Fatigue and Static Structural Analysis of Car Wheel using Finite Element Method

Karan Valetava, Param jethava
PIET, Vadodara

**ABSTRACT:** The purpose of the car wheel rim is to provide a firm base on which tire could be fitted. Its dimensions, shape should be suitable to adequately accommodate the particular tire required for the vehicle. The wheel is designed using 3-dimensional modelling software. In modelling the time spent in producing the complex 3-dimensional models and risk involved in design and manufacturing process can be easily minimized. Then the 3-D model was imported into ANSYS using the IGES format. Using finite element analysis, static structural and fatigue analysis work carried out by considered two different materials namely A356.2 aluminium alloy and carbon fiber and their relative performance have been observed respectively.

The finite element idealization of this model was then produced using the tetrahedron solid element. The analysis was performed in a static condition. The load is applied on the rim. We find out the total deformation, alternative stress and principal stress by using FEA software. And also we find out the life, safety factor and damage of spoke wheel by using S-N curve. In this paper by observing the results of both static and fatigue analysis obtained carbon fiber is suggested as better material for designing of wheel.

**KEYWORDS:** Wheel rim; 3-dimensional model; Finite element analysis; Static structural and fatigue analysis; S-N curve

I. INTRODUCTION

The wheel is a device that enables efficient movement of an object across a surface where there is a force pressing the object to the surface. Early wheels were simple wooden disks with a hole for the axle. Because of the structure of wood a horizontal slice of a trunk is not suitable, as it does not have the structural strength to support weight without collapsing; rounded pieces of longitudinal boards are required.

The spoke wheel was invented more recently, and allowed the construction of lighter and swifter vehicles. Alloy wheels are automobile wheels which are made from an alloy of aluminum or magnesium metals. Historically, successful designs was arrived after years of experience well aided worth extensive field -testing. Since the 1970's several innovative methods of testing and experimental stress measurements have been initiated. In more recent years, the procedures have significantly improved by the emergence of a variety of experimental and analytical methods for structural analysis. Durability analysis, that is: fatigue life prediction and reliability methods, for dealing with various inherent in engineering structures has been used for the study of automotive rims.

In its basic form a wheel is a transfer element between the tire and the vehicle. The main requirements of an automobile wheel are;

- It should be as light as possible so that unsprung weight is least.
- It should be strong enough to perform the above functions.
- It should be balanced statically as well as dynamically.
- It should be possible to remove or mount the wheel easily.
- It material should not deteriorate with weathering and age. In case, the material is suspected to corrosion, it must be given suitable protective treatment.

II. TYPES OF WHEEL

Steel and light alloy are the foremost materials used in a wheel rim however some composite materials together with glass-fibre are being used for special wheels.

A. Wire spoke Wheel

Wire spoke wheel is an essential where the exterior edge part of the wheel rim and the axle mounting part are linked by numerous wires called spokes. Today’s automobiles with their high horse power have made this type of wheel manufacture obsolete. This type of wheel is still used on classic vehicles.

B. Steel Disc Wheel

This is a rim which practices the steel made rim and the wheel into one by joining (welding), and it is used mainly for passenger vehicles especially original equipment tires.
C. Light Alloy wheel

These wheels are based on the use of light metals, such as aluminium and magnesium has come to be popular in the market. This wheel rapidly become standard for original equipment vehicle in Europe in 1960’s and for the replacement tire in United States in 1970’s. The advantages of each light alloy wheel are explained as below.

- **Aluminium Alloy Wheel**
  Aluminium is a metal with features of excellent lightness, thermal conductivity, physical characteristics of casting, low heat, machine processing and reutilizing, etc. This metal main advantage is decreased weight, high precision and design choices of the wheel.

- **Magnesium alloy Wheel**
  Magnesium is about 30% lighter than aluminium and also admirable as for size stability and impact resistance. However its use is mainly restricted to racing, which needs the features of weightlessness and high strength. It is expansive when compared with aluminium.

- **Titanium alloy wheel**
  Titanium is an admirable metal for corrosion resistance and strength about 2.5 times compared with aluminium, but it is inferior due to machine processing, designing and more cost. It is still in developed stage.

- **Composite material wheel**
  The composite material wheel is different from the light alloy wheel, and it is developed mainly for low weight. However this wheel has inadequate consistency against heat and for best strength.

### III. LITERATURE REVIEW

A Review on Modeling and Analysis of Car Wheel Rim using CATIA & ANSYS by Siva Prasad et al.[1] does stress and dynamic analysis of car wheel rim by using CATIA & ANASYS. To determine best material for wheel so that by design and modifications the stresses can be reduces to improve the fatigue life of wheel rim. During this study, they considered two different materials namely aluminium and forged steel and their relative performances have been observed respectively. Aluminium alloy wheel rim are subjected to more displacement and stresses compared to Forged steel and they are suggested forged steel is better material.

