Optimization of a Motorcycle Wheel Rim Using CAD and CAE Software's

Deshabhakt Gavali[#1], Harshwardhan Dhumal[#2], Niranjan Dindore[#3],Parag Betgeri[#4], P.H. Lokhande[#5]

[#]*Department of Mechanical Engineering, Sinhgad College of Engineering, SPPU, Pune-41.*

Abstract **- Wheel Rims form a vital part of two wheeler vehicles. There are two main types of motorcycle rims, namely, Solid Wheels and Spoked Wheels. Solid wheels are the ones in which both the rim and the spokes are manufactured from the same material. While Spoked wheels are the ones in which the motorcycle rim is laced with a number of high tension spokes. This project mainly deals with the optimization of the spoked wheel rim with respect to its weight. The design of the rim is created in a suitable CAD software while, a CAE software is used for analysing the rims for critical conditions. Loading conditions like tyre pressure, radial load and bending load are simulated during the finite element analysis of the component. The geometry of the wheel is optimized until satisfactory results of stress and displacements are obtained. This optimization includes varying the number of spokes and removal of material from the spokes and hub area. ETRTO manual and AIS 073 (Part 2), these standards are referred during the design and analysis stage. Four different models are analysed and compared with each other. The fully optimized model is found to be '23.587 %' lighter than the basic design. While it is also found to be safe in all the testing conditions.**

Keywords **– Alloy Wheel, Optimization, Radial Fatigue Test, Cornering Fatigue Test, Motorcycle Wheel Rim, Weight Reduction, Hyperworks.**

I. INTRODUCTION

The rim is the "outer edge of a wheel, holding the tyre". Its main functions are to rotate over the axle of an automobiles, to use power from engine to take automobile in motion, provide support for braking system over its body, dissipate heat generated in the body of wheel rim to surrounding environment, support whole body weight as well as withstand against impact load due to pot holes and road irregularities.

A wheel rim is the most highly stressed component in an automobile that is subjected to various types of loads. There is an increasing industrial demand for components that are lighterand cheaper to produce, while at the same time maintaining fatigue strength and other functional requirements. The testing of any component for failures during the design stage has become possible with the use of various FEA software packages. The presented work involves design and analysis of a motorcycle wheel rim using suitable softwares. The CAD Model of the motorcycle wheel rim is designed in CATIA. This model is used for CAE . Cornering Fatigue Test (CFT) and Radial Fatigue Test (RFT) are the two testing conditions simulated on the wheel rim model. FEA is done using HYPERMESH – OPTISTRUCT. Eventually the main aim of this study is to optimize the design of the rim in order to reduce the rim weight along with maintaining the strength of the rim. For this purpose various iterations of analysis are carried out by varying the number of spokes of the rim. Also the spokes and hub region was optimized and a final optimized rim with reduced weight is obtained.

II. LITERATURE REVIEW

For the design purpose the ETRTO[1] standard manual is referred while, for the analysis purposes i.e. in order to determine the loading values for RFT and CFT the AIS 073 (Part 2)[2] standard is referred. In 2005, Kinstler [3] studied all the SAE standards and explained evolution of rim testing year by year. This study gives information about the Radial Fatigue Test and Cornering Fatigue Test and evolution of their setup. In RFT, straight ahead driving condition with accelerated load is simulated and the life of tyre and wheel rim is determined. To evaluate cornering, initially a fixture was developed. By using this fixture, straight ahead testing of wheel rim, with tyre on it, at a camber angle was done.J. Stearns et al [4] in 2005 studied the distribution of tyre pressure and radial loads exerted on the wheel rim due to the weight of the vehicle. As the radial load is exerted on the bead seat of the rim similar loading was applied on the model. The maximum von mises stress equal to 15.8 MPa and maximum displacement of 0.223 mm was developed. It was observed that the tyre pressure does not have direct influence the state of stress. The maximum stress was developed at the edge of disk attachment and the maximum displacement was observed at the bead seat. The stresses were higher in the rim than the disk and the critical points of design are bead seat and well.Paropate et al[5] performed analysis on a bajaj pulsar 150cc wheel rim. They considered various materials for the analysis like aluminium, magnesium, carbon fibre and thermoplastic resin, and they performed static, fatigue and modal analysis on the wheel rim model. They concluded that thermoplastic were was the best material for the wheel rim. Rajarethinam. P et al[6] studied the effect of number of spokes on the strength of a motorcycle wheel rim. They also optimized the wheel rim and concluded that there was a weight reduction of 7.3567% as compared to the original design. They used Solidworks and Ansyssoftwares for the same. Similarly Theja et al[7] also performed a comparative study of different models of rim by varying the number of spokes and thee rim material. With the use of magnesium as material and a four spoke wheel model they claimed a considerable reduction in the rim weight as compared the the initial design. Joshi et al[8] in 2016 presented a study on modification of front wheel of motorcycle. The modification process was based on material and topology modification and validation using finite element analysis. The results obtained from modified analysis are compared with the original one. They considered the effect of speed, pressure, gyroscopic effect, braking on the wheel during the analysis procedure while, they also performed modal analysis, impact analysis and RFT on the wheel rim. They achieved a overall weight reduction of 24.45%.

