work: We are going to carry out fatigue life analysis of rim by radial load technique as follows.
2. To carry out material optimization to reduce weight and find fatigue life for each material.
3. To prepare the new automotive wheel rim by suitable casting method.
4. Test the new rim on radial load by experimental radial load fatigue test.
5. To find out fatigue life of new rim.
6. Finite element analysis :
7. To design automotive wheel rim which is now in use.
8. To carry out stress and fatigue analysis of rim in use.
9. To carry out material optimization to reduce weight and find fatigue life for each material.
10. To find out best thickness optimization for rim.
11. Modelling the new rim by catia and importing it into ansys for further analysis

Modelling of Rim Wheel

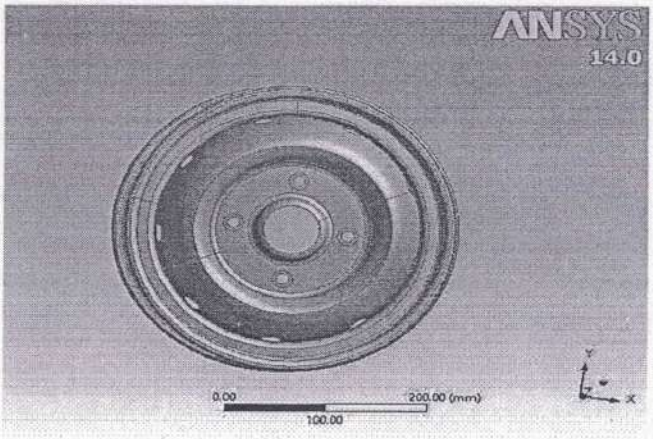


Figure 3D-Model

Finite Element Method

The finite element method is a powerful tool for the numerical procedure to obtain solutions to many of the problems encountered in engineering analysis. Structural, thermal and heat transfer, fluid dynamics, fatigue related problems, electric and magnetic fields, the concepts of finite element methods can be utilized to solve these engineering problems. In this method of analysis, a complex region defining a continuum is discretized into simple geometric shapes called finite elements the domain over which the analysis is studied is divided into a number of finite elements.

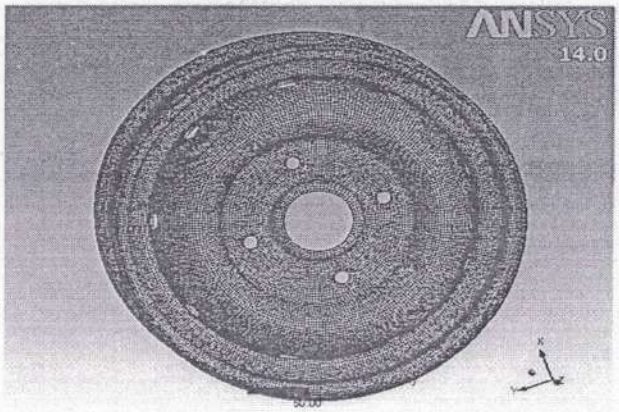


Figure Meshing of wheel rim

Loading and Boundary Condition

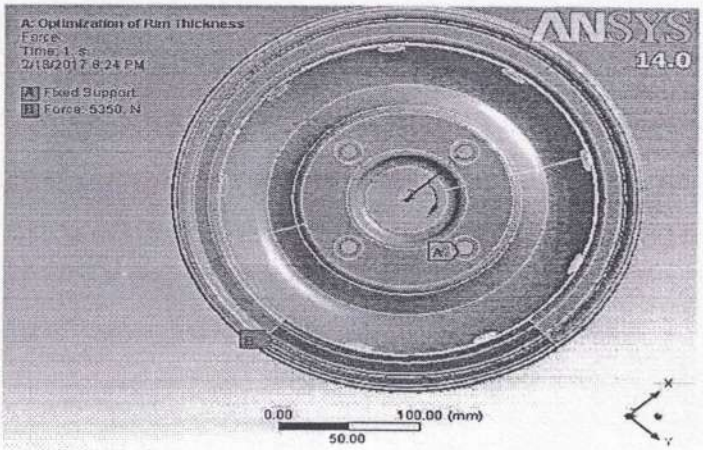
a) Constraints: Wheel Rim is fully constrained at four locations highlighted in fig where it gets clamped to wheel hub during wheel assembly.

b) Loads: The calculated mass distribution on rear wheel = 271 Kg. And considering the 2G acceleration when wheel faces the bump during running condition.

Therefore total Force on wheel Rim = 271 x 2 x 9.81 = 5317 N.

Considering 5350 N load for analysis

80^0 Contact angle is considered for force distribution.



