

# Finite Element Analysis of Cornering Fatigue Test for Wheel Rim

Vidul Vijay Patil<sup>1</sup>, Prof. Mahesh A. Sutar<sup>2</sup>

<sup>1</sup>P. G. Scholar, Department of Mechanical Engineering, Shri Balasaheb Mane Shikshan Prasarak Mandal's Ashokrao Mane Group of Institutions, Kolhapur, 416112, Maharashtra, India.

<sup>2</sup>HOD, Department of Mechanical Engineering, Shri Balasaheb Mane Shikshan Prasarak Mandal's Ashokrao Mane Group of Institutions, Kolhapur, 416112, Maharashtra, India.

DOI: 10.5281/zenodo.20614827

## ABSTRACT

*The automobile wheel rim is a critical component responsible for transmitting torque and supporting vehicle loads under various operating conditions. Ensuring its structural integrity and durability is essential for vehicle safety and performance, especially in modern electric vehicles where weight optimization plays a vital role in improving driving range. This study focuses on the Finite Element Analysis (FEA) of the Cornering Fatigue Test (CFT) for a newly designed 10-inch wheel rim, conducted as per AIS 073 automotive standards. The wheel rim is subjected to complex loading conditions, including static, dynamic, and cornering loads, which may lead to fatigue failure. To reduce the dependency on time-consuming physical testing, nonlinear FEA is performed to predict stress distribution, deformation, and potential failure locations. The simulation incorporates material, geometric, and contact nonlinearities, and is solved using the Newton-Raphson method with an implicit solving scheme. Pre-processing and post-processing are carried out using HyperMesh and HyperView respectively. The results indicate that the baseline design exhibits maximum stress of approximately 368 MPa at the bolt-hole region, exceeding the material yield strength and leading to plastic deformation and crack initiation. An improved design (Design A) reduces plastic strain but still fails under fatigue conditions. The optimized design (Design B) demonstrates significant improvement, with reduced stress levels, elimination of plastic strain, and enhanced durability under cornering fatigue loading. The study confirms that FEA is an effective tool for identifying critical stress regions and predicting failure behavior, thereby reducing design cycle time and minimizing the need for repeated experimental testing. The optimized wheel rim design not only meets safety requirements but also contributes to improved vehicle performance and range in electric vehicles.*

**Keywords:** Finite Element Analysis (FEA), Wheel Rim, Cornering Fatigue Test (CFT), AIS 073 Standard, Stress Analysis, Fatigue Failure, Electric Vehicle (EV), Nonlinear Analysis, HyperMesh, HyperView, Design Optimization, Stress Concentration, Durability, Automotive Components.

## 1. INTRODUCTION

The automobile wheel is a critical component responsible for transmitting torque and ensuring vehicle motion, safety, and driving comfort. It is subjected to various loading conditions such as static, dynamic, radial, and cornering loads. The wheel assembly consists of the tire, wheel rim, disc, bearing, axle, and hub, and is commonly manufactured using steel or alloy materials based on application requirements.

To ensure safety and durability, the wheel must withstand all operating loads without failure. The wheel rim is required to pass standard tests such as the radial fatigue test and cornering fatigue test as per AIS standards. With the growing adoption of Electric Vehicles (EVs), weight optimization has become essential to improve vehicle range, making efficient and durable design more important.

Physical testing of components is time-consuming and increases design cycle time. Therefore, Finite Element Analysis (FEA) is widely used by OEMs and suppliers to predict performance and reduce testing efforts. In this study, FEA of the cornering fatigue test is performed on a wheel rim as per AIS 073 standards to identify high stress concentration regions. Nonlinear analysis considering material, geometric, and contact effects is carried out using an implicit solving scheme. The results help in predicting failure locations and support design optimization while reducing overall development time.

### 1.1 Problem Statement

Improving the driving range of electric vehicles (EVs) is a major challenge in the current automotive industry. One of the key approaches to achieve this is by optimizing the design of vehicle components, particularly the wheel rim. A newly designed 10-inch wheel rim has shown an improvement of 8–10% in vehicle range,

increasing it from 121 km to 137 km per charge. However, this new design must ensure adequate strength and durability under various road loading conditions without compromising safety.

The wheel rim is subjected to complex loads during operation, making it essential to validate its performance through standard physical tests such as Cornering Fatigue Test, Radial Fatigue Test, Durability Test, and Pave Test at both component and full vehicle levels. These experimental tests are time-consuming and lead to increased design cycle time. Therefore, there is a need to develop an efficient approach to predict the structural performance of the wheel rim and identify potential failure regions in the early design stage. In this context, Finite Element Analysis (FEA) of the Cornering Fatigue Test as per AIS 073 standards is carried out to evaluate stress distribution and locate high stress concentration regions. This helps in reducing dependency on extensive physical testing and accelerates the overall design and development process.

**1.2 Objectives**

1. To minimize the design cycle time.
2. Prepare the FE model and apply desired Boundary Conditions.
3. Perform Non Linear static analysis of cornering fatigue test [CFT].
4. Find out the high stress region of wheel for CFT
5. To design and develop a safe and durable design.
6. To validate the virtual analysis with the experimental test.