III. METHODOLOGY

This chapter explains the flow of the entire process and how various steps are taken with the help of CAD and CAE softwares.

3.1. Design and Testing Standards

Design standards are a set of norms which are accepted all over the world while designing any component which specify various design parameters for designing. The design standards that have been studied are ETRTO, IS 10694, JATMA and ITTAC. As per our requirement, the contour MT 3.5 M/C has been selected from ETRTO Manual. Fig 3.1 represents the basic contour of the rim.

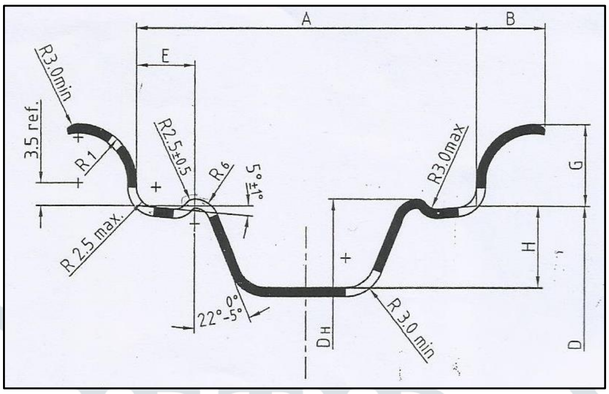


Fig.1 Rim contour from ETRTO manual[1]

Similarly, there exist a number of standards for testing of a component. These testing standards specify various testing parameters like applied load, number of cycles for test, total run time of test, etc. Some of the standards used for testing of wheel rims are AIS-073 (Part 1 and 2), IS 9436, SAE J328, ABNT NBR6750, SNI 1896. The comparative study of these standards is stated in Table 2.1. According to the requirement AIS-073 (Part 2) has been selected for the testing purpose.

AIS-073 (Part 2) specifies two types testing methods namely Cornering Fatigue Test (CFT) and Radial Fatigue Test (RFT).

For CFT, the bending moment M (force x moment arm) in N.m is determined from the formula;

$$M = (R \times \mu + d) F \times S \quad \ldots\ldots(1)$$

Where
R = Maximum static loaded radiusin meters for which wheel rim is designed;
μ = Assumed coefficient of friction developed between a tyre and road;
d = Inset or outset of the wheel rim in meters;
F = Maximum design load of wheel rim in Newtons (N);
S = Accelerated test factor
For values of μ and S, Table 1 has been referred.

Table 1 Test Factors [2]

Test	Accelerated Test Factor
Dynamic Cornering Fatigue Test (μ = 0.7)	S = 1.6
Dynamic Radial Fatigue Test	K = 2.25

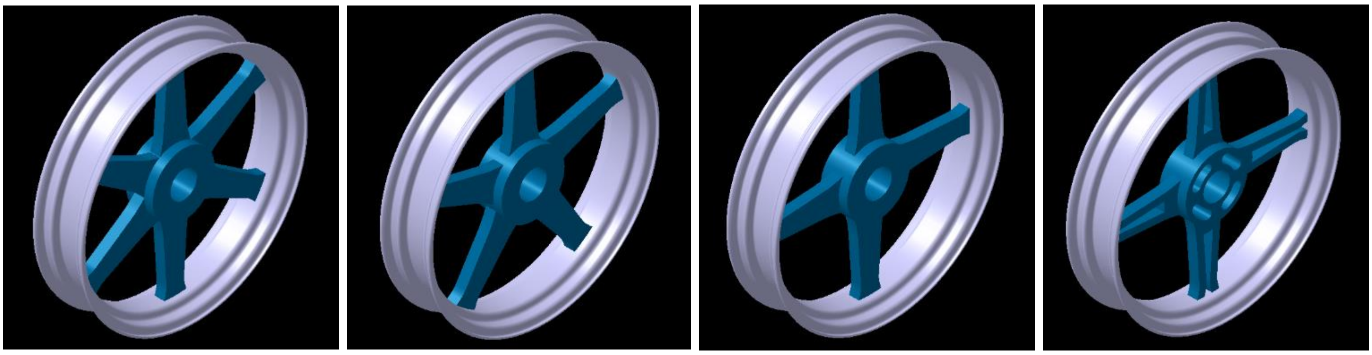
For RFT, the radial load F_r in Newtons is determined as follows;

