Experimental & Finite Element Analysis of Four Wheeler Alloy Wheel by using Strain Gauge Method

Pratibha V. Kendre[1], S.M. Nagure[2], Dr. N. A. Rawabawale[3]

[1]M.Tech student, Department of Mechanical Engineering, M.B.E.S college of Engineering Ambajogai, Maharashtra, India.
[2]Assistant Professor, Department of Mechanical Engineering, M.B.E.S college of Engineering Ambajogai, Maharashtra, India.
[3]Head & Associate Professor, Department of Mechanical Engineering, M.B.E.S college of Engineering Ambajogai, Maharashtra, India.

---***---

Abstract - *The fundamental of vehicle wheel is to give a firm base on which to fit the tire. Its measurements, shape ought to be reasonable to sufficiently oblige the specific tire required for the vehicle. In this undertaking a feel burnt out on vehicle wheel edge having a place with the circle wheel class is thought of. Configuration is a significant mechanical action which impacts the nature of the item. Car application which is completed paying uncommon reference to enhancement of the mass of the wheel. The Finite Element investigation it shows that the improved mass of the wheel could be decreased to around some rate when contrasted with the current strong circle type wheel. In this paper, an examination has been made to streamline the mass of the center point edge using limited component investigation. Test of wheel will be performed on UTM. The similar investigation was completed between the Analytical and test results. By topology optimization technique weight reduction of around 11.1 % is observed along with strain measurement of 116.42 microns and 114 microns by numerical and experimental testing respectively.*

Key Words: 4-Wheler Rim, Topology Optimization, UTM.

1. INTRODUCTION

Car wheels have advanced throughout the decades from early talked plans of wood and steel, level steel plates lastly to the stepped metal designs and present day cast and manufactured aluminum composites edges of the present current vehicles. Generally, fruitful structures showed up following quite a while of experience and broad field testing. As of late, the strategies have been improved by an assortment of trial and logical techniques for auxiliary investigation (strain measure and limited component strategies). Inside the previous 10 years, toughness examination (weariness life predication) and unwavering quality techniques for managing the varieties innate in building structure have been applied to the car wheel. Wheels are obviously wellbeing related parts and thus weariness execution and the condition of worry in the edge under different stacking conditions are prime concerns. Further, wheels keep on getting a lot of consideration as a major aspect of industry endeavors to diminish weight through material replacement and down measuring. Further the current age car has the amalgam wheel. This innovation up degree has given numerous options in regard of material, cross segment for edge and arm interfacing center point and edge. The more current vehicle should have lesser load without bargaining the quality. In this manner, there is a degree for streamlining of wheel plan in regard of geometry of vehicle edge, geometry of arm, material and so forth.

1.2 OBJECTIVES

- Modeling of 4 wheeler rim in CATIA V5 software.
- FEA analysis in ANSYS to determine stresses and deformation in wheel rim of vehicle.
- Topological optimization of the existing model.
- To perform experimental testing of new, optimize wheel rim on UTM.
- Experimental testing and correlating results.

2. LITERATURE REVIEW

Mr. Sushant K. Bawneet.al. Explains that 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. Tire of car wheel rim belonging to the disc wheel category is considered. Design is an important industrial activity which influences the quality of the product. The wheel rim is modeled by using modeling software catiav5r17. By using this software the time spent in producing the complex 3- D models and the risk involved in the design and manufacturing process can be easily minimized. So the modeling of the wheel rim is made by using CATIA. Later this CATIA modal is imported to ANSYS WORKBENCH 14.5 for analysis work. It is the latest software used for simulating the different forces, pressure acting on the component and also calculating and viewing the results. This software reduces the time compared with the method of mathematical calculations by a human. ANSYS static structural analysis work is carried out by considered three different materials namely aluminum alloy, magnesium alloy and structural steel and their relative performances have been observed respectively. In addition to wheel rim is subjected to modal analysis, a part of dynamic analysis is carried out its performance is observed. In this analysis by observing the results of both static and dynamic analysis obtained magnesium alloy is suggested as best material [1].

