International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Design and Analysis of Alloy Wheel Rim Using Finite Element Analysis

[1]Aaksh Patil, [2]Dr. Suman Sharma

[1]PG Scholar, Dept. of Mechanical Engineering, SIRT, Sage University, Indore, India, aakashpatil61@gmail.com

[2] Professor, Dept. of Mechanical Engineering, SIRT, Sage University, Indore, India, drsuman.sharma@sageuniversity.in

ABSTRACT:

In this paper, we are performing Static, Modal, Harmonic, and Fatigue analysis on alloy wheel using different materials Aluminum, Magnesium alloy, and Structural steel. A four-wheeler vehicle wheel is modelled using CATIA software by doing some set of operations and then it is imported into IGES format which can be imported into Ansys software for performing different analysis to choose the best suitable material for the wheel and to find out different factors influencing them. By comparing multiple materials for alloy wheel, we can eventually end up choosing the best and optimal material that can be suitable for the alloy wheel.

Key Words: Alloy wheel, Wheel Rim, CATIA V5, ANSYS, Magnesium alloy, Aluminum alloy.

INTRODUCTION:

Wheels and tyres are obviously crucial components of a car. The vehicle could be pulled if it were without its engine, but without wheels, it would be impossible to move in any direction. Wheels and tyres must support the weight of the vehicle, absorb shock, and cooperate with the steering system. Aluminum and magnesium alloys are often used to create automotive wheel rims, while other metals may also be used. Alloy wheel rims are lighter than standard steel rims, which makes a big difference in handling and speed. Wheel rims for early light-alloy vehicles were built of magnesium alloys. The paper noted that a fatigue lifespan prediction technique of Aluminum alloy wheels was presented to assure their durability at the early design stage, even though they fail approval on conventional automobiles yet stay renowned in the 1960s.

Using ANSYS 19.2 for static structure analysis. The research confirmed what was already known about the location of the highest stress levels, which was across the region covered by the hub bolts. Thus, the finite element model is able to provide findings that are in agreement with the static load test. CATIA V5 is used to create a CAD model of the wheel rim, which is then sent to ANSYS for further analysis. Bolt circles of wheel rims are fastened while a force of 18000 N is applied radially throughout the circumference of wheel rims composed of Aluminum and structural steel.

Figure 1.1: Wheel Rim alloy

LITERATURE SURVEY

P. Meghashyam et al. [1],the rim of an aluminium wheel experiences more stress than one made of forged steel. These pressures are negligible in comparison to the wheel's ultimate strength. In addition, aluminium allows for a greater degree of deflection than forged steel. The forged wheel is the material of choice for the wheel rim in many modern designs.

J. Janardhan et al. [2]to verify the wheel's existence, we looked at the fatigue module. The investigation has spanned a long period of time. Globally, the equivalent (Von-Mises) pressure measured 9.205x1e6 Pa. Deformation is 0.515x1e-1 mm and minimum stress is 0.041x1e6 Pa after undergoing fatigue cycles. It was during the 1.0109 cycle when he found out about eternal life.

Sourav Das et al. [3],In the design space, the arm's mass and shape may be fine-tuned. The damage area is along the flange part of the rim, and it is intended to bear the vehicle's current load for the factor of protection with the least amount of material and manufacturing cost and damages feasible. The fatigue test results prove beyond a reasonable doubt that the material can sustain 1e20 cycles. A maximum of 0.2% was lost during all 1e20 iterations.

Jaspreet Singh et al. [4],The rim's circumference expanded to accommodate the weight. This wheel had a maximum equal tension of 11.283MPa. At 200KN, the design is safe because the equivalent stress is below the yield point of the Al alloy everywhere except for the spokes' corners, where it is greatest. The maximum amount of visible distortion in the alloy wheel was 0.033mm. The diameter of the wheel was the most distorted, while the section of the bolt was the least. Minimum and maximum margins of safety were found at the air pressure hole.

