Design and Optimization of Alloy Wheel Of 2-Wheeler Vehicle

Anupama Chahande, Dr. Laxman V. Kamble
M.E. Design, D. Y. Patil Institute of Technology,
Savitribai Phule Pune University.

Abstract— Now a day's increase in vehicle demand at a lower rate is necessity of vehicle manufacture. Expectation in terms of durability, efficiency and cost is raising exponentially by customers. Over designed components in vehicle loads to increase weight and reduced efficiency. Alloy wheels carry sprung mass of vehicle and are supported by suspension system, they can be redesigned and modified for achieving less weight and thus cost. Existing Alloy component was available in market which is reverse engineered and CAD modeled using CATIA V5 software. Finite element Discretization & Analysis is carried out using ANSYS package. FEA helped in finding out high strain locations in component and also defining area which can be modified and also optimized model is obtained by topology optimization technique. Strain gauging will be done on areas of high strain location indicated by FEA software. Fixture is designed for mounting component on UTM. Vertical loading simulating steering linkage leading to cantilever conditions is achieved using UTM. Machining has been done to remove excess material from component. Comparative analysis is done between FEA & Experimental strains.

Keywords—Alloy wheels, 2- Wheeler, topology optimization, FEA.

I. Introduction

Alloy wheels were initial developed within the last sixties to satisfy the demand of course enthusiasts who were perpetually searching for a position in performance and styling. It had been associate unorganized business then. Since its adoption by Original Equipment manufacturers (OEM's), the alloy wheel market has been steady growing. Today, because of a lot of refined and environmentally aware client, the employment of alloy wheels has become progressively relevant [1]. With this increased demand came new developments in style, technology and producing processes to supply a superior with a large style of designs. Within the fatigue life analysis of aluminium wheel design, the unremarkably accepted procedure for carriage wheel producing is to pass 2 durability tests, specifically the radial fatigue take a look at and cornering fatigue test [2]. Since alloy wheels area unit designed for variation stylish and have a lot of complicated shapes than regular steel wheels, it's troublesome to assess fatigue life by exploitation analytical strategies [3]. Wheels have very important importance for the protection of the vehicle and a special care is required so as to make sure their durability. The event of the vehicle industry has powerfully influenced the look, the fabric choice and therefore the producing processes of the wheels [4]. The wheels loading manner may be a complicated one, additional improvement and economical wheel style are attainable provided that they are loading are higher understood. So as to realize associate optimum style of the wheel, 2 needs area unit needed: the precise information of the loading and therefore the mechanical properties and allowable stresses of the material that depend upon the vehicle characteristics, service conditions and producing processes [5]. Another risk is to use the finite element methodology so as to ascertain the stresses within the wheel rim and to check the various style solutions.

Fig. 1: Alloy wheel of 2 wheeler vehicle

Done Modeling of existing 2-wheeler vehicle alloy wheel in CATIA software. Determined stresses and deformation using ANSYS software using static structural analysis Formulated optimized model using topological optimization technique.
Optimized model used to determine for stresses and deformation. Machined of existing wheel as per optimized model obtained from topology optimization. Designed and manufactured fixture to hold wheel firmly in testing. Experimental testing is done and correlating results has find out.

II. LITERATURE REVIEW

Madhu K Sa et al. [1]This paper state that, the industries in the automobile sector are going to explore the composite material to achieve reduction of weight without significant decrease in vehicle quality and reliability. Reduction of weight leads to more precise handling and minimizing the fuel consumption. Aluminum alloys is the material presently used material for manufacture of two wheeler alloy wheel. Composites are the only materials that cater to the never ending demand of the material technology. Aluminum are the most widely used composite materials in auto mobile sectors due to their light weight and superior strength. Presently aluminum alloys are replaced for steel wheels. Main aim of the paper is presenting the new material for two wheeler alloy wheel by using composite materials.

Sandeep Kumaret al. [2], this paper state that, accurate finite element model of a structure is required in order to carry out its dynamic design at the computer level. It has been observed that from this paper due to various approximations made in the Finite Element model the predicted values of the dynamic characteristics differs from that of the actual structure. In view of this, the FE model is updated in the light of the test data from Experimental modal analysis (EMA) so as to improve its correlation with the test data. In this paper updating FE model of a two wheeler chassis structure is presented. Chassis is considered in the assembled condition with engine, suspension and tires. First an approximate FE model is constructed followed by modal analysis on the structure.

