International Conference on Latest Research in Engineering Science
12 & 13 August 2017
www.ijrugc.org/Conference

DESIGN AND TOPOLOGY OPTIMIZATION OF CAR WHEEL RIM USING FEA

[1]G.KISHAN, [2]N.SADANANDAM, [3]B.V.S. RAO, [4]Dr. P. RAVINDER REDDY
[1]Academic consultant, University College of Engineering & Technology for Women, K.U campus, Warangal, Telangana.
[2]Asst.professor, Kamala Institute of Technology and Science (Singapur), Huzurabad, Telangana
[3] Asst.professor, CBIT Gandipet, Hyderabad, Telangana
[4]Professor &Head, CBIT Gandipet, Hyderabad, Telangana
1,2,3 and 4: Department of mechanical Engineering

ABSTRACT: In the field of automobile industries the part of the vehicle manufacturer from cast alloys they have been utilizing widely due to high strength to weight ration, better fuel consumption and low cost and weight reduction of vehicle utilized to reduce the fuel consumption. So this paper deals with the design and topological optimization of aluminum alloy wheel. In this paper optimized results are compared with the existing reference model from piaggio manufacturer wheel. Topology optimization concentrates on the distribution of the material and structural connectivity in the design domain, Which helps to reduce the weight of the structural component in less time. In this work wheel rim in the modeled by using SOLID WORKS V.10 software and analyzed to see the variation in the different analysis i.e., torsion and rotating bearing test by using altair optistruct.

Keywords: Aluminum alloy, wheel rim, topology optimization, stress analysis, transient dynamic analysis.

1. INTRODUCTION: 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 [1]. Wheel is one of the most important components of an automobile [2] Wheel is generally composed of rim and disc. [3] The essential of car wheel rim is to provide a firm base on which to fit the tire. Its dimensions, shape should be suitable to adequately accommodate the particular tire required for the vehicle [4]. Wheels have a vital importance for the safety of the vehicle and special care is needed in order to ensure their durability. The development of the vehicle industry has strongly influenced the design, the material selection and the manufacturing processes of the wheels. They are loaded in a complex manner and further improvement and efficient wheel design will be possible only if their loading will be better understood. In order to achieve an optimum design of the wheel, two requirements are needed: the precise knowledge of the loading and the mechanical properties and allowable stresses of the material, which depend on the vehicle characteristics, service conditions and manufacturing processes. Today, most manufacturers develop the wheel design based on results of the traditional dynamic radial fatigue test, also called the rim roll test, and on the dynamic cornering fatigue test, also called the rotating bending test. Another possibility is to use the finite element method in order to establish the stresses in the car rim and to compare the different design solutions. However, the modeling of the real loading of the rim cannot be accurate enough and that is why the results should be use with precaution. Anyhow, this method is very useful for comparing different design solutions and, therefore, selecting the car rim that should be tested further.In this paper, the car rim is analyzed with the finite element method. The static stresses are studied in order to find the zones with higher stress concentration and to suggest the better design solution. The results have been compared to those obtained using an experimental stand. The wheel is a device that enables efficient movement of an object across a surface [5] where there is a force pressing the object to the surface. Common examples are a cart pulled by a horse, and the rollers on an aircraft flap mechanism. Wheels are used in conjunction with axles; either the wheel turns on the axle, or the axle turns in the object body.

The low resistance to motion (compared to dragging) is explained as follows (refer to friction):

1. The normal force at the sliding interface is the same.
2. The sliding distance is reduced for a given distance of travel.
3. The coefficient of friction at the interface is usually lower.

Bearings are used to help reduce friction at the interface. In the simplest and oldest case the bearing is just a round hole through which the axle passes (a "plain bearing").The rim is the "outer edge of a wheel, holding the tire. It makes up the outer circular design of the wheel on which the inside edge of the tire is mounted on vehicles such as automobiles. For example, on a bicycle wheel the rim is a large hoop attached to the outer ends of the spokes of the wheel that holds the tire and tube. In the 1st millennium BC an iron rim was introduced around the wooden wheels of chariots.

2. Design Process of Wheel Rim

Aluminum Alloys in Wheel Manufacturing: Characteristics of Aluminum

Face-Centered Cubic unit cell promotes ductility & machinability. Ductility promotes bending failure rather than fractures. Chemical properties promote corrosion resistance. Machinability promotes excellent finishes. Chemical characteristics allow anodizing. This metal main advantage is decreased weight, high precision and design choices of the wheel [6] .

