



e-ISSN: 2319-8753 | p-ISSN: 2347-6710

IJIRSET

International Journal of Innovative Research in
SCIENCE | ENGINEERING | TECHNOLOGY



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

Volume 12, Issue 6, June 2023

ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA

Impact Factor: 8.423

9940 572 462

6381 907 438

ijirset@gmail.com

www.ijirset.com

Design and Analysis of Car Alloy Wheel Using Finite Element Analysis (Static and Fatigue)

Chinni Hemanth¹, Yalamanchili Suresh²

P.G. Student, Department of Mechanical Engineering, Gonna Institute of Information Technology & Sciences, Visakhapatnam, India¹

Assistant Professor, Department of Mechanical Engineering, Gonna Institute of Information Technology & Sciences, Visakhapatnam, India²

ABSTRACT: In the present study report we have a subject on automotive four wheeler car wheel which is one of the important components of the car. Basically they manufactured on the combination of alloy metals, such as aluminium and magnesium in large Era. The main concept of this project, Analysis is to prepare the car wheel of magnesium, aluminium alloys. Design of the wheel carried out in cad software (Catia V5 R21) ,simulate the design in ansys 19.2.The analysis over a wheel rim is done with the maximum load applied on the rim. finally to constitute which allow of metal is fine suited for wheels under different loads and stress values. For this analysis strength of material, von-mises stress and various engineering processes are main considerations.

KEYWORDS: Wheel Rim, FEA analysis, CATIA, ANSYS.

I. INTRODUCTION

Many years ago was the beginning of the wheel the people started to travel or move logistics form one place to the other. At the starting or the initial stages the wheel is used to be like round slices of a log and it absolutely was step by step re-in forced and were employed in this manner for hundreds of years on both carts and wagons vehicles. Later on under several experiments the solid disc is modified to a style having a spokes. The main purpose of the spokes is to support the outer part of the wheel and to keep the wheel equal from the centre of the wheel. Almost in 1920 mostly steel truck rims are created within the same manner with a solid and tough hub. The rim is balanced correctly so given a sleek end. Alloys wheels are usually available in present market. Whereas coming to truck wheels made of several spokes along the wheel because of their steady and tough carrying loads. Later on given preference to the casted solid wheel rims which are made of different varying conditions. Lighter wheels will improve handling by reducing sprung mass which constitute the overall reduction of mass of the vehicle which reduces the fuel consumption. Good Heat conductivity of the metal will facilitate the heat from the brakes which results in better braking performance.

1.1 Basic and Common types of Wheel Rim And their Material.

- (1) Steel Disc Wheel
- (2) Wire Spoke Wheel
- (3) Light Alloy Wheel
 - (a) Aluminium Alloy Wheel
 - (b) Magnesium Alloy Wheel
 - (c) Titanium Alloy Wheel
 - (d) Composite Material Wheel

II. ALLOY WHEEL

The wheels / Rims are the one which are made of Commonly Named Alloy metals i.e Magnesium, Aluminium, titanium. Sometimes there may be a composite mixture of both the Alloys which is Combination of Aluminium and Magnesium. The Main Concept of using this alloy wheels which reduce the weight and for better heat conductivity on metal which constitute the better braking performances. Reduction will also reduce the overall fuel consumption of the vehicle.

III. MATERIALS AND MANUFACTURING PROCESSES

The wheel is one of the important component in an automotive Which experiences tremendous amount of heat on the surface of the wheel rim that subjected to bending and torsional loads due to long cyclic stresses. The main usage of Alloy wheels is very light in weight and cost effective and that can be easily casted. Which commonly result in high strength and long life. There was a huge competition among the materials and the manufacturing processes in order to produce the good quality product with a reduced cost according to present situations. Same way to maintain good fatigue, life and other functional requirements.

In The present study report we have considered two alloy wheels that made of Aluminum, Magnesium for the better understanding of various stresses involved and deformation experienced on the rim.

Material type	Young's Modulus	Poisson's Ratio	Density	Bulk Modulus	Shear Modulus
Aluminium	7.1E+10 pa	0.33	2770 kg m ⁻³	6.9608E+10 pa	2.6692E+10 pa
Magnesium	4.5E+10 pa	0.35	1800 kg m ⁻³	5E+10 pa	1.6667E+10 pa

Table 1: shows the material properties of the of aluminium, magnesium

IV. MODELING AND ANALYSIS

The Early Computer Aided Design (CAD) system uses geometry construction and editing techniques to create the part model. This techniques creates the wireframe and surfaces that can be easily edited be geometric editing. They do not have relationship between the geometry. Later on parametric modelling system introduced which can build relationship between geometric entities. Modelling of wheel rim carried out in CATIA V5 R21.