Poonam Meshram et al. [3] from this paper they learn about material which is used for alloy wheels in automobile. Wheels are made from an alloy of aluminum or magnesium metals or sometimes a mixture of both. These wheels differ from normal steel wheels because of their lighter weight, which improves the steering and the speed of the car. Alloy wheels will reduce weight of a vehicle compared to one fitted with standard steel wheels. They find the benefit of reduced unstrung weight is more precise steering as well as a nominal reduction in fuel consumption. Alloy is an excellent conductor of heat. This improves heat dissipation from the brakes and reducing the risk of brake failure under demanding driving conditions. At present four-wheeler wheels are made from Aluminum Alloys. In this study they compare aluminum alloy with another alloy.

M. Vamsi Krishna et al. [4] Wheel is important in automobile is. The most ordinarily utilized materials for making wheel spokes are with highlights of phenomenal softness, warm conductivity, erosion opposition, qualities of throwing, frigidity, high damping property, machine handling and reusing, and so forth. This metal has main advantage is to reduce weight and increase accuracy.. This metal is useful for energy conservation because it is possible to re-cycle. Spokes make vehicles look great but they require attention in maintenance. To perform their functions best, the spokes must be kept under the right amount of tension. They use Aluminum alloy wheels. Aluminum alloy is replaced by magnesium alloy .

Sourav Das et al.[5] This paper they use aluminum alloy wheel for automobile application for optimization of the mass of the wheel. They do the optimization for mass of wheel and it reduced to around 50% as compared to the existing solid disc type Al alloy wheel. The FE analysis shows that the strain generated within the optimized component is well below the particular yield stress of the Al alloy. The Fatigue life estimation by finite element analysis, under radial fatigue load condition, is administered to research the strain distribution and resulted displacement within 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 fabric and safe for the appliance. The FE analysis indicated that even after a fatigue cycle of 1020, the damage on the wheel is found only 0.2%.

X Jiang et al. [6] in this paper they talk about light weight in automobile vehicle. Lightweight is a significant application trends, using magnesium alloy wheels is a valuable way. They discuss the design of a new model of automobile wheel. They do the bending test and radial test finite element model. They use the three different materials namely magnesium alloy, aluminum alloy and steel, and find out the stress and strain performances of each material. After testing and analyzing the model in bending test and radial test, they find reasonable and superior results for magnesium alloy wheel. They compare the results of the equivalent stress and deformation, the magnesium alloy wheel practicality has been confirmed.

Daniel Antony Cet al. [7] this paper is all about composite material. Automobile wheels which are made from an alloy of aluminum or magnesium metals (or sometimes a mixture of both). These wheel differ from normal steel wheels because of their lighter weight, which improves the speed of the vehicle, in the design of automobile, they replace the aluminum alloy to reduce the weight without significant decrease in vehicle quality and reliability by polymeric composite material. By using this material they achieve the mass optimization. The reduction of weight of a vehicle directly impacts fuel consumption.

K.Sankara Narayanan et al. [8] in this study they talk about the wheel rim. In two wheel , there are two types of wheels used, one is alloy wheel and another one is spokes wheel. Most of the time alloy materials are used for fabricate the wheel rim. Reason of using alloy wheel is alloy material is increase the efficiency of the two wheeler by reducing the weight. This project is to create the motor bike wheel rim model by using SOLIDWORKS software and the created model is analyzed by the ANSYS workbench software. The structures and system are evaluated by the ANSYS tool. They do the analysis for complex structures. Visualization, pre-processing and analysis are the three process involved in the ANSYS.

N.V.Dhandapani et al. [9] this paper give the information about design optimization of vehicle rim using Finite element analysis. They concentrated on weight optimization without compromising its performance and strength. They use reverse engineering technology for design 3-dimensional model of the alloy wheel. They use existing alloy wheel dimensions in CAD model and the part is completed with Solid works 2015.

Problem Statement

Wheel is one of the most important components of an automobile. Currently, wheel contains excess material, leads to increase in weight of the vehicle. Directly affects the mileage and cost. In this thesis, modeling existing wheel in CAD software and analyzing it for induced structural stresses and deformation in CAE software. Then using Topology optimization excess material will be removed. It is also tested experimentally for the structural stresses and results were correlated with analysis results.

III METHODOLOGY

Optimization is done by some methodology, analysis and optimization which is done by experimental testing and FEA analysis. Taken existing alloy wheel of two wheeler from market. Removed optimized material manually in optimized area by adding some holes. Designed and manufactured the fixture considering the size of wheel for mounting wheel on UTM. Vertical loading simulating steering linkage leading to cantilever conditions is achieved using UTM. And data is collected.

