International Journal of Research in Engineering, Science and Management
Volume-2, Issue-6, June-2019
www.ijresm.com | ISSN (Online): 2581-5792

Static Structure and Fatigue Analysis of Automotive Vehicle Rim

Praveen Bhajantri[1], Prem Kumar Magundanavar[2], Santosh Kumar Malipatil[3]
[1,2]B.E. Student, Department of Mechanical Engineering, Maratha Mandal Engineering College, Belagavi, India
[3]Assistant Professor, Dept. of Mechanical Engineering, Maratha Mandal Engineering College, Belagavi, India

***Abstract*: The project discusses the use of finite element technique for simulation test of functionality of automotive wheel rim behavior during operation. A specific design wheel is used for this purpose. The purpose of the car wheel rim is to provide a firm base on which tire could be fitted. Its dimensions, shape should be suitable to adequately accommodate the particular tire required for the vehicle. The wheel is to be designed using 3- dimensional modeling software. The 3d model thus designed would consider the manufacturing feasibility and ease of assembly into the application. The load is applied on the rim to find out the total deformation, alternative stress and principal stress by using FEA software. This in turn leads to find out the life, safety factor and damage of spoke wheel by using S-N curve. In this project, by observing the results of both static and harmonic analysis, the comparison between materials is suggested as for designing of the wheel rim. In this thesis, study will be carried out on an Aluminum Alloy Wheel.**

***Keywords*: Aluminum Alloy Wheel rim, FEA, CATIA, Ansys Workbench**

1. Introduction

The wheel is a device that enables efficient movement of an object across a surface where there is a force pressing the object to the surface. Early wheels were simple wooden disks with a hole for the axle. Because of the structure of wood a horizontal slice of a trunk is not suitable, as it does not have the structural strength to support weight without collapsing; rounded pieces of longitudinal boards are required [3], [4].

2. Methodology

The broad methodology adopted in this project is as shown in the flow chart. This projects aims to include Fem concepts and composite designs in understanding the way to study the fatigue characteristics of a wheel rim using finite element methods.

3. Analysis results

A. Linear Static Analysis

A linear static analysis is an analysis where a linear relation holds between applied forces and displacements. In practice, this is applicable to structural problems where stresses remain in the linear elastic range of the used material. In a linear static analysis the model's stiffness matrix is constant, and the solving process is relatively short compared to a nonlinear analysis on the same model. Therefore, for a first estimate, the linear static analysis is often used prior to performing a full nonlinear analysis [1], [2].

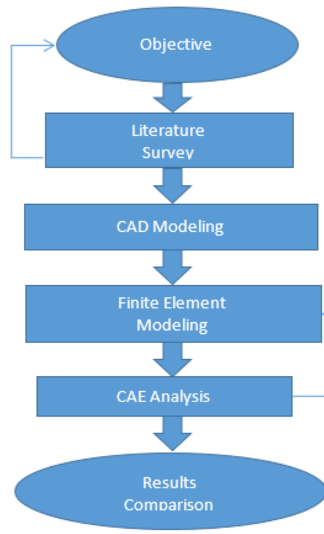

Fig. 1. Methodology flowchart

Table 1
Material Properties

Material	Density (kg/m^3)	Mass (kg)	Young's modulus (Mpa)	Poisson's Ratio	Volume m^3
AL-6061	2700	19.78	68900	0.33	0.00732

- *Loading and boundary conditions*

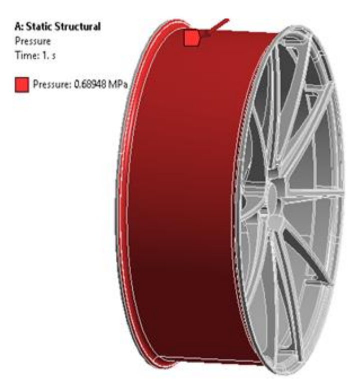

Fig. 2. Loading and boundary conditions

Pressure of 0.68948MPa applied on the Rim Radial load of 23153N applied X-direction.

