Fatigue Life Estimation of An Aluminium Wheel Rim Using Finite Element Analysis

Mandage A.P.[1], Sharma M.H.[2], Rayate A.K.[3], Kange P.N.[4], Hirulkar N.S.[5]
[1, 2, 3, 4, 5] Department of Mechanical Engineering
[1, 2, 3, 4, 5] Sinhgad Institute of Technology, Lonavala, Maharashtra-410401, India.

Abstract-*In this era, there is a call for sustainable designs with higher life expectancy and so automotive industry demands flawless wheel rims. The paper deals with the design of aluminium wheel rim for automobile application. The Fatigue life estimation by finite element analysis, under radial fatigue load condition, is carried out to analyse the stress distribution and resulted displacement in the alloy wheel. S–N curve approach is taken for envisaging the fatigue life of aluminium wheels by simulating transient structural analysis with cyclic loads as they are found to converge with experimental results, safety factor for fatigue life is also assessed. Paper also deliberates about experimental designs and response surface of discussed experiment using ANSYS to achieve validate results in optimum range.*

Keywords-Aluminium Wheel Rim, Ansys, Automobile, Fatigue Life, Finite Element Analysis, Response Curve, S-N Curve, Transient Structural.

I. INTRODUCTION

This is an exciting time to be a part of the automobile industry which is witnessing major structural changes. Even though the demands on the industry have never been greater. Customer expectations of vehicle quality, safety, reliability and utility are at an all-time high. There is a call for sustainable designs with higher life expectancy thus the automobile manufacturers are increasingly investigating and developing new design tools to help improve the quality of their products. Computer aided engineering helps reduce the time necessary to produce a new design. It also improves the quality of design.

The wheel is a critical component in the automobile and bears the weight of the car as well as helps the tire to maintain contact between the car and the road. The wheel is exposed to very hazardous environmental conditions. For high and optimal performance, the wheel is designed to meet some safety and engineering criteria. The wheel should be able to withstand the impact of shock and vibrations and be able to bear the weight of the car and the passengers; it should be light in weight but highly durable.

Fatigue fracture, like yielding, is due to formations of cracks at the microscopic level and lengthened by continued applications of stress. It differs however in the manner the stress is applied. Fatigue fracture is instigated by cyclical stresses on the material, which occurs as a result a process of crack nucleation and pre-existent cracks in a material.

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.

II. TRANSIENT STRUCTURAL ANALYSIS

Transient structural analysis is used to determine the dynamic response of a structure under the action of any general time-dependent loads. You can use it to determine the time-varying displacements, strains, stresses, and forces in a structure as it responds to any transient loads. The time scale of the loading is such that the inertia or damping effects are considered to be important.

A. Material Properties

Material being used is Aluminium Alloy.

Table: 1

Sr. No.	Property	Value
1.	Density	2770 Kgm^-3
2.	Young's Modulus	7.1e^10 Pa
3.	Poisson's ratio	.33
4.	Yield strength	2.8e^8 Pa
5.	Ultimate strength	3.1e^8 Pa

The properties were given as input for the model. To calculate fatigue life alternating stress ratio against life cycles is required. The graph is as follows:

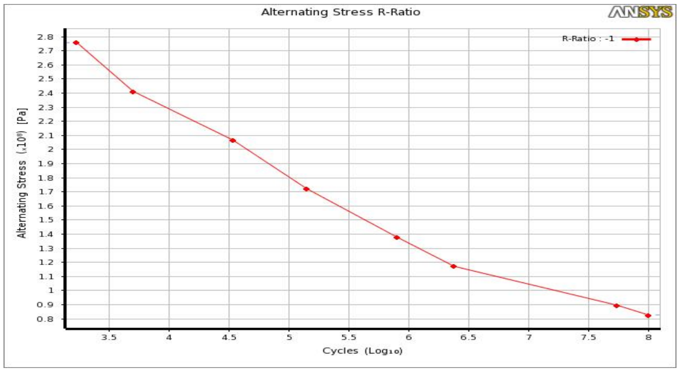

Figure 1: Alternating stress vs cycles

B. CAD Modelling

For better analysis, the wheel is modelled using a CAD software, Solidworks, using different features ranging from revolve features, sweep features, extrude boss base and extrude cut, fillet and surface features. To model the wheel a picture sketch was used. This was to ensure that the actual shape of the rim is used while modelling so as to reduce errors. Figure 2 is the CAD model of the rim.

