International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 09 Issue: 06 | June 2022 www.irjet.net p-ISSN: 2395-0072

# COMPUTER AIDED DESIGNING AND SIMULATION OF RADIAL FATIGUE TEST OF AUTOMOBILE WHEEL RIM USING ANSYS

**JITHIN RAJ[1], SACHIN SUNIL[2], RENITH P R[3] ,AMAL KRISHNA[4]**

*[1,2,3,] B -Tech Student, Department of Mechanical Engineering Mangalam College of Engineering,Kerala,India-686631*

*[4]Assistant Professor, Dept. of Mechanical Engineering, Mangalam College of Engineerimg,Kerala,India-686631*

---

**Abstract -** *In automotive engineering, the wheels are one of the most critical components especially rims and their function is of vital importance for human safety. Predicting the fatigue life of the wheel accurately is one of the major challenges lying ahead for the wheel manufacturers. Fatigue life is a key consideration in designing an automotive wheel. In this study, an automotive wheel rim has been modeled in order to analyze stress and fatigue analysis using the finite element method. Simulation of fatigue loading is done by S-N curve and the expected life of rim is predicted. The expected life of any automotive component is to be well studied beforehand so that the risk of accidents during real service conditions can be reduced to a larger extent. Finite Element Method (FEM) has evolved as a resourceful tool for analyzing various components under a variety of operating conditions." It is being used not only to predict the critical points bearing the highest stress in a wheel but also to predict its fatigue life". The 3-dimensional model of the wheel was designed using CATIA. Then the 3-D model was imported into ANSYS. Static structural analysis is carried out to estimate the fatigue life, total deformation, and equivalent stress developed in the wheel rim. The results were optimized among three different materials - Structural Steel, Aluminum alloy, and Magnesium Alloy.*

***Key Words***: (Finite Element analysis, ANSYS, Stress-life, Static analysis, Car rims, Fatigue, Stress concentration, CATIA)…

## 1. INTRODUCTION

The tire is an important part of the car as it supports the weight of the car and helps the tire to stay in touch with the road. The tire is under very dangerous conditions. The wheel is designed to meet certain safety and engineering standards to provide high performance and efficiency. The wheel should be able to handle the shock and vibration, as well as the weight of the car and its occupants; it should be lightweight but extremely strong.

In this case, the tire is put into a series of testing during production to ensure that only the highest quality tires are properly removed from the market. In the design and design of the wheels, three main tests are used. Circular bending tests, radial fatigue tests, and impact assessments are three. They are used to set the model wheel by its movement in terms of fatigue and durability.

Fatigue fractures, as a result, are caused by the formation of small cracks that extend by adding long-term stress. However, the nature of stress is different. Fatigue fractures are caused by cyclical loads on objects, resulting from the process of crack nucleation and existing cracks [1].

The tension caused by circulatory stress is usually sinusoidal with the letter and varies with the frequency with which it is used [2]. Each height is marked by the mass of the main stress or the minimum of the minimum pressure, both with equal or opposite sign (thickness and pressure respectively). The pressure applied can be torsional, tensile, or flexural.

Equilibrium for the biaxial notch type loading equity was proposed by some researchers [3], and was used to measure the fatigue life of a passenger car tire during a fatigue test under air pressure conditions. They realized that when the analytical results from the computer program were compared with the actual fatigue test results, the results were positive.

Lu and Wei [4] used the Cosmos software to model a fixed feature of the HS6061T6 bus rack, which was used to replicate circulating fatigue tests. The life cycle of the rim has been improved to more than 1.0 x 105, indicating that the proposed method of analyzing the limiting factor was a good and effective way to predict the fatigue life of aluminum rims.

Some researchers [5] have used ABAQUS software to mimic circular fatigue testing by constructing a specific feature model for the fixed load of aluminum wheels. The benchmark test results of a rotating aluminum tire tire revealed that failure and implementation of fractures occurred near the hub hole area, which was consistent with the findings.