Sourav das et al.[2] gives design of aluminum alloy wheel for automobile application which is carried out paying special reference to optimization of the mass of the wheel. The Finite Element analysis it shows that the optimized mass of the wheel rim could be reduced to 26Kg to 12.15kg as compared to the solid disc type Al alloy wheel. The FE analysis indicated that even after a fatigue cycle of $1 \times 10^5$, the damage on the wheel is found only 0.2%. And the damage region is found the flange portion of the rim.

Rajarethinam P et al. [3] presented paper on motorcycle wheel spokes and in this paper wheel rim designed by using designing software SOLID WORKS and later, for analysis 3-D model is imported into ANSYS.

- The maximum stress area was located at Spoke-Rim contact.
- Stresses induced in 5 Spokes Alloy wheel are less as compared with Al-Alloy of the 6 Spokes.
- Material reduction can be done by reducing number of Spokes

Liangmo Wang et al.[4] gives analysis to improve the quality of aluminum wheels, a new method for evaluating the fatigue life of aluminum wheels is proposed in this paper. The ABAQUS software was used to build the static load finite element model of aluminum wheels for simulating the rotary fatigue test. The results from the aluminum wheel rotary fatigue bench test showed that the baseline wheel failed the test and its crack initiation was around the hub bolt hole area that agreed with the simulation. Using the method proposed in this paper, the wheel life cycle was improved to over $1.0 \times 10^7$.

M. Saran Theja et al. [5] presented paper deals with the static and fatigue analysis of wheel to analyse the safe load of the alloy wheel A typical alloy wheel configuration of Suzuki GS150R commercial vehicle is used for study. This design is 60% lighter and the overall dimensions are controlled by reducing number of spokes to the alloy wheel with same functioning stability and less weight. The stress and displacements in 4 spoke alloy wheel are lesser than six and five spokes alloy wheels.

N. Satyanarayana et. al.[6] gives a detailed "Fatigue Analysis of Aluminum Alloy Wheel under Radial Load". During the part of project a static and fatigue analysis of aluminum alloy wheel A356.2 was carried out using FEA package. The 3 dimensional model of the wheel was designed using CATIA. Then the 3-D model was imported into ANSYS using the IGES format. The analysis was performed in a static condition.

- The total deformation of wheel maximum is 0.2833mm and minimum is 0.031478 at hub portion.
- The equivalent stress is 163.97MPa and 0.038MPa.
Sunil N. Yadav et al.[7] gives effect of slip angle on stress distribution and fatigue life of wheel rim of passenger car under radial load condition which arises due to off road field area and road unevenness. The finite element analysis as well as experimental analysis of passenger car wheel rim performed for radial load with the effect of slip angle on stress distribution and fatigue life.

- The stresses are much higher in the disc area than the rim area.
- The likely failures locations identified in the wheel rim by finite element analysis are stud holes, stiffening bulge and ventilation holes.
- The stresses in wheel rim are directly proportional to slip angle i.e. the life of wheel rim decreases as slip angle increase.

S Vikranth Deepak et al.[8] does static structural and fatigue analysis of four wheeler vehicle by using finite element analysis. A typical alloy wheel configuration of ford fiesta is used for study. The analysis results showed that the maximum stress area was located in the hub bolt whole area. For all comparing the three materials (aluminium, magnesium and zinc) of stress, displacement, total life, load factor and damage factor they was suggested that aluminium alloy is the best material for the alloy wheel.

P. V. Ravi Kumar et al.[9] have studied paper describes impact test and topology optimization of cast aluminium alloy wheel with constrain of plastic strain. the fail value of plastic strain for standard cast aluminium alloy wheel is 4.0%, cracks will appear if the Plastic Strain value is greater than 4%. This analysis will predict the plastic strains induced during impact testing. Topology optimization is carried out by increasing the thickness of the rim until the plastic strain value is below 4%. Impact analysis is carried out using LS-Dyna software to predict the plastic strains during impact test. They concluded that thickness of cast aluminium alloy wheel should be 5.9mm which will be perform satisfactory.

S. Ganesh et al.[10] gives analysis of alloy wheels which are made from an alloy of aluminum or magnesium metals or sometimes a mixture of both. They use four wheeler wheels are made of Aluminum Alloys. They are collecting data from reverse engineering process from existing model.

In the case of bending test normal stress along Y-axis shows compression on the top rib and tension on the bottom rib and compression on the bottom rib. In the case of pressure loading, normal stress along X-axis shows compression on the top rim and on the inside portion of the rim there is a gradual transition from compression to tension.