OBJECTIVE

- To compare the strength between different types of wheel rims.
- To make the test of ending endurance and radial test.
- To select best suitable material for wheel rim.
- To give the consumers both safety and comfort of a car with good quality rim.
- To increase the quality and durability rim for consumers.

DESIGN METHODOLOGY

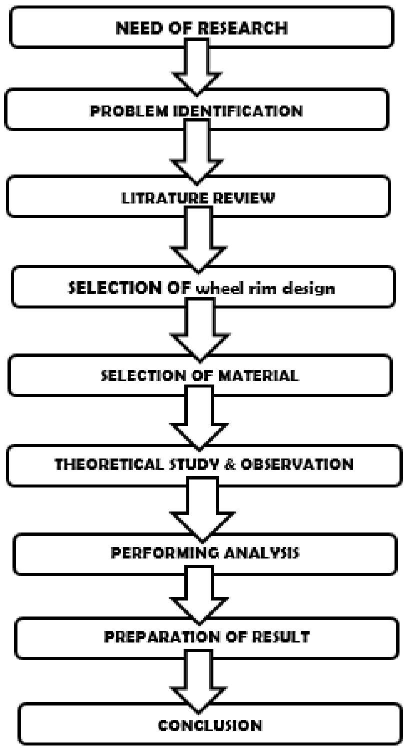
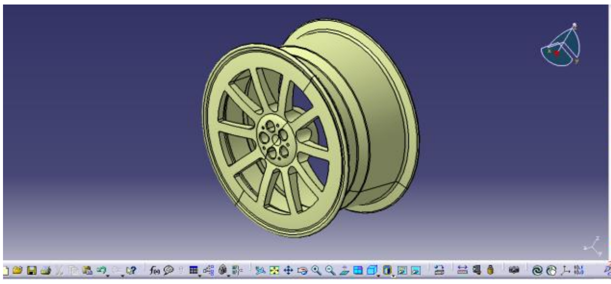


Figure 2: Methodology adopted

DESIGN OF WHEEL RIM

Dassault Systems CATIA (acronym for computer-aided three-dimensional interactive application) is a multi-platform software package for CAD, CAM, CAE, PLM, and 3D modelling built for a variety of platforms. As a result of Dassault's Mirage fighter jet's usage of the CADAM software at the time, the French aviation company AVIONS MARCEL DASSAULT developed CATIA in-house in 1977. It was subsequently adopted by a variety of other sectors, including aircraft, automobiles, and shipbuilding.



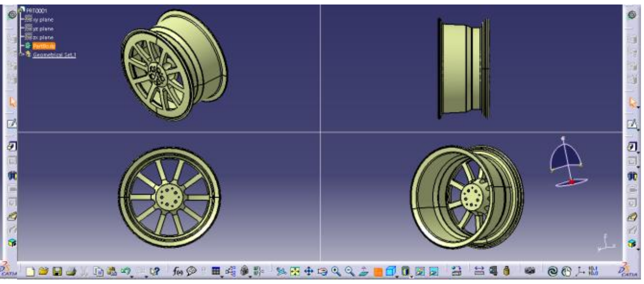


Figure 3: Isometric view of rim **Figure 4:** Multi view of Wheel rim

FINITE ELEMENT ANALYSIS

It is also known as the Finite Element Analysis (FEA). An analytical approach known as the Finite Element Method (FEM) may be used to solve and replace complex issues with simpler ones, resulting in approximate solutions. Because of its versatility, the finite element approach is used across a wide range of sectors to address a wide range of real-world engineering challenges. Relative results may be generated using the finite element approach. An engineering simulation software called ANSYS is available for download (computer aided Engineering). Its techniques include finite element analysis for thermal, static, dynamic, and fatigue, amongst others, all of which are intended to aid in the product's development. Dr. John A. Swanson started Swanson Analysis Systems, Inc. SASI in 1970. Primary goals were the creation and distribution of structural physics finite element analysis software capable of simulating static, dynamic, and heat transport (thermal) issues.