The current study shows a direct method for modeling a motor rim in CATIA, as well as a systematic simulation of rim pressure, bending, and radial fatigue testing.

## 2. CAD Model and Rim Size Designation

Table 1 summarizes wheel rim specifications. The CAD model of the rim is called the mathematical model here. The CATIA-built wheel uses a number of features such as rotating features, sweeping features, boss extrude base and extrude cut, fillet and overhead features for better analysis. A drawing is used to make a wheel model. This is done to ensure that when modeling the paint, the real state of the rim is applied, minimizing imperfections.

CATIA is software used to create and edit objects. In CATIA and the design and modeling feature is available. Design refers to the process of creating something new or repairing an existing one. To write an outline means a representation or idea of something. Modeling means creating and converting 2D into 3D. Using the CATIA software, create a wheel rim model

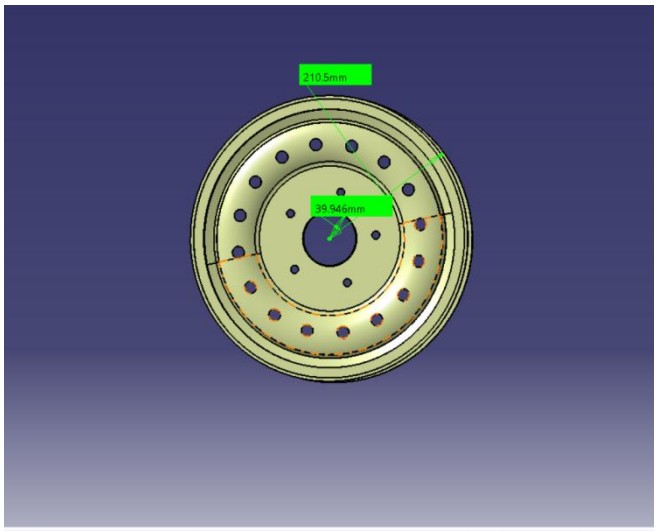


**Fig -1**: Wheel Rim With Dimension

| | |
|---|---|
| Rim diameter | 421mm |
| Rim width | 174.909mm |
| Offset | 145.595mm |
| PCD | 70mm |
| Hub diameter | 39.946mm |

**Table -1**: Wheel Rim Dimension

### 2.1 Steps Involved In Design

1. Draw a profile picture of the wheel rim.

2. Now rotate the profile body relative to the y axis.

3. By selecting the wheel surface, the required design drawn on the surface is removed using the POCKET function.

4. Using a circular pattern a specific design is found throughout the line.

5. By re-selecting the face draw a circle and rotate it using a circular pattern.

6. From the holes use the POCKET option.

7. And finally using the EDGE FILLET option the edges have been fitted to complete the final.

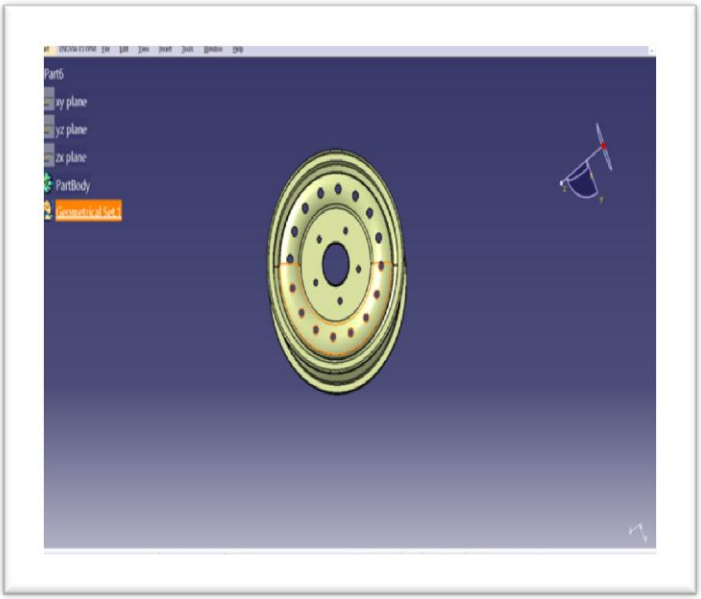


**Fig -2:** 3D Wheel Rim

## 3. ANALYSIS

Stable stress analysis is used to determine the dynamic pressure. The radial load (Fr) load and the estimated wheel pressure cause the pressure in the wheel (Pi) to be distributed. Fatigue stress found in static stress analysis was found to be reliable [6]. ANSYS was used for this project. Workbench is used for this statistical analysis.