P. Meghashyam et al.[11] gives a tire of car wheel rim belonging to the disc wheel category is considered. The wheel rim is designed by using modelling software catiav5r18. Later this CATIA model is imported to ANSYS for analysis work. ANSYS static analysis work is carried out by considered two different materials namely aluminium and forged steel and their relative performances have been observed respectively. In this paper by observing the results of both static and modal analysis obtained forged steel is suggested as best material.

Numerical simulation of steel wheel dynamic cornering fatigue test by Shu-Qin Pan et al. [12] presented a computational methodology is proposed for fatigue life and failure prediction of automotive steel wheel by the simulations of dynamic cornering fatigue test. They concluded that principle planes variation changes a little, varying from -40° to 30°, and the stress states of automotive steel wheel are in biaxial tensile and compression stresses during dynamic cornering fatigue test, and it is conservative and considerable to observe that fatigue test cycles and crack initiation locations are predicted using Brown–Miller damage criterion, which are close to the actual test results, and the minimum error is -4.4%.

Hongyu Wang et al. [13] gives the parametric three dimensional model of the rim section is built based on Solid Works. The optimization methods which combined multi-island genetic algorithm (MIGA) with sequential quadratic programming (NLPQL) is used for exploration. By adjusting control parameters of the rim shape quality is optimized.

IV. MODELING OF WHEEL

Creo is software which is used for creation and modifications of the objects. In Creo design and modeling feature is available. Design means the process of creating a new object or modifying the existing one. Drafting means the representation or idea of the object. Modeling means create and converting 2D to 3D. By using Creo software, create the model of the wheel rim.

Step Involved In Design
1. Draw the profile diagram of the wheel rim.
2. Now revolve the profile body about respect to axis.
3. By selecting the face of wheel, the required design is drawn on the surface is removed by using Cut operation.
4. Now selecting face draw circle and rotate them using circular pattern about axis so, spokes are obtained all over the rim.
5. And finally using round option the side edge are made filleted for final finishing.
Table 1. Wheel Specification of Hyundai i20(15)

<table>
<thead>
<tr>
<th>Specification</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rim Diameter</td>
<td>432mm</td>
</tr>
<tr>
<td>Rim Width</td>
<td>178mm</td>
</tr>
<tr>
<td>Positive Offset</td>
<td>35mm</td>
</tr>
<tr>
<td>Bolt PCD</td>
<td>100mm</td>
</tr>
<tr>
<td>Rim Thickness</td>
<td>5.9mm</td>
</tr>
<tr>
<td>Hub Diameter</td>
<td>54.1mm</td>
</tr>
<tr>
<td>Hole Diameter</td>
<td>12mm</td>
</tr>
<tr>
<td>Spoke Fillet Radius</td>
<td>5mm</td>
</tr>
</tbody>
</table>

V. FINITE ELEMENT ANALYSIS

The Finite Element Technique produces many simultaneous algebraic equations, which are generated and solved on a digital computer. The Finite Element Method originated as a method of stress analysis. The Finite Element Method is firmly established as a powerful and popular analysis tool. It is a numerical procedure for analyzing structures and continua. Finite element procedures are used in design of buildings, electric motors, heat engines, ships, airframes and spacecrafts.

The Finite Element Method, in general, models a structure as an assemblage of small elements. FEA has become a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material and allowing designers to see all of the theoretical stresses within. This method of product design and testing is far superior to the manufacturing costs, which would accrue if each sample was actually built and tested. FEA consists of a computer model of a material or design that is stressed and analyzed for specific results.

It is used in new product design, and existing product refinement. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition. FEA uses a complex system of points called nodes, which make a grid called a mesh. This mesh is programmed to contain the material and structural properties, which define how the structure will react to certain loading conditions.

A. Mesh Generation

The mesh was meshed with tetrahedron structural solid elements. The wheel was meshed using an element edge length is 5mm. The total number of nodes and elements are 379471 and 230824 respectively. The finite element realization of the wheel obtained is shown in fig.2.
B. Material Properties

Fatigue and static structural analysis of wheel using Finite element analysis is carried out to analyse zones with higher stress concentration and to improve the life cycle of wheel. For analysis, two materials are selected namely A356.2 aluminium alloy and carbon fiber and to show their relative performance using finite element analysis. This materials properties are shown in table 2.