**1.3. Methodology**

1. To build CAD model of new styled 10” wheel rim using Cre-o parametric.
2. Study Wheel Rim Cad & Specifications
3. To build the Finite element model in HyperMesh 21
4. To perform nonlinear static simulation using Altair Optistruct solver.
5. Post processing
6. Following steps are following to perform FE Analysis

Table 1: Methodology to perform FE Analysis

<b>Pre Processing</b>	
1	Import CAD & Geometry Cleanup
2	Meshing: 2D, 3D, 1D
3	Mandril 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
<b>Analysis</b>	
8	Create Non Linear Static Subcases
9	Debug Error & Perform Implicit Analysis
<b>Post Processing</b>	
10	Post Process results Displacement, Stresses, Plastic Strain

Table 2: Wheel Rim Description

<b>Wheel Rim Description</b>	
Size	3.5*10 Inch
Max. Design Load 100OL	280 kg
Inset of wheel rim	16.5 mm
Max static loading radius	234 mm
Wheel Rim Weight	3.5 kg
Rim Thickness	2.4 mm
Disc Thickness	3.3 mm
Rim Material	SAPH550
Disc Material	SAPH550
Bolt	M10*1.5

**2.DETAIL STRUCTURAL ANALYSIS OF WHEEL RIM:**

**2.1. Pre Processing- Import CAD data & Geometry Cleanup: Unit System mm, tone, sec.**

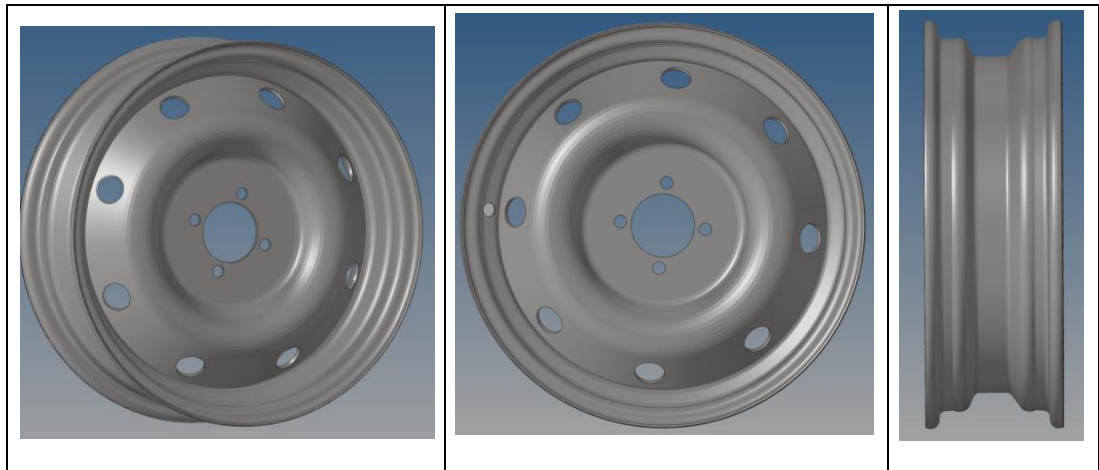


Fig 1- Wheel Rim CAD geometry

**Meshing: 2D, 3D, 1D**

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

**Mesh Quality Checks**

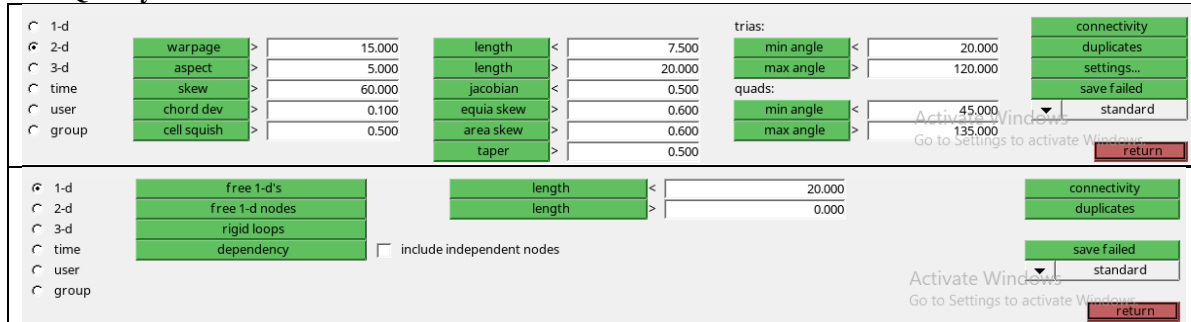


Fig 2-Mesh Quality Criteria

Table 3: Mesh Count

Component	Mesh type	count	Total
Rim	3D Solid	107960	<b>233051</b>
Disc	3D Solid	50748	
Mandrill	3D Solid	74330	
Bolt	1D Beam	4	
Rigid	Rb2	9	