$$F_r = F_v \times K \quad \ldots\ldots(2)$$

Where,
F_v = Maximum design load of wheel rim in Newtons (N);
K = Accelerated test factor
For values of K, Table 1 has been referred.
During analysis procedure, loading calculations are done using Equation 1 and 2.

3.2 Design

Now, as per the contour selected from the ETRTO Manual[1], the CAD model has been prepared. The CAD Software used for the designing is CATIA. Hence CAD models of 4 spokes, 5 spokes and 6 spokes wheel rim has been prepared as show



in Fig. 2.

Fig. 2 CAD Models (From left to right 6 spokes, 5 spokes, 4 spokes and 4 spokes optimized wheel rim)

3.3 Analysis

The next aim is to simulate the tests which will be performed on the rim after the manufacturing for validation purposes. Also, it would provide a platform for comparing various models of the rim and optimizing them. These simulations will be run in CAE software. The preferred CAE Software is HYPERWORKS.

HYPERMESH Software is used for the meshing purpose.This software is selected because it allows the user to manually mesh any geometry easily as it provides the entire control of the mesh to the user and one could achieve the required quality of the mesh. In our study we have given utmost priority to the meshing procedure because the accuracy of the results depend on the quality of mesh. Hence by keeping the mesh as fine as possible and giving proper connections between different models we would be able to obtain more accurate results.The mesh of the design consists of 2D elements quad and tria (mixed) extruded over the geometry forming 3D elements. The average element size is 4 mm. The FREEZE contact is specified between the rim and the disc and also between the spokes and the hub.The property considered for the elements is 3D PSOLID. The materials considered for this study is an Aluminium alloy (201.0-T43 Insulated mould casting) [7].The mechanical properties of this material are given in the Table 2.

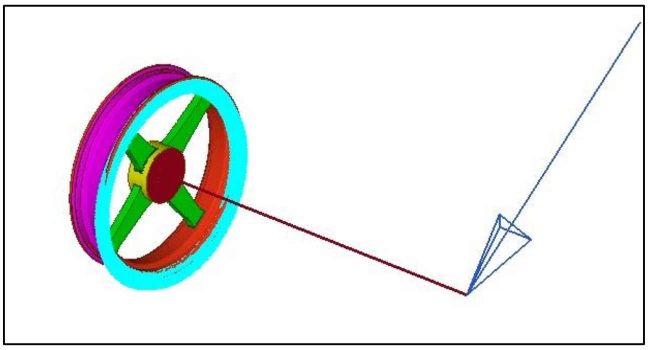
Table 2 Material Properties- Al Alloy (201.0-T43 Insulated Mold Casting (SS))

Elastic Modulus (GPa)	Poisson's Ratio	Mass Density (Kg/m3)	Tensile Strength (MPa)	Yield Strength (MPa)	Thermal Expansion Coefficient (/K)	Thermal Conductivity (W/mK)	Specific Heat (J/kgK)
71 GPa	0.33	2800	273	232	1.9e-005	121	963

The tests which are simulated using the software are:-

1) Cornering Fatigue Test (CFT):-

Using the equation (1) and using the values from table (1) the value of bending moment to be applied is calculated to be *M = 1507 N-m.* The test simulation is done as per the description given in AIS-073 (Part 2). To simulate the exact test conditions, along with the wheel rim moment arm is also modelled. The moment arm is modelled using the rigid (RBE2) elements. The length of the moment arm is taken as 1000mm while a force of 1507 N is applied at the end of the moment arm. This would simulate the required loading condition and would create a combined effect of a moment of 1507N-m on the rim. Once the Force is applied the next step is to apply constraints. As per the observations made from the actual setup of the test the rim was constrained circumferentially on one side using clamps. Similar condition is simulated in the software by restricting all the degrees of freedom of the circumferential points. The loading conditions i.e. the load collectors are being shown in the Fig 3. After defining the load collectors the load step is created and the model is solved using OPTISTRUCT solver. The results obtained for different models are given in the Fig.5 and Fig.6.