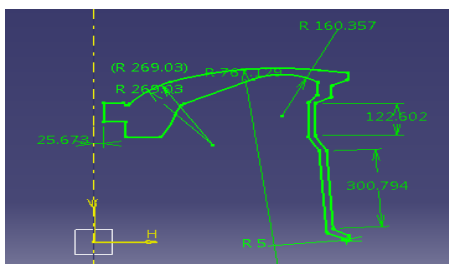


Fig 1. Sketch for the rim

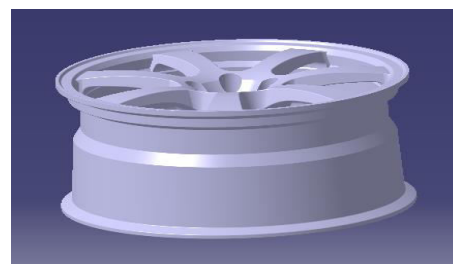


Fig 2. Solid rim after rotation of sketch

The procedure followed in Catia for modelling the rim is as follows: Since CATIA is very user friendly we can create models easily Initially project the sketch by taking the required axis and necessary measurement. Care should be taken while constraining the sketch and then rotate the sketch which converts wireframe to solid body. After modeling the wheel rim save it in CATIA part and IGES which is used for meshing purpose. IGES means Initial Graphic Exchange Specification; this file can be easily imported to any software package. The CATIA part file can also be saved as STEP file.

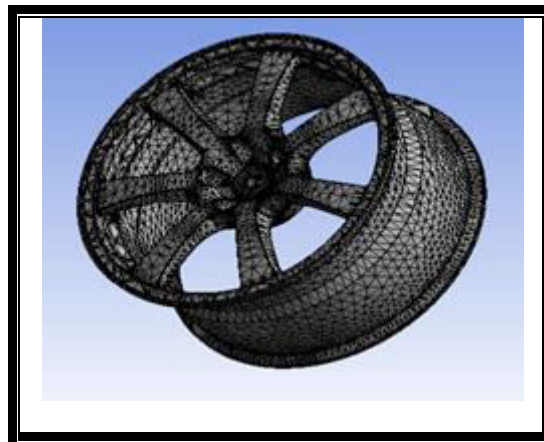
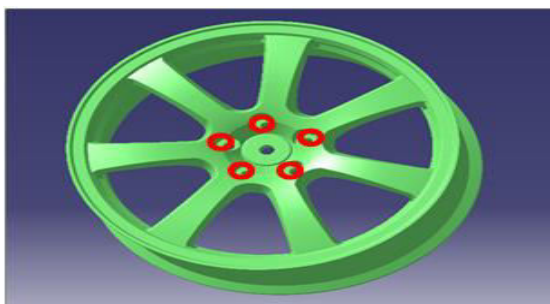


Fig. 3 Meshing of Wheel rim

ANSYS is general-purpose Finite Element Analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user designed size) called elements. The software implements equations that govern the behaviour of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. The ANSYS Workbench environment is an intuitive up-front finite element analysis tool that is used in conjunction with CAD systems and/or Design Model. ANSYS Workbench is a software environment for performing structural, thermal, and electromagnetic analyses. The Workbench focuses on attaching existing geometry, setting up the finite element model, solving, and reviewing results.

V. LOADING AND BOUNDARY CONDITION

In an Automotive vehicle the wheel experiences lot of friction Due to the contact of tyre and wheel when actually the vehicle in motion.



The red highlighted part in the wheel rim has been constrained in all Degrees of freedom. And the arrow represents the remote force acting along the x-direction with a magnitude of 1000N.



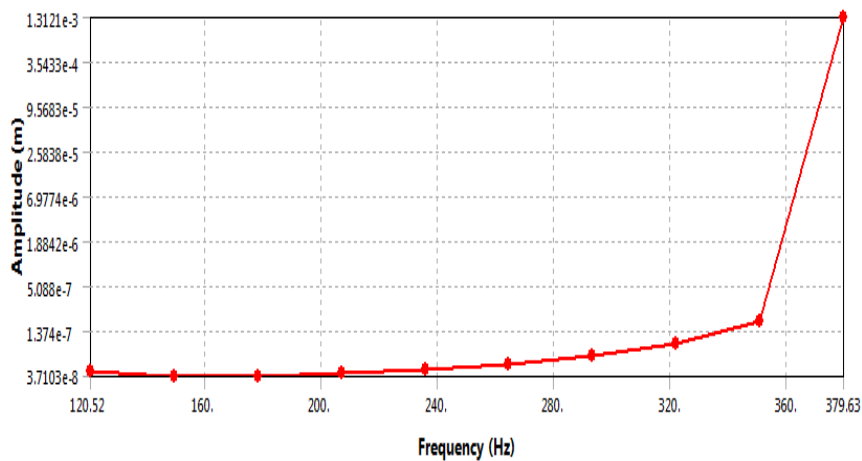
VI. EXPERIMENTAL RESULTS AND DISCUSSION

Comparing the Maximum And minimum natural frequencies of For 10 mode 91.733 Hz, 379.63 Hz Modal analysis design wheel rim are shown in below figure.

