Structural Stress Analysis for Cornering Fatigue Test of Wheel Rim as per AIS 073 using Finite Element Method

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*: All vehicle components are undergone with the physical testing to certify the safety standard and to design validation. Experimental testing is taking too much time and so the overall design process is getting slow. Now a days to minimize the design cycle time most of suppliers and OEM are perform the FE simulation. FE simulation is commonly used for the full vehicle analysis and component level. FEA simulation can be able to provide the results with minimum % error. In this project I have perform the finite element analysis of the cornering fatigue test for the E vehicle automobile wheel rim as per the automotive industry standard AIS 073. The objective of this analysis is to find out the critical and high stress concentration region for the CFT loading for further design modification and to minimize the design cycle time.**

Keywords: Automobile Wheel Rim, AIS 073, comparative Cornering Fatigue Test, Finite Element Analysis, Non-Linear Stress Analysis.

I. INTRODUCTION

High stress location and the permanent deformation location are marked by performing the non-liner static analysis of automobile wheel. Material, geometric and contact nonlinearities are used in the simulation to obtain the correct results of max stress. Newton Rap son's methods is used for the for-FE calculation. The loading and the boundary condition are applied as per the AIS Standard. The analysis is done by the Implicit solving scheme and carried out in three steps. Preprocessing in done in Hypermesh and post processing is done in HyperView. The references and for the analysis are taken from the above research paper, internet sources and Automotive standard to achieve the accuracy in results.

II. LITERATURE REVIEW

Ashwini T Vinchu: In this paper they have conducted two tests on wheel rim radial fatigue and CFT test. The regulation which is follow for the testing is as per the automotive industry standard. The simulation is done for the multiple material. They have also optimized the overall weight of the wheel. They have performed the simulation in two stages. The experimental validation is done with the physical testing of same test they have found 15 % error between the two-test method. In experimental validation they have done the strain gauging to find out the strain value at the critical location on wheel. The simulation is done in the ANSYS software.

Emmanuel M. Adigio In this Paper they have perform the computer aided simulation of automobile wheel for the radial fatigue test. First, they have performed the static analysis of automobile wheel to find out the high stress location. The analysis is carried out over the Ansys software the tetra meshing scheme is used for the wheel rim meshing. Next, they perform the physical test the automobile is rotated at the speed of 1790rpm in that they found the cracks are propagated from the location which are found in the virtual simulation. Outcomes from the paper is the critical and stress concentration region can be found from virtual testing which can be helpful to minimizes the experimental testing time.

III. OBJECTIVE AND METHODOLOGY:

1. To minimize the design cycle time, perform comparative stress analysis of baseline design & Design.
2. Prepare the FE model and apply desired Boundary Conditions.
3. Perform Non-Liner static analysis of cornering fatigue test [CFT].
4. Find out the high stress region of wheel for CFT

IV. FE ANALYSIS PROCEDURE:

- To build CAD model of new styled 10" wheel rim using Cre-o parametric.
- Study Wheel Rim Cad & Specifications
- To build the Finite element model in Hypermesh
- To perform nonlinear static simulation using Altair Opti struct solver.
- Post processing

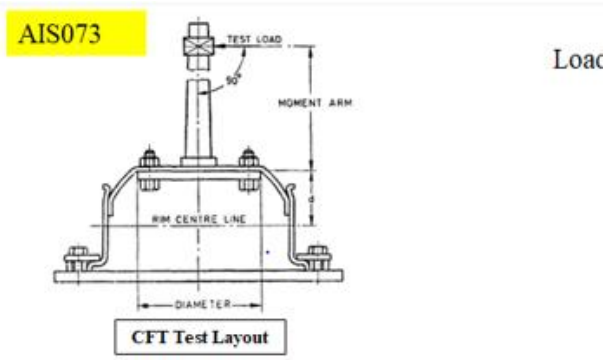
Pre Processing	
1	Import CAD & Geometry Cleanup
2	Meshing: 2D, 3D, 1D
3	Mandrill Modelling As per AIS 073
4	Connections Bolting Rb2, Rb3
5	Assign Non Linear contact
6	Assign Nonlinear Material and Properties
7	Apply Boundary Condition
Analysis	
8	Create Non Linear Static Subcases
9	Debug Error & Perform Implicit Analysis
Post Processing	
10	Post Process results Displacement, Stresses, Plastic Strain

Fig. 1.

- Analysis Setup:

FE model is built as per AIS 073 Standard.

The bottom side is fixed with the fixture and the mandrill is bolted to the wheel



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

Fig. 2.

- Material Properties

TABLE I.