Validation

1) Stresses

We have

Circumferential stress Qc = $\frac{p \times d}{2 \times t}$

Where,

p = pressure of wall rim = 0.68Mpa

d = diameter of rim = 570mm

t = thickness of rim = 15mm

$$Qc = \frac{0.68 \times 570}{2 \times 15}$$

Qc = 12.92Mpa

FEA model stress = 13.16Mpa

$$\% \text{ difference} = \left(\frac{13.16 - 12.92}{13.16}\right) \times 100$$

= 1.8% < 5% their for acceptable

2) Deformation

Deformation of thin cylinder is given by

$$D = \frac{Pi \times Di \times Do^2}{E \times (Do^2 - Di^2)}$$

$$D = \frac{0.689 \times 565 \times 570^2}{68900 \times (570^2 - 565^2)}$$

D = 0.32 mm

FEA deformation = 0.4mm

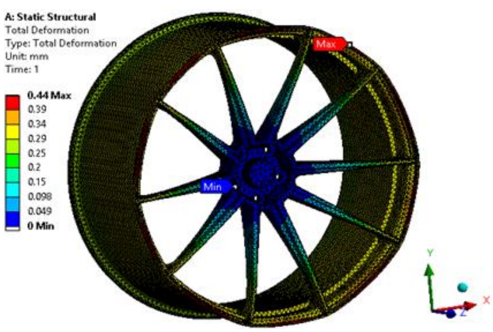

Fig. 3. Total deformation observed in the rim is 0.44mm

Von –Mises Stress

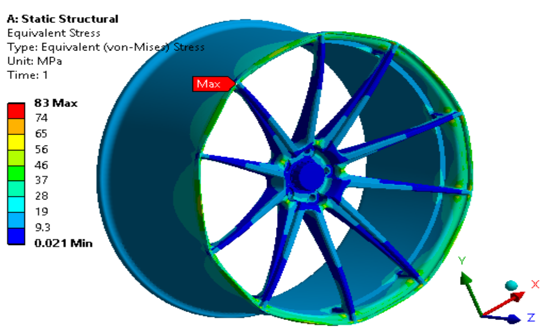

Fig. 4. Max Equivalent stress of 83 MPa is observed which a singular stress @ Fillet corner Location

Maximum Principal Stress

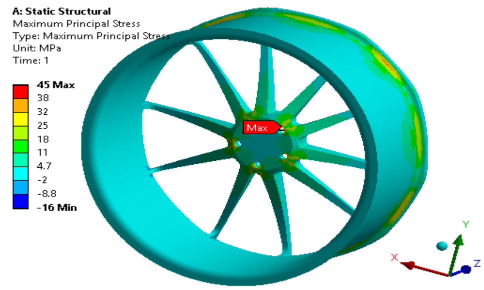

Fig. 5. Max principal stress of 45 MPa is observed which a singular stress @ Bolt Location

Minimum Principal Stress

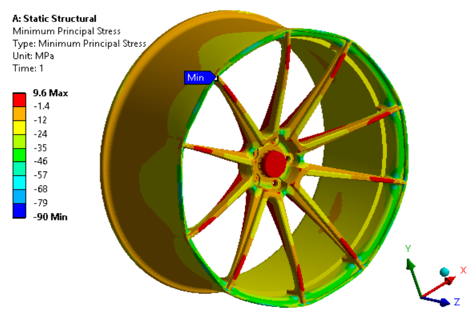

Fig. 6. Min principal stress of -90 MPa is observed which is a singular stress @ Fillet corner Location

B. Modal Analysis

Modal analysis is the study of the dynamic properties of systems in the frequency domain.

