Comparative Study of FEA with CFT for Wheel RIM

Rakesh Balwant Thakare
Sandip Polytechnic, India

Abstract— Wheels rim should not fail during its working. From design point of view the strength and fatigue life of rim are critical issues. In order to reduce costs, design for light-weight and limited-life is increasingly being used for all vehicle components. In the actual product development, the rotary fatigue test is used to detect the strength and fatigue life of the wheel rim tests are much expensive and consumes lot of time. Computer simulation and Finite Element analysis of these tests can reduce the time and cost required to perform a wheel rim design at significant level. In this paper we discussed the simulation and Finite Element Analysis of wheel rim using Dynamic Cornering Fatigue test and compared to the experimental results.

Keywords: Wheel Rim, Simulation, Finite Element Analysis, Dynamic Cornering Fatigue Test

I. INTRODUCTION

The importance of wheel and tires in the automobile cannot be challenged. Without engine, car may tow, but without the wheels, this is not possible. The wheel with tires takes full load, and reduces friction, and provides cushioning effect to passenger by absorbing vibration due to road surface unevenness and assist in steering control. The alloy of conventional disc wheel in case of car and wire wheel as in case of motorbike has better aesthetic looks and easy of manufacturing.

The wheel rim is the outer edge of the wheel, holding the tire. It makes up the outer circular design of wheel on which the inside edge of tire is mounted. It should not fail during its working. Moreover, since other requirements such as lighter weight or more attractive design make the configuration of the wheel more complicated and sophisticated, it has become necessary to perform rigorous strength evaluations of the wheel in detail when a new wheel design is developed. A well designed wheel is the foundation which adds strength, stability and durability to a tire. Hence, the increased urge to make them safer and reliable. As a safety related components, the wheel must fulfill its function reliably throughout the entire life of vehicle. The total weight of a car is balanced with a vertical reaction force from the road through the tire. This load constantly compresses the wheel radially. While the car is running, the radial load becomes a cyclic load with the rotation of the wheel. Hence, the evaluation of wheel fatigue strength under radial load is an important performance characteristic for structural integrity. From design point of view the strength and fatigue life of rim are critical issues. In order to reduce costs, design for light-weight and limited-life is increasingly being used for all vehicle components. In the actual product development, the rotary fatigue test is used to detect the strength and fatigue life of the wheel. Therefore, a reliable design and test procedure is required to guarantee the service strength under operational conditions and full functioning of the wheel. Loads generated during the assembly may cause significant levels of stress in components. Under test conditions, these high levels of stress alter the mean stress level which in turn, alters the fatigue life and critical stress area of the components as well. The inclusion of clamp load improves the prediction of the critical stress area and fatigue life of aluminum wheels for a new wheel, the failure probability of the dynamic radial fatigue test can be read directly from this probability contour drawn from the test data. A bi-axial load-notch strain approximation for proportional loading to estimate the fatigue life of a passenger car wheel during the cornering fatigue test. The elastoplastic strain components were calculated analytically using the total deformation theory of plasticity. Wheel is generally composed of rim and disc. Rim is a part where the tire is installed. Disc is a part of the rim where it is fixed to the axle hub. Offset is a distance between wheel mounting surface where it is bolted to hub and the enterline of rim. The flange is a part of rim, which holds the both beads of the tire. Bead seat comes in contact with the bead face and is a part of rim, which holds the tire in a radial direction. Hump 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. Well is a part of rim with depth and width to facilitate tire mounting and removal from the rim. The randomness of fatigue prediction owing to the inherent uncertainties in loading, manufacturing variability, and material properties has been commonly recognized

FEA has been developed to an incredible precision. There are generally two types of analysis that are used in industry: 2-D modeling, and 3-D modeling. While 2-D modeling conserves simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results. 3-D modeling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively.

A. Problem Definition

Manufacturing of wheel rims in automotive industries demands no-flaw wheel rims. This component must not fail. However, process as well as material deficiency in varying ratios of influence has been counteracting factors in producing flawless wheel rims. During manufacturing and forming process cracking of wheel rims is a major problem, also at weld joints the cracks are normally seen. The various cyclic loads are act on wheel rim in dynamic conditions which initiates the crack. Various experimental methods are used to study the fatigue failures of the rim, out of which dynamic cornering fatigue test is important. But experimental analysis on this test not gives exact stress conditions. When moment applied on rim we don"t know exact location of stresses arise on disc or on mounting holes or both. Also the test is costly and time consuming. This dissertation will focus on identifying the critical locations, which agree well with the actual crack locations by computer simulation methods for simplification and saving cost and time and suggest the better method of FEA.