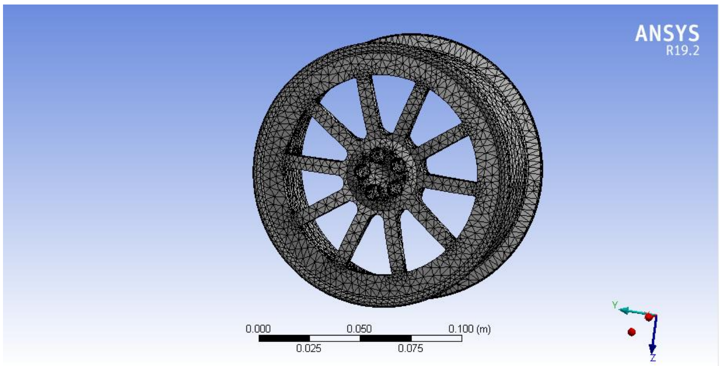


Figure 4: Meshing of wheel rim

The exact result In the ANSYS software the internal command setting can be available for mesh generation and above figure shows the process of the meshing and show the model for meshing.

TABLE 8.1: MESHAING DETAILS

NAME	ELEMENTS	NODES
CAM SHAFT	52771	94613

ANALYSIS OF WHEEL

STRUCTURAL STEEL WHEEL RIM

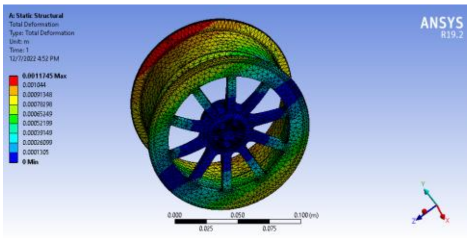


Figure: Total deformation

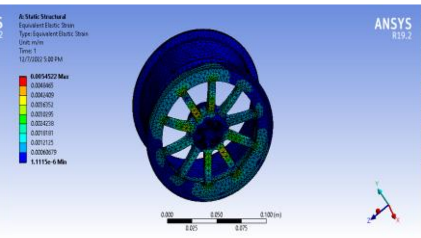


Figure: Equivalent Strain

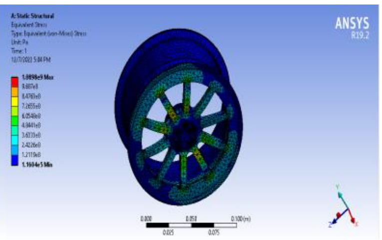


Figure: Equivalent Stress

ALUMINUM ALLOY WHEEL RIM

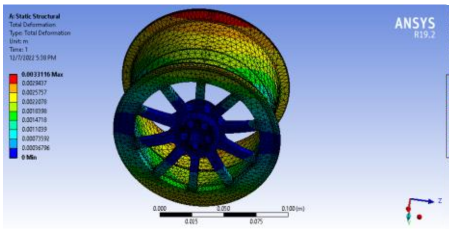


Figure: Total deformation

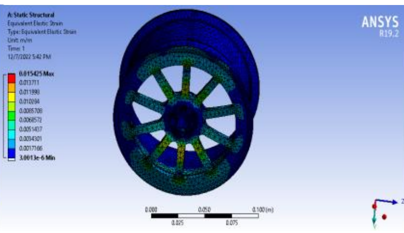


Figure: Equivalent Strain

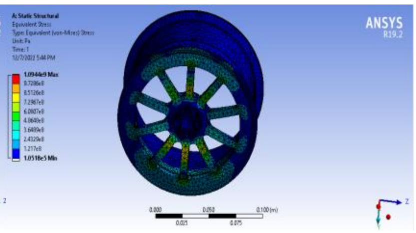


Figure: Equivalent Stress

MAGNESIUM ALLOY WHEEL RIM

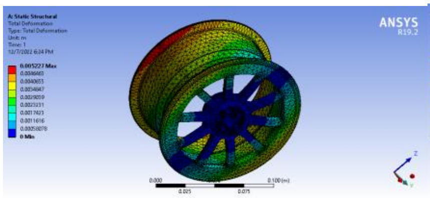


Figure: Total deformation

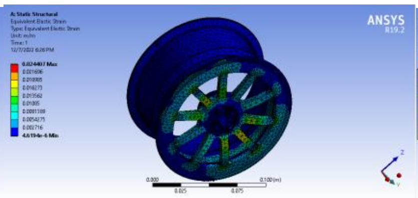


Figure: Equivalent Strain

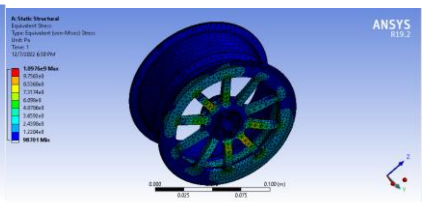


Figure: Equivalent Stress

RESULT SUMMARY

STRUCTURAL STEEL WHEEL RIM

Results	Minimum	Maximum	Units	Time (s)
Total Deformation	0.	1.1745e-003	m	1.
Equivalent Elastic Strain	1.1115e-006	5.4522e-003	m/m	1.
Equivalent Stress	1.1604e+005	1.0898e+009	Pa	1.

ALUMINUM ALLOY WHEEL RIM

Results	Minimum	Maximum	Units	Time (s)
Total Deformation	0.	3.3116e-003	m	1.
Equivalent Elastic Strain	3.0013e-006	1.5425e-002	m/m	1.
Equivalent Stress	1.0518e+005	1.0944e+009	Pa	1.

MAGNESIUM ALLOY WHEEL RIM

Results	Minimum	Maximum	Units	Time (s)
Total Deformation	0.	5.227e-003	m	1.
Equivalent Elastic Strain	4.6194e-006	2.4407e-002	m/m	1.
Equivalent Stress	98701	1.0976e+009	Pa	1.

CONCLUSION