**Mandrill Modeling as per AIS 073**

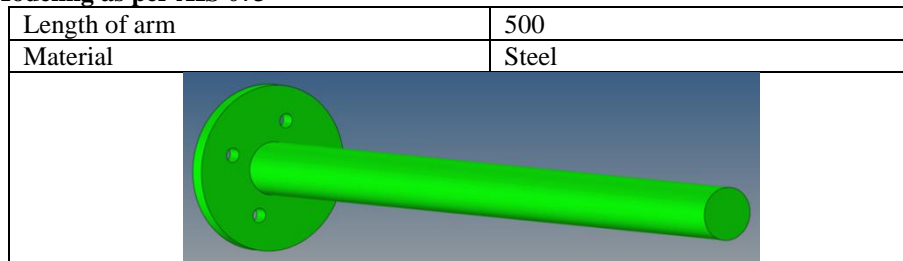


Fig 3 - Mandrill

**Connection: Rb2, Rb3, Bolting**

<b>M10 Bolt 1D</b>	<b>Rb2</b>
--------------------	------------

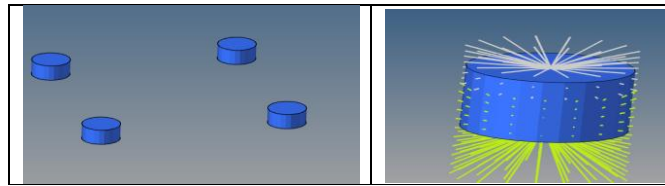


Fig 4- Bolt Connection

Assign Non-Linear Contact between Mandrill and Disc

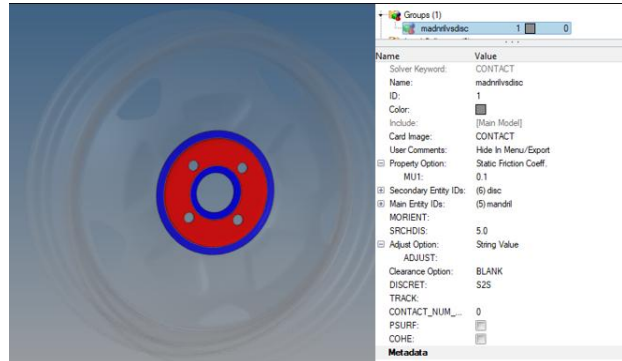


Fig 5- Contact Modelling

Assign Non-Linear Material and Properties

Properties	Steel	SAPH 540	Unit
Density	7.8e-9	7.8e-9	Ton/mm3
Poisson's ratio	0.3	0.3	-
Young's Modulus	210000	210000	N/mm2
Yield Strength		350	Mpa
Ultimate Tensile Strength		540	Mpa
% of elongation		33	%

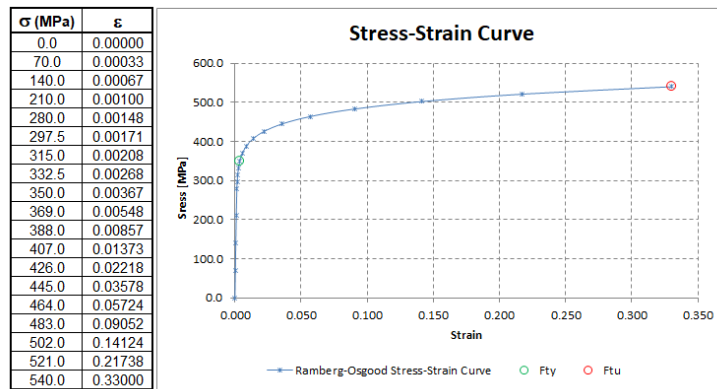


Fig 6- Material Modelling

Table 4: Material & Properties

Component	Material	Properties
Rim	SAPH 540	Sec Solid
Disc	SAPH 540	Sec Solid
Bolt M10	Steel	Sec Beam
Mandrill	Steel	Sec Solid

Boundary Condition: SPC Constraint 6 DOF

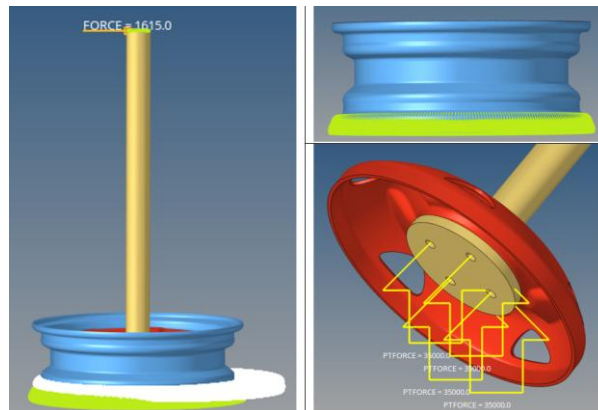


Fig 7- Boundary Conditions

Bolt Pretension: Input Torque 70 N.m

Pretension Manager

View: By Bolt

	Bolt Type	Bolt Id	EID/SURFID	Load Type	Loadcol	Load Id	Load Magnitude
<input type="checkbox"/>	1D	1	2552314	Force	PRETENS_1(1)	4505	35000.0
<input type="checkbox"/>	1D	2	2552313	Force	PRETENS_1(1)	4504	35000.0
<input type="checkbox"/>	1D	3	2552312	Force	PRETENS_1(1)	4503	35000.0
<input type="checkbox"/>	1D	4	2552311	Force	PRETENS_1(1)	4502	35000.0

Fig 8- Bolt Preload

Bolt Pretension Calculation:

Bolt Design		
Bolt Size	M10	
Nominal Dia	Pitch	Yield Stress
10	1.2	900
Area (mm <sup>2</sup> )	61.81931183	
Proof Strength:	85-90% Yield Strength	
Proof Stress (N/mm <sup>2</sup> )	765	
Preload (N)	Permanent Joint	42562.59619
	Reuse Bolt	35468.83016
K: Nut Factor	0.2	
Torque (N.m)	KDF/1000	
Permanent Joint	85.12519239	
Reuse Bolt	70.93766032	

Fig 9- Bolt Preload Calculation

Force Calculation: As Per AIS 073

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

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

Where:

- R = Maximum static loaded radius in metres for which wheel rim is designed;
- $\mu$  = 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	Accelerated Test Factors
Dynamic cornering fatigue $\mu = 0.7$ (see 6.2.3.3)	S = 1.60
Dynamic radial fatigue (see 6.2.4.3)	K = 2.25

Fig 10- AIS 073 Standard

CFT Static Force Calculation		
Bending Moment Calculation	$M = (R \cdot \mu + d) \cdot F \cdot S$	Unit
M ( moment*force)	$M = (234 \cdot 0.7 + 16.5) \cdot 2800 \cdot 1.6$ <b>807744</b>	N.mm
Arm Length	500	mm
Static Force	Bending moment/Length of arm	N
	$807744/500$ <b>1615.488</b>	

Fig 11- Force Calculation & Load Increment

2.2. Analysis Set Up:

Implicit Scheme and Newton Rap son Method is used for solving the nonlinear Simulation.

