FE Analysis of Cornering Fatigue Test for Wheel Rim of Light Commercial Vehicle

Chaudhari Dhananjay D.
ME 2nd Year
Sahyadri Valley College of Engineering and Technology
Rajuri Pune
chaudharidhananjay96@gmail.com

Bhagwat Randhavan
Head, Mechanical Engineering Department
Sahyadri Valley College of Engineering and Technology
Rajuri Pune

***Abstract*: Automobile wheels are an essential and important part of any automobile from their invention to today they have taken on various designs changes, structures, and styles. Main function of automobile wheels is to carry the overall weight of vehicle, resist the forces & stresses created during driving and transmit the driving torque to achieve the required speed & torque while driving. Wheels must have well strength stiffness and having durable design to withstand running at high speeds. Also they are capable to absorb various shocks and vibration on various road load condition like braking, bump, cornering.**

Automobile wheel assembly consist of various parts like tire, wheel rim, disc, and bearing. Wheel is mounted on axle hub to provide vehicle motion through bearing. Tires reduce shock and provide traction & cushioning. They can be considered part of the suspension system. They transmit engine power, as well as braking and cornering efforts, to the road. Tires prevent wheel rim from various force which are directly comes from roads while driving.

In this paper the FE analysis of wheel rim is done through the automotive industry standard. AIS 073. Static nonlinear analysis is perform for the designed road load condition of wheel rim. The High Stress location are observed on wheel rim assemble.

Keywords: Wheel Rim, AIS 073, Cornering Fatigue Test, Finite Element Analysis, Stress Analysis.

I. INTRODUCTION

To minimize the physical testing cycle time of wheel rim perform non liner static analysis of cornering fatigue test [CFT] using finite element method to obtain the a safe and durable design. While designing the new wheel rim the basic criteria is wheel must having good strength and durable design and also withstands with various road load condition without any failure and provide the safety to vehicle and passenger. To achieve to above parameter it is mandatory to carry out the following test Cornering fatigue test, Radial fatigue test, Durability test. The physical testing are taking too much process time and hence it impossible to develop in minimum timeline. To achieve the required design and testing timeline it is possible to carry the testing virtually by using finite element analysis. Basically there are two mandatory test for light commercial vehicle as per automotive Industry standard. First is cornering fatigue test and is second is radial fatigue test. This paper describe the virtual Stress analysis of Wheel rim for cornering fatigue test using the Altair HyperMesh software. The methodology is for above test is as per the AIS 073 standard provided by the Automotive Research Association of India (ARAI).

II. LITERATURE REVIEW:

Rakesh B. Thakare in this paper they have perform the Finite element analysis for the CFT test. They have predicted the fatigue life of wheel rim on the basis of equivalent stress approach. The comparative analysis is carried out for the various sample. They have perform the physical testing on the same sample and compare the virtual analysis results to the physical testing. Also they have minimizes the error between the both physical and virtual testing for the further improvement of finite element analysis.

Jithin Raj The paper is describe the radial static analysis of wheel rim by using ansys software they apply static pressure on rim which is equivalent to design load. They have perform the simulation for various material and choose the comparatively safe one. The comparative analysis is done on the basis of the strain energy Max stress and strain level. Following material are used in the analysis structural steel aluminum alloy magnesium alloy. Structural steel is comparatively robust in nature and hence it uses in commercial vehicles.

Rahul K. Jape In this paper they have done the weight optimization of the aluminum alloy wheel rim. By using FE analysis method. The analysis is carried out in the ansys software. They have calculate the moment as the design load and perform the analysis for different road load condition like 50% loading condition 75% loading condition 100 % loading condition. They have done 300 g weight reduction.