Mr. P. H. Yadav et.al. Explains that Automobile industries require manufacturing vehicles at optimum cost and with greater safety. For this purpose every component of vehicle is analyzed for critical conditions. In this work car rim is considered for optimization by using finite element method. The optimization is carried to minimize the weight of rim without exceeding allowable strain. The intention is to create the geometry utilizing parameters for all the variables, deciding which variables to use as design, state and objective variables to obtain an accurately converged solution. FEM software OptiStruct is used for optimization of rim. If the stress & strain values are within the permissible range, then certain dimensions are modified to reduce the amount of material needed. The procedure is repeated until design changes satisfy all the criteria. For experimental verification similar type of object is used and stresses in it are calculated for set load with suitable instrumentation. Then same is calculated in OptiStruct software and Experimental results are compared with FEM results [2].

H. Akbulut et.al. Explains that the optimization of an octopus-type car rim for which critical zones were found first and then optimum thickness was investigated using an elasto-plastic analysis. In this study, three-dimensional finite element method was used for conducting elasto-plastic analysis. In the finite elements analysis, the elements forming the meshes are hexahedral linear elements with eight nodes. Twelve different meshes were used. A quadrant of the rim was utilized due to its symmetric shape. The theoretical results were compared with experimental ones. It seems that the theoretical results are in agreement with the experimental ones [3].

Sourav Das et.al. Explains that the design of aluminum alloy wheel for automobile application which is carried out paying special reference to optimization of the mass of the wheel. The Finite Element analysis it shows that the optimized mass of the wheel rim could be reduced to around 50% as compared to the existing solid disc type Al alloy wheel. The FE analysis shows that the stress generated in the optimized component is well below the actual yield stress of the Al alloy. The Fatigue life estimation by finite element analysis, under radial fatigue load condition, is carried out to analyze the stress distribution and resulted displacement in the alloy wheels. S-N curve of the component depicts that the endurance limit is 90 MPa which is well below the yield stress of the material and safe for the application. The FE analysis indicated that even after a fatigue cycle of 1020, the damage on the wheel is found only 0.2% [4].

S. Phani Kumar et.al. Explains that Automobile wheels have in the time period spanning the last five decades progressively evolved from the early spoke design of wood and steel the carryovers from wagon and bicycle technology, flat steel discs, and more recently stamped metal configurations. The metal configurations are made from either cast or cast plus forged aluminum alloys in the present and newer generations of ground vehicles. This project work summarizes the application of Finite Element Techniques for analyzing stress and displacement distribution in vehicle wheels subjected to the conjoint influence of inflation pressure and radial load. Wheel strength with regard to the fractures on edges and other critical points when the wheel strikes an obstacle shall be checked. In order to show the sufficient resistance to fractures it is necessary to carry out an impact test. An impact load (as per SAE standards) is applied on the wheel at a determined angle. The modal analysis is carried out for observing the natural frequencies. Static analysis for the given radial load and the angular velocities of 40, 60, 80, and120, with the appropriate constraint set of conditions in order to check the wheel strength. Yield strength of the alloy wheel material is checked with the Von-Mises stress obtained, (as per the ISO 7141 Road Vehicles Wheels Impact Test Procedure, and SAE J175 Impact Test Procedures standards).Therefore the study states that the innovative design of the wheel is safe for its operating conditions for the given loads specified [5].

3. METHODOLOGY

Methodology used in this paper is as follow,

Step 1: Started the work of this project with literature survey. I gathered many research papers which are relevant to this topic.

Step2: After that the components which are required for our project are decided.

Step 3: After deciding the components, the 3D Model and drafting will be done with the help of CATIA software.

Step 4: The Analysis of the components will be done with the help of ANSYS using FEA.

Step 5: The Experimental Testing will be carried out.

Step 6: Comparative analysis between the experimental & analysis result & then the result & conclusion will be drawn.

4. CATIA MODEL OF ALLOY WHEEL

This is existing alloy wheel cad model in CATIA software with its drafting.

