International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 04 Issue: 07 | July -2017 www.irjet.net p-ISSN: 2395-0072

Structural Simulation of Car Rim Using Finite Element Method

Suraj L. Gondhali[1], Milind M. Banubakode[2], Ankush K. Biradar[3]

[1] Research Scholar, Department of Mechanical Engineering, , B. R. Harne College of Engineering and Technology(BRHCET), Thane, Maharashtra, India
[2] Assistant Professor, Department of Mechanical Engineering, , B. R. Harne College of Engineering and Technology(BRHCET), Thane, Maharashtra, India
[3] Head of the Department, Department of Mechanical Engineering, , B. R. Harne College of Engineering and Technology(BRHCET), Thane, Maharashtra, India

---

Abstract :- The essential of car wheel rim is to provide a firm base on which to fit the tyre. Its dimensions, shape should be suitable to adequately accommodate the tyre required for the vehicle. In this project, a tyre of car wheel rim belonging to the disc wheel category is considered. Design is an important industrial activity which influences the quality of the product. The wheel rim is modelled by using modeling software SolidWorks. 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 modeling of the wheel rim is to be made by using SolidWorks. Later this SolidWorks model will be imported to ANSYS WORKBENCH 14.5 for analysis work. ANSYS WORKBENCH 14.5 is the latest software used for simulating the different forces, pressure acting on the component and calculating and viewing the results. By using ANSYS WORKBENCH 14.5 software reduces the time compared with the method of mathematical calculations by a human. ANSYS WORKBENCH 14.5 static structural analysis work will be carried out by considering three different materials namely aluminium alloy, magnesium alloy and structural steel and their relative performances will be observed respectively. In addition to wheel rim is subjected to modal analysis, a part of dynamic analysis will be carried out to observe its performance. In this analysis by observing the results of both static and dynamic analysis suitable material will be preferred that will give optimum performance depending on different parameters.

Key Words: - Finite Element Method (FEM), modeling, rim, structural simulation etc...

INTRODUCTION :-

Automotive wheels have evolved over the decades from early spoke designs of wood and steel, 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. 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). Within the past 10 years, durability analysis (fatigue life predication) and reliability methods for dealing with the variations inherent in engineering structure have been applied to the automotive wheel.

Wheels are clearly safety related components and hence fatigue performance and the state of stress in the rim under various loading conditions are prime concerns.
Further, wheels continue to receive a considerable amount of attention as part of industry efforts to reduce weight through material substitution and down gauging. Although wheels are loaded in a complex manner and are highly stressed during their rolling duty, light weight is one of the prime requirements, hence cast and forged aluminium alloys are essential in the design.

Further the current generation automobile has the alloy wheel. This technology up gradation has given multiple choices in respect of material, cross section for rim and arm connecting hub and rim. The newer cars are supposed to have lesser weight without compromising the strength. Therefore, there is a scope for optimization of wheel design in respect of geometry of car rim, geometry of arm, material etc. The car rim is subjected to static as well as dynamic loading condition. it undergoes bending, twisting, circumferential loading and impact loading. Therefore, it is justified to have a detailed analysis using the technique like FE for the stresses developed during used. It is proposed to analyse the car rim using FE approach for varied geometry parametric parameter for optimization of its weight.

1.1 Major findings from literature review:-

One the latest and most comprehensive pieces of research on the finite analysis of car wheel rim was performed by

Nakka et al. [1]. Using data provided by previous authors, Nakka attempted to parameterize stress and displacement distribution in vehicle wheel based on thickness to diameter ratio for numerous wheel geometries.

Jaspreet Singh et al. [2] have studied static analysis of aluminium alloy wheel by reverse engineering. Reverse engineering is a good method to redesign the old component. The wheels should pass different tests for best performance like static analysis, vibration analysis etc. In this project, a wheel was considered for the analysis. In the analysis, the results of the equivalent stress, safety factor, and deformation were calculated.

T. Siva Prasad et al. [3] have studied static structural analysis of two wheel having different materials. In this project, a tyre of car wheel rim belonging to the disc wheel category is considered. ANSYS static analysis work is carried out by considered two different materials namely aluminium and forged steel and their relative performances have been observed respectively.