In FEA analysis, CAD modelling is done on existing wheel by CATIA V5, apply material and property for discretization Meshing is done from cad model apply vertical load and boundary condition to the wheel using some constraints. Results are came out in the form of stress strain plots. On this results topology optimization is done on these results to remove some materials again did analysis in the form of plot of stress strain result comparison is done and concluded the results,

CAD MODEL OF 2-WHEELER ALLOY WHEEL

The CAD design of wheel is prepared based on the standard nomenclature at the outer and the hub region of the wheel. Figure 3. Shows the CAD design of the wheel rim before optimization

The material used is die casted aluminum. The composition is used in this alloy are silicon, copper, magnesium, iron, manganese, and zinc.

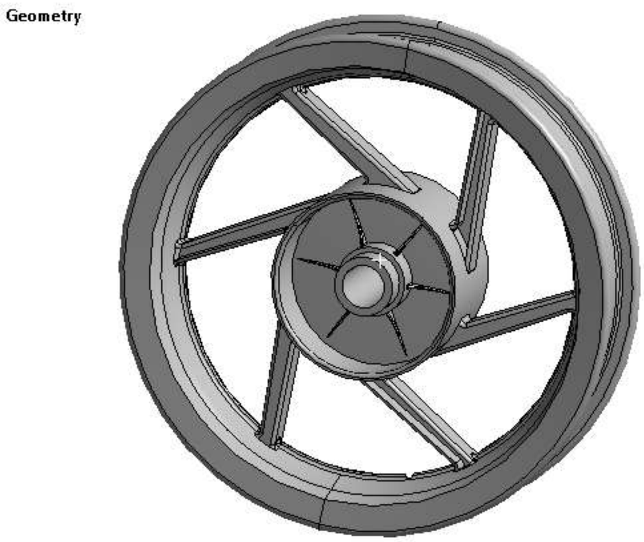


Fig. 3: CATIA model of existing fender

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		
6	Isotropic Elasticity		
7	Derive from	Young's Modulus and Poiss...	
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

Details of "Body Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	8.0 mm

FEA ANALYSIS OF 2-WHEELER ALLOY WHEEL

MESH

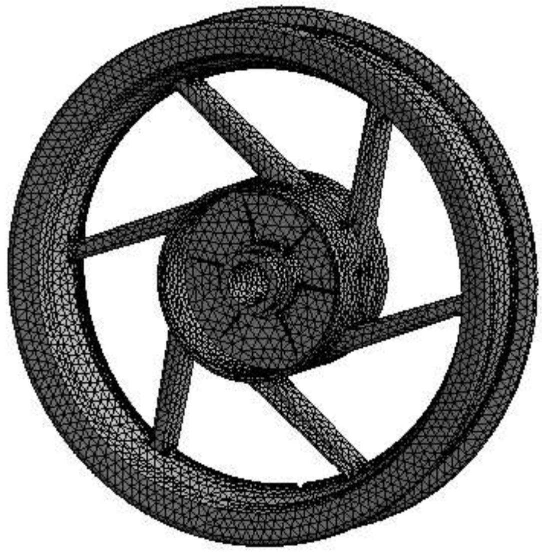
Design of existing 2-wheeler alloy wheel is designed in CATIA software and FEA analysis is performed using ANSYS software for determination of stress, strain and Topology optimized design using ANSYS topology optimization module.

Initially CATIA designed model is imported in ANSYS as per above pic mentioned for analysis.

Meshing is a general-purpose, intelligent, automated high-performance product. It produces the most appropriate mesh for accurate, efficient Multiphasic solutions. A mesh well suited for a specific analysis can be generated with a single mouse click for all parts in a model. Full controls over the options used to generate the mesh are available for the expert user who wants to fine-tune it.

The power of parallel processing is automatically used to reduce the time you have to wait for mesh generation. Creating the most appropriate mesh is the foundation of engineering simulations. ANSYS Meshing is aware of the type of solutions that will be used in the project and has the appropriate criteria to create the best suited mesh. ANSYS Meshing is automatically integrated with each solver within the ANSYS Workbench environment.

The FE model is prepared for 36° of its circumference as the remaining part can reflected exactly. The required modifications can also be done in the same portion. Tetrahedral elements are used for modelling.