Properties	**Steel**	**Steel SAFH**
Density	7.8e-9	7.8e-9
Poisson's ratio	0.3	0.3
Young's Modulus	210000	210000
Yield Strength	196	350

Load Calculation As per AIS 073:

Bending Moment Calculation: For 100% Overload condition the vehicle load

M= (R*u+d)F*s

CFT Static Force Calculation		
Bending Moment Calculation	**M=(R*u+d)F*S**	**Unit**
M (moment*force)	**M=(234*0.7+16.5)*2800*1.6** **807744**	**N.mm**

Fig. 3.

V. RESULTS AND CONCLUSION:

TABLE II.

Results	**Baseline Design**	**Design Rim A**	
	Actual Value	Actual Value	% Change wrt. Baseline
Displacement (mm)	**0.69**	**0.47**	31
Von Misses Stress (MPa)	**369**	**350**	5.15
Plastic Strain (%)	**0.5**	**0.1**	80

Baseline Design

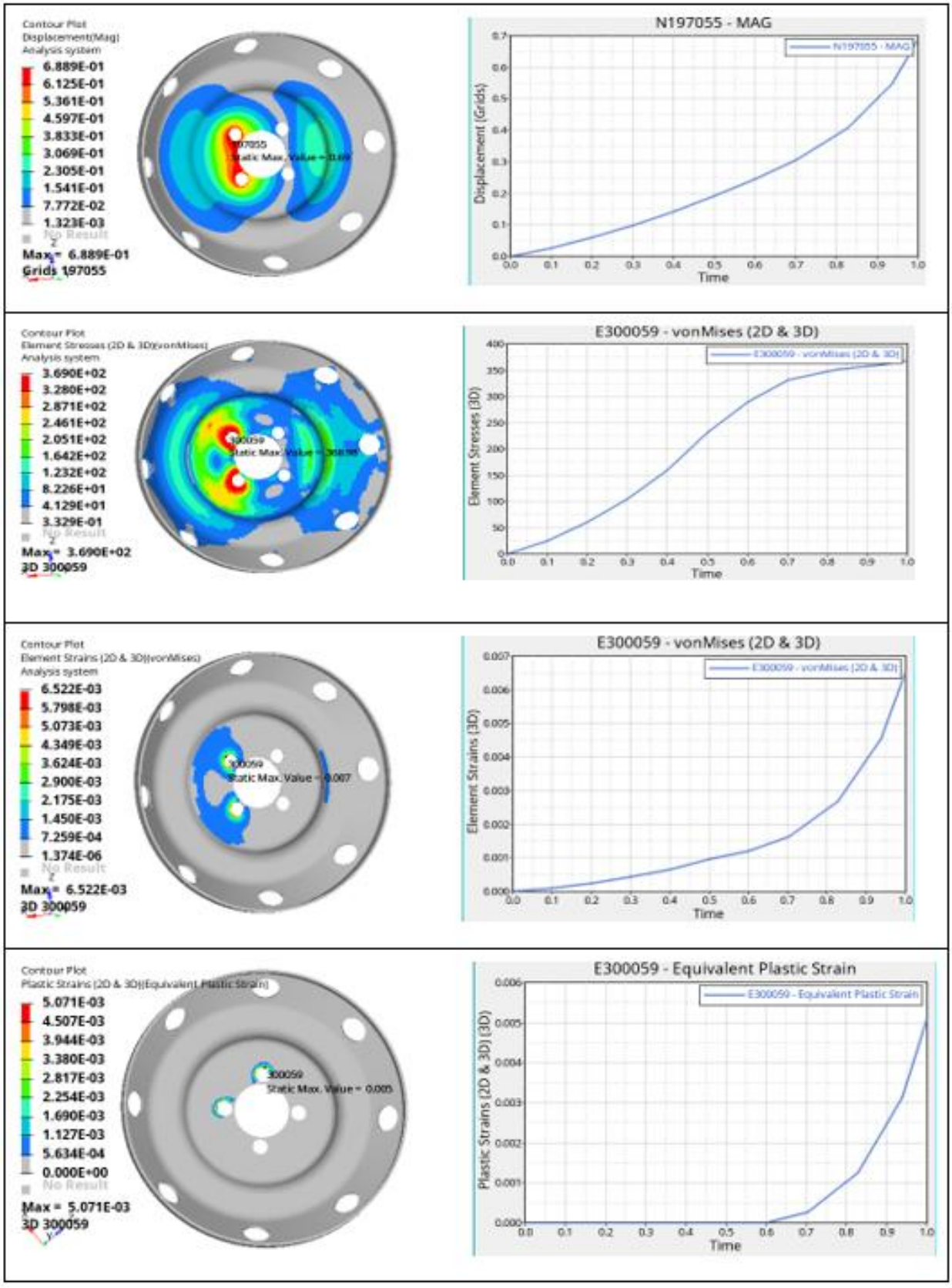


Fig. 4.