III. SCOPE AND METHODOLOGY:

1. To build CAD model of wheel rim using cre-o parametric.
2. To build the Finite element model in HyperMesh
3. Apply the boundary condition and the loading as per AIS standard of cornering fatigue test.
4. Perform the nonlinear Static analysis using Altair Optistruct
5. While post processing highlight the critical stress and strain location for further modification in design

IV. STRUCTURAL ANALYSIS PROCEDURE

- Build the cad model in Cre-O parametric as per desired dimension

This work is licensed under a Creative Commons Attribution-Noncommercial 4.0 International License

- **Finite Element Modelling:**

3D Hex Meshing is done as per standard quality criteria. Following quality criteria is maintain for the brick meshing

War page: 15

Aspect: 5

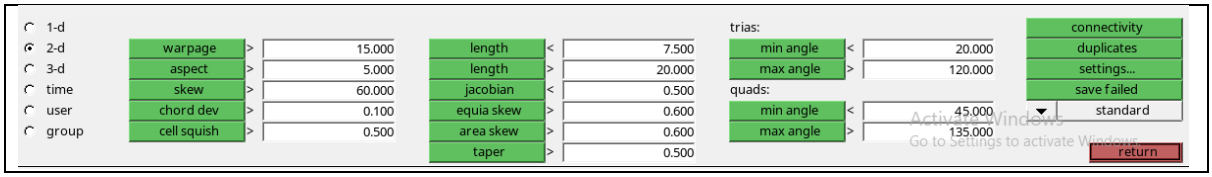
Skew: 60

Jacobean: 0.5

Min Angle for quadrilateral: 45

Mix Angle for quadrilateral: 135

TABLE I.



- Analysis Setup:

The Finite element Model is built in Altair Hypermesh as per the AIS 073 Standard.

The bottom side is constraint with the fixture and the arm is bolted to the disc

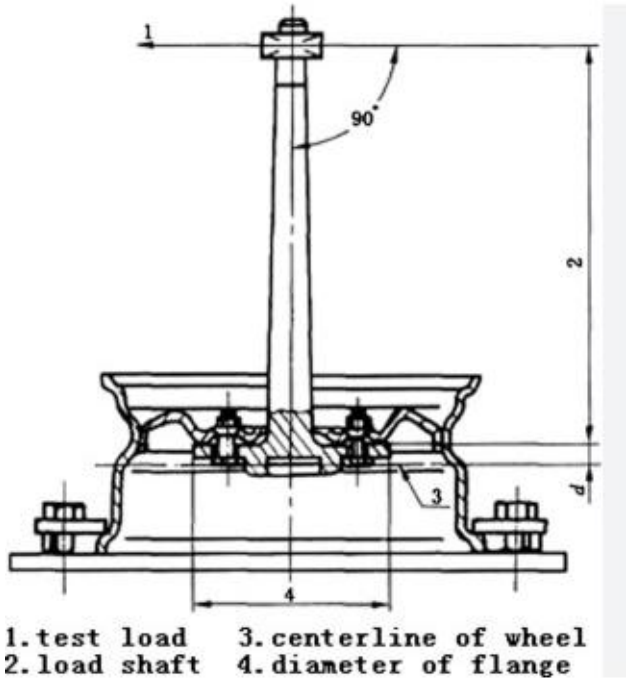


Fig. 1.

- Material Properties

TABLE II.

Material for wheel Rim Steel Properties		Unit
Density	7810	kg/m3
Poisson's ratio	0.3	
Elasticity	2.10E+05	MPA

Yield Strength of Martial Is 350MPa

- **Apply Moment As per AIS 073: Yield Strength of Martial Is 350MPa**

Bending Moment Calculation:

M= (R*U+D)F*S

Bending moment determination- The bending moment M (force x moment arm) in newton metres, is determined from the formula:

$$M = (R.\mu + d)\ F \times S$$

Where

R = Maximum static loaded radius in metres for which wheel rim is designed;

μ = Assumed coefficient of friction developed between tyre and road;

d = Inset or Outset of the wheel rim in metres;

F = Maximum design load of wheel rim in Newtons (N);

S = Accelerated test factor

Test	
Dynamic cornering fatigue (see 6.2.3.3)	μ = 0.7
Dynamic radial fatigue (see 6.2.4.3)	

For 100% Overload condition the design load is 280 kg

TABLE III.

