Fatigue Failure Analysis of Rim

Kunal H. Borase[1], Dr. E.R.Deore[2]

1 PG Student, Mechanical Engineering Department, S.S.V.P.S.'s. B.S.D. Dhule ,Maharastra, India
2 Prof., Mechanical Engineering Department, S.S.V.P.S.'s. B.S.D. Dhule ,Maharastra, India

---

Abstract:*A Finite Element Method methodology is proposed for fatigue life and failure prediction of automotive steel wheel by the simulations of dynamic radial fatigue test (RFT). 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.*
Keywords: Finite Element Method, Dynamic Radial Fatigue Test, elasto – plastic, Wheel.

Introduction

Fatigue is an important consideration for components and structures subjected to repeated loadings is one of the most difficult design issues to resolve. Experience has shown that large percentage of structural failure are attributed to fatigue and as a result, it is an area which has been and will continue to be the focus of both fundamental and applied research. Fatigue design provisions are only recently included in the aluminum association specialization. Related loadings of a component or structure at stresses above the design allowable for static loadings may cause a crack or racks to form. Under cyclic loading these cracks may continue to grow and precipitate a failure. When the remaining structure can no longer carry the loads, the mechanism of crack formation and growth is called fatigue. [12]

Radial Fatigue Test Experimentation

The radial fatigue test is a standard SAE test, which is to be carried dynamically by exerting the radial loads to the wheel. RFT also takes Tire Air Pressure present over the wheel rim. Fig 5.1 shows the test system in which the test wheel is mounted to the rotating shaft, 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.

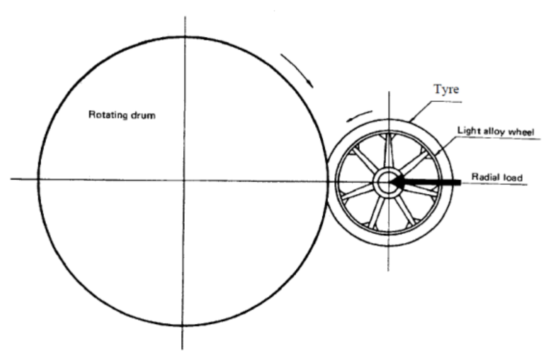

Fig1. Schematic Diagram of RFT

Finite Element Fatigue Analysis

We have studied the rim Analysis with constant tire inflation pressure 40 Psi (275.79Kpa) and

Variation in radial load applied on rim. As tire inflation pressure is one of the constant force acting on rim throughout its working conditions, while the radial load acting on rim is variable. In order to evaluate the performance of rim for different radial load, analysis carried out with constant tire inflation pressure o 275.79 Kpa and different radial load employed on rim as follows:

1. 150 kg
2. 175 kg
3. 200 kg

A1] Analysis under Inflation Pressure of 275.79 kpa and Load 150kg-From displacement plot it can be seen that the displacement or strain in Rim is negligible, the maximum strain value is 0.26334 which is observed in well area of Rim. While the maximum Von Mises **stress value is 47.5 MPa and minimum stress value is 0.117 MPa.**

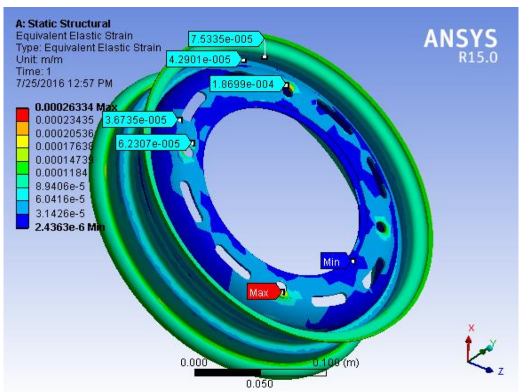

Fig.2 Displacement plot at 275.79 kpa pressure 150kg load

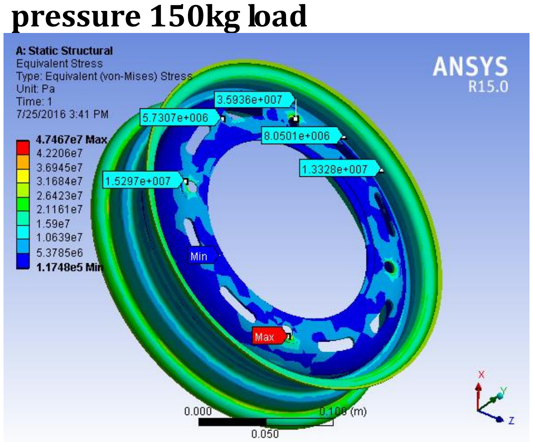

Fig.3. Von Mises Contour plot at 275.79 kpa pressure and load 150kg.