Wheel Examples and Piaggio Specifications: In addition to many examples shown you, the following are offered. These are typically forged or pressure cast, then finish-machined as needed. The increased pressure during casting minimizes the loss of ductility that is common to castings.

Fig 1: Different types of wheels

Piaggio Beverly 300 ie - Technical specifications

Engine	Piaggio single-cylinder 4-stroke
Engine capacity	278 cc
Bore x stroke	75 mm / 63 mm
Power	22.2 hp at 7,250 rpm
Torque	23 Nm at 5,750 rpm
Timing system	Single overhead camshaft (SOHC) - 4 valves
Fuel system	Electronic injection
Cooling	Liquid
Lubrication	Wet sump
Starter	Electric with oil-bath free wheel
Transmission	Twist-and-go CVT with torque server
Clutch	Automatic, centrifugal dry clutch
Frame	Double cradle in high strength tubular steel
Front Suspension	Telescopic hydraulic fork with 35-mm stanchions – wheel travel 90 mm
Rear suspension	Double hydraulic shock absorbers with adjustable preload with 4 settings – wheel travel 81 mm
Front brake	300-mm disc brake with two-piston floating calliper
Rear brake	240-mm disc brake with two-piston floating calliper
Front wheel rim	Aluminium alloy 16" x 3.00
Rear wheel rim	Aluminium alloy 14" x 3.50
Front tyre	Tubeless 110/70 - 16"
Rear tyre	Tubeless 140/70 - 14"
Length/Width	2,150 mm / 780 mm
Wheelbase	1,535 mm
Seat height	790 mm
Fuel tank capacity	12.5 litres
Kerb weight	165 Kg
Emissions compliance	EURO 3

Fig 2: Specifications of Piaggio bike

Solution methodology:

Property (1)	Symbol (2)	Value (3)
Rim Radius	R_r	309.5 mm
Bending Moment of Inertia	I_{zz}	795 mm^4
Bending Moment of Inertia	I_{rr}	1200 mm^4
Torsional Moment of Inertia	J	1139 mm^4
Hub Flange Radius	R_h	22.2 mm
Plane of Rim to Left Flange	H_L	36.7 mm
Plane of Rim to Right Flange	H_R	14.1 mm
Spoke Diameter	D_s	1.83 mm
Spoke Modulus	E_s	206 GPa
Rim Modulus	E_r	69 GPa
Rim Shear Modulus	G	26 GPa
Number of Spokes	N_s	36 spokes

Table I: Common Properties of wheels

Modelling of the Rim

Solid Worksv.10 is an advanced high-end CAD/CAM/CAE software package developed by Solid Works Corporation, USA is a part of Dassault Systems USA.Solid works Corporation was founded in 1993 by Jon Hirsctick to make the process of product design easier. This easy to learn tool makes it possible for mechanical designers to quickly sketch ideas, experiment with features and dimensions and produce models and detailed drawings. It is a feature based parametric solid modeling mechanical design and automation software. The model is designed by considering the following Dimensions

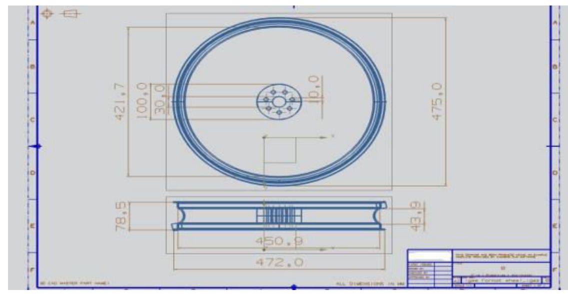


Fig 3: 2D Drawing dimensions are in mm

Drafting Views

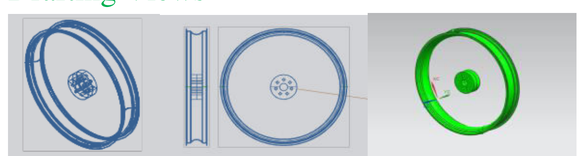


Fig 4: 3d Rim model

METHODOLOGY OF THE PREPROCESSING

Altair Hyper mesh is a high-performance finite element pre-processor to prepare even the largest models, starting from import of CAD geometry to exporting an analysis run for various disciplines.Importing the Cad model in the Preprocessing Software and meshing is done for the model and property and material is assigned to the model and Torsional analysis is carried out. Using the Solid Map option the 3D mesh is generated.