Statistics	
Nodes	121300
Elements	71230

Fig.4. Meshing of 2-Wheeler Alloy Wheel

Boundary Condition

A boundary condition for the model is the setting of a known value for a displacement or an associated load. For a particular node you can set either the load or the displacement but not both.

The main types of loading available in FEA include force, pressure and temperature. These can be applied to points, surfaces, edges, nodes and elements or remotely offset from a feature. In 2-wheeler alloy wheel outer hub rim is fixed as it is stationary. Load on inner circumference is applied as per above defined and pressure of tire are applied on rim.Fixed support is applied at edge support Pressure of tire nearly 30 psi is applied in the form of 0.206 mpa. Weight of bike is considered around 110 kg so, on each wheel weight acts is 55 kg in which 30 % is absorbed by suspension so 30% of weight is applied 1650 n

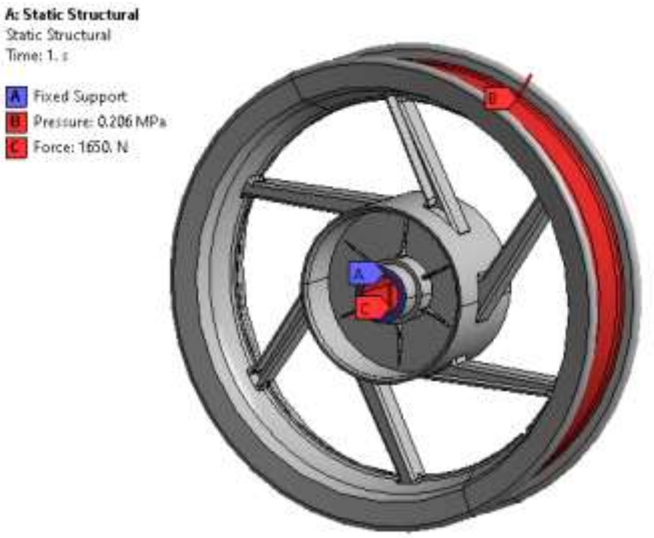


Fig. 5 boundary condition of 2-Wheeler Alloy Wheel

Total Deformation

The total deformation & directional deformation are general terms in finite element methods irrespective of software being used. Directional deformation can be put as the displacement of the system in a particular axis or user defined direction. Total deformation is the vector sum all directional displacements of the systems.

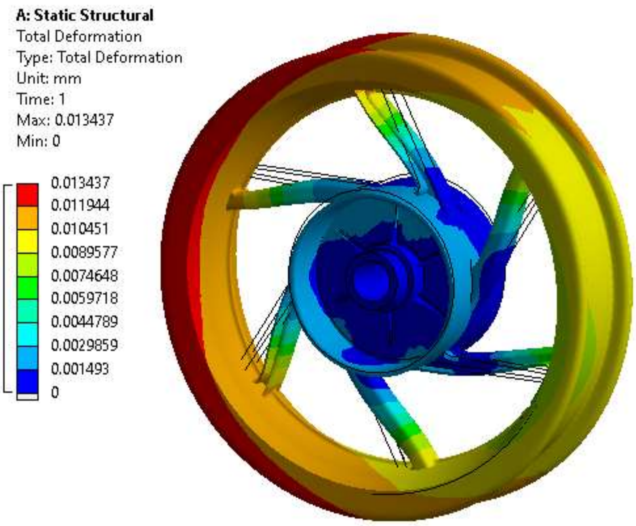


Fig. 6. Deformation results for existing material for 2 wheeler alloy wheel

Maximum deformation of 0.0134 mm is observed with existing material on application of boundary condition as per mentioned in above section.

EQUIVALENT STRESS RESULTS

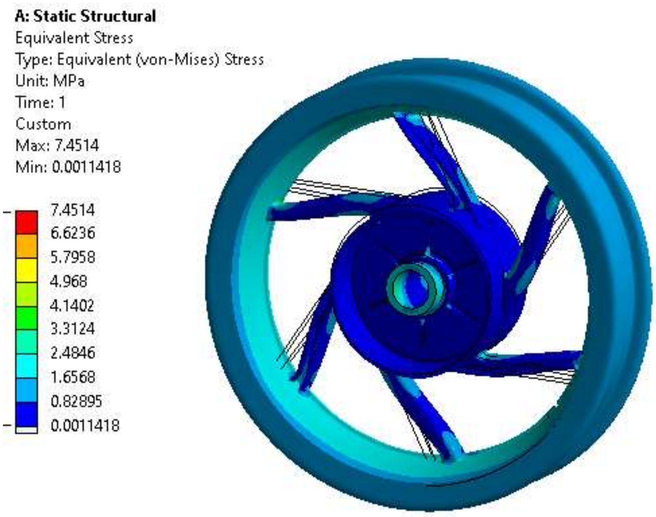


Fig. 7: Von misses stress results for existing material for 2 wheeler alloy wheel

Maximum equivalent stress is observed around 7.45 MPa with existing material on application of boundary condition as per mentioned in above section. It is observed that design is safe as stress are not beyond yield strength of material that is for aluminium alloy yield strength is around 280 MPa and ultimate strength is 310 MPa.