Results for Original Geometry

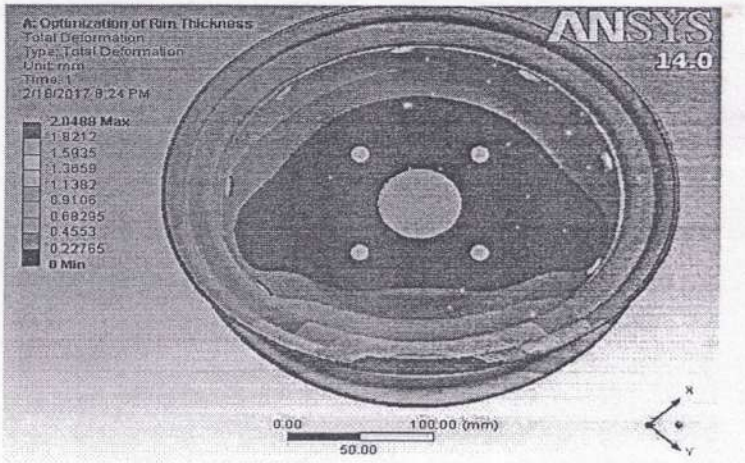


Figure Total Deformation

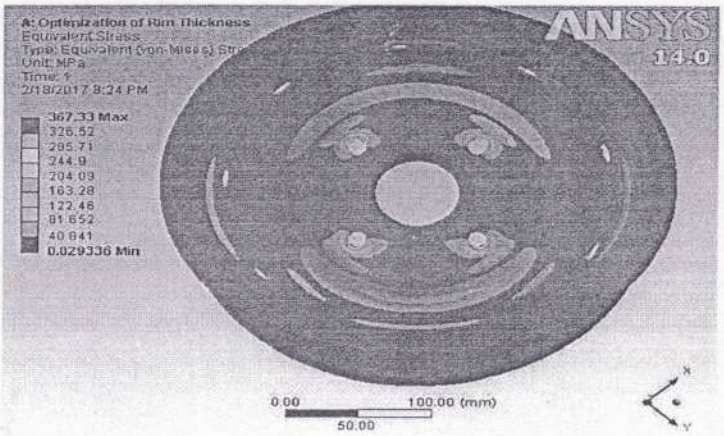


Figure Equivalent Von Mises stress

Solution

It is necessary to carry out experimental and finite element analysis of wheel rim to avoid the failure of the rim by improving the geometry and material optimization with the help of design of experiment. Results are to be validated by radial load test on wheel rim to check its improved fatigue life.

References

1. M.M.Topac,S.Ercan,N..Kuralay, "fatigue life prediction of heavy vehicle steel wheel under radial loads by using finite element analysis", Dkuz Elul university, izmir, Turkey, 2011.
2. P. Ramamurthy Raju, B. Satyanarayana, K. Ramji, K. Suresh Babu, "Evaluation of fatigue life of aluminum alloys wheels under radial loads", Engineering Failure Analysis 14, pp.791-800,2007
3. Y.H Kim, T.K.Ryou, H.J.Choi "An analysis of the forging processes for 6061 aluminum-alloy wheels", journal of material processing technology 123 (2002) 270-276,2002.
4. Sunil N. Yadav, N. S. Hanamapure,"Analyze the Effect of Camber Angle on Fatigue Life of Wheel Rim of Passenger Car by Using Radial Fatigue Testing", International Journal of Engineering Science and Innovative Technology (IJESIT) Volume 2, Issue 5, September 2013.
5. Mehmet Firat, Recep Kozan, Murat Ozsoy, O. Hamdi Mete, "Numerical modeling and simulation of wheel radial fatigue tests." Engineering Failure Analysis 16, pp.1533-1541, 2009.
6. N.Satyanarayana,"fatigue analysis of aluminium alloy wheel under radial load", International journal of mechanical and industrial engineering(IJMIE),Vol-2,Issue-1,2012.
7. U.Kocabicak, M.Firat,"Numerical Analysis Wheel Cornering Fatigue Tests", Engineering Failure Analysis 8(2001), 339-334, 2000.
8. Vivek Asnani, Damon Delap, Colin Creager,"The development of wheels for lunar roving vehicle",Journal of terramechanics 46 (2009) 89-103,2009
9. Ravi Lidoriya, Sanjay, Chaudhary and Anil Kumar Mohopatra,"Design and Analysis of Aluminum Alloy Wheel using PEEK Material", International Journal of Mechanical Engineering and Research. ISSN No. 2249-0019, Volume 3, Number 5 (2013), pp. 503-516, 2013.
10. V.B.Bhandari,"Design of Machine Elements", Tata McGraw Hill Publication, third edition.