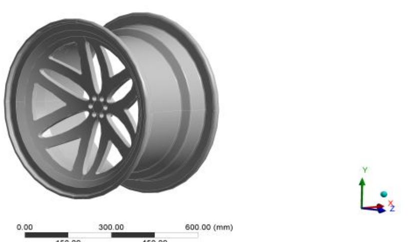

Figure 2: CAD Model

C. Meshing

The ANSYS workbench uses a finite element method to discretize the model into finite elements. Finer the size accurate will be the answer. But at the same time the software needs more memory and time for processing. Thus optimum meshing is used.

Figure 3 shows the discretised model of the rim.

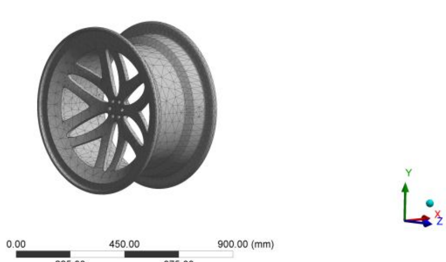

Figure 3: Discretised Model

D. Analysis Setup

The tyre pressure was kept 36 psi which is equivalent t0.248 MPa as per the SAE norms. The front face of the wheel was exposed to the atmospheric pressure of 0.101 MPa. The wheel was given a rotational velocity. Initial time step for the analysis of 1 second was given as 1e^-04 second. Minimum time step was kept same as initial time step whereas maximum time step was 0.1 second.

E. S-N curve

Many non-ferrous metals and alloys, such as aluminium, magnesium, and copper alloys, do not exhibit well-defined endurance limits. These materials instead display a continuously decreasing S-N response. In such cases a fatigue strength for given number of cycles must be specified.

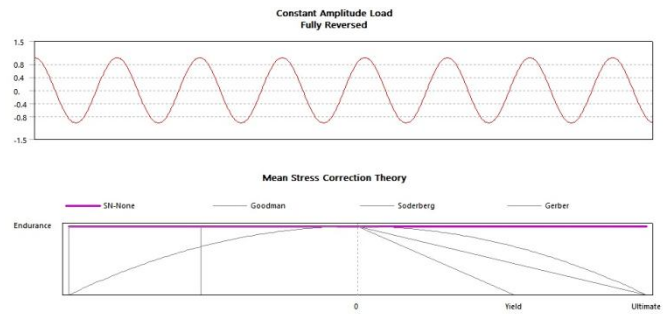

Figure 4: SN curves

III. RESULTS

A. Deformation

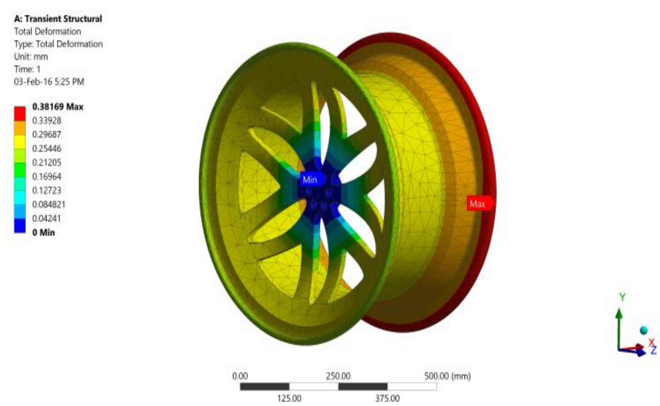

Figure 5: Total Deformation

The analysis bequeathed the total deformation about 0.38mm and the location for the same was the inner edge of the wheel. The maximum deformation in the bead seat is found to be 0.28mm. The location of bolts is free from deformation.