TOPOLOGY OPTIMIZTION TECHNIQUE

The work in this paper makes use of the grid perturbation approach.

Topology optimization is being used more frequently in recent studies to find preliminary, and sometimes completely innovative, structural configurations that meet specific conditions (i.e., objective function and constraints).

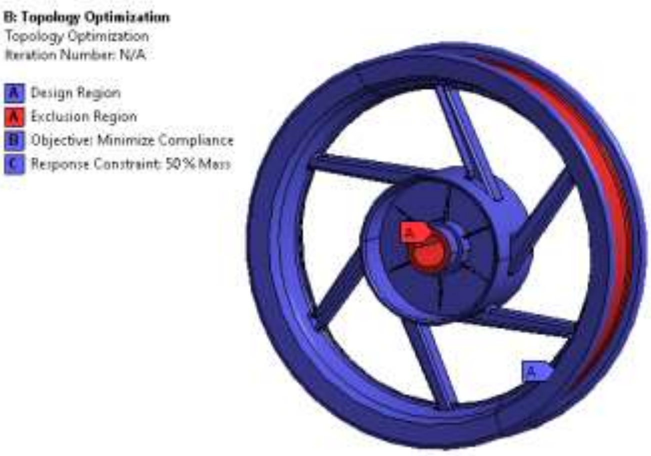
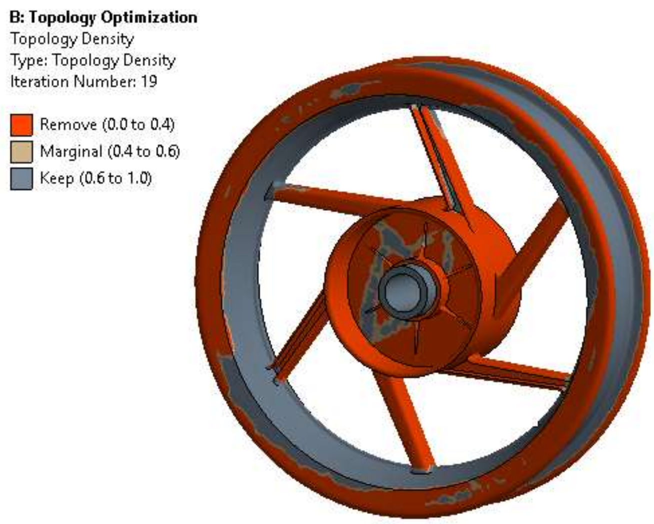


Fig. 8: Boundary condition of topology optimization of 2 wheeler alloy wheel

In ANSYS using topology optimization module it is linked with static structural analysis to obtain boundary condition and result for optimization. Initially blue region indicates as design space along with red region as exclusion area which consider boundary condition namely pressure, load and fixed support.



- Red region indicates material removal area.

Fig. 9: Optimized model of existing 2 wheeler alloy wheel

After performing topology optimization red region colour indicates material removal area along with marginal and keep region to avoid material removal from that area. Material is removed with proper shape and defined dimension geometry from red region and reanalysis is performed on optimized design to check stress intensity on component.

MATERIAL REMOVAL PROCESS

Material is removed by drilling the holes by particular diameters.

OPTIMIZED DESIGN

Fig. 10. Optimized design of wheel

In optimized design holes of diameter 10mm, 8mm and 6 mm along with 22 mm centre holes are drilled in topology optimized area.

Details of "Body Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	8.0 mm

Statistics	
Nodes	122314
Elements	71340
Mass	6.554 kg

Fig. 11. Details of meshing and initial mass

Second stage of FEA analysis is discretization that is breaking of component or model in small parts for simulation or analysis. In this process tetrahedral mesh is preferred with mesh size as 8 mm with nodes and element count as 121314 and 71340. In meshing solution are solved at nodes so mesh size is important factor for stress and strain determination.