Subcase 01 Solve Bolt pretension nonlinear load case		Subcase 02 Solve Main CFT Force nonlinear load case along with Subcase 01	
Name	Value	Name	Value
ID:	1	Name:	01_CFT_X
Include:	[Main Model]	ID:	2
User Comments:	Do Not Export	Include:	[Main Model]
<b>Subcase Definition</b>		<b>Subcase Definition</b>	
Analysis type:	Non-linear static	Analysis type:	Non-linear static
SPC:	(2) SPC	SPC:	(2) SPC
LOAD:	<Unspecified>	LOAD:	(3) CFT_x
NLPARAM:	(4) NLPARAM1	NLPARAM:	(4) NLPARAM1
NLPARAM(L...):	<Unspecified>	NLPARAM(L...):	<Unspecified>
SUPPORT1:	<Unspecified>	SUPPORT1:	<Unspecified>
DEFORM:	<Unspecified>	DEFORM:	<Unspecified>
PRETENS...	(1) PRETENS_1	PRETENS...:	<Unspecified>
Name:	PRETENS_1	MPC:	<Unspecified>
ID:	1	STATSUB (...)	(1) Pretension
Color:	[Yellow]	Solver ...	SUBCASE
Include:	[Main Model]	Name:	Pretension
Card Im...:	<None>	ID:	1
Load T...:	PTFORCE(4)	Include:	[Main Model]
		User C...:	Do Not Export

Fig 12- Analysis Setup

2.3. Post Processing

Table 5 Baseline Design Rim Detail FE Results

Load Cases	Sub Case 01 (Pretension)	Sub case 02 (Pretension + CFT Force)
<b>Displacement</b>		
<b>Max Von Mises Stress</b>		
<b>Plastic Strain</b>		