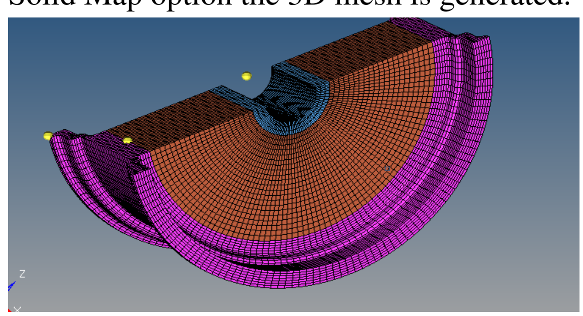


Fig 5: 3D solid map panel is used for tetra mesh

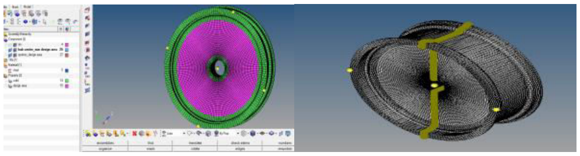


Fig 6: Complete model is meshed, Element qualityconnectivity checking using tool page edges.

3. Analysis:

Torsional and bending load Analysis is carried out for reference model, along with stress analysis of reference model and transient dynamic analysis is carried. Based on these analysis will improve the safety of reference wheel rim using new technology called Optistruct, stiffest design is achieved. By achieving new design which again will be re-designed with solid works using same reference dimensions of wheel rim. Torsional and bending load Analysis is carried out for reference model, stress analysis of reference model and transient dynamic analysis is carried out for new optimized design model. Finally will compare the reference results and optimized results by which will conclude the best and stiffest design. Below are the following steps which require solving the problem in RADIOSS software. To this meshed model material and Property is assigned and Load collector that is Constraints and load is applied and performed the Static analysis to the model and results are viewed in the Hyper view.Torsion and Rotating Analysis of Wheel Rim

Material Database:

Aluminum Die Casting Material is used for wheel rim, young's modulus; Poissons Ratio and Density are defined.

Property collector:

Property is assigned to the Model. Element type is mentioned in Hyper mesh as PSOLID..

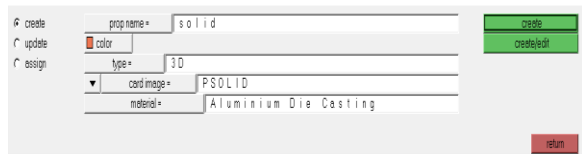


Fig 7: Complete model is meshed and Element Property is applied Fixed Constraints

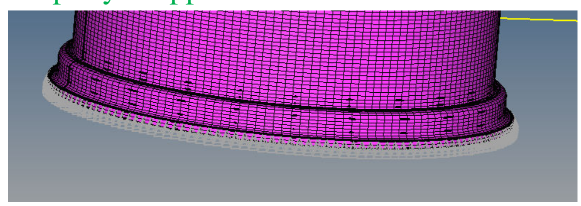


Fig 8: Complete model is meshed and fixed at the bottom rim

Load Applying

Load type of Force is applied to the model, This Force rotate the Shock Absorbers.

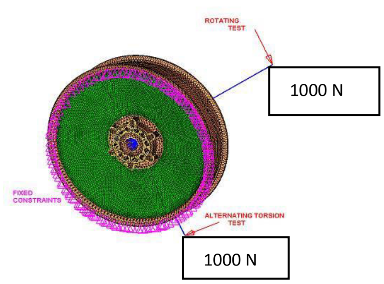


Fig 9: Complete model is meshed and fixed at the rim location, force is applied.

Rotating Bending Test and Alternative Torsion Test are carried out for the Base model with the force of 1000N and as above figure Constraints and loads are applied for the model.