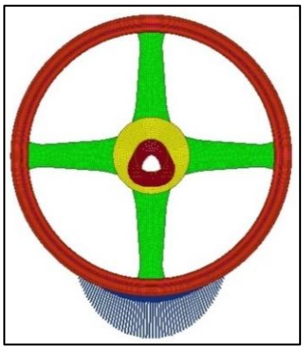


Fig.3 CFT CAE ModelFig.4 RFT CAE Model

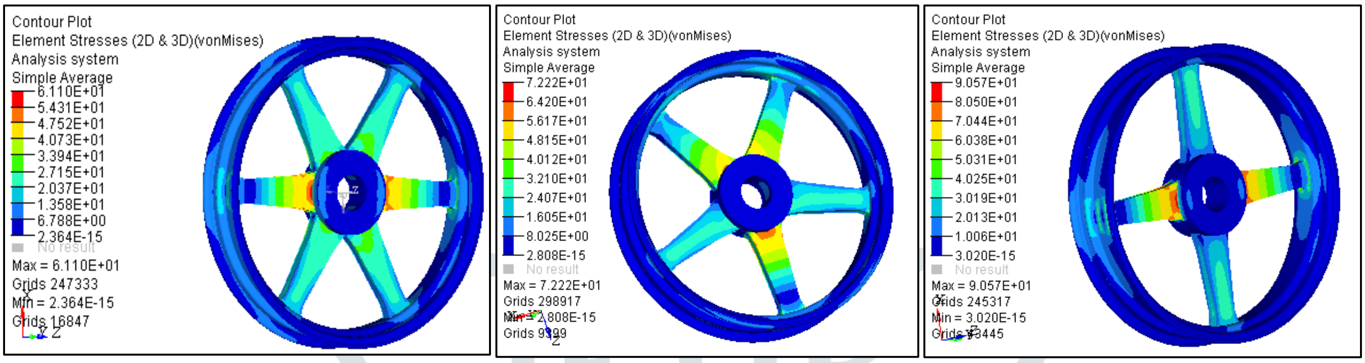


Fig.5 Stress results for CFT loading condition

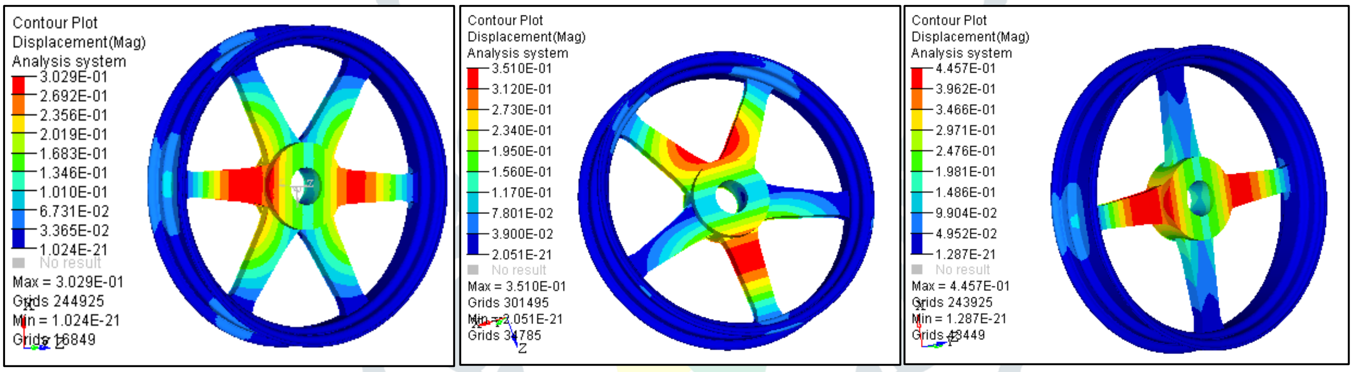


Fig.6 Displacement results for CFT loading condition

2) Radial Fatigue Test (RFT):-

This test is mainly used to determine the conjoint influence of tyre inflation pressure and radial load on stress and concomitant displacement distribution of the wheel rim. The radial load mentioned above is the vertical reaction force exerted on the two tyres in order to balance the entire weight of the vehicle. Each of these forces act normally on the tyre and tends to compress the wheel rim in the vertical direction. As the vehicle is motion this radial force becomes cyclic in nature. Due to this reason the wheel fatigue strength should be evaluated very carefully in order to maintain the structural integrity of the wheel. As prescribed by AIS-073 (Part 2), the wheel rim should maintain its structural integrity, without developing any cracks or experiencing excessive plastic deformation, for more than 4×10^{6} cycles, under the radial load (F_r), which is given by equation (2). In this equation the value of accelerated test factor (K) is taken from Table 1 as, K=2.25 , while the value of Maximum design load of wheel rim (F_v) is taken as, F_v=3924N(400kg). Now substituting the above values in equation we get the value of radial load as, F_r= 2.25 × 3924 = 8829N. In this study the loading condition is applied by referring the research presented by J. Stearns .et al. [8]. It is observed that in an actual wheel, the radial load is applied to the wheel at the bead seat where the tyre actually rests on the rim. For this analysis the radial load is applied as variable pressure distribution. This pressure distribution, along the circumferential direction, is assumed to follow a cosine function distribution. This pressure distribution is applied in the software using the equation option and a user defined cylindrical co-ordinate system. This equation is given by $W_r = 1.5766\ cos(2.5717)$.Along with the Radial Load, uniform pressure is also applied on the model in order to simulate the tyre pressure exerted on the rim. The tyre pressure is taken as 206.843KPa (30psi). After applying both these loads, the constraints are applied on the model. For this purpose, all the nodes of the hub hole are constrained i.e. all their degrees of freedom are set to zero. Fig.4 shows the model which has been created for the simulation of Radial Fatigue Test. Once the model is ready it is solved using OPTISTRUCT solver to obtain the required stress and displacement values. The obtained results for the 4, 5 and 6 spokes wheels are given in Fig.7 and Fig.8.