B. Stress study

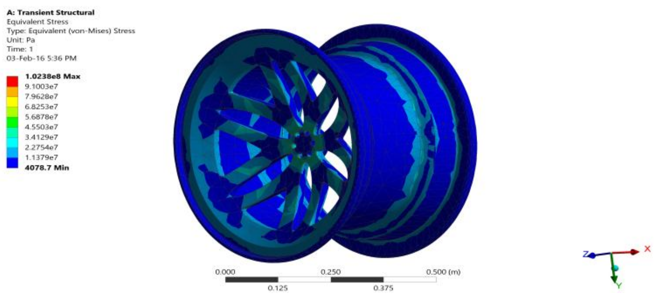

Figure 6: Equivalent Stress

The maximum equivalent stress in the rim is found to be 102.38 MPa which less than the yield strength for aluminiumalloy, the material used i.e. 280 MPa. The maximum stress concentration is on the edges of hole provided for bolts.

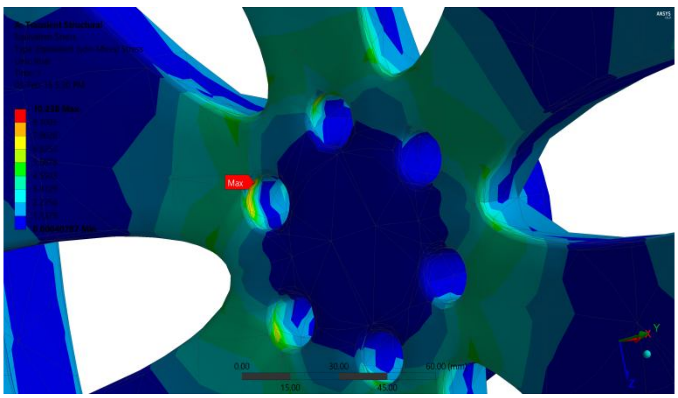

Figure 7: Maximum stress location

C. Fatigue Tool

To estimate the fatigue life of the rim, the fatigue tool is used. For the given test load conditions and material properties, the minimum life is found to be 1.29e^7 cycles. The weakest point is same where the stress concentration is maximum.

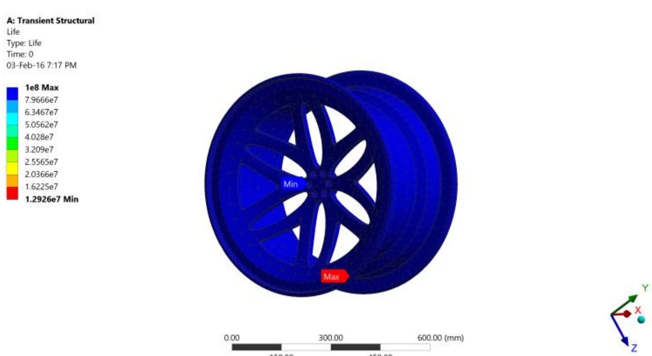

Figure 8: Fatigue Life

The safety factor was also deduced from the analysis.

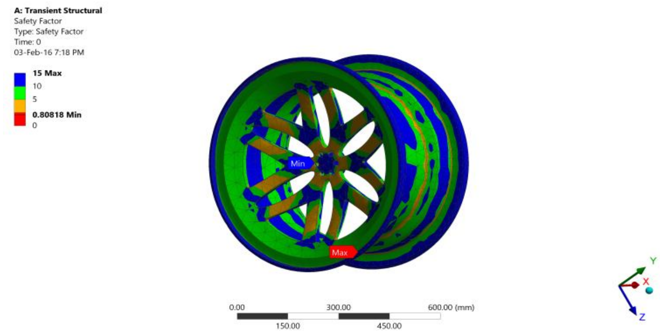

Figure 9: Safety Factor

IV. DESIGN OF EXPERIMENTS

To save the time and labour of experimenting multiple times with different variable for getting to obtain the required results, we utilised Design of experiment tool availaible in ANSYS. The tool is capable to perform the experiment by linking input and output parameters.