Stress Analysis of Wheel Rim:

Stress analysis is carried out when the wheel rim has sudden impact to the ground or a rough road due to that deflection of wheel rim is checked. Using finite element method for reference model, descritized using meshing techniques like tetramesh. Material used is aluminum die casting; loads are applied about 5000 N from the ground and fixing at centre of the wheel rim. According to original equipment manufacturer like Piaggio uses same loads as mentioned above. As per any two wheeler wheel which had an impact of 2000 N from the ground but in this thesis factor of safety is taken more for comparison of two designs like reference and new optimized design. Steps involved to solve stress analysis using Hypermesh are explained below:

Preprocessing:

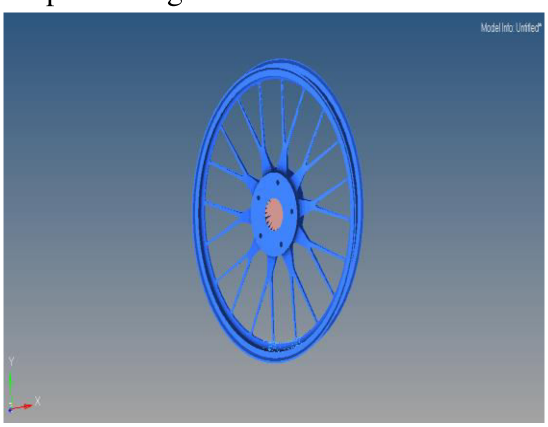


Fig 10: Reference model is imported in to Hyper mesh for preprocessing

Reference wheel rim is imported in to Altair Hypermesh for meshing using tetramesh panel which is very easy to mesh the complex model like above wheel rim. Enter in to 3d page which is in Hypermesh select the tetramesh option for meshing it. By mentioning the size of element is 10mm and minimum size 0.8mm and meshed using tetramesh.

Fig 11: Reference meshed model using tetramesh

Transient Dynamic analysis

Impact is a transient response dynamic analysis- load varying with the function of time, hence this analysis type will yield the forced response of the system ie. Tire, wheel, suspension and chassis (if modeled) to a varying load/time function. Sharp impacts produce impulse effects- that it as amplication of load due to the short time period in which it is applied. In transient response analysis loads are applied with respect to time. For 8 seconds transient response is calculated and checked the deflection of wheel rim compared with time.

Loads and Boundary Conditions:

Loads and Boundary conditions are specified according to the problem. Load is about 5000 N applied from the ground side, fixed at the centre of the wheel rim. The load about 5000 N is assumed when vehicle bumps into the uneven road surface; how the load impact is on the wheel rim the stress is calculated.

Fixed Constraints and Load Applying

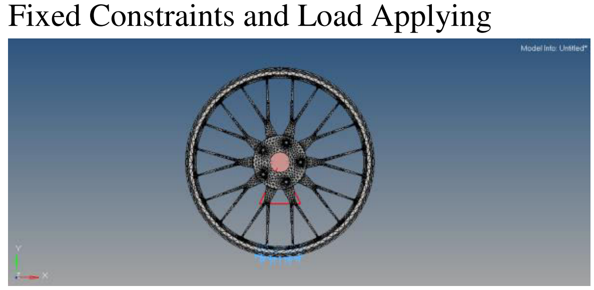


Fig 12: Meshed Model and Fixed at the centre of the rim and load is applied from

the ground

Piaggio Wheel Rim Results

Base Run using torsion and rotation test results is below:

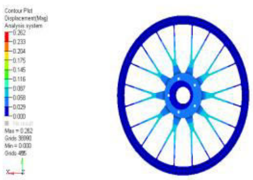


Fig 13: Reference Model Results Displacement 0.262mm

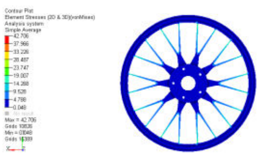


Fig 14: Reference Model Results Stress 42MPa

The CAD model has been validated according to the alternated Loadings conditions the maximum stress for the alternated bending is 42MPa acceptable for the required fatigue life (124MPa).To get a better design and stiffer design of wheel rim, topology optimization is carried out for the reference model. The results given by optimization is taken for new designing of wheel rim and using solid works the 3d cad model is designed and solved for same loading conditions using Altair Hypermesh for preprocessing, Radioss for solving and Altair Hyperview for post processing.