B Raghupathi et al. [4] have studied Static and modal analysis of light commercial vehicle alloy wheels of materials Structural steel and Aluminium alloy. In this study, the model correlation between (Computer Aided Engineering), CAE simulations and tests is performed. For this purpose, mode shapes and their natural frequencies obtained from CAE simulations are compared with experimental modal analysis results. After the correlation is provided, wheel design optimization proposals are given by considering and durability criteria.

Sunil N. Yadav et. al. [5] has studied the effect of camber angle on wheel rim. The aim of the paper is to investigate the effect of camber angle on stress distribution and fatigue life of wheel rim of passenger car under radial load condition which arises due to off road field area and road unevenness. Finite element analysis (FEA) is carried out by simulating the test conditions to analyse stress distribution and fatigue life of the steel wheel rim of passenger car. Experimental analysis performed by radial fatigue testing machine for evaluation of fatigue life under influence of camber angle. For radial fatigue testing SAE J328 standard is used.

M. Topaç, S. Ercan, N.S. Kuralay[6] have studied radial dynamic fatigue tests for newly designed heavy commercial vehicle steel wheel the cause of this damage was studied via finite element analysis. To determine the reason of the fatigue failure, stress analysis was performed via the finite element method. In this way, stress concentrated regions, where fatigue failure is expected, were determined. Mechanical properties of the wheel material were determined by tensile tests and hardness measurements. The fatigue life of the damaged wheel was estimated using the stress–life (S–N) approach, utilising the ultimate tensile strength of the processed wheel material and the main factors determined for the critical regions. To extend the life of the wheel disc and delay the onset of fatigue, design enhancement solutions were applied.

Liangmo Wang et al. [7] have studied fatigue life analysis of aluminium wheels. To improve the quality of aluminum wheels, a new method for evaluating the fatigue life of aluminum wheels is proposed in this paper. The ABAQUS software was used to build the static load finite element model of aluminum wheels for simulating the rotary fatigue test the results indicated that the proposed method of integrating finite element analysis and nominal stress method was a good and efficient method to predict the fatigue life of aluminum wheels.

Muhammet Cerit [8] have studied simulation of impact test for a cast aluminium alloy wheel by using 3–D explicit finite element methods. The analyses results are presented as a function of time. The maximum value of the displacement and stress on the wheel and tire are shown. As a result, the use of explicit finite element method to predict the performance of new products design is replacing the use of physical test.

Chia-Lung Chang and Shao-Huei Yang [9] have studied nonlinear dynamic finite element is used to simulate the SAE wheel impact test. The wheel modelled as an elastic–plastic body is mounted at an incline of 130 to horizontal, and the striker is prescribed an initial velocity to simulate a drop height. The total plastic work concept of ductile fracture mechanics is used to predict the impact failure of wheel. Three-dimensional finite element method is employed to obtain the strain energy density of wheel at impact, and the critical strain energy density is based on the total plastic work of wheel material. Compared with actual tests, the finite element results show the total plastic work approach can be used to predict the wheel fracture during the impact test.

J. Stearns, T. S. Srivatsan, X. Gao, and P. C. Lam [10] have highlighted the use of the finite element technique for analysing stress and displacement distributions in wheels of automotive vehicles when subject to the conjoint influence of inflation pressure and radial load. The most commonly used considerations in the design of the rotating body are elucidated. A potentially viable technique for finite element modeling of radial wheel,

subjected to loading, is highlighted. The extrinsic influence of inflation pressure on performance of the rotating body, that is, the wheel, is rationalized.

H. Akbulut [11] have studied an octopus-type car rim for which critical zones were found first and then optimum thickness was investigated using an elasto-plastic analysis. In this study, three-dimensional finite element method was used for conducting elasto-plastic analysis.

S. S. Bhavikatti[12] deals with optimum shape design of the rotating disks by nonlinear programming method. The shape of the cross section is defined by 5th degree polynomial which is completely determined by the boundary conditions and four design variables. The stress analysis of the disk is carried out by finite element method using parametric elements. The optimization technique used is with improved move limit method of sequential linear programming. Optimum shapes are obtained for different speeds and for different fit pressures from hub.

While many have researched wheel rims over the years, the following comparison study will seek to contribute insight into the use of FEA to predict wheel rim performance factors, especially in specifying the level of agreement that can be expected at various operating conditions, and discuss meshing methods to improve results.