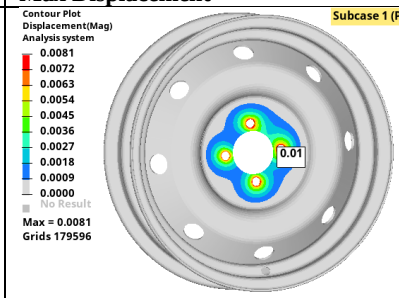
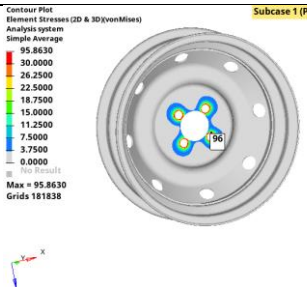
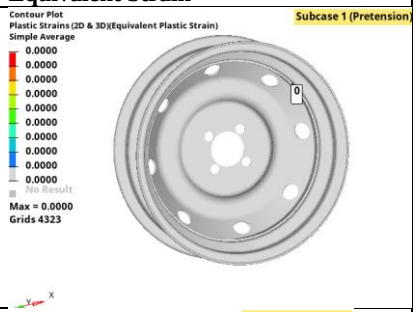
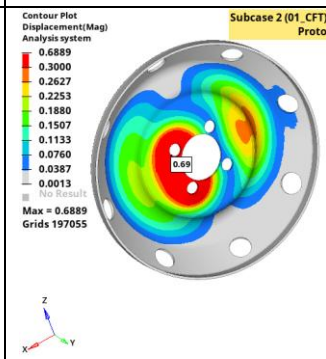
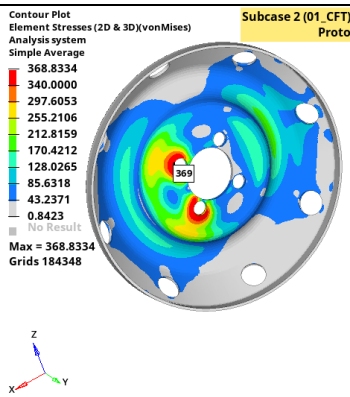
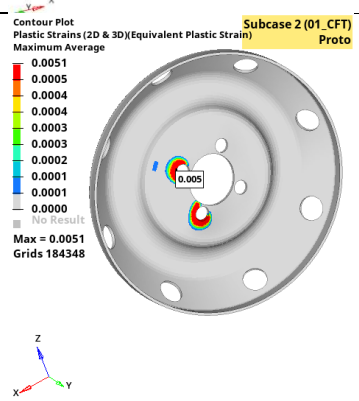
CFT Moment Calculation		R	Static Loading radius	234	mm
M= (R*u+d)F*s		u	Coeffi Of Friction	0.7	
		d	Inset Of Wheel rim	16.5	mm
80774.4	N.mm	F	Design Load	280	N
80.7744	N.m	S	CFT Test Factor	1.6	

V. RESULTS AND DISCUSSION:

Post processing is done in the Hyperview to find out the High Stress Location:

In Finite element Analysis the max stress 368 MPa is observed at the bolting location which is the above yield strength of material that is 350 MPa. And the effective plastic strain is almost 0.005. Showing the high stress concentration location. While doing physical testing the failure can be observed at the bolting location

TABLE IV.

Results	Max Displacement	Max Von Misses Stress	Equivalent Strain
Bolt Pretension			
Cornerning Fatigue test			

REFERENCES:

[1] Jithin Raj, Computer Aided Designing And Simulation Of Radial Fatigue Test Of Automobile Wheel Rim Using Ansys

[2] N. Satyanarayana Fatigue Analysis Of Aluminum Alloy Wheel Under Radial Load

[3] Tushar Y. Badgujar Badgujar, Analysis Of Stresses In Wheel Rim By Using Dynamic Cornering Fatigue Test

[4] Rahul K. Jape, Cad Modeling And Fea Analysis Of Wheel Rim For Weight Reduction

[5] Gaurav Machave, Study Of Influence Of Pressure And Load On Wheel Rim By Radial Fatigue Test

[6] AIS – 073 (Part 1) Automotive Vehicles – Wheel Rims For Two And Three Wheeled Vehicles - Light Alloy Wheel Rims –Methods Of Test And Requirements