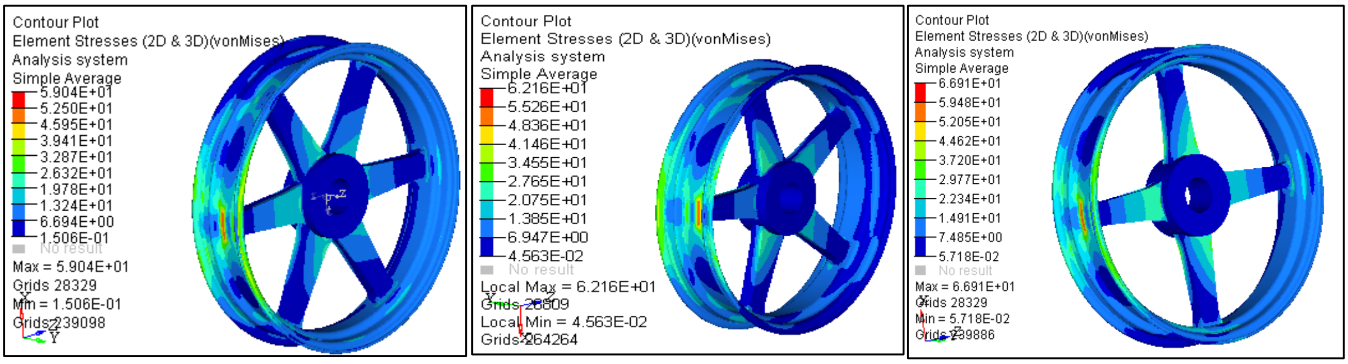


Fig.7 Stress results for RFT loading condition

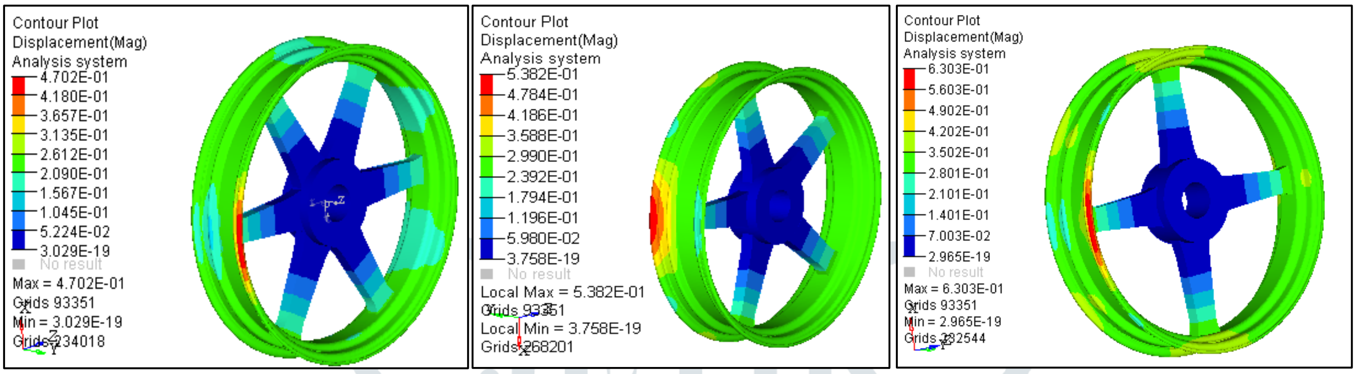


Fig.8 Displacement results for RFT loading condition

3.4 Optimization

After studying the results for the above two simulated tests, it was observed that there was reduction in weight as we go from 6 spokes wheel to the 4 spokes wheel. Also it was noted that strength and displacement values for the 4 spoke wheel rim were well under the permissible limit of the given material. Due to thus there is further scope to reduce the material and optimize the rim. Hence a new model of optimized 4 spokes wheel rim was designed and analysis procedure is carried out on it as it was done in the previous steps. The results obtained for CFT test simulations are given in Fig.9 while those for RFT test simulations are given in Fig.10.