A2] Analysis under Inflation Pressure of 275.79 kpa and Load 175kg.

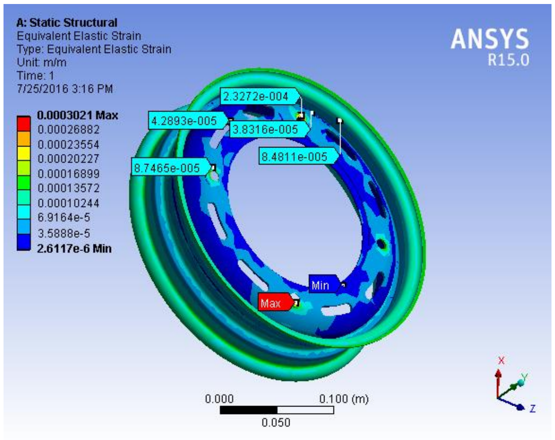

Fig.4 Displacement plot at 275.79 kpa pressure and load 175kg .

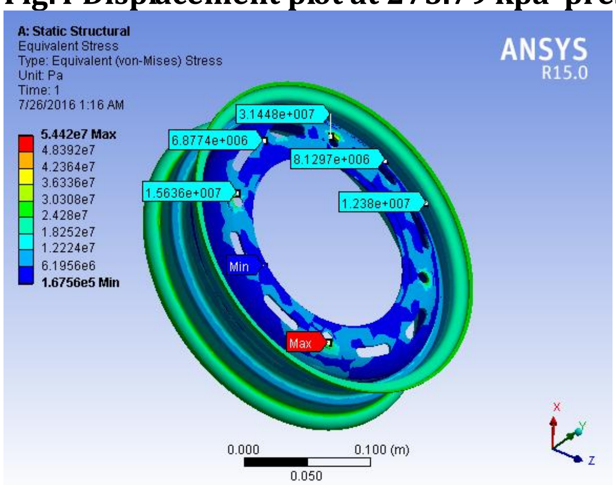

Fig.5.Von Mises Contour plot at 275.79 kpa pressure and load 175kg.

From displacement plot it can be seen that the displacement or strain in Rim is negligible, the maximum strain value is 0.3021 which is observed in well area of Rim. While the maximum Von Mises stress value is 54.42 MPa and minimum stress value is 0.167 MPa.

A3] Analysis under Inflation Pressure of 275.79 kpa and Load 200kg .

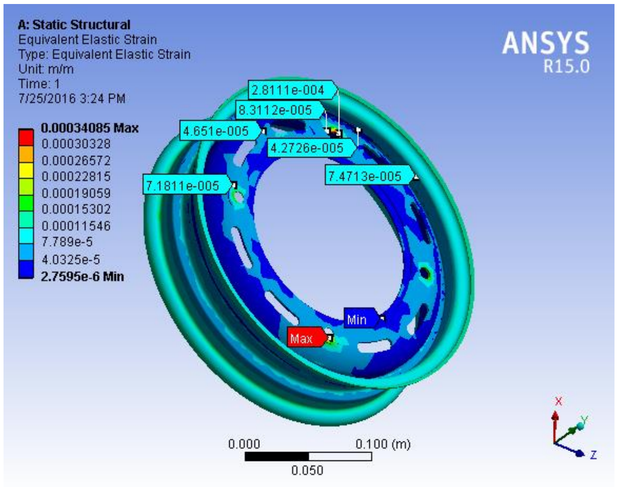

Fig.6 Displacement plot at 275.79 kPa
Pressure load 200kg.

From displacement plot it can be seen that the displacement or strain in Rim is negligible, the maximum strain value is 0.3408 which is observed in well area of Rim. While the maximum Von Mises stress value is 61.37 MPa and minimum stress value is 0.223 MPa.

