TOPOLOGY OPTIMIZATION OF TWO WHEELER WHEEL RIM

Mogali Gangdhar Teja[1], G M Prasada Rao[2]

[1]Student,Department of Mechanical Engineering, Nova College of Engineering & Technology, Jupudi, Ibrahimpatnam 521456, Andhra Pradesh, India
[2]Associate Professor, Department of Mechanical Engineering, Nova College of Engineering & Technology, Jupudi, Ibrahimpatnam 521456, Andhra Pradesh, India

---***---

Abstract - *This dissertation provides a succinct compilation of research related to analysis of stress and displacement in an aluminum alloy two wheeler wheel rim to reduce the weight of the rim by changing the topology of the rim. Finite element analysis modelling of rim response under the conjoint influence of radial load and inflation pressure is critically examined and discussed. The influence of optimizing opening geometry so as to reduce the deformation in the rim under defined load condition. Detailed and elaborately discussed are: (a) measurement of deformation on the rim under known loading conditions; (b) measurement of the stressed on the rim under the known loading conditions & (c) Optimization to reduce the material consumption hence to reduce the weight of the wheel. The FE analysis shows that the stress generated in the optimized component is well below the actual yield stress of the Al alloy. Design in an important industrial activity which influences the quality of the product. The significance of topology will help insizing and shape optimisation to develop light weight components which are used in manufacturing. In modelling the time spent in producing the complex 3-D models and the risk involved in design and manufacturing process can be easily minimised. So the modelling and analysis of the wheel rim is made by using ANSYS WORKBENCH 12.1.The aim of this work is to develop an optimization procedure in order to reduce the weight of rim component. The focus of this project has been the weight reduction of an aluminium wheel by means of a topology analysis.*

Key Words: Topology, Finite element analysis, ansys work bench, spokes, wheel rim

1. INTRODUCTION

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 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 wheel 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 used with precaution. Anyhow, this method is very useful for comparing different design solutions and, therefore, selecting the rim that should be tested further.

The ever increasing demand for petroleum based fuels and the uncertainty in their availability has been a matter of concern world over. The huge outflow of foreign exchange on one hand and increasing emissions causing environmental hazards on the other, have triggered interest in alternatives to gasoline and diesel. Oil provides energy for 95% of transportation and the demand continues to rise, particularly in rapidly developing countries like India and China. The requirement of gasoline and diesel is expected to be about 13 MMT and 66 MMT by 2011-2012. The domestic supply of crude oil in India will satisfy only about 22% of the demand and the rest will have to be met from imported crude oil. Crude oil prices and availability are subject to great volatility depending upon the international situation and relationships between the countries. Moreover, import of petroleum is a major strain on a country‟s foreign exchange resource. Hence, steps are being taken to reduce dependence on oil imports.

The wheel 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 enables efficient movement of an object across a surface 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.

Fig 1: Wheel rim

1.1 Problem Statement

To optimize the design of wheel rim and perform structural analysis of two wheeler wheel rim using 3D FEA and to verify the design of components in strength and stiffness point of view under various types loads. Initial design is considered and the deformation and stress of initial design is taken as reference. By resizing and shaping the design to obtain an optimal model, which can withstand for the defined load condition. To reduce the deformation in the optimal design and performing the modal analysis. Weight reduction in the optimal design is compared with the reference modal.

2. MODELLING AND LOADS CALCULATIONS

ANSYS WORKBENCH is mechanical design software, addressing advanced process centric design requirements of the mechanical industry. We have modelled desired model in the Geometry module. We used the Geometry cell to create the model file with require dimensions. New Geometry cell launches the Design modeler where we can build a new design. In the geometry module the model are generated with taking reference with base model. ANSYS WORKBENCH design solutions provide tools to help you implement sophisticated standards based architecture. This enables collaborative design and offers digital mock-ups and hybrid designs. The part design workbench is a parametric and feature based environment and in which the solid objects are designed. The part design workbench is a parametric and feature based environment and in which the solid objects are designed. The sketcher is the basic required tool for this workbench. In the sketcher only the 2d sketches for the given solid object was done and it will be transformed into 3d by using sketch based feature tool in the part design. The modelling in performed in three geometry cells.

3 Models are generated with reference with base model, where the weight reduction is taken as the key parameter to optimize the structure of the models. The hub and wheel rim are not change in shape and size aspect as they need to be assemble with other part of the motor cycle and rim is not changed as the tyre specification are standard. Our key focus was kept on the spokes where the spoke are alter with their shape and size and mass distribution.