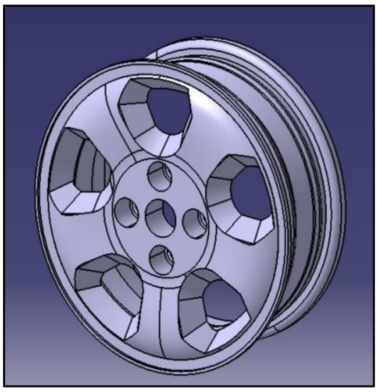


Fig -1: CATIA Model of Alloy Wheel

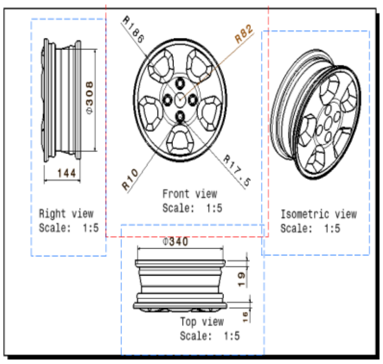


Fig -2: Drafting of Alloy Wheel

5. FEA ANALYSIS

Material Selection – Aluminium Alloy

Properties of Outline Row 3: Aluminum Alloy

	A	B	C
1	Property	Value	Unit
2	Material Field Variables	Table	
3	Density	2770	kg m^-3
4	Isotropic Secant Coefficient of Thermal Expansion		
5	Coefficient of Thermal Expansion	2.3E-05	C^-1
6	Isotropic Elasticity		
7	Derive from	Young's Modulus and Pois...	
8	Young's Modulus	7.1E+10	Pa
9	Poisson's Ratio	0.33	
10	Bulk Modulus	6.9608E+10	Pa
11	Shear Modulus	2.6692E+10	Pa

Fig -3: Drafting of Alloy Wheel

5.1 Boundary Condition

In this analysis following boundary conditions were used:

- At point A apply pressure which is pressure of tire.
- Point B considered as fixed support because hub connected
- At point C apply force in down word direction (consider static condition)

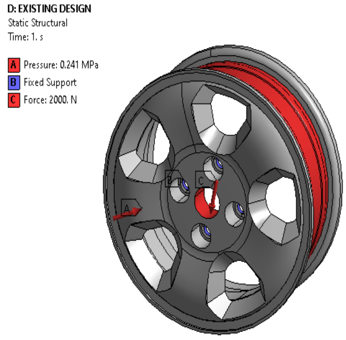


Fig -4: Details of Boundary conditions

5.2 Meshing

ANSYS Meshing may be a general-purpose, intelligent, automated high-performance product. It produces the foremost appropriate mesh for accurate, efficient Multiphysics solutions. A mesh compatible for a selected analysis are often generated with one click for all parts during a model. Full controls over the choices want to generate the mesh are available for the expert user who wants to fine-tune it. The facility of multiprocessing is automatically wont to reduce the time you've got to attend for mesh generation. After meshing of steering upright nodes are 183069 and elements 105877.Initia weight of wheel is 8.3699 kg.

5.3 Analysis Results

Maximum deformation under static condition of steering upright 0.0148 mm is observed &Maximum stress observed around 7.52 MPa.

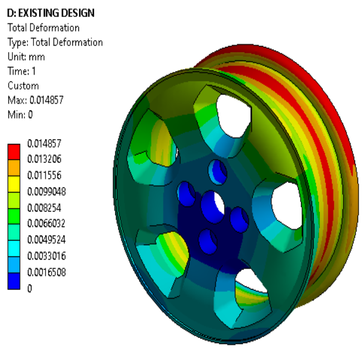


Fig -5: Total deformation

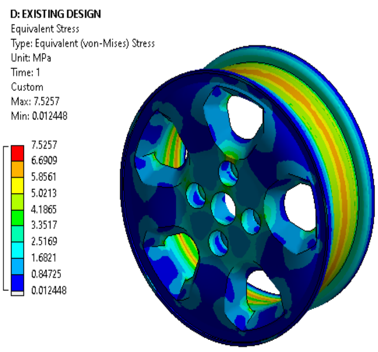


Fig -6: Equivalent stress

6. TOPOLOGY OPTIMIZATION

Topology optimization may be a mathematical approach that optimizes material layout within a given design area, for a given set of loads and boundary conditions such that the resulting layout meets a prescribed set of performance targets.