Strain Gauge Location	Applied load Tire inflation pressure 275.79 kpa and 200 Kg load		
Trial 1	**ESA strain**	**FEA strain**	**% Diff**
1	0.075042	0.071811	4.50
2	0.048361	0.046510	3.98
3	0.297133	0.281110	5.70
4	0.044828	0.042726	4.92
5	0.078030	0.074713	4.44

Table1. Comparison of experimental observation and FEA results

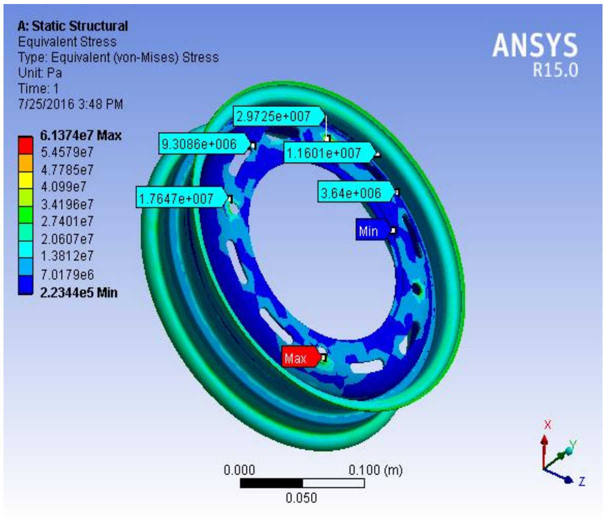

Fig.7 Von Mises Contour plot at 275.79 kPa (40Psi) pressure and load 200kg.

By utilizing strain gauges the experimental analysis of rim us carried out to investigate the strain induced in rim under defined loading conditions. The comparison between experimental strain values with FEA strain values shows variation in results ranging from 0.6% to 6%.

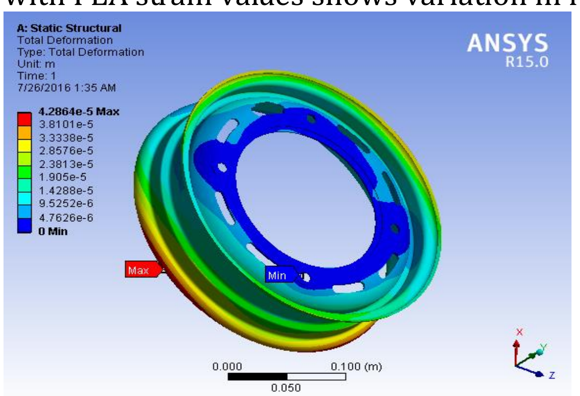

Fig.8 Total Displacement plot at 241.32 kPa inflation pressure load 200kg.

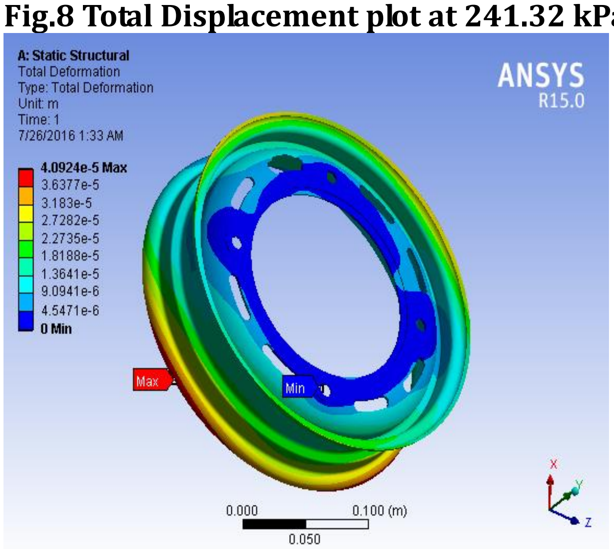

Fig.9 Total Displacement plot at 220.632 kpa and 200kg load

Strain Gauge Location	Applied load Tire inflation pressure 275.79 kpa and 200 Kg load		
Trial 2	ESA strain	FEA strain	% Diff
1	0.074575	0.071811	3.85
2	0.048630	0.046510	4.56
3	0.297049	0.281110	5.67
4	0.044571	0.042726	4.32
5	0.077290	0.074713	3.45

Table2. Comparison of experimental observation and FEA results

.

Conclusion

Following the described objective, the state of stress and mechanical response of aluminum automobile rims has been established, and the effects of inflation pressure are now well understood, in addition to the imposed radial load. Analysis of opening geometry and the effects on the state of stress in the wheel is now well defined. Provided within are the observations of the effects of environmental degradation rims has been documented, in addition to the established corrosion testing procedures which provided the reader a well-rounded background on the subject.The extensive literature search revealed the need to further investigate the effect of inflation pressure on the state of stress in the rim, and to provide a finite element analysis using the more accurate brick element rather than the shell or plate element as in past publications. Below provides a summary on each unique objective stated.

Experimental results were compared to finite element results, and to the published literature. The contribution of this research was not to bench mark finite element stress analysis of the rim, this has been done by others in the references cited.