Base Run using stress analysis of wheel rim results is below:

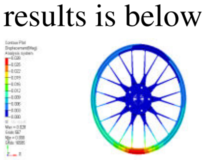


Fig 15: Reference Model Results Displacement is 0.028mm for stress analysis

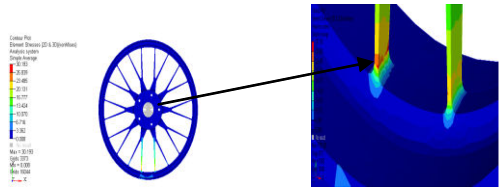


Fig 16: Reference Model Results Stress is 30MPa for stress analysis

As per above results stress is below the yield stress value of Aluminum die casting material, displacement is very less which is about 0.028mm. By this stress analysis of wheel rim is successfully completed and it is having more strength which is not failing.

Base Run using Transient Dynamic analysis of wheel rim results is below:

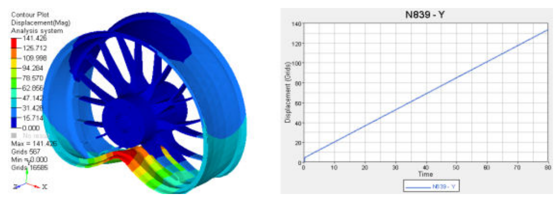
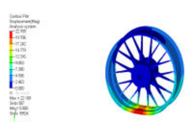


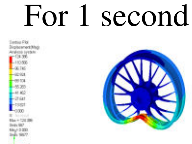
Fig 17: Time versus Displacement curve for reference model in Hyper graph

As shown in above figure displacement for reference model is about 141mm, which is failed and having less strength and tends to bend easy on the sudden impact from the ground. Time versus Displacment images are shown below:

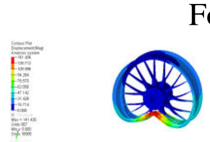


For 1 second

For 2 seconds



For 7 seconds



For 8 seconds

Fig 18: Time versus Displacement values in Hyper view

At 1 second displacement is about 22 mm, 2 second displacement is about 39 mm, at 7 second displacement is 124mm and at 8 seconds 141mm.As per dynamic analysis results for the reference wheel rim model is failed because deflection is more and rim tends to bend. To get a better design to this wheel, using optimization technique new design shape will be given by Optistruct. To do this optimization procedure is carried out in later chapter. By getting new design again torsion, rotating, stress analysis and transient dynamic analysis will be carried out. Comparing both the designs, final shape is considered for better safety.

OPTIMIZATION PROCEDURE

The primary objective of this study is to carry out the weight optimization of a car rim by using finite element analysis [7] in order to reduce the weight of vehicle components. The focus of this paper has been the weight reduction of an aluminum wheel by means of a topologic analysis. The topologic optimization result has realize the CAD model used for following structural validation

Design constraints:

1. Maximum Von Mises Stress < 30 MPa corresponding to the fatigue limit (0.3*R) of this material for 10^7 cycles.
2. % of mass reduction: 75 %

Manufacturing constraints:

Minimum rib thickness: 6 mm, Structure with polar symmetry (3, 4, 5, 6 spokes)

Geometric constraints:

Alternating Torsion and Rotating Bearing Test: the wheel must withstand the applied 1000 N load. Stress analysis of wheel rim, the impact from the ground is 5000 N.

Transient dynamic analysis, the impact from the ground is 5000 N.

Topology Optimization

Topology optimization generates an optimized material distribution for a set of loads and constraints within a given design space. The design space can be defined using shell or solid elements, or both. The classical topology optimization set up solving the minimum compliance problem, as well as the dual formulation with multiple constraints are available. Constraints on von Mises stress and buckling factor are available with limitations. Manufacturing constraints can be imposed using a minimum member size constraint, draw direction constraints, extrusion constraints, symmetry planes, pattern grouping, and pattern repetition. A conceptual design can be imported in a CAD system using an iso-surface generated with OSSmooth, which is part of the Optistruct package. Free-size optimization is available for shell design spaces. The shell thickness or composite ply-thickness of each element is the design variable.

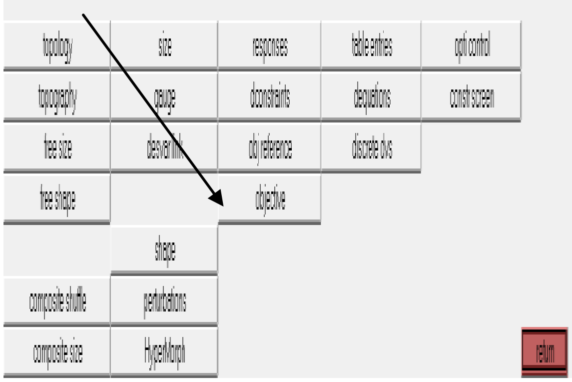


Fig 19: Optimization problem setup in Analysis page.