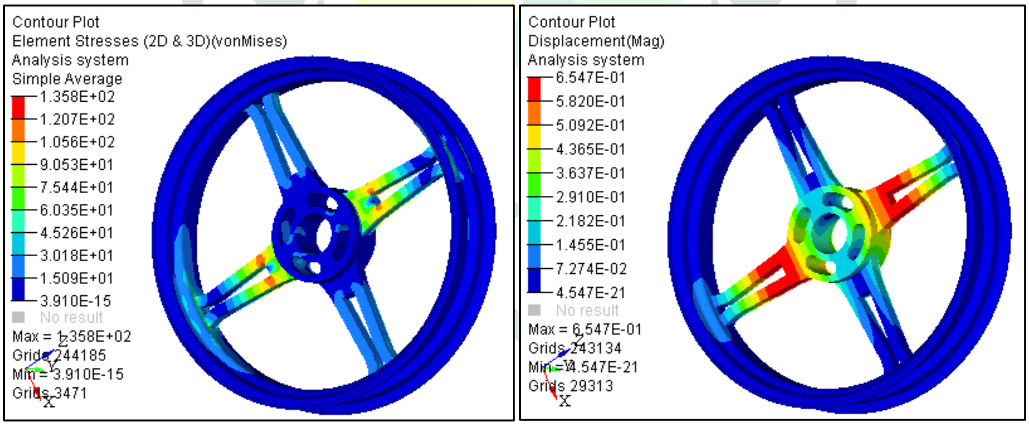


Fig.9 Stress and Displacement results for CFT

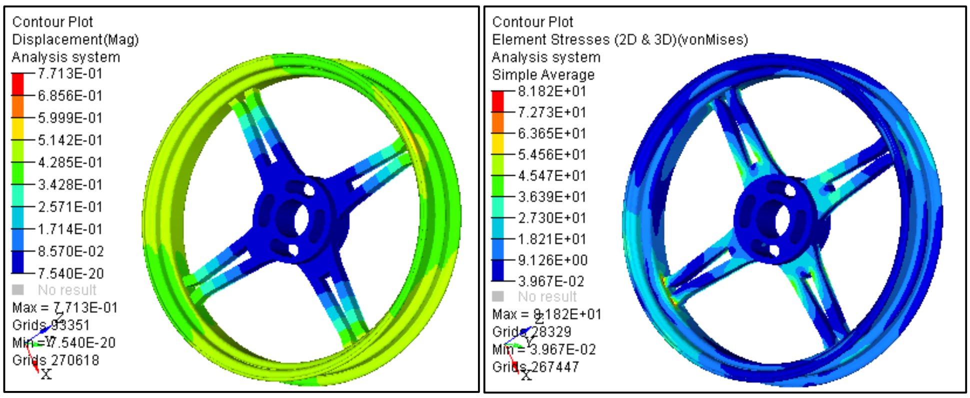


Fig.10 Stress and Displacement results for RFT

IV. RESULTS AND DISCUSSION

All the different models are been thoroughly analysed and comparative study is represented using graphs in the Fig.11 . The graphs give a comparative representation of the stress and displacement values for RFT and CFT. Also the Table 3gives the comparison of weight between the different types of models. It is observed that the weight of the initial 6 spokes wheel design i.e. 7.88 kg reduces to 5.88kg for the optimized 4 spokes wheel rim. Hence an overall weight reduction of 23.587% is achieved. Also the stress and displacement result for this wheel rim are within the permissible range of the material with a minimum safety factor of 1.707. So it can be concluded that this design of the wheel rim is safe.

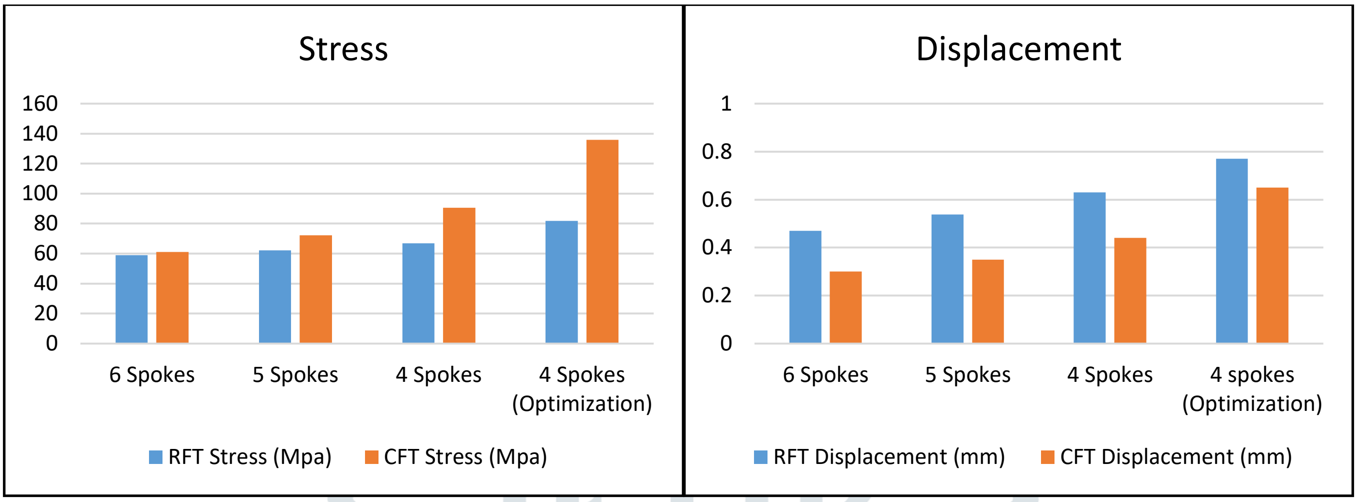


Fig.11 Comparative analysis of Stress and Displacement Results