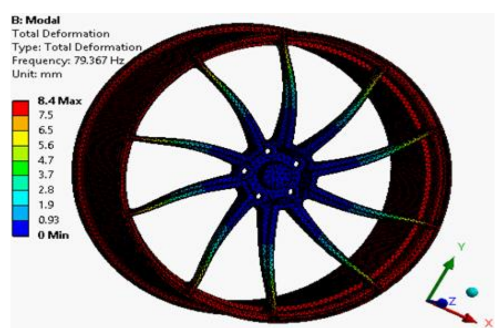

Fig. 7. System level Twisting mode along Z axis @79.4 Hz

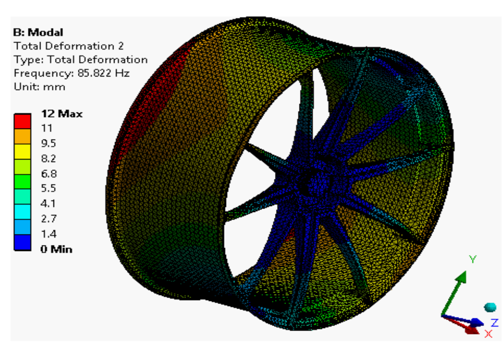

Fig. 8. System level Bending mode along Z axis @85.5 Hz

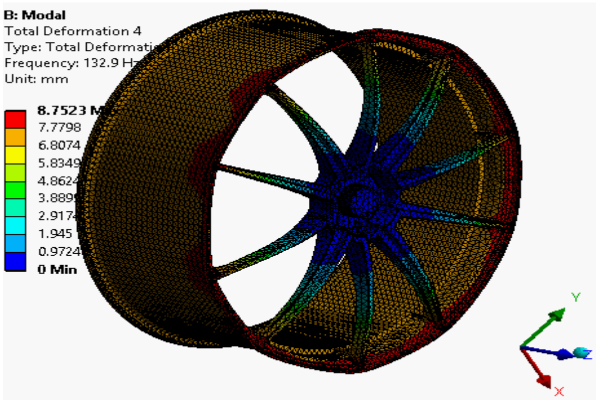

Fig. 9. System level Bending mode along Z axis @ 132.9 Hz

C. Harmonic Analysis

Harmonic analysis is branch of mathematics concerned with the representation of functions or signals as the superposition of basic waves, and the study of and generalization of the notions of Fourier series and Fourier transforms (i.e. an extended form of Fourier analysis).

Harmonic stress

Von-Mises Stress

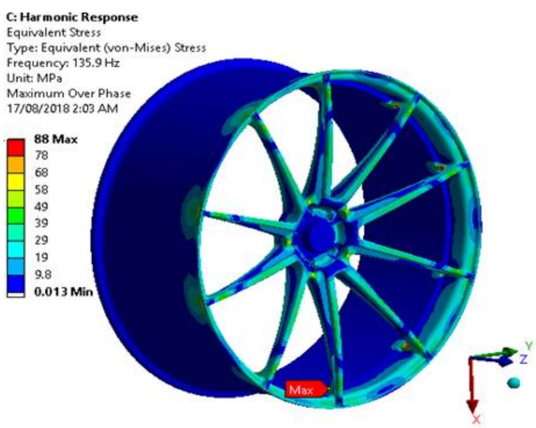

Fig. 10. Max equivalent stress of 88 MPa is observed which a singular stress @ Rib Location

Maximum Principal Stress

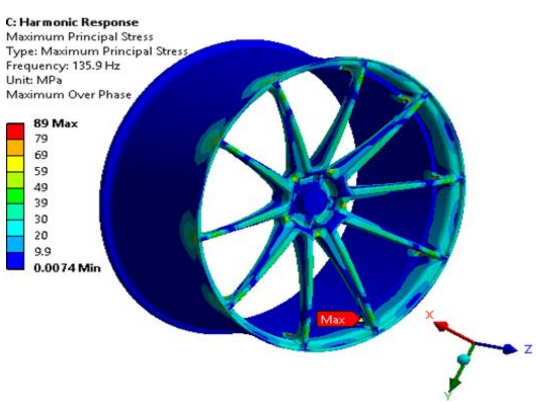

Fig. 11. Max. principal stress of 89 MPa is observed which is a singular stress @ Rib Location

Minimum Principal Stress

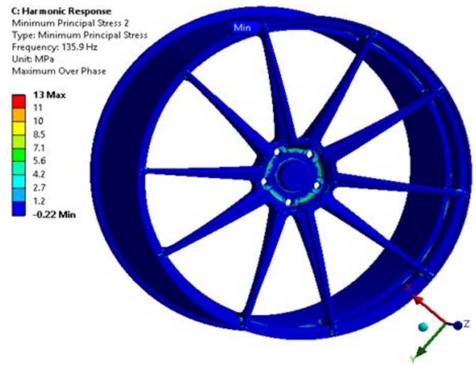

Fig. 12. Min. principal stress of -0.22 MPa is observed

Table 1
Stress conditions

	Static Structural	Yield Strength	Factor of Safety
Equivalent Stress (MPa)	66.1	276	4.2
Max. principal Stress (MPa)	20.5	276	13.5
Min. Principal Stress (MPa)	-78.9	276	3.5

Component is having FOS of 3.5 considering maximum stress conditions.