Fig 2: ISOMETRIC VIEW OF WHEEL RIM MODEL 1(curved spokes)

Fig 3: ISOMETRIC VIEW OF WHEEL RIM MODEL 2(inclined spokes)

Fig 4: ISOMETRIC VIEW OF WHEEL RIM MODEL 3(adjacent spokes)

2.1: LOAD CALCULATION:

To ensure the reliability of the analysis in ANSYS WORKBENCH, the total weight of the motorcycle including weight of two persons was applied to the rear wheel alone. Considering, the maximum applicable load is equal to weight of motorbike and weight of two persons. Tyres used in the analysis are of common type with inner tube filled with air pressure and uniformly distributed on the exterior ring surface of wheel.

Fig 5: Boundary Condition with Pressure

Here the constraint is given at the center of the HUB as the wheel is fixed at the hub, we input this as single point constraint. The pressure on the wheel is applied on the wheel rim surface.

Considering load is distributed uniformly over the rim surface. The maximum applicable loads are weight of two wheeler and maximum person sitting on the vehicle (comfortably two persons). Assuming load is distributed to only one wheel. The following calculations are used for analysis

Dead weight of vehicle = 200 Kg
Extra loads= 20 kg
Consider the average weight of person= 80 Kg
Total Weight in Kg's= 200+40+2*80=400 Kg
Total weight in Newton's = 400*9. 81=3965.3 N
Assuming 30% of total weight reduced by the suspension system
Therefore, Net weight=WNet= 3965.3*0.7=2775.7 N
Hence Reaction forces are =Nr=2775.7N
Considering net weight acting on only one wheel rim

Rim surface area $= Circumference * d$
Where d is the width of the rim

Rim surface area $= (461.71*\pi)*58.7$
=85144 mm2
Rim surface are is calculate with r

Pressure on rim $= \frac{FORCE}{AREA}$

Here the force is the total weight acting on the rim surface and area is the rim surface area where the pressure load is acting
Pressure on rim =2775.7/85144
= 0.0326 N/mm2
=0. 0326MPa
The total pressure on the rim is applied on the 5 faces of the rim which is show in the fig 3.

3. ANALYSIS OF WHEEL RIM:

3.1: INTRODUCTION TO FEM:

The finite element method is numerical analysis technique for obtaining approximate solutions to a wide variety of engineering problems. Because of its diversity and flexibility as an analysis tool, it is receiving much attention in engineering schools and industries. In more and more engineering situations today, we find that it is necessary to obtain approximate solutions to problems rather than exact closed form solution.

It is not possible to obtain analytical mathematical solutions for many engineering problems. An analytical solutions is a mathematical expression that gives the values of the desired unknown quantity at any location in the body, as consequence it is valid for infinite number of location in the body. For problems involving complex material properties and boundary conditions, the engineer resorts to numerical methods that provides approximate, but acceptable solutions.

The fundamental areas that have to be learned for working capability of finite element method include:

- Matrix algebra.
- Solid mechanics.
- Variation methods.
- Computer skills.

Matrix techniques are definitely most efficient and systematic way to handle algebra of finite element method. Basically matrix algebra provides a scheme by which a large number of equations can be stored and manipulated. Since vast majority of literature on the finite element method treats problems in structural and continuum mechanics, including soil and rock mechanics, the knowledge of these fields became necessary. It is useful to consider the finite element procedure basically as a Variation approach. This conception has contributed significantly to the convenience of formulating the method and to its generality.

BASIC APPROACH TO FEA SOFTWARE:

Basic approach for any finite element analysis (FEA) can be divided in to three parts

- Pre-processors
- Solver
- Post – Processor

3.2: INTRODUCTION TO ANSYS WORKBENCH:

In our analysis we have used ANSYS WORKBENCH 12.1. ANSYS is a finite element modeling software for analyzing ample collection of mechanical problems which comprise static/dynamic structural analysis (linear and nonlinear), thermal and fluid problems and also acoustic and electromagnetic problems.

It is a numerical technique for solving complex problems by discretizing the complex structure into a number of small pieces called "Elements". In ANSYS the equations that govern the behavior of these elements can be solved by creating the comprehensive explanation which can then be represented in tabular or graphical forms. Design and optimization of system that is too complex to solve can be analyzed. Complexity of structures due to their geometries, scale or governing equations can be taken in to consideration.