Fig. 12. Boundary condition

In 2-wheeler alloy wheel outer hub rim is fixed as it is stationary. Load on inner circumference is applied as per above defined and pressure of tyre are applied on rim. Fixed support is applied at edge face support. Pressure of tyre nearly 30 psi is applied in the form of 0.206 MPa. Weight of bike is considered around 110 kg so, on each wheel weight acts is 55 kg in which 30 % is absorbed by suspension so 30% of weight is applied 1650 N. After application of boundary condition FEA analysis is solved to determine stress and deformation.

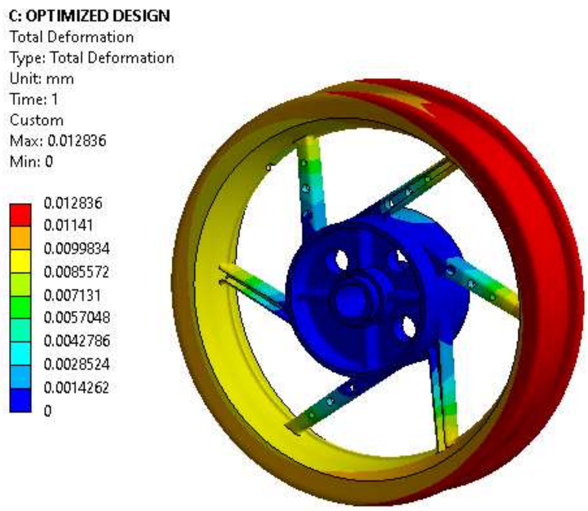


Fig. 13. Total deformation results

Maximum deformation of 0.012 mm is observed with existing material on application of boundary condition as per mentioned in above section.

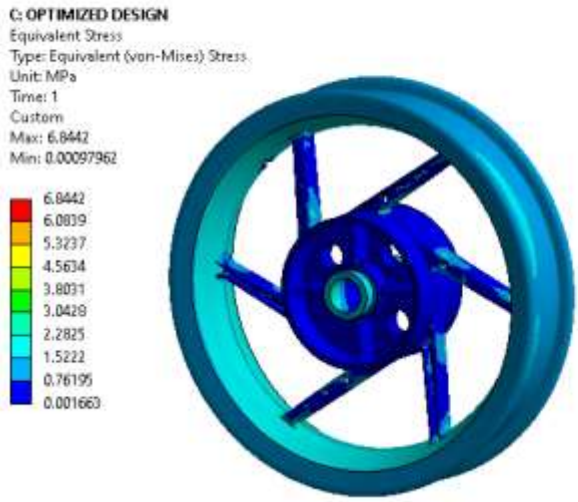


Fig. 14. Equivalent stress results

Maximum equivalent stress is observed around 6.84 MPa with existing material on application of boundary condition as per mentioned in above section. It is observed that design is safe as stress are not beyond yield strength of material that is for aluminium alloy yield strength is around 280 MPa and ultimate strength is 310 MPa.

EXPERIMENTAL TESTING FEA

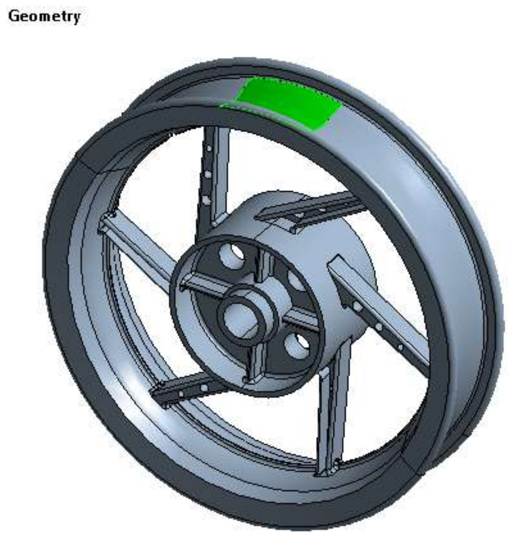


Fig.15. Optimized geometry imported in ANSYS

FORCE – PRESSURE X AREA

FORCE – 0.201MPa x 5107. 5 mm2

FORCE – 1052 N

During experimental testing tyre pressure of 30 psi on specified patch is applied in the form of force by multiplying pressure by area to determine force.

Then this force is applied in ANSYS to determine strain and after determination of maximum strain area strain gauge is attached to that location for validation of experimental is performed with application of 1052 N force by upper arm of UTM and recording strain using DWESOFT software.