Table 6: Baseline Design Pretension Load Case Plots

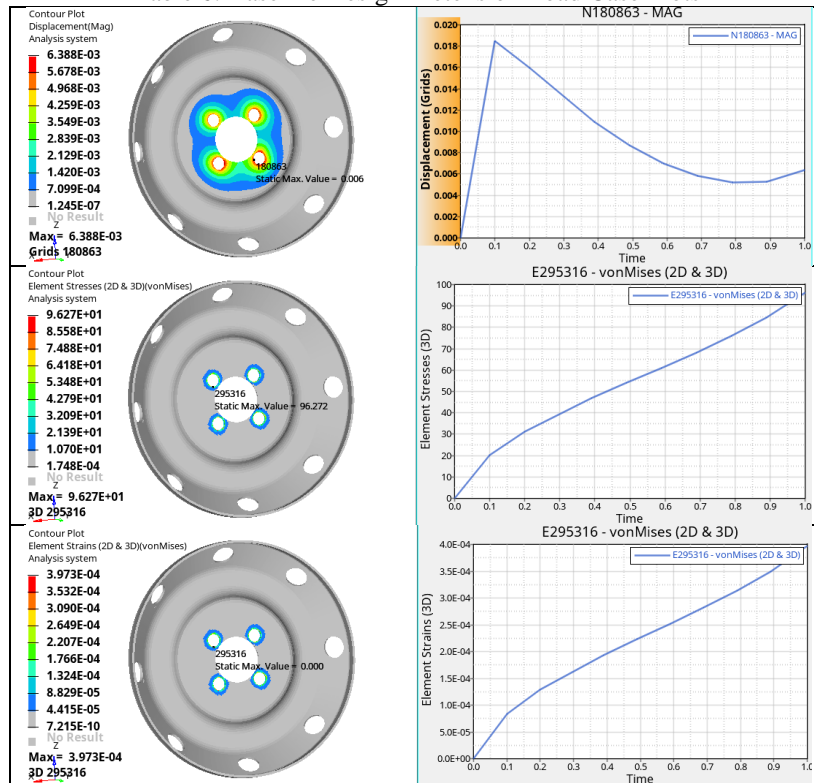
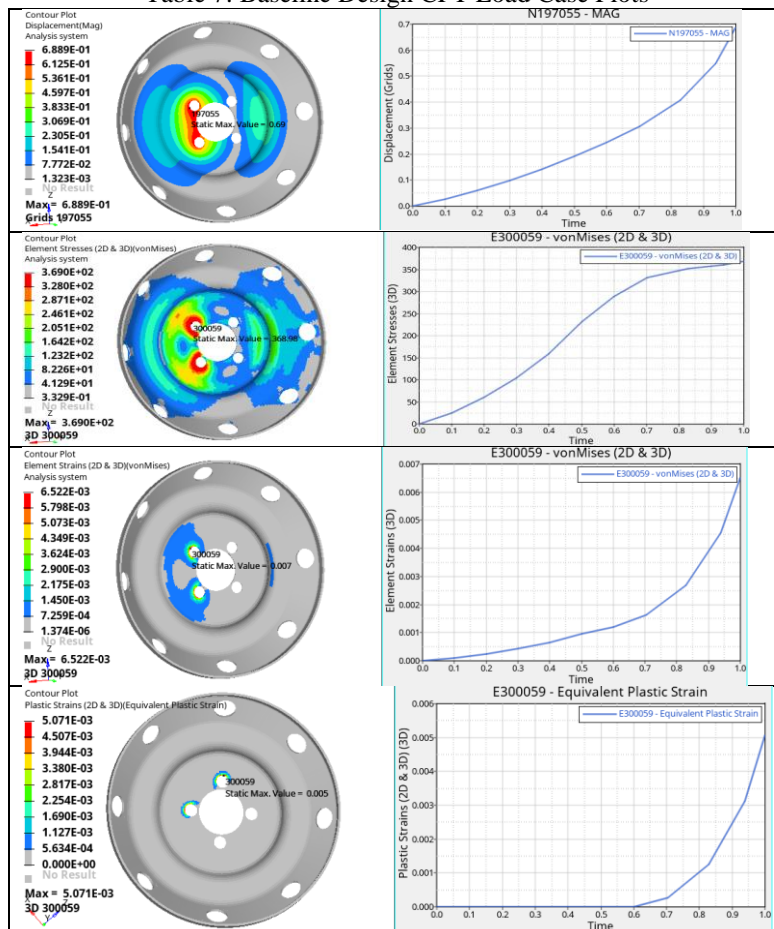
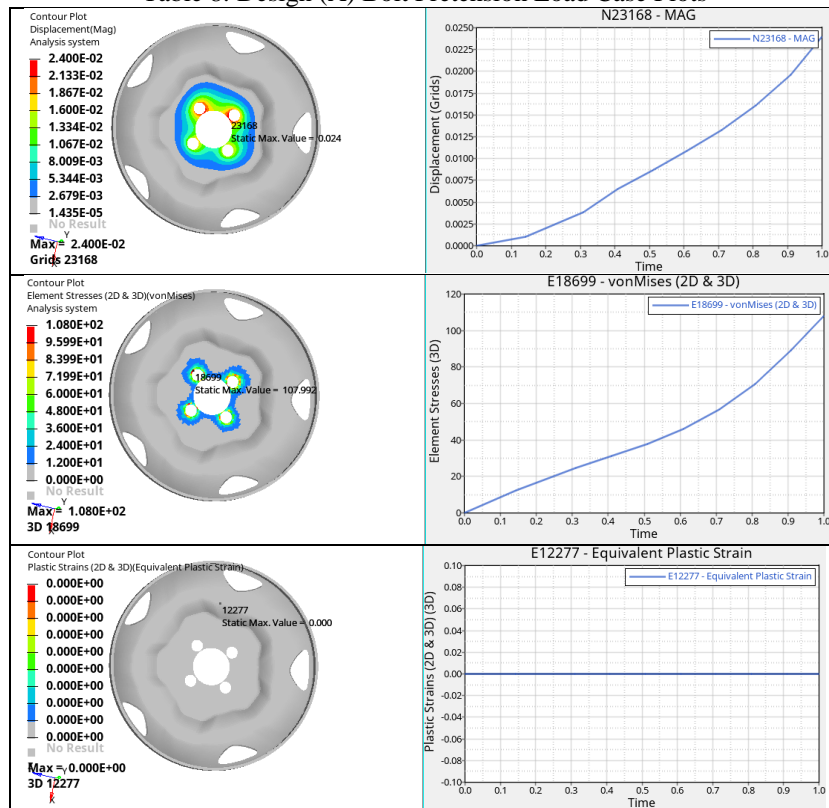