II. LITERATURE REVIEW

Zhanguang Zheng et al (2015) study to reveal the fatigue failure mechanism of the fractured steel wheel after the cornering fatigue test. High power scanning electron microscopy is performed to analyse the fractured surfaces of the wheel and to examine the fatigue cracks in greater detail. The results show that the failure locations are mainly around the cooling hole and the bolt hole of the disc.[1]

Zhan-Guang Zheng et al (2014) studies A computational methodology is proposed to simulate wheel dynamic cornering fatigue test and estimate its" multi-axial fatigue life. The technique is based on the critical plane theory and the finite element methods. The prediction of fatigue life is found to be in close agreement with the corresponding experiment. The stress states of wheel are basically biaxial tensile and compression normal stresses during the prototype test. The principal stresses are not proportional and the unstable principle plane is changing with loading direction, which indicates that the fatigue crack may occur first in the circumferential direction of steel wheel.[2]

T. Siva Prasad et al (2014) Materials to produce these wheels have become has sophisticated as a design and material can range from steel to non-ferrous alloys like magnesium and aluminium. Automotive wheels have evolved over the decades from early spoke design of wood and steel. Today"s modern vehicles are uses the stamped metal configuration and modern cast and forged aluminium alloys rims. Aluminum alloy wheel rim is subjected to more displacement compared to Forged steel. [3]

Sharma et al. (2013) studied failure analysis of wheel during manufacturing process and conduct some tests on wheel such as weld shear strength test, nut rigidity test, air leak test, wheel static and dynamic test etc. they focus on corner fatigue test and radial fatigue test. Blank size reduction process is implemented for eliminating failure from wheel. This process helps in elimination of failure in manufacturing process and also to detect the problem. After analysis the failure, they expose the problem area and give appropriate solution and suggestion which results in the elimination of problem in wheel. [4]

Alexandru Valentin rt al (2012) says Wheels have vital importance for the safety of the vehicle and special care is needed in order to ensure their durability. The development of the vehicle industry has strongly influenced the design, material selection and manufacturing processes of wheels. The wheels loading manner is a complex one; further improvement and efficient wheel design will be possible only if the loading will be better understood. In this paper, the car rim is analyzed with finite element method, using the 400 loading test. The static stresses are studied in order to find the zones with higher stress concentration and to suggest the better design solution. The results have been compared with those obtained by using an experimental stand. Finally, the Wöhler curve for the car rim is obtained.[5]

Satyanarayana and Sambaiah (2012) studied the detailed "Fatigue Analysis of Aluminum Alloy Wheel under Radial Load". During the part of project a static and fatigue analysis of aluminum alloy wheel A356.2 was carried out using FEA package. The 3 dimensional model of the wheel was designed using CATIA. Then the 3-D model was imported into ANSYS using the IGES format. The finite element idealization of this modal was then produced using the 10 node tetrahedron solid element. The analysis was performed in a static condition. This is constrained in all degree of freedom at the PCD and hub portion. The pressure is applied on the rim. We find out the total deformation, alternative stress and shear stress by using FEA software. And also we find out the life, safety factor and damage of alloy wheel by using S-N curve. S-N curve is input for a A.356.2 material. [6]

Prabha and Pendyala (2012) studied analyzing the stress and the displacement distribution in vehicle wheels subjected to conjoint influence of inflation pressure and the radial load this project work generalizes the application of Finite Element Analysis Techniques. The most commonly used considerations in Alloy wheels are illustrated. The analysis is carried out by using "ANSYS" and the model is done by using "PRO/E". The wheel is modeled by using ten nodded tetrahedron solid elements; for the analysis of linear elastic with isotropic conditions the constitutive material model is selected.[7]

A. *Tested Method*

The test equipment shall be designed to produce a constant bending moment on the center of the light-alloy wheel which rotates with a constant velocity. The dynamic cornering fatigue test is a standard SAE test,

Fig. 1: Set up of CFT

Which simulates cornering induced loads to the wheel. In the test system the test wheel is mounted to the rotating table, the moment arm is fixed to the wheel outer mounting pad with the bolts and a constant force is applied at the tip of the moment arm by the loading actuator and bearing, thus imparting a constant rotating bending moment to the wheel. If the wheel passes the dynamic cornering fatigue test, it has a good chance of passing all other required durability tests. The fatigue life of steel wheel rims was predicted by using the equivalent stress amplitude and steel alloy wheel rim S-N curve. The results from the steel wheel rim dynamic cornering fatigue test showed that the baseline wheel rim failed the test and its crack initiation was around the hub bolt hole area that agreed with the simulation. Using the method proposed in this paper, the wheel rim life cycle was improved to over 1.0×10^6 and satisfied the design requirement. 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 steel To apply bending moment to the wheel rim, a force may be applied either perpendicular or parallel to the plane of the mounting surface of the wheel at a specified distance (moment arm).

1) Test Conditions: Bending Moment

The bending moment M, in Nm applied in accordance with shall be determined by the following equation:

M = Sm./ p . W. r

where

Sm is a coefficient.

p is the friction coefficient between tyre and road, equal to o-7;

W is the maximum design load marked on the wheel, in decanewtons; and

r is maximum static loaded radius among those tyres that can be fitted to the wheel, in metres.