	Mode	✓ Frequency [Hz]
1	1.	91.733
2	2.	92.223
3	3.	92.654
4	4.	92.99
5	5.	141.32
6	6.	162.25
7	7.	212.83
8	8.	214.36
9	9.	379.45
10	10.	379.63

Table 2.shows the natural frequencies at mode

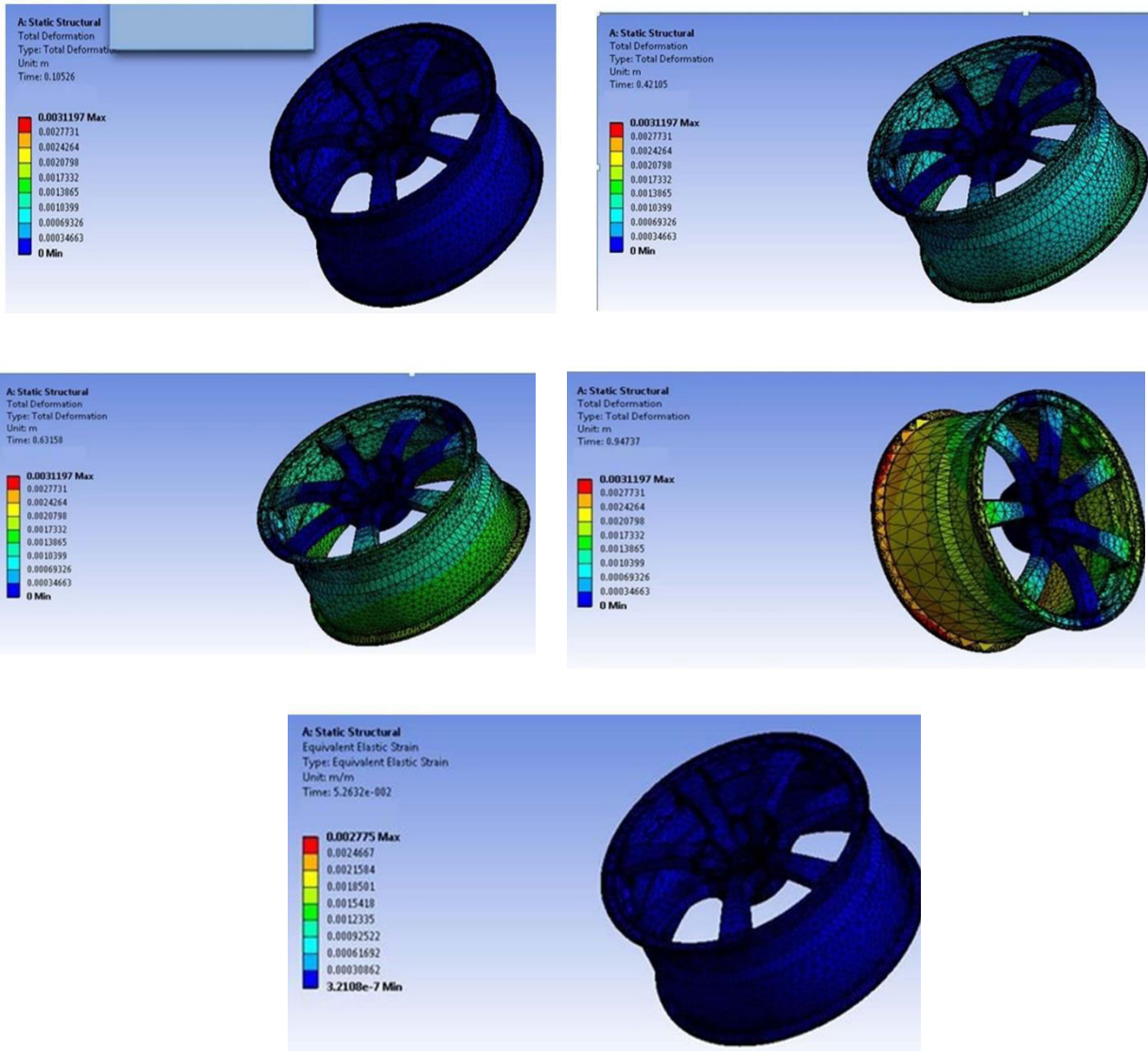
Comparing the natural frequencies for the harmonic analysis