Topology optimization is selected

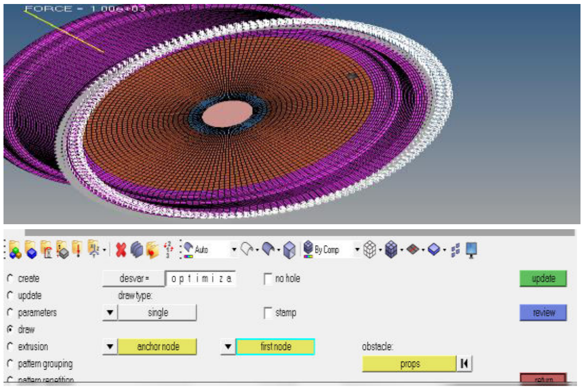


Fig 20 :Topology view

By using above options Topology setup is prepared and solved using Optistruct and seen the analysis results.

Optimization Results:

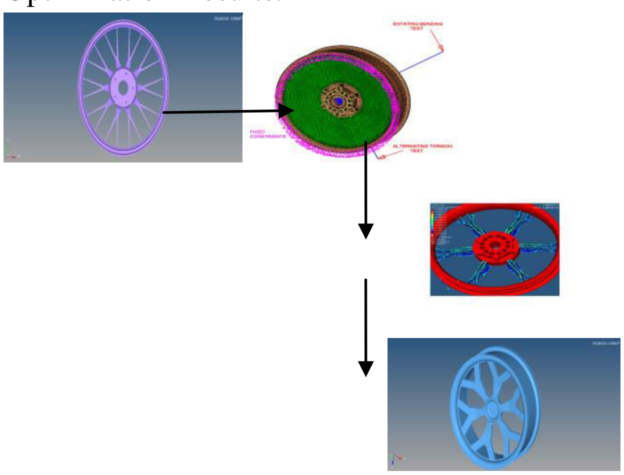


Fig 21: Optimized Model Results (six spokes)

Redesign of the Optimized model

The model is Redesigned based on the optimization results and performed the base run analysis for that model and results are compared to the original model and the optimized model.

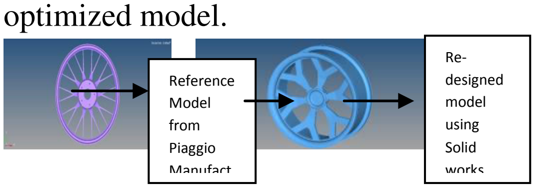


Fig 22: Optimized Model redesigned using Solid works

Again this model is meshed and boundary conditions are applied. To these model again the material and property is assigned and load steps are applied as per earlier application model and performed the base run analysis.Optimized Model Results carried out for torsion and rotating test:

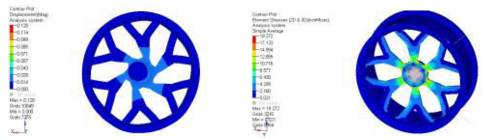


Fig 23: Optimized Model Results

Displacement 0.128mm, Stress 19MPa

Optimized model Run using stress analysis of wheel rim results is below:

Fig 24: Optimized Model Results Displacement is 0.049m

As per above results stress is below the yield stress value of Aluminum die casting material, displacement is very less which is about 0.049mm. By this stress analysis of wheel rim is successfully completed and it is having more strength which is not failing. Compare to the reference model displacement little bit high but stress of optimized wheel rim is having 15MPa.\Optimized model Run using Transient Dynamic analysis of wheel rim results is below:

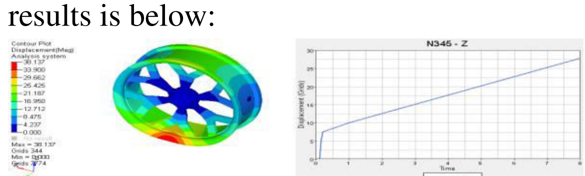
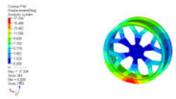


Fig 25: Time versus Displacement curve for optimized model in Hyper graph

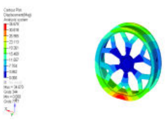
As shown in above figure displacement for reference model is about 38mm, which is having less deflection compared to reference model. The optimized model is having more strength compare to reference design.