At the end of the study and design process, we choose the most reliable and long-lasting wheel rim. This programme automates the design and analysis processes. Modal analysis, dynamic analysis, and performance evaluation of the rim were also conducted. There, they looked at the outcomes of static and model analysis, and they decided that forged steel would be the best option.

REFERENCE

[1.] Liangmo Wang, Yufa Chen, Chenzhi Wang, Qingzheng Wang, Fatigue Life Analysis of Aluminum Wheels by Simulation of Rotary Fatigue Test, Journal of Mechanical Engineering, Volume 57, 2011

[2.] S Vikranth Deepak, C Naresh and Syed Altaf Hussain, International Journal of Mechanical Engineering and Robotics Reasearch, Volume 1, 2012

[3.] P. Meghashyam, S. Girivardhan Naidu and N. Sayed Baba, International Journal of Application or Innovation in Engineering & Management, Volume 2, 2013

[4.] N. Satyanarayana & Ch.Sambaiah, International Journal of Mechanical and Industrial Engineering, Volume2 , 2012

[5.] Subba Rao Vlisi And Tippa Bhimasankar Rao, International Journal of Mathematical Sciences, Technology and Humanities, Volume 64, 2012

[6.] Ravi Lidoriya, SanjayChaudhary and Anil Kumar Mohopatra, International Journal of Mechanical Engineering and Research, Volume 3, 2013

[7.] Currie and Wilson, Finite Element Analysis of an Automotive Wheel—A Case History, National Conference Publication, Institution of Engineers, Australia, pp. 16–20, April 1981.

[8.] Institution of Mechanical Engineers, Ed., Automobile Wheels and Tyres, Institution of Mechanical Engineers, Ed., I. Mech. E Conference Publications, Conference Code 03757, London, UK, December 1983.

[9.] M. Reisner and R. DeVries, “Finite Element Analysis and Structural Optimization of Vehicle Wheels,” SAE Technical Paper Series, pp20.

[10.] N. Mizoguchii and A. Nishimura, “Stress analysis and fatigue strength of sheet fabricated Aluminum wheels,” R & D Magazine, vol. 32, no. 2, pp. 8–12, 1982.