B. Meshing of the Wheel

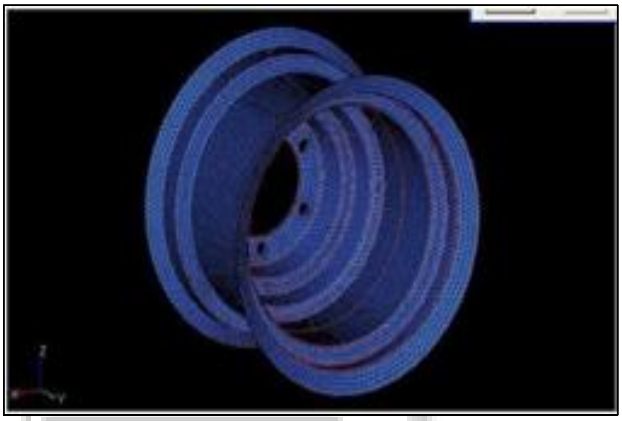

Fig. 2: Meshing of wheel Rim

Analysis: Fatigue analysis is used to determine the life, safety and damage of any component. The present work involves the determination of the life, safety factor and damage of alloy wheel and corresponding deformation, shear stress and alternative stress.

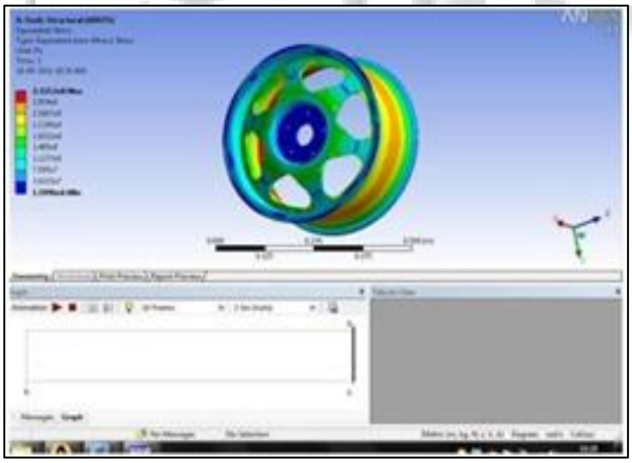

Fig. 3: Analysis of wheel Rim using ansys

III. CONCLUSION

A computational methodology is proposed for fatigue life and failure prediction of automotive steel wheel by the simulations of dynamic cornering fatigue test. Following with a short review of theoretical models, numerical simulation models were described in conjunction with bilinear elasto-plastic finite element stress analysis under wheel rotating loading. The following conclusions are drawn based on all study results above: 1. The fatigue failure critical locations are estimated, and the nodal points are identified as the critical locations, which agree well with the actual crack locations. 2. According to stress analysis of the key locations based on the critical plane theory, two principle stresses are not proportional and unstable principle planes are changing with loading direction. Principle planes variation changes a little, varying from 40° to 30° and the stress states of automotive steel wheel are in biaxial tensile and compression stresses during dynamic cornering fatigue test. 3. With the help of Cornering Fatigue Test (CFT) and blank size reduction process we eliminated some unused area from the wheel which helps in the prevention of wheel from failure.

REFERENCES

[1] Zheng, Zhanguang, Shuai Yuan, Teng Sun, and Shuqin Pan. "Fractographic study of fatigue cracks in a steel car wheel." Engineering Failure Analysis 47 (2015): PP, 199-207.

[2] Zheng, Z.G., Sun, T., Xu, X.Y., Pan, S.Q. and Yuan, S.,. Numerical simulation of steel wheel dynamic cornering fatigue test. Engineering Failure Analysis,. (2014) 39, pp.124-134

[3] T. Siva Prasad,T. 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] Sharma T, Shrivastava M & Jayaswa P,„Failure Analysis Of Wheel Rim‟, International Journal Of Automobile Engineering, Research & Development, vol. 3, issue 1, Mar 2013, 97-106

[5] Radulescu, Alexandru Valentin, Sorin Cananau, And Irina Radulescu. "MECHANICAL Testing Methods Concerning The Stress Analysis For A Vehicle Wheel Rim."The 3rd International Conference On Diagnosis And Prediction In Mechanical Engineering Systems Dipre 2012.

[6] Satyanarayana.S & Sambaiah C. „Fatigue Analysis of Aluminum Alloy Wheel Under Radial Load‟ International Journal of Mechanical and Industrial Engineering (IJMIE) ISSN No. 2231 –6477, Vol-2, Issue-1, 2012

[7] Prabha And Pendyala, „Design And Development Of Aluminium Alloy Wheels,‟ International Journal Of Advanced Science, Engineering And Technology. Issn 23195924 vol 1, issue 2, 2012, pp 55-60.