Table 7: Baseline Design CFT Load Case Plots

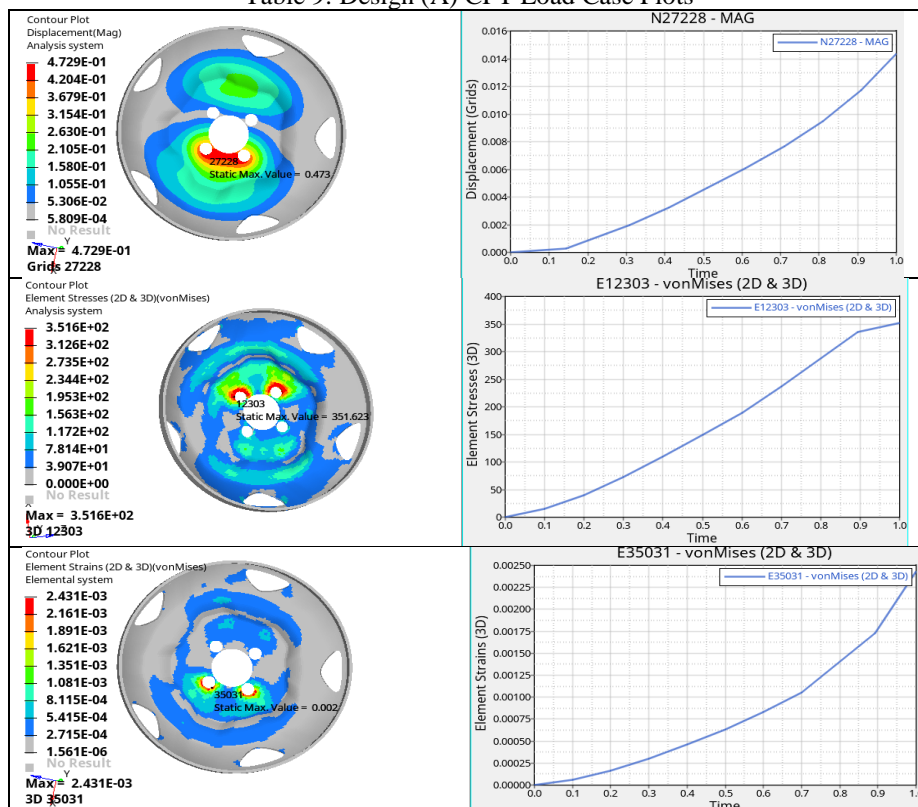


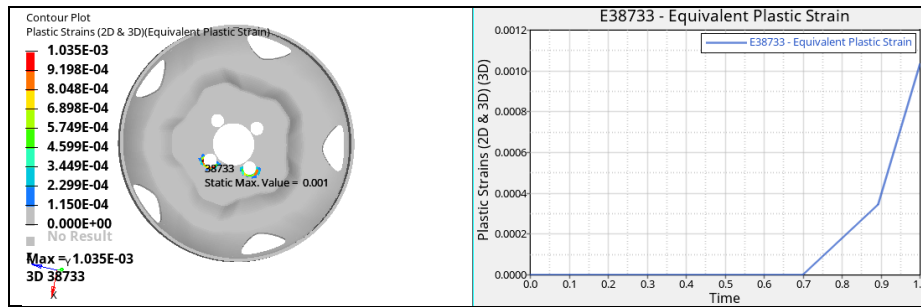
**Perform Same Analysis For Design (A) & (B) Wheel Rim:  
Design A Rim Detail FE Results:**

**Table 8: Design (A) Bolt Pretension Load Case Plots**



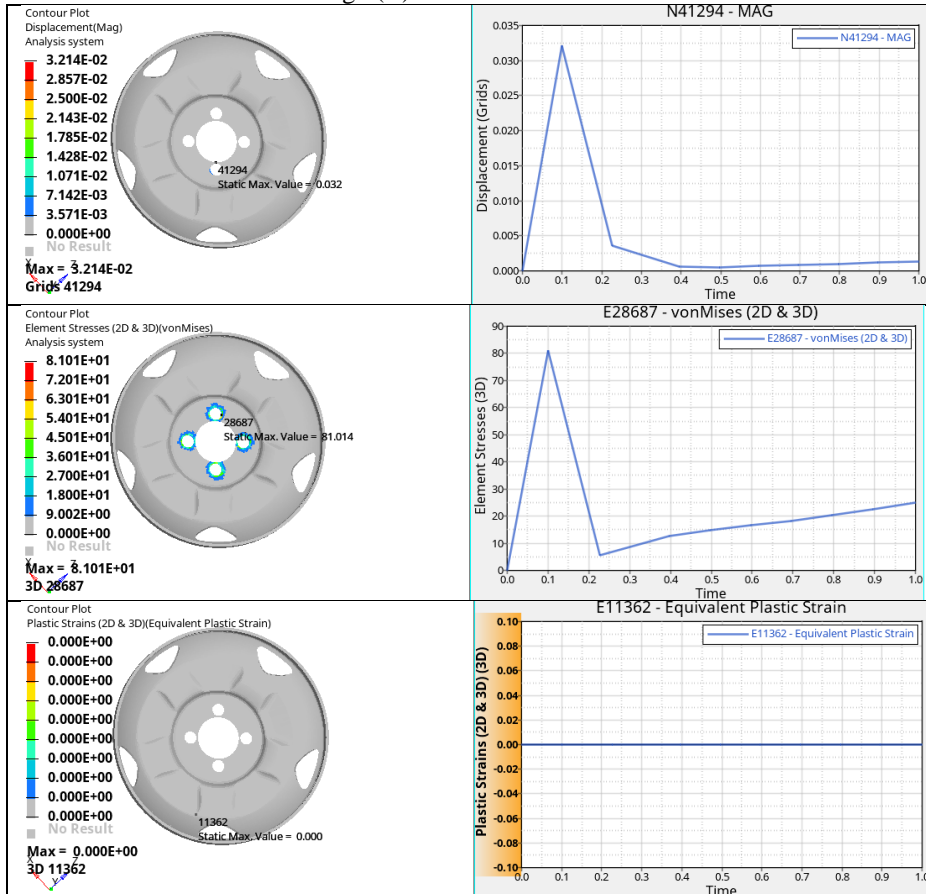
**Table 9: Design (A) CFT Load Case Plots**