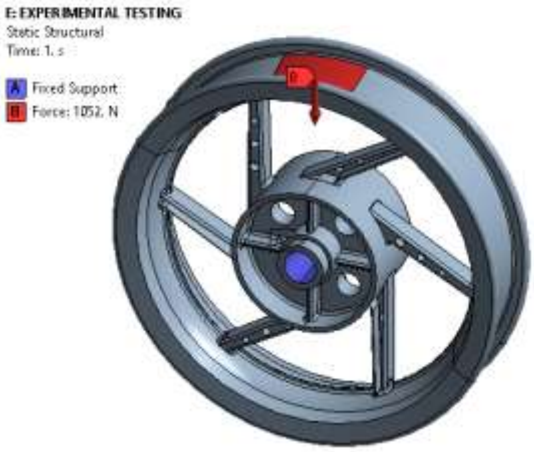


Fig. 16. Boundary conditions

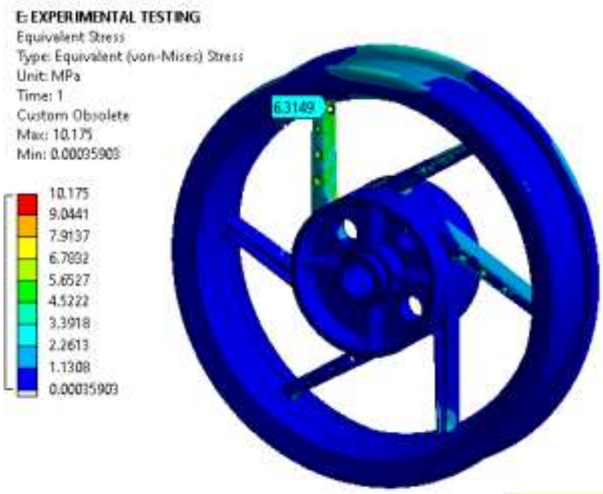


Fig. 17. Equivalent stress result

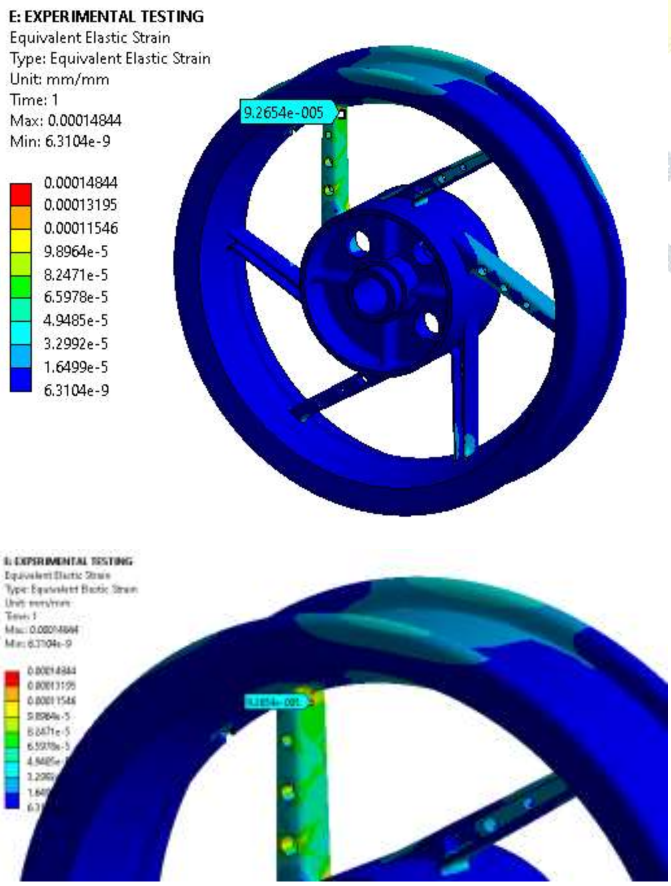


Fig. 18. Equivalent strain result

- Strain is observed around 92.65 microns using FEA.

Experimental procedure

Fixture is manufactured according to component designed. Considering the wheel innner diameter (hub) 50mm, Sodil shaft is used 50mm diameter and 200mm long from Market. After turning it becomes 49.5mm. On C channel of diamension70x40x5mm having 400mm long welded by arc wedling the two L channels. The diamension of the these L channel is 40x40x5mm and 300 mm long each.

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 `
the strain obtained by numerical and experimental results. During strain gage experiment two wires connected to strain gage is connected to micro controller through the data acquisition system and DAQ is connected to laptop. Strain gage value are displayed on laptop using DEWESOFT software. Wheel is mounted on this fixture on some height on UTM fot testing.