### 3.1 STRUCTURAL STEEL

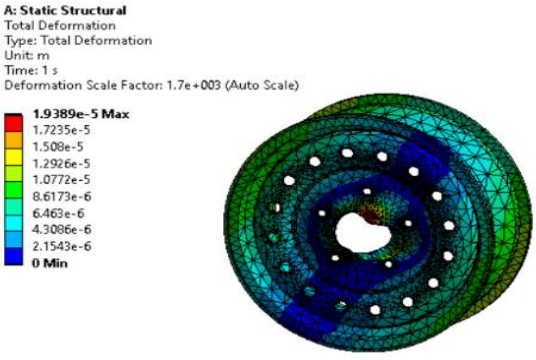


Fig-3 TOTAL DEFORMATION

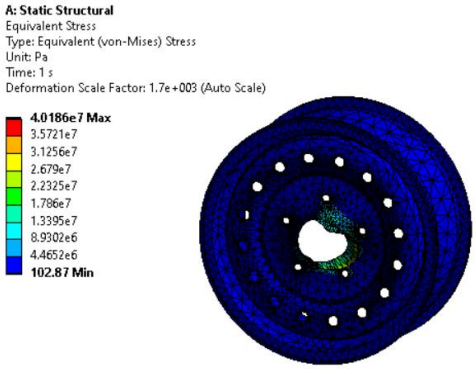


Fig -4 EQUIVALENT STRESS

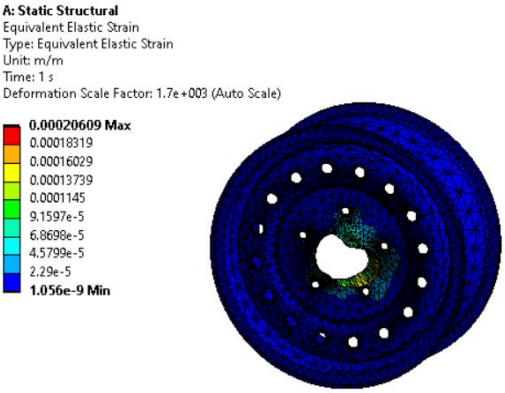


Fig -5 EQUIVALENT STRAIN

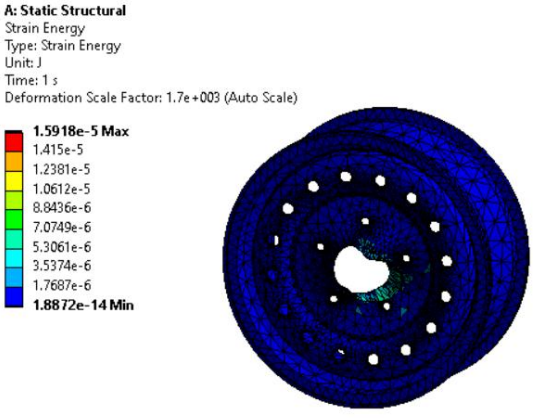


Fig -6 STRAIN ENERGY

### 3.2 MAGNESIUM ALLOY

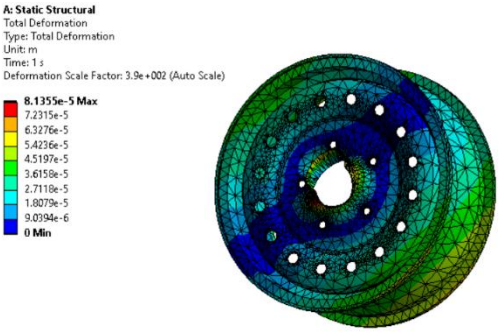


Fig -7 TOTAL DEFORMATION

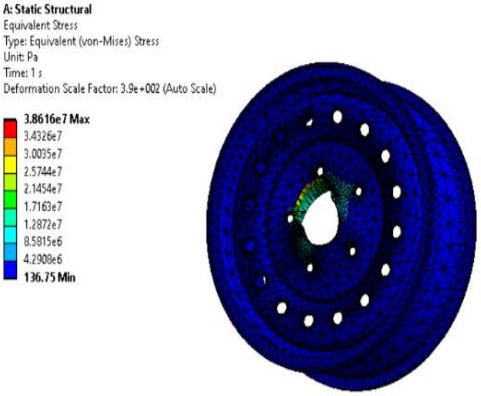


Fig 8-EQUIVALENT STRESS

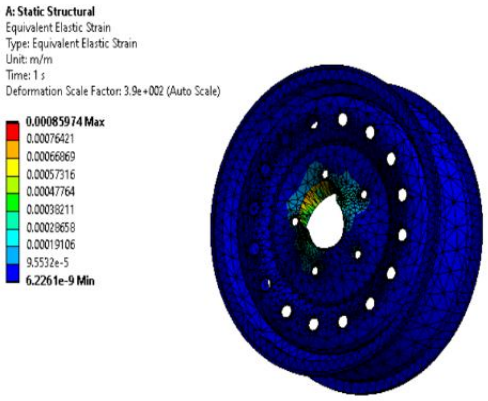


Fig 9-EQUIVALENT STRAIN

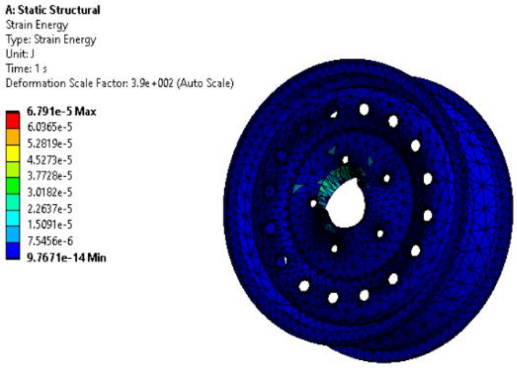


Fig 10 -STRAIN ENERGY

### 3.3 ALUMINIUM ALLOY

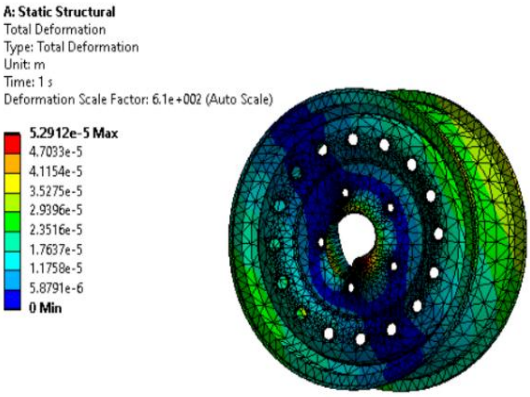


Fig 11- TOTAL DEFORMATION

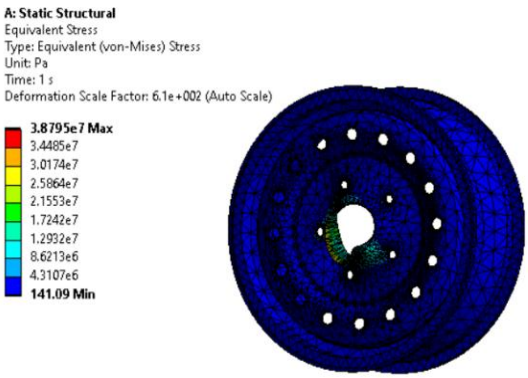


Fig 12- EQUIVALENT STRESS

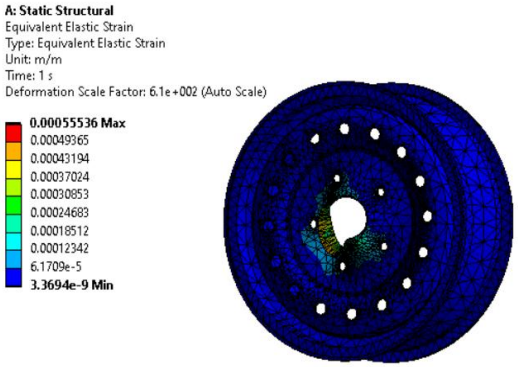


Fig 13- EQUIVALENT STRAIN

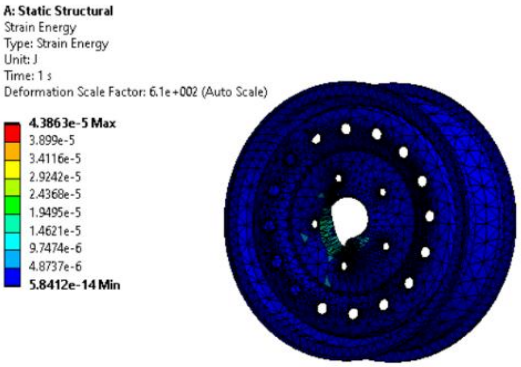


Fig 14- STRAIN ENERGY

## 4. RESULTS

| Material | Total Deformation (mm) | Equivalent Stress (MPa) | Equivalent Strain (m/m) | Strain Energy (J) |
|---|---|---|---|---|
| Structural Steel | 0.013064 | 4406.930 | 0.00020609 | 0.0107254 |