Table 3 Comparative study of the Weights of different CAD Models

Model	6 Spokes	5 Spokes	4 Spokes	4 Spokes (Optimization)
Weight (kg)	7.88	7.43	6.96	5.88

REFERENCES

[1] ETRTO Manual.

[2] AIS – 073 (Part 2).

[3] John Kinstler, 'The Science and Methodology of SAE Wheel Fatigue Test Specifications', SAE World Congress, 2005.

[4] J. Stearns, T. S. Srivatsan, X. Gao, and P. C. Lam,' Understanding the Influence of Pressure and Radial Loads on Stress and Displacement Response of a Rotating Body:The Automobile Wheel', International Journal of Rotating Machinery, 2005.

[5]SaurabhM Paropate Sameer J Deshmukh,'Modelling And Analysis Of A Motorcycle Wheel Rim ', 2013

[6]Rajarethinam. P, Periasamy. K,'Modification Of Design And Analysis Of Motor Cycle Wheel Spokes', 2014.

[7] M. Saran Theja, M. Vamsi Krishna,'Structural and Fatigue Analysis of Two Wheeler Lighter Weight Alloy Wheel', 2013.

[8] Hrishikesh Joshi, Ashish Powar, SanketKhuley and D.P.Yesane,'Analysis and Topological Optimization of Motorcycle Front Wheel',2016.