There are three kinds of structure optimization,

- Size optimization
- shape optimization
- Topology optimization

Three optimization ways that correspond to the three stages of the product design methodology, significantly the detailed design, basic design and conceptual design. Size optimization keeps the structural form and topology structure invariant, to optimize the various parameters of structure, like thickness, section size of beam, materials properties; shape optimization maintains the topology structure, to vary the boundary of structure and form, search for the foremost applicable structure boundary scenario and shape; topology optimization is to hunt out the most effective path of materials distribution throughout never ending domain that meet the displacement and stress conditions in structure, produce a selected performance optimum. Thus, compared to size and shape optimization, topology optimization with more freedom degree and larger design area, its greatest feature is below unsure structural form, in keeping with the well-known condition and a given load to figure out the cheap structure, every for the abstract variety of recent product and improvement design for existing product, it's the foremost promising side of structural optimization. For continuous structure topology optimization, there are some mature ways like: uniform technique, evolutionary structural optimization technique, variable density technique etc. Uniform technique introduced cell structure of micro structure (unit cell) at intervals the elements of the structure, each unit has three forms, significantly non-material voids (size = 1), isotropic-material entity medium (size = 0) and orthotropic-material opening-hole medium (0 < size < 1). Whereby the distribution of each form are able to describe the form of topology and conjointly the form of structure; evolutionary structural optimization technique believes that stress in any elements of the structure should beneath the identical level in an ideal structure. Which suggests the native material with a low stress state isn't entirely used, thus you'll be able to delete the material artificially. Thus bit by bit remove material that in a low stress state, then delete the update rate, thus optimized structure becomes more uniform. Variable density technique is used to conduct optimization throughout this paper.

6.1 Boundary Condition for optimization

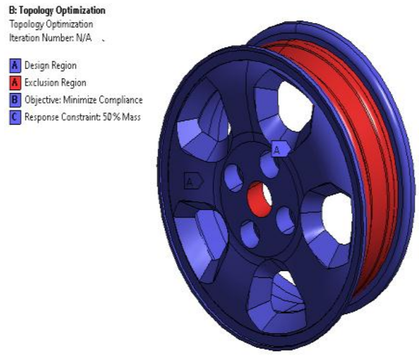


Fig -7: Boundary condition for topology optimization

In boundary condition non design area indicated in red region include the boundary condition in static structural analysis and design area is indicated in blue region.

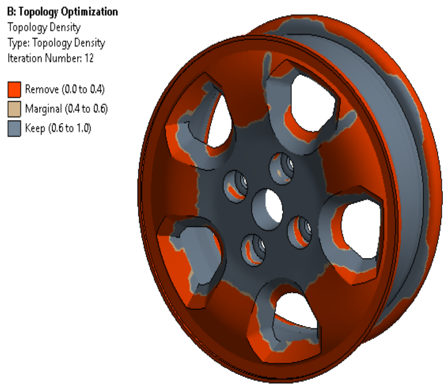


Fig -8: Topology optimization results

Topology optimization is performed after static structural analysis with existing boundary condition to determine material removal area. After, performing topology optimization red region indicates the material removal area from which material can be removed as per our need. So, in our case original mass is 7.04 kg but removal of material is about 55 % which lead to 4.03 kg as per software. But it depends on us to removal of material by proper design and reanalysis as per existing conditions to sustain boundary condition.

6.2 Optimized model & Meshing

By considering ease manufacturing holes created on the spokes of the wheel so that it will not create any obstacles to tire mounting and hub.

Fig -9: Optimized design

After meshing of steering upright nodes are 181274 and elements 104233.Initia weight of wheel is 7.4088 kg. That means 11.1% weight is optimized.