Design A

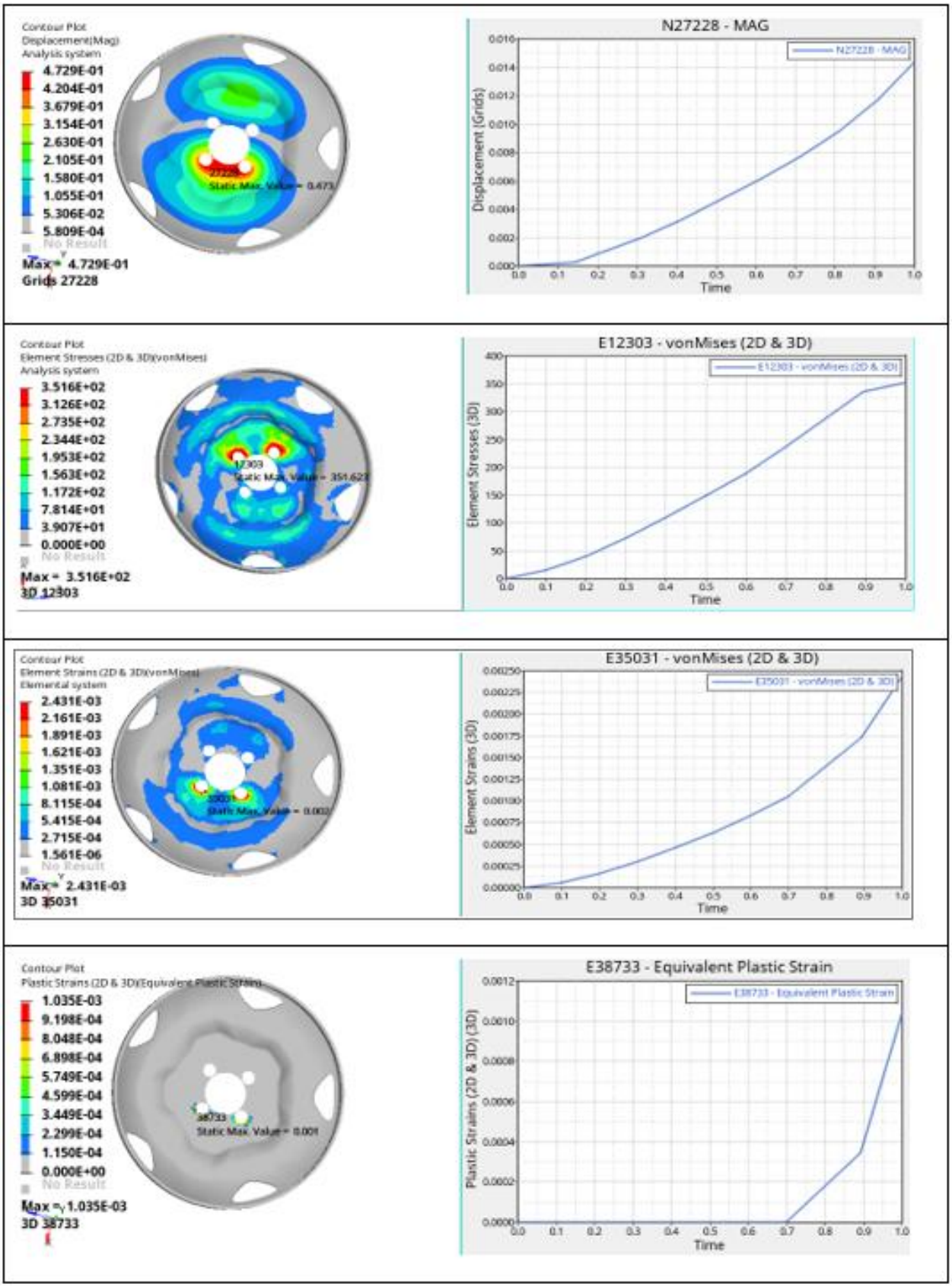


Fig. 5.

1) Post processing is done to find out the comparative High Stress Location in Finite element Analysis the max von 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 0.5%. Showing the high stress concentration location. While doing physical testing the failure can be observed at the bolting location.
2) In Design A the plastic strain is reported 0.1% the % reduction of strain then the baseline design is 80%
3) So, it is possible to find out the high stress location to minimize the design cycle time and testing time through finite element analysis.

REFERENCES

[1] Ashwini T. Vinchu, Review Study of Automotive Wheel Fatigue Tests
[2] Emmanuel M. Adigio Computer Aided Design and Simulation of Radial Fatigue
[3] Prof D. S. Chaudhari,Weight Optimization of Vehicle Rim with Fatigue TestsRahul K. Jape, Cad