1.2 Objective & Scope:-

The various car rim geometry for the exiting car rim and certain variation of them in regard to cross section of arm and rim geometry is modeled and the suitable constrained and loading condition will imposed.

These models are analysed using slandered commercial FE software. The result for each of the case are compared for the various design parameter like stress and weight, stress weight and cost this shall further then be compared to chosen the optimum design.

In all cases described in methodology, FE static analysis is carried out and result are compared with analytical calculation for some cases.

The proposed analysis will be given an opportunity to study the various loading that would come on the car rim during its working and give an insight to the stresses the rim is exposed to.

Further the variation in the geometry and the arm shape are providing different possible approach for design of rim. The optimization is providing the comparison between the variety of design existing and proposed.

The proposed work included following steps:

1) Study of literature review of various work reported.
2) Selecting some of the commercially available rim design and certain variation of them.
3) CAD model is created using various tool in SolidWorks i.e. Extrude, revolve, Mirror, etc.
4) FE analysis of the above geometry for different loading and boundary condition will be carried out.
5) Comparison of the result obtained in above step for different materials.
6) Choosing the optimum design based on the material and other parameter if any.
7) Natural frequencies and modal shapes are also to be determined.

REFERENCES:-

[1] Mr. Sunil Kumar Nakka et al. "Structural analysis of car alloy wheel using aluminum and magnesium alloys" International Journal for Development of Computer Science and Technology (IJDCST) Issue- V-4, I-2, SW-09 in March 2016.
[2] Jaspreet Singh et al. "Static analysis of alloy wheel using ANSYS15.0" International Journal of Research in Engineering and Technology (IJRET).ISSN No. 2319-1163,Volume 4, Issue 7 (July 2015)
[3] T.SivaPrasad.Krishnaiah, J. Md. Iliyas, M.Jayapal Reddy, "A Review on modeling and analysis of car wheel rim using CATIA & ANSYS" International Journal of Innovative Science and Modern Engineering (IJISME) ISSN:2319-6386,Volume-2,Issue-6,May 2014.
[4] B Raghupathi et al. "Design and analysis of car wheel rim using fem technique" journal International Journal of Computer Science information and Engg Technologies (IJCSIET) Vol. 3, Series 4, Issue. 4, Dec. 2014
[5] 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" journal International Journal of Engineering Science and Innovative Technology (IJESIT) Volume 2, Issue 5, September 2013.
[6] M.M. Topaç, S. Ercan, N.S. Kuralay, "Fatigue life prediction of a heavy vehicle steel wheel under radial loads by using finite element analysis" Engineering Failure Analysis 20 (2012) 67-79 : Elsevier Ltd.
[7] Liangmo Wang et al. "Fatigue life analysis of aluminum wheels by simulation of rotary fatigue test" Journal of Mechanical Engineering Volume 1 Pages 31-39(2011)
[8] Muhammet Cerit, "Numerical simulation of dynamic side impact test for an aluminium alloy wheel" Scientific

Research and Essays (SRE) Volume 5, ISSN 1992-2248 (Sept. 2010), Page No. 2694-2701
[9] Chia-Lung Chang, Shao-Huei Yang "Simulation of wheel impact test using finite element method" Engineering Failure Analysis 16(2009) 1711–1719: Elsevier Ltd.
[10] J. Stearns, T. S. Srivatsan, X. Gao, and P. C. Lam, "Understanding the influence of pressure and radial loads on stress and displacement response of a rotating body: The automobile wheel" International Journal of Rotating Machinery Volume 2006, Article ID 60193, Pages 1–8 DOI 10.1155/IJRM/2006/60193
[11] H. Akbulut, "On optimization of a car rim using finite element method" reported in the journal of Finite Elements in Analysis and Design 39 (2003) 433–443: Elsevier Ltd
[12] S.S. Bhavikutti, C.V. Ramkrushnana "Optimum shape design of disc" CAS Vol II No.5-C pp.397-401 Pergamon Press Ltd. (1980)
[13] Indian Standard for Automotive Vehicles "Rims-General requirements", IS10694 (1): 1993.
[14] Strength of Materials, chapter 19 stresses due to rotation by Er. R. K. Rajput, S. CHAND Publication Fifth Revised Edition. Page. No. 1140,1141,1142.