6.3 Boundary Condition for optimized model

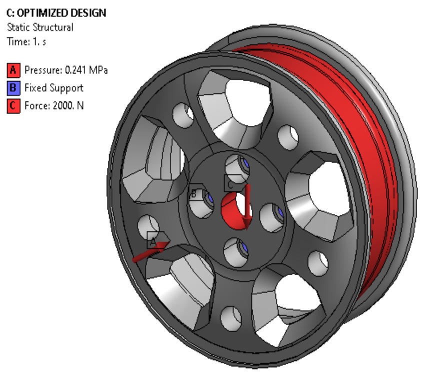


Fig -10: Optimized design boundary condition

6.4 Analysis Results

Maximum deformation 0.0153 mm is observed & Maximum stress observed around 7.4004 MPa.

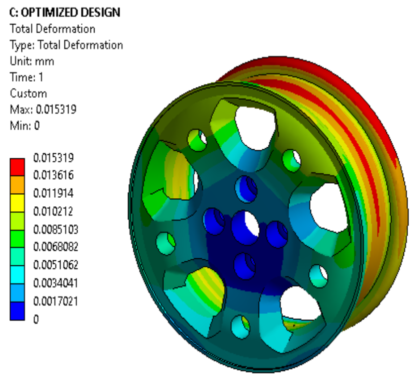


Fig -11: Optimized design deformation result

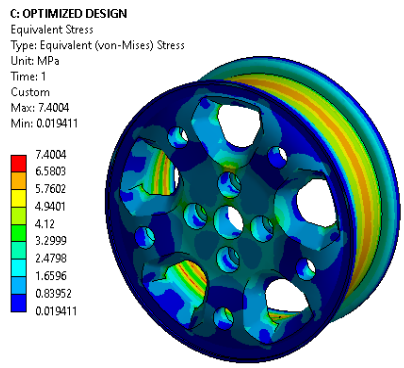


Fig -12: Optimized design equivalent stress result

7. EXPERIMENTAL SETUP FOR UTM

A universal testing machine (UTM), is used to test the tensile strength and compressive strength of materials. "Universal" part of the name reflects that it can perform many standard tensile and compression tests on materials, components, and structures. The set-up and usage are detailed in a test method, often published by a standards organization. This specifies the sample preparation, fixturing, gauge length, analysis, etc. The specimen is placed in the machine between the grips and an extensometer if required can automatically record the change in gauge length during the test. If an extensometer is not fitted, the machine itself can record the displacement between its cross heads on which the specimen is held. However, this method not only records the change in length of the specimen but also all other extending or elastic components of the testing machine and its drive systems including any slipping of the specimen in the grips. Once the machine is started it begins to apply an increasing load on specimen. Throughout the tests the control system and its associated software record the load and extension or compression of the specimen.

Table -1: Specification of UTM.

1	Max Capacity	400KN
2	Measuring range	0-400KN

3	Least Count	0.04KN
4	Clearance for Tensile Test	50-700 mm
5	Clearancefor Compression Test	0- 700 mm
6	Clearance Between column	500 mm
7	Ram stroke	200 mm
8	Power supply	3Phase, 440Volts, 50 cycle. A.C
9	Overall dimension of machine (L*W*H)	2100*800*2060
10	Weight	2300Kg