The main contribution is

1) The experimental results and the modifications and identification of the proper methods for applying the radial load,

2) Comparisons of uniquely identified rim geometry's and the effects of inflation pressure on the state of stress,

3) Applying polynomial and Fourier series interpolations to actual tire rim nonlinear finite element model output data, and using the polynomials as input loading for the finite element model,

4) Observations and documentation of environmental degradation in a large rim inventory of a local test center and,

5) Determining the effect of drilling a hole in the rim, and its effect on the state of stress at critical locations.

The study is helpful towards:

i) Reducing the cost of calibration and maintenance of testing equipment.
ii) Providing the useful database for the product development.
iii) Replacing the complex trial and error methods like Taguchi Method etc.
iv) Obtain the optimization of topology and size of cooling holes towards minimization of weight and material.
v) Could leads to robustness in design with high accuracy.
vi) Providing the interface for stress analysis, useful for confirmation of new set up of testing rig or after calibration.

References

1. Wan, Xiaofei, Yingchun Shan, Xiandong Liu, Haixia Wang, and Jiegong Wang. "Simulation of biaxial wheel test and fatigue life estimation considering the influence of tire and wheel camber." *Advances in Engineering Software* 92 (2016): 57-64.
2. D'Andrea, Antonio, and Cristina Tozzo. "Interface stress state in the most common shear tests." *Construction and Building Materials* 107 (2016): 341-355.
3. Esmaeili, F., A. Rahmani, S. Barzegar, and A. Afkar. "Prediction of fatigue life for multi-spot welded joints with different arrangements using different multiaxial fatigue criteria." *Materials & Design* 72 (2015): 21-30.
4. Zheng, Zhanguang, Shuai Yuan, Teng Sun, and Shuqin Pan. "Fractographic study of fatigue cracks in a steel car wheel." *Engineering Failure Analysis* 47 (2015): 199-207.
5. Fang Gang, Wei-Ran Gao, and Xiao-Ge Zhang. "Finite element simulation and experiment verification of rolling forming for the truck wheel rim." *International Journal of Precision Engineering and Manufacturing* 16, no. 7 (2015): 1509-1515.
6. Fanelli, Pierluigi, and Francesco Vivio. "A general formulation of an analytical model for the elastic–plastic behaviour of a spot weld finite element." *Mechanics Research Communications* 69 (2015): 54-65.
7. Zheng, Zhan-Guang, Teng Sun, Xi-Yong Xu, Shu-Qin Pan, and Shuai Yuan. "Numerical simulation of steel wheel dynamic cornering fatigue test." *Engineering Failure Analysis* 39 (2014): 124-134.
8. Zhao, Dawei, Yuanxun Wang, Xiaodong Wang, Xuenong Wang, Fa Chen, and Dongjie Liang. "Process analysis and optimization for failure energy of spot welded titanium alloy." *Materials & Design* 60 (2014): 479-489.
9. Xu Fengxiang, Guangyong Sun, Guangyao Li, and Qing Li. "Failure analysis for resistance spot welding in lap-shear specimens." *International Journal of Mechanical Sciences* 78 (2014): 154-166.
10. Zhang, Hongqiang, Xiaoming Qiu, Fei Xing, Jie Bai, and Jianhe Chen. "Failure analysis of dissimilar thickness resistance spot welded joints in dual-phase steels during tensile shear test." *Materials & Design* 55 (2014): 366-372.
11. Senatore, C., and K. Iagnemma. "Analysis of stress distributions under lightweight wheeled vehicles." *Journal of Terramechanics* 51 (2014): 1-17.
12. Topac, M. M., S. Ercan, and 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.
13. Chang, Chia-Lung, and Shao-Huei Yang. "Simulation of wheel impact test using finite element method." *Engineering Failure Analysis* 16, no. 5 (2009): 1711-1719.
14. Bhattacharyya, Sandip, M. Adhikary, M. B. Das, and Sudipto Sarkar. "Failure analysis of cracking

in wheel rims–material and manufacturing aspects." *Engineering Failure Analysis* 15, no. 5 (2008): 547-554.

15. Carboni, M., S. Beretta, and A. Finzi. "Defects and in-service fatigue life of truck wheels." *Engineering Failure Analysis* 10, no. 1 (2003): 45-57.

16. J.Stearns, T.S.Srivatsan, A.Prakash, P.C.Lam, 'Modeling the Mechanical response of an aluminium alloy automotive rim', Univ. Akron, 2003,Pp.1-4.