Analyze the Fatigue life of Aluminium Wheel Rim by using Rotary and Radial Fatigue Testing

[1]G.Arivazhagan, [2]D.Kalaimegam
[1]M.E. Student, [2]Assistant Professor
[1]CAD CAM Engineering, Department of Mechanical Engineering,
[1]The Kavery College of Engineering, Salem, India

Abstract - **This work aims to investigate the effect of slip angle on stress distribution and fatigue life of wheel rim. 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 from the aluminum wheel rotary fatigue bench test showed that the baseline wheel failed the test and its crack initiation was around the hub bolt hole area that agreed with the simulation. 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.**

IndexTerms - **Aluminum wheel, Fatigue analysis, Radial fatigue test (RFT), Wheel Rim, Fatigue life.**

I. INTRODUCTION

Wheels are one of the most important components of automobiles from the view point of structural safety. Aluminum wheels should not fail during service. Their strength and fatigue life are critical. 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.

A well designed wheel is the foundation which adds strength, stability and durability to a tyre. Hence, the increased urge to make them safer and reliable. 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.

II. ROTARU FATIGUE TEST

For predicting the wheel fatigue life, the nominal stress method was integrated into the CAD/CAE technology to simulate the rotary fatigue test. In the rotary fatigue test, a wheel was spun to bear a moment to simulate the process of turning corner continued the wheel's ability bearing the moment.

According to the rotary fatigue test condition as specified in the SAE test procedure, a wheel was mounted on a rotating table. A shaft was attached to the center of the wheel where a constant normal force was applied as shown in Fig.1.

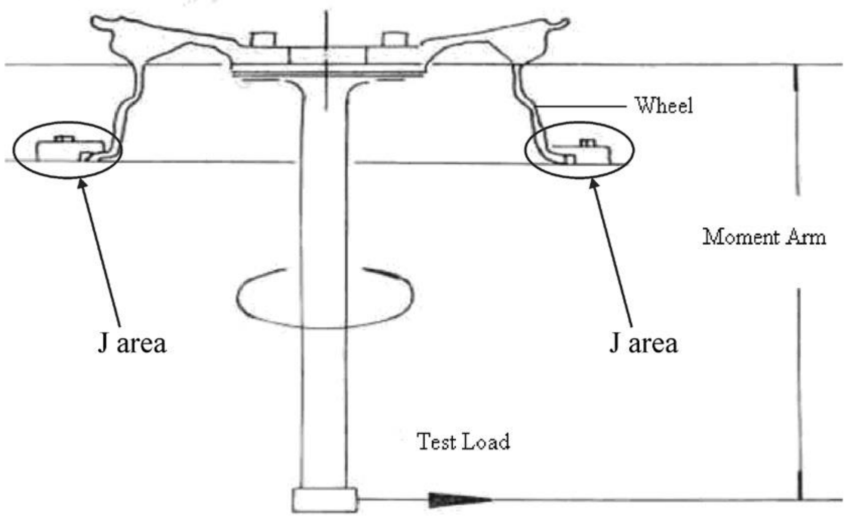


Fig.1. Layout of wheel rotary fatigue test

Test moment is calculated as follows

$$\mathbf{M} = (\mu\mathbf{R} + d)F$$

Where,

M is the moment (Nm), it is the strengthening moment the real vehicle bears;
μ is the friction coefficient between tires and the road;
R is the tire static load radius (m);
d is the offset of wheel (m);
F is the maximum rated load (N), which can be obtained by standards;
λ is the strength coefficient

III. Wheel rim nomenclature

The wheel rim is actually the name for the cylindrical part where the tyre is installed. A wheel is the name for the combination between the rim and disc plate.

Once the disc plate is installed inside the cylinder this assembly becomes a wheel or wheel rim.