7.1 Experimental testing in FEA

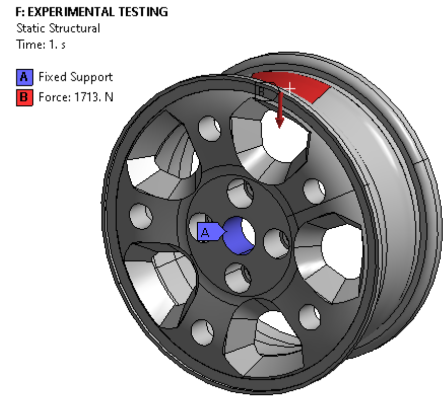


Fig -13: Experimental testing boundary condition

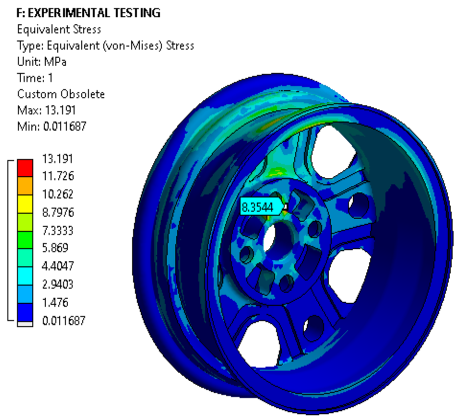
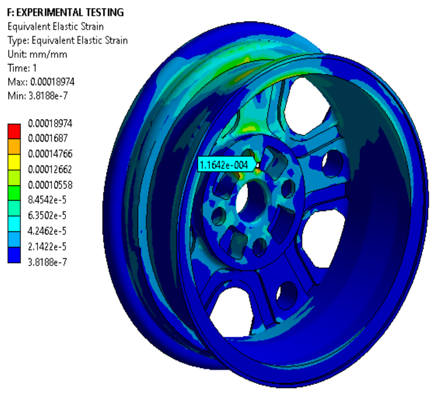


Fig -14: Experimental testing deformation results



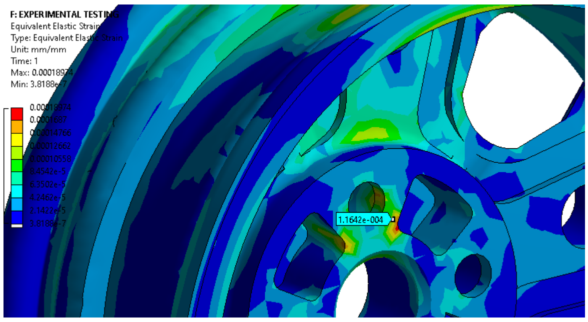


Fig -15: Experimental testing strain results

By FEA analysis strain is observed around 116microns.

7.2 Experimental testing using strain gauge

Testing of wheel on UTM fixture is manufactured according to component designed. Single force is applied as per FEA analysis and reanalysis is performed to determine strain by numerical and experimental testing. Strain gauge is applied as per FEA results to maximum strained region and during experimental testing force is applied as per numerical analysis to check the strain obtained by numerical and experimental results. During strain gauge experiment two wires connected to strain gauge is connected to micro controller through the data acquisition system and DAQ is connected to laptop. Strain gauge value are displayed on laptop using DEWESOFT software.

Fig -16: Experimental testing setup

8. CONCLUSIONS

Static structural analysis of existing 4 wheeler alloy wheel. It is observed that around maximum deformation is 0.014 mm and equivalent stress is 7.52 MPa. By topology optimization technique weight reduction of around 11.1 % is observed along with strain measurement of 116.42 microns and 114 microns by numerical and experimental testing respectively.

ACKNOWLEDGEMENT

The author likes to thank to Mr. Kunal Shingare, Engineer Bolt Technology, Pune for their valuable time and guidelines.

REFERENCES

[1]Mr. Sushant K. Bawne, Prof. Y. L.Yenarkar "Optimization of Car Rim" Int. Journal of Engineering Research and Applications ISSN: 2248-9622, Vol. 5, Issue 10, October 2015

[2]Mr. P. H. Yadav, Dr. P. G. Ramdasi "Optimization of Car Rim Using OptiStruct" IOSR Journal of Environmental Science, Toxicology and Food Technology (IOSR-JESTFT) ISSN: 2319-2402, ISBN: 2319-2399. Vol. 2, Issue 3 (Nov-Dec 2012)

[3]H. Akbulut, on optimization of a car rim using finite element method, Finite Elements in Analysis and Design 39 (2003) 433–443

[4]Sourav Das "Design and Weight Optimization of Aluminium Alloy Wheel" International Journal of Scientific and Research Publications, Vol 4, Issue 6, June 2014 ISSN 2250-3153

[5]S. Phani Kumar, M. Raja Roy, M. Sailaja, Finite Element Analysis of Alloy Wheel International Journal of Engineering and Management Research ISSN: 2250-0758 Volume-5, Issue-2, April-2015