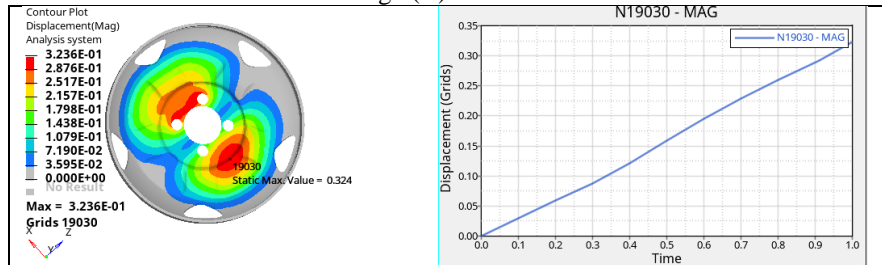


**Design B Rim Detail FE Results:**

**Table 10: Design (B) Bolt Pretension Load Case Plots**



**Table 11: Design (B) CFT Load Case Plots**



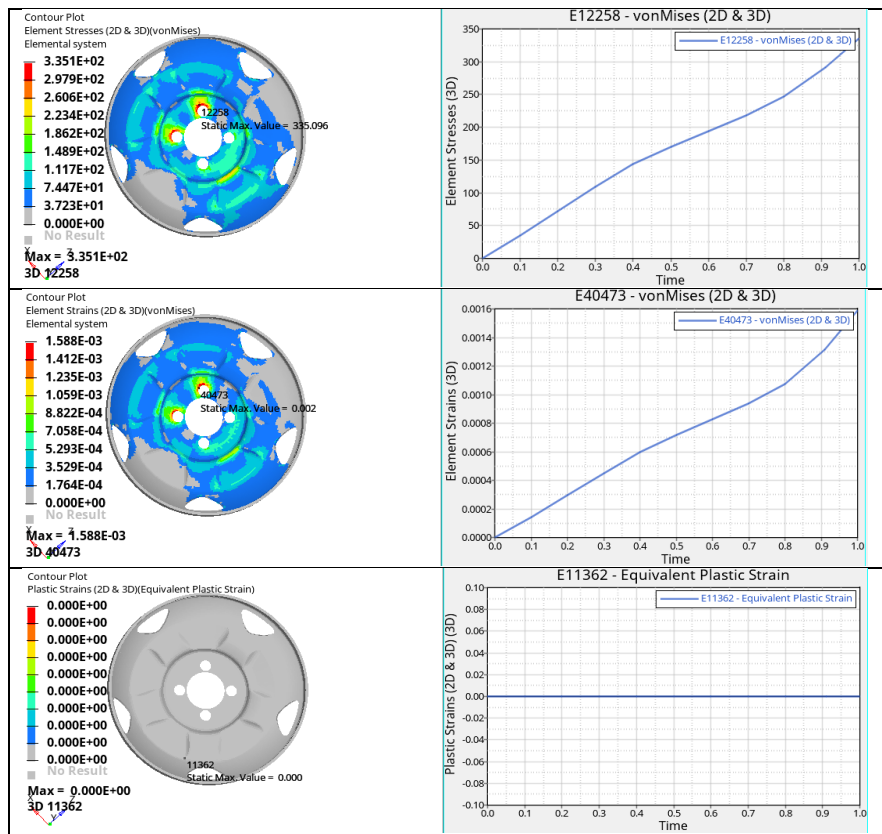
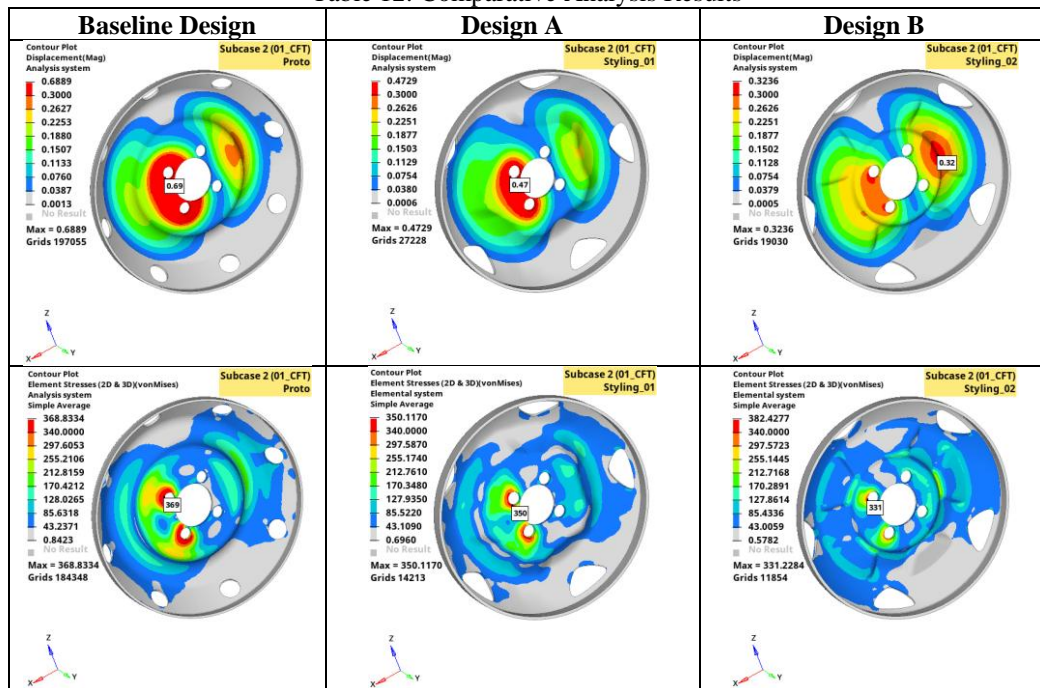


Table 12: Comparative Analysis Results



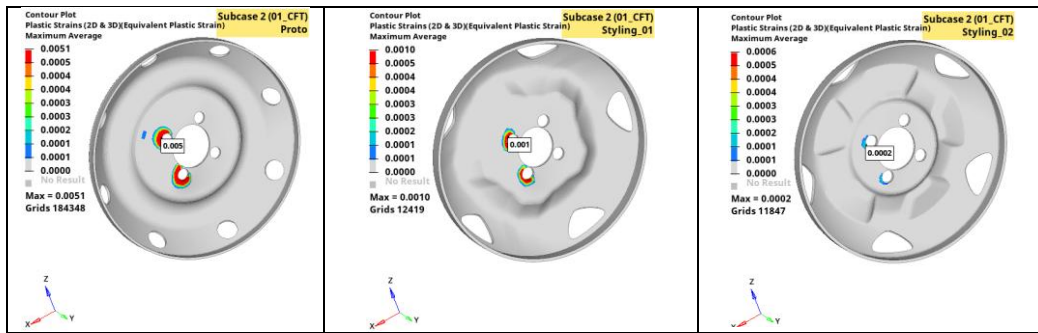


Table 13: Results Table

		Baseline	Design A	Design B
Displacement mm	Pretension	0.006	0.024	0.032
	CFT	0.69	0.47	0.32
Max Stress MPa	Pretension	96	108	81
	CFT	368	352	335
Element Strain %	Pretension	0.0	0.0	0.0
	CFT	0.7	0.2	0.16
Plastic Strain %	Pretension	0.0	0.0	0.0
	CFT	0.5	0.1	0.0