| Aluminum Alloy | 0.035651 | 4254.388 | 0.000555 36 | 0.0295546 |
| Magnesium Alloy | 0.054816 | 4234.758 | 0.000859 74 | 0.0457473 |

Table 2-RESULTS

## 5. CONCLUSION

- Static structural and fatigue analysis was done on three different materials namely Magnesium alloy, Structural Steel and Aluminium alloy and the results were recorded.
- The total deformation of Magnesium alloy is 0.054816m which is higher as compared to Structural Steel and Aluminium.

- The maximum stress of Steel is recorded to be 4406.930 Mpa which is higher as compared to Magnesium and Aluminium.
- Higher strength is for Magnesium alloy which is of 0.0457473 Joules.
- Based on deformation Magnesium alloy is better but considering the feasibility of the three materials Structural Steel is the most viable option.
- Hence Structural Steel is commonly used in commercial vehicles.

## REFERENCES

[1] Emmanuel M. Adigio and Ebughni O. Nangi (2014) Computer Aided Design and Simulation of Radial Fatigue Test of Automobile Rim Using ANSYS, Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320- 334X, Volume 11, PP 68-73

[2] Jaspreet Singh, Siddhartha Saha (2015) "Static analysis of alloy wheel using ANSYS",International Journal of Research in Engineering and Technology (IJRET),Volume 4 Issue 7,ISSN 2319-1163

[3] K. Srinivasa Rao, M .Rajesh, G.Sreedhara Babu(2017) "Design And Analysis of Alloy Wheel", International Research Journal of Engineering and Technology (IRJET), Volume 4 Issue 6,ISSN 2395-0072

[4] Gamachisa Mitiku Tadesse(2017) "Modeling and Analysis of Car Wheel", International Research Journal of Engineering and Technology (IRJET),Volume 4 Issue 2 ISSN 2395-0072.

[5] Dr. M. S. Hebbal, Mukunda Dabair(2019) "Static Structural, Modal and Harmonic Analysis of Alloy Car Wheel Rim using ANSYS Workbench", International Journal of Engineering Research & Technology (IJERT), Vol 8 Issue 7, ISSN 2278-0181

[6] R. Ashok Kumar(2019) "Experimental Studies of Optimization of Automotive wheel Rim using ANSYS", International Journal of Trend in Scientific Research and Development (IJTSRD),Volume 3 Issue 3,ISSN 2456 - 647