<table>
<thead>
<tr>
<th>Properties</th>
<th>A356.2 Al alloy</th>
<th>Carbon Fiber</th>
</tr>
</thead>
<tbody>
<tr>
<td>Possion's Ratio</td>
<td>0.33</td>
<td>0.10</td>
</tr>
<tr>
<td>Density (g/cm$^3$)</td>
<td>2.7</td>
<td>1.60</td>
</tr>
<tr>
<td>Yield Strength (MPa)</td>
<td>225</td>
<td>450</td>
</tr>
<tr>
<td>Compressive Yield Strength (MPa)</td>
<td>250</td>
<td>570</td>
</tr>
<tr>
<td>Ultimate Tensile Strength (MPa)</td>
<td>279</td>
<td>600</td>
</tr>
<tr>
<td>Young's Modulus (GPa)</td>
<td>69</td>
<td>110</td>
</tr>
<tr>
<td>Thermal Expansion ($10^6 \times K^{-1}$)</td>
<td>21.5</td>
<td>2.10</td>
</tr>
<tr>
<td>Thermal Conductivity (W/mK)</td>
<td>210</td>
<td>180</td>
</tr>
</tbody>
</table>

C. Static Structural Analysis

- Import developed Creo model into Ansys
- Select the type of analysis require.
- Applying the material
- Create the mesh for the imported model
- Fixing the geometry
- Apply the loads (18.345KN)
- Run the analysis

1. A356.2 Al Alloy Results

Total Deformation

Von-mises Stress
2. Carbon Fiber Results

Total Deformation

Von-mises Stress
D. Fatigue Analysis

1. A356.2 Al Alloy Results

Life

S-N Curve
2. Carbon Fiber Results

3. Life

Fig. 11 Safety Factor of A356.2 Wheel

Fig. 12 Damage in A356.2 Wheel

S-N Curve

Fig. 14 S-N Curve for Carbon Fiber Wheel

Fig. 13 Life of Carbon Fiber Wheel

Safety Factor

Damage
E. Result Comparison

<table>
<thead>
<tr>
<th>Material</th>
<th>Total Deformation(mm)</th>
<th>Von-mises Stress(MPa)</th>
<th>Principal Stress(MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A356.2 Al Alloy</td>
<td>105.505</td>
<td>0</td>
<td>103.54</td>
</tr>
<tr>
<td>Carbon Fiber</td>
<td>0.97196</td>
<td>0</td>
<td>102.97</td>
</tr>
</tbody>
</table>

Table III. Static Structural Results Comparison

<table>
<thead>
<tr>
<th>Material</th>
<th>Life</th>
<th>Safety Factor</th>
<th>Damage</th>
</tr>
</thead>
<tbody>
<tr>
<td>A356.2 Al Alloy</td>
<td>10^{10}</td>
<td>1.51 \times 10^6</td>
<td>15</td>
</tr>
</tbody>
</table>

Table IV. Fatigue Analysis Result Comparison

In table 3 and table 4 result of static structural and fatigue analysis is given. Also it is graphically denoted in fig.3 to fig. 16. After completion of meshing we apply load 18.345KN at rim. For A356.2 Al Alloy, the total deformation of wheel maximum is 1.5505mm and minimum is 0 at hub portion. The equivalent stress is 103.54MPa and 0.043245MPa. The principal stress maximum is 107.83MPa and minimum is -29.248MPa. The life of wheel maximum 1e10 cycles and the minimum cycles of wheel is 1.51e6 at a bolt area of wheel. For carbon fiber, the total deformation of wheel maximum is 0.97196mm and minimum is 0 at hub portion. The equivalent stress is 102.97MPa and 0.030863MPa. The principal stress maximum is 106.23 and minimum is -29.63MPa. The life of wheel maximum more than 1e10 cycles and the minimum cycles of wheel is 1.0988e6 at a bolt area of wheel. For both materials, total deformation maximum at rim flange and minimum at hub and equivalent stress maximum at bolt and minimum at rim flange and The wheel safety maximum at a hub portion because the load is maximum acting at a rim. Minimum load is acting at a hub. Finite element analysis is carried out by simulating the test conditions to analyze stress distribution and fatigue life, safety and damage of alloy wheel.
VI. CONCLUSIONS

Model of the wheel rim is generated in Creo and this is imported to ANSYS for processing work. An amount of load 18.345KN is applied along the circumference of the wheel rims made of both A356.2 Al Alloy & Carbon Fiber and hub circle of wheel rim is fixed. Following are the conclusions from the results obtained:

1. A356.2 Al alloy wheel rim is subjected to more total deformation compared to carbon fiber.
2. A356.2 Al alloy spoke wheel is subjected more von-mises stress and principle stress compared to carbon fiber.
3. Since in both cases stress generated is less than the yield strength, hence design is safe.
4. Carbon fiber has a more life compared to A356.2 Al alloy.
5. Weight of carbon fiber is 40% less compared to A356.2 Al alloy.
6. By comparing all result we are suggested that carbon fiber is better material than A356.2 Al alloy for designing of wheel.

REFERENCES