3.3: ANALYSIS OF WHEEL RIM:

In ANSYS WORKBENCH we do the meshing of the geometry model. This model are assigned with the material properties, meshing type, type of load acting on the model, constrain applied on the model for restricting the degrees of freedom and the type of analysis performed on the model.

3.3.1: Material Used for the Structural Analysis:

Aluminum Alloy-201.0-T43 material is used for our model, which is presently using to manufacture the alloy wheel of two wheeler. The aluminum is widely used material in the automobile industry

S.l No	Parameters	Al Alloy-201.0-T43
1	Ultimate tensile strength (Mpa)	273
2	Yield strength (Mpa)	225
3	Young's modulus(Mpa)	71000
4	Poisson's ratio	0.33
5	Density (Kg/m3)	2770

Table 1: Material Properties

3.3.2: Meshing for structural analysis:

Meshing is an integral part of the computeraided engineering simulation process. The mesh influences the accuracy, convergence and speed of the solution. Furthermore, the time it takes to create and mesh a model is often a significant portion of the time it takes to get results from a CAE solution. Therefore, the better and more automated the meshing tools, the better the solution. Powerful automation capabilities ease the initial meshing of a new geometry by keying off physics preferences and using smart defaults so a mesh can be obtained upon first try. Rim is was modeled as a solid, solid element named SOLID187 was used to mesh the model. Fine meshing of the model should be done for the better results. SOLID187 element is a higher order 3-D, 10-node element. Solid187 element is used for the meshing in our analysis. SOLID187 has a quadratic displacement behavior and is well suited to modeling irregular meshes. The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

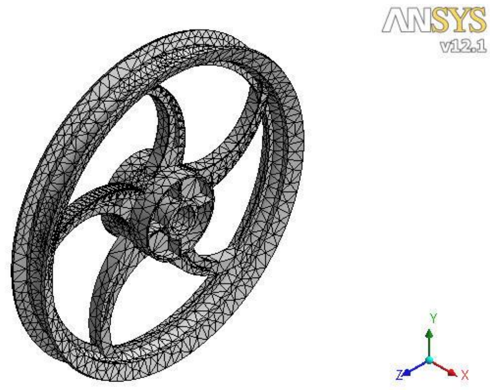

Fig 6: Meshed model 1

PARAMETERS	MODEL 1 (CURVED SPOKES)
VOLUME	1.2549e+006 mm^3
NODES	29884
ELEMENTS	15415
ELEMENT TYPE	SOLID 187

Table 2: Model 1 Parameters

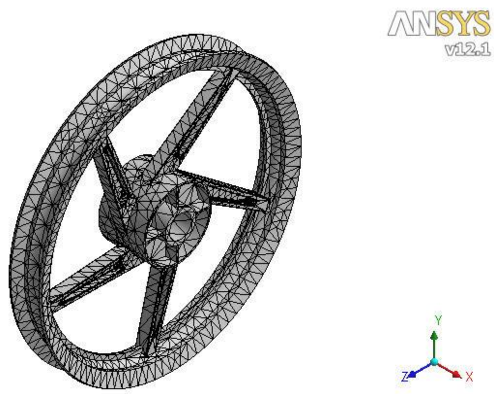

Fig 7: Meshed model 2

PARAMETERS	MODEL 2(INCLINED SPOKES):
VOLUME	1.1885e+006 mm^3
NODES	32695
ELEMENTS	18253
ELEMENT TYPE	SOLID 187

Table 3: Model 2 Parameters

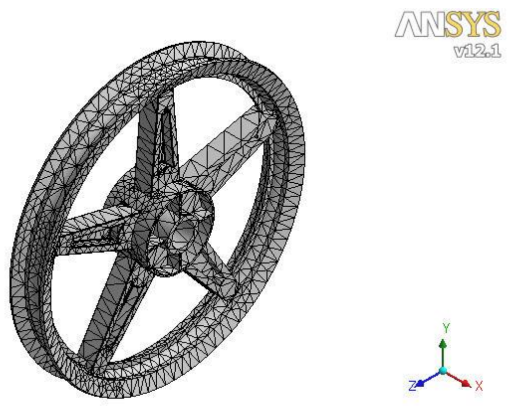

Fig 8: Meshed model 3

PARAMETERS	MODEL 3 (ADJACENT SPOKES):
VOLUME	1.7758e+006 mm^3
NODES	24861
ELEMENTS	13236
ELEMENT TYPE	SOLID 187

Table 4: Model 3 Parameters

4: RESULTS AND DISCUSSIONS:

All the new designs are carried out and total 6 modes are simulated with each design. This results are all tabulated for comparison. The total deformation and VON MISES stress are tabulated

	TOTAL DEF (mm)	VON MIS (MPa)
MODEL 1 (CURVED SPOKES)	5.88E-03	2.3417
MODEL 2(INCLINED SPOKES)	6.72E-03	1.8803
MODEL 3 (ADJACENT SPOKES)	5.14E-03	2.0206

Table 5: Stress and deformation comparison

4.1.1 Weight calculation of designs

MODEL 1 (CURVED SPOKES)

Volume of design = 1254900mm^3=1254900*e^-9
Density of material = 2770 Kg/m3
Mass = volume*density
= (1254900*e^-9)*(2770)/1000
=3.47 kg

MODEL 2 (INCLINED SPOKES)

Volume of design = 1188500mm^3= 1188500*e^-9
Density of material = 2770 Kg/m3
Mass = volume*density
= (1188500*e^-9)*(2770)/1000
=3.29 kg

MODEL 3 (ADJACENT SPOKES)

Volume of design = 1775800mm^3= 1775800*e^-9
Density of material = 2770 Kg/m3
Mass = volume*density
= (1775800*e^-9)*(2770)/1000
=4.92 kg

The weight of the base reference design is 4.42 kgs MODEL 1 (CURVED SPOKES) and MODEL 2 (INCLINED SPOKES) wheel rim are have less weight compared base reference design. The MODEL 3 (ADJACENT SPOKES) wheel rim is more than the reference design.

MODAL ANALYSIS

OPTIMIZED DESIGN

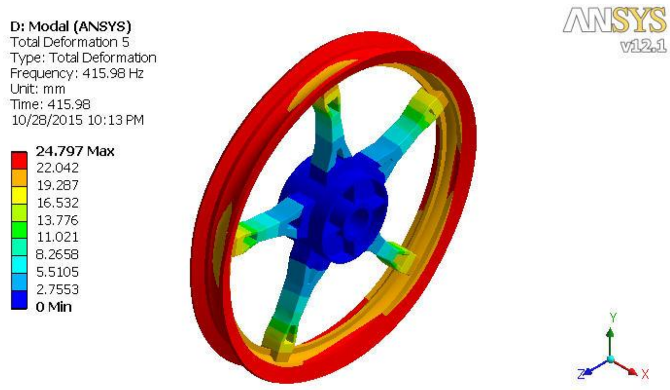

Fig 9: MODE 1 (OPTIMIZED DESIGN)

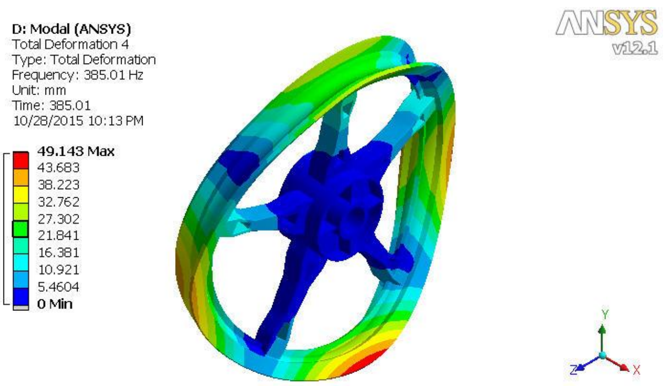

Fig10: MODE 2 (OPTIMIZED DESIGN)

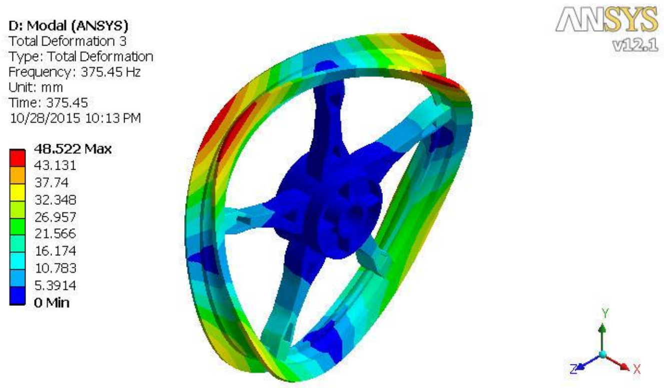

Fig 11: MODE 3 (OPTIMIZED DESIGN)