- **Wheel rim or wheel:** Wheel is generally composed of rim and disc.
- **Rim:** This is a part where the tyre is installed. It is outermost radial part of the wheel serves to accommodate the tyre.
- **Disc:** This is a part of the rim where it is fixed to the axle hub. It serves as the interface between rim and hub.
- **Offset:** This is a distance between wheel mounting surfaces where it is bolted to hub and the centerline of rim.
- **Flange:** The flange is a part of rim which holds the both beads of the tyre.
- **Bead Seat:** Bead seat comes in contact with the bead face and is a part of rim which holds the tyre in a radial direction (i.e) it is the portion of the wheel rim below the rim flange providing radial support to the bead of the tyre.
- **Hump:** It is bump what was put on the bead seat for the bead to prevent the tyre from sliding off the rim while the vehicle is moving.
- **Well:** This is a part of rim with depth and width to facilitate tyre mounting and removal from the rim.

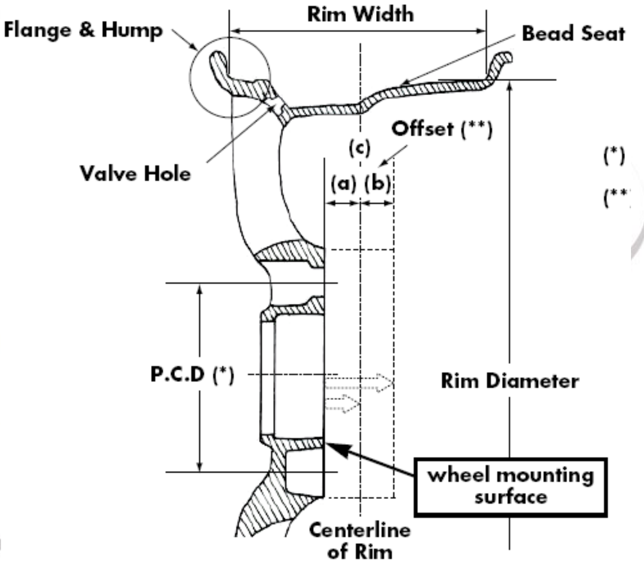


Fig. 2: Wheel rim nomenclature

IV. Finite element analysis

Generating CAD model is first step towards CAE analysis of any product. In the present case, CAD model of the complete wheel rim has been generated using CATIA V5R17 software. The CAD model of the wheel rim used for analysis.

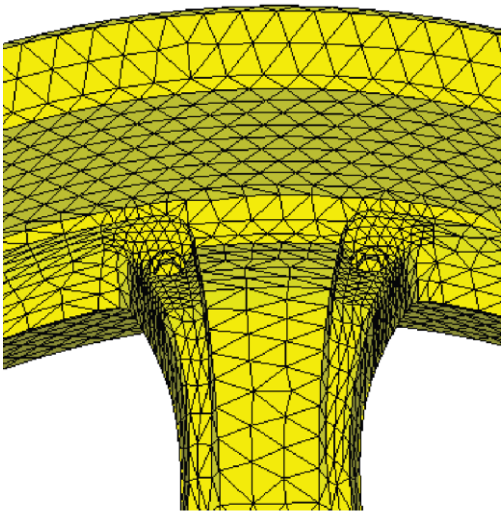
The stress analysis was performed by using a commercial FEA software ABAQUS. ABAQUS is powerful engineering finite element simulation software with a library of rich elements that can be used to simulate any geometry and typical engineering properties of materials. The software provides solutions to problems ranging from relatively simple linear analysis to many complex non-linear problems.

Finite element models of the wheel have been created in OPTISTRUCT using HYPERMESH v 9.0.

For fatigue life prediction Msc.Fatigue v 2005 use. The CAD model in IGES format is imported in Hyper Mesh for the preparation of FE model. Meshing is performed on surface by using shell element and these elements are offset on other surface by "ELEMENT OFFSET" process. These shell elements converted in solid brick element by "LINEAR SOLID" command. The three dimensional models are used hexahedral or brick elements.

Wheel Meshing

When the wheel is meshed, in estimated data change gradient big spot, it needs to adopt more intensive grid to better reflect the changes of data. In the wheel hub, the danger zones are rim, junction with rim and rib, and the areas around bolt hole. The stress concentration region corresponding grid distribution should be dense; but the rim the stress cannot consider nearly in the entire parsing process, the corresponding grid distribution should be sparse



Wheel meshing: junction with rim and rib

Loads and Boundary Conditions

In the FEA model, loading and boundary conditions were set up similarly to those in the bench test. The wheel was constrained around flange edge of the rim and loaded with a constant force at the end of the shaft. The load shaft and wheel were connected by bolts. Due to the main concern being wheel deformation, the load shaft in the FEA analysis was defined as a rigid body, using tie connection with wheel.