Connected to laptop. Strain gage value are displayed on laptop using DEWESOFT software.

Conclusion

- In the optimization of wheel rim, the wheel structure and its features are divided into two parts, namely design space and non-design space. The non-design space is the standard design and cannot be modified.

- The design space is the region for optimizing the weight and shape of the arms. The wheel design space is optimized in order to withstand the existing load of the vehicle with the factor of safety with a least quantity of material and manufacturing cost and losses.
- Numerical analysis of existing 2-wheeler alloy wheel is performed with conventional material with the help of ANSYS software.
- Optimized design of 2-wheeler alloy wheel is obtained from topology optimization technique.
- The weight of rim is optimized from 7.227 Kg to 6.554 Kg using topology method. The shape of the arm's cross section is made easier to manufacture and to distribute the stress induced in the rim.
- The optimized design is analysed to withstand all the loading conditions acting upon it, such as: Vertical load used is 1650 N to 1052 N and the Maximum equivalent stress is observed around 7.45 MPa to 6.84 MPa, which is less than the yield stress of the material suggested i,e., 280MPa
- Weight reduction of around 9.2 % is observed along with strain measurement of 92.65 microns and 92 microns by numerical and experimental testing respectively.

References:

1. Madhu K Saet, Ravi Prakash Mb, & Somashekar H Kc ”*Structural Analysis and Optimization of Two Wheeler Alloy Wheel by Using FEA Approach”*

2. Sandeep Kumar, T. K. Kundra and S. V. Modakre *”Model Updating Studies on a Two Wheeler Chassis”,*

3. Poonam Meshram , Ashish Sinha *”design & optimization of alloy wheel rim using ansys”,*

4. M. Saran Theja, M. Vamsi Krishna *”Structural and Fatigue Analysis of Two Wheeler Lighter Weight Alloy Wheel”,*

5. 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.

6. X Jiang, R Lyu1, Y Fukushima, M Otake and D Y Ju *”Lightweight design and analysis of automobile wheel based on bending and radial loads”,*

7. Daniel Antony C, Prince Jerome Christopher J *”Design and Analysis of Two Wheeler Alloy Wheel Rim Using Composite Materials”*

8. J.L.Miren Kisshan, K.Sankara Narayanan, Ben Mathew Augustine, Dr. D.Vijayaganapathy *”design and analysis of wheel rim”,*

9. R.Masilamani, N.V.Dhandapani, R.Ratheesh Kumar, Nived Devadas, M.Elangovan *Design Modification And Optimization of Vehicle Rim using Finite Element Analysis – A Pot Hole Approach”,*

10. S. Chaitanya, and B.V.RamanaMurty, *“Mass Optimization of Automobile Wheel Rim”* International Journal of Engineering Trends and Technology (IJETT) – Volume 26 Number 3- August 2015.

11. Ch. P. V. Ravi Kumar, and R. Satya Meher, *“Topology Optimization of Aluminium Alloy Wheel”*, International Journal of Modern Engineering Research (IJMER), Vol. 3, Issue. 3, May.-June. 2013 pp-1548-1553 ISSN: 2249-6645

12. TurakaVenkateswara Rao, and KandulaDeepthi, *“Design & Optimization of a Rim Using Finite Element Analysis”,* International Journal of Computational Engineering Research (IJCER), ISSN (e): 2250 – 3005, Vol, 04, Issue, 10, October – 2014.

13. Mr. Sushant K. Bawne, and Prof. Y. L. Yenarkar, “*Optimization Of Car Rim”*, International Journal of Engineering Research and Applications, ISSN: 2248-9622, Vol. 5, Issue 10, (Part - 2) October 2015.

14. BGN Satya prasad, M Anil kumar, *“Topology Optimization of Alloy Wheel”*, Altair Technology Conference, India-2013.

15. D. H. Burande, and T. N. Kazi, *“Fatigue Analysis of Alloy Wheel for Passenger Car under Radial Load”*, International Journal of Engineering Research and General Science Volume 4, Issue 2, March-April, 2016, ISSN 2091-2730.

16. H. N. Kale, and Dr. C. L. Dhamejani, *“COMPARATIVE STUDY OF WHEEL RIM MATERIALS”*, IJARIIE-ISSN(O)-2395-4396, Vol-1 Issue-5 2015.

1. Hypermesh Help Desk for Topology Optimization.

2. Dassault Systems user manual.