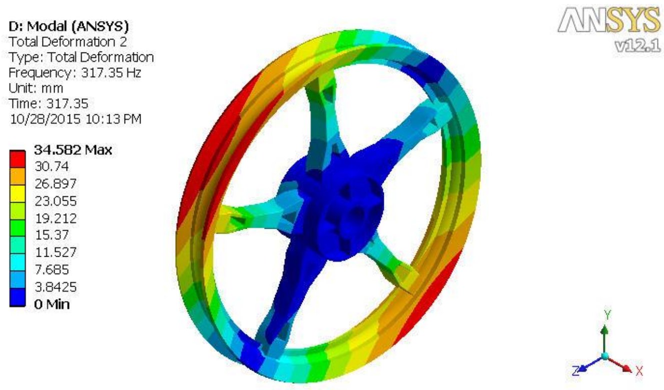

Fig 12: MODE 4 (OPTIMIZED DESIGN)

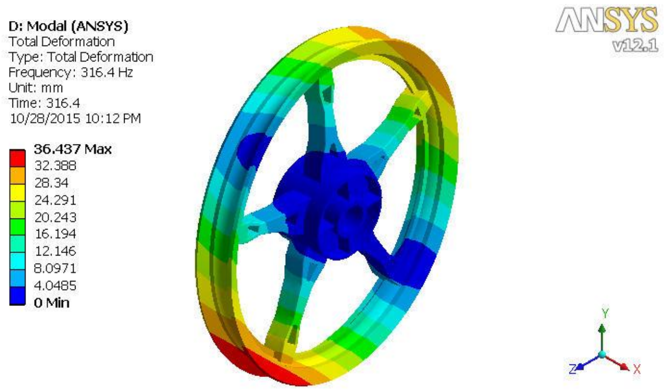

Fig 13: MODE 5 (OPTIMIZED DESIGN)

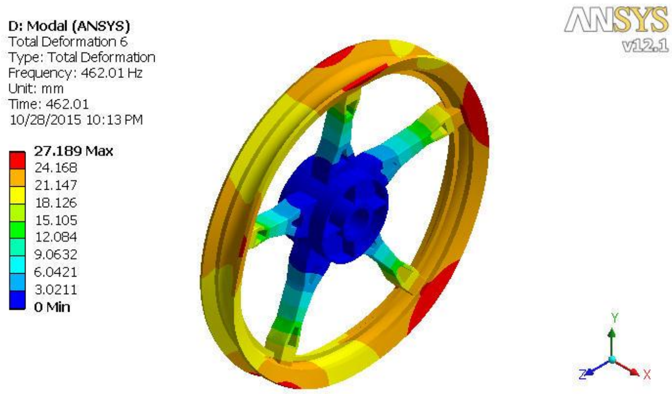

Fig 14: MODE 6 (OPTIMIZED DESIGN)

Weight calculation

OPTIMIZED MODEL (CURVED SPOKES)

Volume of design = 1395500mm^3=1395500*e^-9
Density of material = 2770 Kg/m3
Mass = volume*density
= (1395500*e^-9)*(2770)/1000
=3.86kgs

The weight of optimized mode is 3.86 whereas the reference base model weight is 4.42kg.For the newly designed model of the wheel rim the modal analysis carried out, a total number of six normal modes are simulated along with the base model. The simulation results are tabulated below.

	MODEL 1	MODEL 2	MODEL 3	OPTIMIZED MODEL
MODE 1	209.9	195.83	303.15	316.4
MODE 2	211.06	197.24	304.52	317.35
MODE 3	272.91	265.58	383.65	375.45
MODE 4	279.11	267.82	395.25	385.01
MODE 5	285.83	272.74	446.86	415.98
MODE 6	407.12	364.7	476.6	462.01

Table 6: Modal analysis results

Base on the static structural analysis and calculation, the following data is obtained and calculated.

MODELS	DEFLECTION (mm)	STRESS (MPa)	WEIGHT (Kg)	FS	% REDUCTION IN WT
REFERENCE DESIGN	4.91E-03	1.569	4.42	143.4	NA
MODEL 1 (CURVED SPOKES)	5.88E-03	2.3417	3.47	96.8	21.4

MODEL 2(INCLINED SPOKES)	6.72E-03	1.8803	3.29	119.6	25.5
MODEL 3 (ADJACENT SPOKES)	5.14E-03	2.0206	4.92	111.3	-11.3
OPTIMIZED MODEL	4.55E-03	1.4762	3.86	153	12.6

Table 7: Structural analysis results

4.1.2 Stiffness calculations:

The stiffness, *k*, of a body is a measure of the resistance offered by an elastic body to deformation.

$$K = \frac{F}{\delta}$$

Where *F* is the force applied on the body

δ Is the displacement produced by the force along the same degrees of freedom.