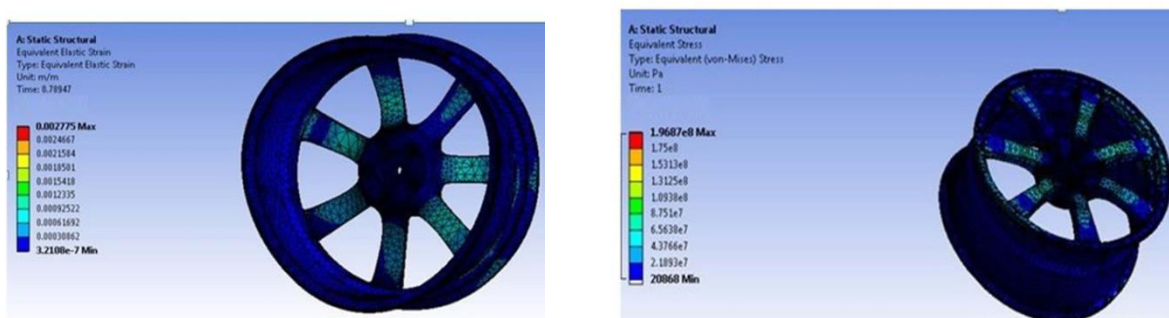
1.Graph represents the curve through Amplitude and frequency HZ

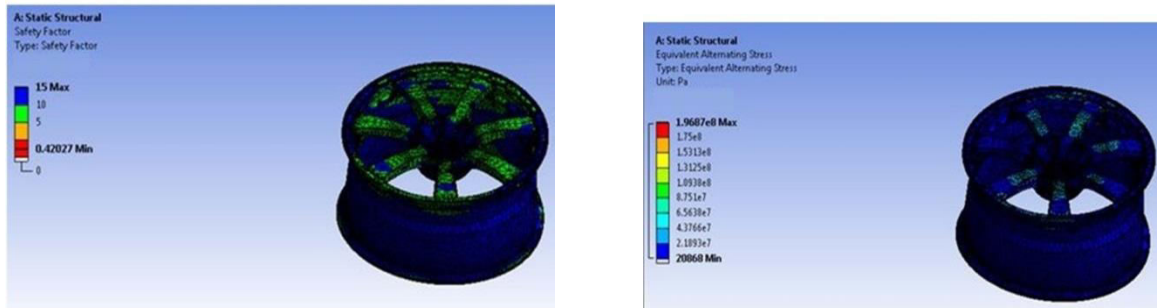
Comparing the Experimental Results Obtained when the wheel rim experiences the remote force of 1000N along X-direction and with pressure of 248000pa with is equal to 0.248Mpa.

Total Deformation, Equivalent elastic strain, Equivalent Stress, Safety factor, Equivalent alternating stress that experiences on wheel rim made of aluminium alloy



Total Deformation, Equivalent elastic strain, Equivalent Stress, Safety factor, Life that experiences on wheel rim made of aluminium alloy as follows





V. CONCLUSION

The maximum deformation can be seen in the wheel rim when analysis carried out using Magnesium Alloy. The maximum stress appears at the top of the rim. The value of von-mises stresses that comes out from the analysis is far less than material yield stress. While Comparing the metal alloys i.e Aluminium, Magnesium Allows fatigue life is more and with a good life, safety factor. It is observed the Aluminium alloys shows the best results compared to magnesium where analysis carried out at 248000pa which is equal to 0.24Mpa and with a remote force of 1000N towards –X direction.

REFERENCES

- [1] K. Mahadevan and Balaveera Reddy, “Design Data Hand Book”.
- [2] “Finite Element Analysis”, Chandra Pautla.
- [3] “Strength Of Materials”, Ramambrutham.
- [4] “Ansys User Manual”,
- [5] “Metal Fatigue” , Ralfh Stefunson, Ali Fatemi & A.O.Cuph.
- [6]“MSC Fatigue User Manual”,
- [7]Metal fatigue in engineering by Stefan.
- [8]Fatigue Life Analysis of Aluminum Wheels by Simulation of Rotary Fatigue Test Liangmo Wang* - Yufa Chen - Chenzhi Wang - Qingzheng Wang School of Mechanical Engineering, Nanjing University of Science & Technology, China
- [9]Fatigue properties of a cast aluminum alloy for rims of car wheels. C.Bosi,G.L.Garagnani



Impact Factor: 8.423



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

 9940 572 462  6381 907 438  ijirset@gmail.com



www.ijirset.com

Scan to save the contact details