Table 14: % Reduction With respect to baseline Design

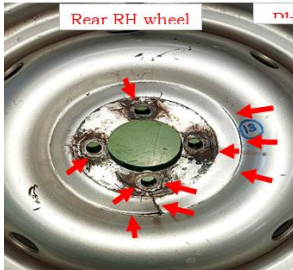
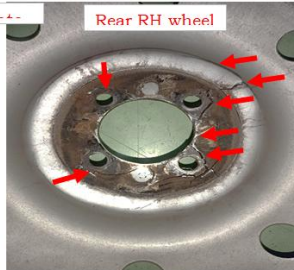


Results	Baseline Design	Design Rim A		Design Rim B	
	Actual Value	Actual Value	% Change wrt. Baseline	Actual Value	% Change wrt. Baseline
Displacement (mm)	0.69	0.47	31	0.32	53.63
Von Misses Stress (MPa)	369	350	5.15	335	10.3
Plastic Strain (%)	0.5	0.1	80	0.02	96

### 3. EXPERIMENTAL TEST RESULTS:

- The experimental results compare the performance of three wheel rim designs: Baseline Design, Design A, and Design B, under different testing conditions.
- In the Durability Test (30000 km), all three designs performed satisfactorily and were found to be safe, indicating that each design can withstand long-term usage under normal operating conditions.
- However, differences were observed in more critical tests. In the Pave Pot Hole Test, the Baseline Design showed a crack at the disc region, indicating failure under severe road conditions. In contrast, both Design A and Design B successfully passed this test without any failure, showing improved strength and resistance to impact loading.
- For the Cornering Fatigue Test (CFT) at 100% overload condition, the Baseline Design was not tested or data was not available. Design A exhibited a crack at around 3980 cycles, indicating that it could not withstand repeated cornering loads for a longer duration. On the other hand, Design B successfully passed the test without any crack, demonstrating superior fatigue strength and durability.
- The failure images further support these observations. The Baseline Design showed visible cracks during the pothole test, while Design A failed during the CFT test due to crack formation. Design B did not show any failure in the tested conditions.
- Overall, the results indicate that Design B is the most reliable and optimized design, as it meets all safety and performance requirements, while the Baseline Design and Design A show failure under critical loading conditions.

Table 15: Experimental test results

Test Method	Baseline Design	Design A	Design B
Durability (30000 Km)	Safe	Safe	Safe
Pave Pot Whole	Crack Observed at Disc	Safe	Safe

CFT (For 100 % OL Loading)	-	Crack Observed @ 3980	Safe
Baseline Design Rim Failure Images @ pot whole test			
			
Design A Failure @ CFT Test Cracked observed			
			

#### 4.RESULTS DISCUSSION AND CONCLUSION:

- 1) 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 0.5%. 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% also the crack is observed at wheel at fatigue testing.
- 3) In design no plastic strain is observed and stress are within the limiting value also is safe in all physical testing and meet the safety criteria.
- 4) So it is possible to find out the high stress location to minimize the design cycle time and testing time through finite element analysis.
- 5) The baseline wheel rim exhibited stress levels exceeding material yield strength (~368 MPa) with plastic strain ~0.5%, leading to failure at bolt-hole regions, which closely matched experimental crack locations.
- 6) Design A showed reduced plastic strain (~0.1%) but still failed during cornering fatigue testing.
- 7) Design B demonstrated stress reduction (~10%), plastic strain elimination, and significant displacement reduction (~54%), remaining safe under all physical tests.
- 8) The study confirms that nonlinear FEA can accurately predict critical stress zones and failure behavior, effectively reducing dependence on repeated physical testing and shortening the design cycle.

#### 5.REFERENCE

- [1]. Jithin Raj, Sachin Sunil, Renith P R ,Amal Krishna, “Computer Aided Designing and Simulation of Radial Fatigue Test of Automobile Wheel Rim Using Ansys”, *International Research Journal of Engineering and Technology (IRJET)* eISSN: 2395-0056. Volume: 09 Issue: 06 | June 2022
- [2]. N. Satyanarayana & Ch.Sambaiah “Fatigue Analysis of Aluminum Alloy Wheel Under Radial Load”, *International Journal of Mechanical and Industrial Engineering (IJMIE)*, ISSN No. 2231 –6477, Vol-2, Issue-1, 2012
- [3]. Tushar Y. Badgujar Badgujar, Analysis of Stresses in Wheel Rim by Using Dynamic Cornering Fatigue Test
- [4]. Jape Rahul K, Jadhav S. G, “CAD Modeling and FEA Analysis of Wheel Rim for Weight Reduction”, *International Journal of Engineering Science and Computing (IJESC)*, Volume 6 Issue 6 pp 7404 - 7411.Gaurav Machave, Study of Influence of Pressure and Load On Wheel Rim By Radial Fatigue Test
- [5]. AIS – 073 (Part 1) Automotive Vehicles – Wheel Rims for Two And Three Wheeled Vehicles - Light Alloy Wheel Rims –Methods Of Test And Requirements