This tool is utilised to give us a safe range of operation for the wheel rim and the design points obtained are as follows in table 4.1.

Table 2: Design Points

	Tyre Pressure Magnitude (MPa)	Total Deformation Maximum (mm)	Equivalent Stress Maximum (MPa)	Life Minimum
1	0.248211	0.381693	102.3777	12925939.79
2	0.248211	0.369434	96.71414	24596345.3
3	0.248211	0.395554	108.6379	6347686.599
4	0.22339	0.370035	102.7929	12330479.47
5	0.273032	0.393417	101.962	13550995.21
6	0.22339	0.357643	97.12933	23463188.59
7	0.22339	0.384157	109.0531	6055299.65
8	0.273032	0.381284	96.29839	25785833.5
9	0.273032	0.407083	108.2223	6654606.954

The response chart allows you to graphically view the impact that changing each input parameter has on the displayed output parameter. Based on the above design points a 3-Dimensional response surface is acquired.

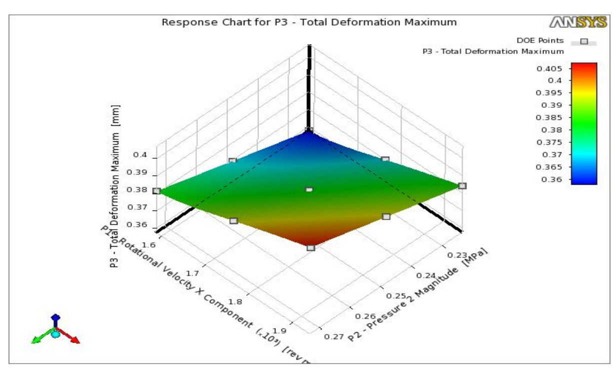

Figure 10: Response Surface

V. CONCLUSION

The finite element based approach, using ANSYS, is an effective method of predicting the failure mode of an automobile wheel rim during the design stage. Finite element-based stress analysis and fatigue life prediction obtained from the ANSYS software showed that the fatigue crack initiation regions on the wheel rim are subjected to stress concentration. Fatigue crack initiation occurs at the most stress concentrated regions of the wheel spokes and bolting holes which are the critical regions of the wheel. Considering these results minimum fatigue life is found to be 1.29e^07 cycles for current design

REFERENCES

[1] G. Arivazhagan, D.Kalaimegam, Analyze the Fatigue life of Aluminium Wheel Rim by using Rotary and Radial Fatigue Testing, International journal of engineering development and research (IJEDR) , 2015, vol-3, Issue 2, pp 89-93.

[2] E. M. Adigio, E. O. Nangi,Computer Aided Design and Simulation of Radial Fatigue Test of Automobile Rim Using ANSYS, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), 2014, e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 11, Issue 1 Ver. IV (Feb. 2014), PP 68-73.

[3] L. Wang, Y. Chen, C.Wang, Q. Wang, Fatigue Life Analysis of Aluminum Wheels by Simulation, Journal of Mechanical Engineering , 2011, 57-1, pp. 31-39.

[4] N. Furian , M. O'Sullivan , C. Walker , S. Vössner , D. Neubacher, A conceptual modeling framework for discrete event simulation using hierarchical control structures, Simulation Modelling Practice and Theory , 2012, 56, pp. 82-96.

[5] Wang, Y. Chen, C. Wang, Q., Wang, Fatigue Life Analysis of Aluminum Wheels by Simulation, Journal of Mechanical Engineering , 2011, 57-1, pp. 31-39.

[6] N. Satyanarayana&Ch.Sambaiah, Fatigue Analysis of Aluminum Alloy Wheel Under Radial Load, International Journal of Mechanical and Industrial Engineering (IJMIE), 2012,ISSN No. 2231-6477,Vol-2, Issue-1, pp 44-48.