V. FATIGUE ANALYSIS

The main methods to determine the model fatigue life are fatigue life test and fatigue life analysis. Fatigue life test has a high-cost and long cycle. Fatigue life analysis is based on the fatigue properties of materials and load time histories.

It can predict fatigue life at the product design stage, reduce the number of experimental prototypes, and shorten the development cycle. Fatigue life analysis includes nominal stress analysis, crack initiation analysis, stress-strain field intensity and energy law.

A method that combined nominal stress analysis and static analysis was used to predict wheel fatigue life. To validate the prediction of fatigue life, wheel rotary fatigue test was conducted. The test results showed that the prediction of fatigue life using FEA was consistent with the result of bench test.

VI. CONCLUSION

A fatigue lifetime prediction method of aluminum alloy wheels was proposed to ensure their durability at the initial design stage. To simulate the rotary fatigue test, static load FEM model was built using ABAQUS.

The finite element analysis as well as experimental analysis of passenger car wheel rim performed for radial load with the effect of slip angle on stress distribution and fatigue life. Aluminum alloy wheel rotary fatigue bench test was conducted.

The test result showed that the prediction of fatigue life was consistent with the physical test result. These results indicate that the fatigue life simulation can predict weakness area and is useful for improving aluminum alloy wheel. These results also indicate that integrating FEA and nominal stress method is a good and efficient method to predict aluminum alloy wheels fatigue life.

VII. ACKNOWLEDGMENT

I proposed my deep gratitude and sincere thanks to my supervision **Mr.D.Kalaimegam.,M.E., Assistant Professor, Department of Mechanical Engineering at The Kavery College of Engineering** for his valuable, suggestion, innovative ideas, constructive, criticisms and inspiring guidance had enabled me to complete the paper present work successfully.

REFERENCES

[1] Vatroslav Grubisic, Gerhard Fischer, "Automotive Wheels, Method and Procedure for Optimal Design and Testing," SAE Technical Paper Series, 830135, 1983.
[2] Miloslav Reisner, Richard I. DeVries, "Finite Element Analysis and Structural Optimization of Vehicle Wheels," SAE Technical Paper Series, 830133, 1983.
[3] Hsu, Y.-L., Wang, S.-G., Liu, T.-C. (2004). “Prediction of fatigue failure of aluminum disc wheel using the failure probability contour based on historical test data”. Journal of the Chinese Institute of Industrial Engineers, vol. 21, no. 6, p. 551-558.
[4] Kocabicak, U., Firat, M. (2001). “Numerical analysis of wheel cornering fatigue tests, Engineering Failure Analysis”, vol. 8, p. 339- 354.
[5] D. H. Wright, "Test Methods for Automotive Wheels," I Mech E Conference Publications, Conference Code 03757, 1983.

[6] John Kinstler, "The science and methodology of SAE wheel fatigue test specifications" SAE technical publication 2005 -01-1826,2005
[7] 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.
[8] Xiaofeng, W., Zhao, L., Xiaoge, Z. (2007)."Finite element analysis of a wheel based on the wheel dynamic cornering fatigue test". Tractor & Farm Transporter, vol. 34, no. 1, p. 45-47.
[9] Tenglun, Z. (2007). "The application of ABAQUS 6.6 in mechanical engineering", China Water Power Press, Beijing.
[10] Karandikar, H.M. (1990). "Fatigue life prediction for wheels by simulation of the rotating bending test". Society of Automotive Engineer, Warrendale, Pennsylvania.
[11] Bannantine, J.A., Corner, J.J., Handrock, J.L. (1990). "Fundamentals of metal fatigue Analysis", Prentice Hall, New Jersey.
[12] Pook, L. (2007). "Metal fatigue: what it is, why it matters, solid mechanics and its applications", Springer Verlag, Berlin.