Table 2
Result summary

	Harmonic Stress
Equivalent Stress (MPa)	66
Max. principal Stress (MPa)	56
Min. Principal Stress (MPa)	-0.22

Acceleration plot

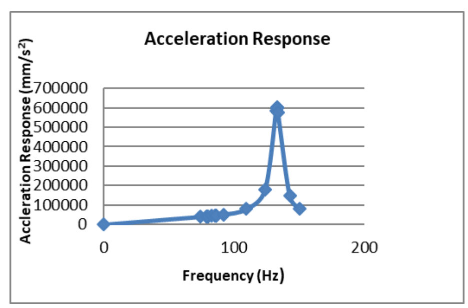

Fig. 13. Maximum Response observed of 6.0114e+005 mm/s² @135 Hz

D. Fatigue Analysis

- Gross Vehicle Weight – 2320 Kg
- Maximum Loading weight – 480 Kg
- Minimum Kerb Weight - 1860
- The total weight on wheels =4660Kg.
- Which is distributed over 4 tires, therefore 1165 Kg per tire.
- But we assumed for worst case condition considering Braking, Cornering etc. as 1180 kg of Weight
- Therefore the Maximum vertical load is

$$F= 1180x9.81=11575.8 \text{ N}$$

The radial load is calculated by the expression

$F_r = S. F$

Where, F is the highest load carried by the wheel, and S 2.0 (Accelerated test factor)

Fatigue Assessment

For Correction from R= -1 to R=0

$$\sigma ac = \frac{\sigma a}{1-(\frac{\sigma m}{\sigma ts})}$$

$$\sigma a = 66\ MPa$$

$$\sigma m = 66.1\ MPa$$

$$\sigma Uts = 324 MPa$$

$$Ultimate\ Tensile\ Strengt$$

$$\sigma ac = \frac{66}{1-(\frac{66.1}{324})}$$

$$\sigma ac = 82.91\ MPa$$

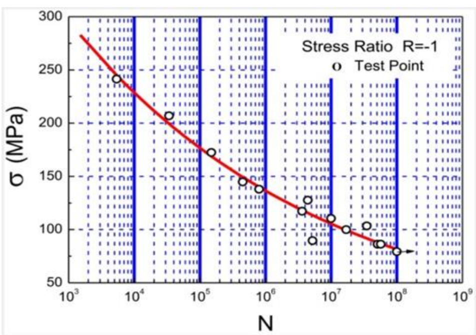

Fig. 14. S-N curve for AL-6061 Alloy for R= -1

4. Conclusion

This project deals with the design of metal and composite wheel rim for automobile application which is carried out paying special reference to optimization of the mass of the wheel. The Finite Element analysis from literature survey shows that the optimized mass of the wheel rim could be reduced to around 20% as compared to the existing solid disc type Al alloy wheel. The FE analysis is carried out for the optimized to study its behavior for static load. The Fatigue life estimation by finite element analysis, under radial fatigue load condition, is carried out to analyze the stress distribution and resulted displacement in the alloy wheels. S-N curve of the component is used to study the endurance limit.

References

[1] 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, Volume 2, Issue 6, May 2014.

[2] Sourav Das, "Design and Weight Optimization of Aluminium Alloy Wheel", International Journal of Scientific and Research Publications, Volume 4, Issue 6, June 2014.

[3] Rajarethinam P., Periasamy K., "Modification of Design and Analysis of Motor Cycle Wheel Spokes", International Journal of Modern Engineering Research, pp. 123-127.

[4] Liangmo Wang - Yufa Chen - Chenzhi Wang - Qingzheng Wang, "Fatigue Life Analysis of Aluminum Wheels by Simulation of Rotary Fatigue Test", Journal of Mechanical Engineering, vol. 57, no. 1, pp. 31-39, 2011.