Time versus Displacment images are shown below:

For 1 second



For 2 seconds



For 7 seconds

For 8 seconds

Fig 26: Time versus Displacement in Hyper view At 1 second displacement is about 13 mm, 2 second displacement is about 17 mm, and at 7 second Displacment is 34mm and at 8 seconds 38mm.

Conclusions:

1. Initial Weight of the reference model was 5.7 Kg, and the Final Weight obtained is 5.1 Kg, therefore percentage weight reduction is 11% achieved.
2. Displacement of reference model is 0.262mm, where the displacement of the Optimized model is 0.128mm achieved, for torsion and rotating test.
3. Stress value for reference model is 42MPa and the stress value for Optimized model is 19MPa. This is comparatively less value achieved, for torsion and rotating test.
4. On Second analysis ie. Stress analysis done on reference and optimized model the displacement and stress values are also achieved, the displacement for reference model is 0.028mm and for optimized model is 0.049 mm, the stress for reference model is 30MPa and for optimized model is 15MPa.
5. In transient dynamic analysis when load of 5000 N applied for a span of 8 seconds the displacement values obtained for 1 sec=22mm; 2secs=39mm to 7secs=124mm; 8secs=141mm for reference model.

The displacement values for optimized model are for 1 sec=13mm; 2secs=17mm to 7secs=34mm; 8secs=38mm.With respect to the above results obtained the reference model and optimized model results are under the fatigue life of the metal, reference model can also be considered but in the transient dynamic analysis the reference model fails as it tends to bend for the applied load and optimized model design does not bend and gives the best values and

Ana	Reference mode	Optimized mo	% Re
1.Torsic	1.Weight =5.7kg	1.Weight =5.1k	11%
Rotat	2.Displacement=	2.Displacement	51%
Anal	3.Stress=42MPa	3.Stress=19MP	54%
II. Stres	1. Displacement	1. Displacemen	42%
	2. Stress=30MPa	=0.049mm	50%
III. Tran	Displacement fo	2. Stress=15MF	
Dyr	1 sec=22m	Displacement f	40%
Ana	2 sec=39m	1 sec=13	56%
	7 sec=124	2 sec=17	72%
	8 sec=141	7 sec=34	73%
		8 sec=38	

is of good strength and better durability.

References

1. S. Karthick et al "Modeling and Analysis of Car Wheel Rim by using Peek Material" International Journal for Scientific Research & Development| Vol. 3, Issue 03, 2015,ISSN: 2321-0613,pp.3372-

3374.

2. BGN Satya prasad "Topology Optimization of Alloy Wheel" Altair technology conference-2013, Hyundai Motor India Engineering , Hyderabad.
3. J. Janardhan et al "Radial Fatigue Analysis of An Alloy Wheel" Int. Journal of Engineering Research and Applications, ISSN : 2248-9 622, Vol. 4, Issue 12, Part 6, December 2014, pp.253-258
4. Mr. Sushant et al "Optimization Of Car Rim" Int. Journal of Engineering Research and Applications, ISSN: 2248-9622, Vol. 5, Issue 10, (Part - 2) October 2015, pp.01-08
5. Daniel Antony C et al "Design and Analysis of Two Wheeler Alloy Wheel Rim Using Composite Materials" International Journal of Innovative Research in Science, Engineering and Technology, ISSN: 2319-8753 Vol. 5, Issue 5, May 2016.
6. Sourav Das, "Design and Weight Optimization of Aluminium Alloy Wheel", International Journal of Scientific and Research Publications, Volume 4, Issue 6, June 2014, ISSN 2250-3153
7. P. H. Yadav et al "Optimization of Car Rim Using OptiStruct" Journal Of Environmental Science, Toxicology And Food Technology, ISSN: 2319-2402, ISBN: 2319-2399. Volume 2, Issue 3 (Nov. - Dec. 2012), PP 10-15
8. H. N. Kale et al "Comparative Study of Wheel Rim materials", International Journal of Advance Research and Innovative Ideas in Education, ISSN:2395-4396 1437, Vol-1 Issue-5 2015,pp:257-260.