The stiffness of the reference model is 900.38 kg/mm

MODEL 1 (CURVED SPOKES)

Stiffness $K = \frac{F}{\delta}$

Force =3.47 kg

Deflection =0.00588

Stiffness,,k =3.47/.00588

=590.13 kg/mm

MODEL 2 (INCLINED SPOKES)

Stiffness $K = \frac{F}{\delta}$

Force =3.29 kg
Deflection =0.00672
Stiffness,,k =3.29/.00672

=489.58 kg/mm

MODEL 3 (ADJACENT SPOKES)

Stiffness $K = \frac{F}{\delta}$

Force =4.92 kg
Deflection =0.00514
Stiffness,,k =4.92/.00514

=957.19 kg/mm

MODEL 4 (OPTIMIZED MODEL)

Stiffness $K = \frac{F}{\delta}$

Force =3.86 kg
Deflection =0.00455
Stiffness,K =3.86/.00455

=848.35 kg/mm

Comparing with the reference base design the stiffness for MODEL 3 (ADJACENT SPOKES) is more as the weight of the model is also more.

5: CONCLUSIONS AND FUTURE SCOPE OF WORK:

Modelling and analysis of the two wheeler wheel is carried out by taking the reference base model to optimize the design. Deformation and stresses are measure for all the models. Modal analysis is performed for the 6 natural frequencies. The weight of the optimized wheel rim is compared with reference base model. Optimized design is achieved with less weight and less stress.

Based on the analysis results the stress of initial reference modal is 1.59 MPa and the final optimized model has maximum stress 1.4762. Which is much better than the reference model. Initial weight of base model is 4.42kg and Final weight obtained for the optimized model is 3.86 kg. Therefore % of weight reduction is 12.6%.

The future scope of work will be focus on the material parametric analysis of the model as emphasis of automobile industries are on the cost saving by reducing the weight of the model or by using the other material with same properties and less cost.

6. REFERENCES:

[1] S. Ganesh, Dr.P.Periyasamy "Design and Analysis of Spiral Wheel Rim for Four Wheeler". A thesis submits to The International Journal of Engineering and Science (IJES).

[2] P. Meghashyam, S. Girivardhan Naidu, N. Sayed Baba "Design and Analysis of Wheel Rim using CATIA & ANSYS". A thesis submits to International Journal of Application or Innovation in Engineering & Management (IJAIEM)

[3] N. Satyanarayana&Ch.Sambaiah "Fatigue Analysis of Aluminum Alloy Wheel under Radial Load". A thesis submits to International Journal of Mechanical and Industrial Engineering (IJMIE)

[4] Saran theja. M, Shankar. G, Vamsi Krishna. M, "Design analysis of two wheeler lighter weight alloy wheel". A thesis submits to Indian journal of engineering.

[5] Rajarethinam. P, Periswamy. K "Modification of design and analysis of motor cycle wheelspokes" A thesis submits to "International OPEN ACCESS Journal of Modern Engineering Research (IJMER)."

[6] Sourav Das, Design and Weight Optimization of Aluminum Alloy Wheel". A thesis submitted to International Journal of Scientific and Research Publications

[7] BGN Satya Prasad, M Anil kumar, "Topology Optimization of Alloy Wheel" paper presentation at altair technology conference

[8] Ch. P. V. Ravi Kumar1, Prof. R. SatyaMeher, " Topology Optimization of aluminum Alloy Wheel" paper presented to International Journal of Modern Engineering Research (IJMER)

[9] 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 AutomobileWheel" A thesis submits to International Journal of Rotating Machinery

ACKNOWLEDGEMENT:

I take this opportunity to express my sincere gratitude to my guide Mr. G M PRASADA RAO for his valuable patience, encouragement, guidance and support throughout this work The work presented in this dissertation was conducted in the Department of Mechanical Engineering. My gratitude is also addressed to Mr. R S DHEERAJ, Head of Department and to the technical staff of the Department. Finally, I extend my thanks to all who have been a great help in my project work, directly or indirectly.

BIOGRAPHIES

Mogali Gangdhar Teja, student, Department of Mechanical Engineering, Nova College of Engineering & Technology