MODELING AND STRUCTURAL ANALYSIS OF ALLOY WHEEL USING ANSYS

Mr. Sasank Shekhar Panda*, Mr. Dibya Narayan Behera, Mr. Satya Narayan Tripathy

*Research scholar, Department of Mechanical Engineering, G.I.E.T, Gunupur, Rayagada, India
Research scholar, Department of Mechanical Engineering, G.I.E.T, Gunupur, Rayagada, India
Research scholar, Department of Mechanical Engineering, G.I.E.T, Gunupur, Rayagada, India

DOI: 10.5281/zenodo.47607

ABSTRACT

Wheel spokes are the supports consisting of a radial member of a wheel joining the hub to the rim with Carbon Fiber, Magnesium Alloy, Titanium Alloy and Aluminum Alloy. The two main types of motorcycle rims are solid wheels, in which case the rim and spokes are all cast as one unit, usually in Aluminum or magnesium alloys and the other spoke wheels, where the motorcycle rims are laced with spokes which require high spoke tension, since the load is carried by fewer spokes. If a spoke does break, the wheel generally becomes instantly un-ridable also the hub may break. Presently, for high cc bikes Magnesium wheels are used, due to its low heat resistance and micronisation of crystal grains, replacing it with Aluminum alloy. This Simulation work attempts to model the wheel of a two wheeler racing by using the CATIA Software, and conducting the tests: Static and Fatigue analysis using the ANSYS software by reducing the number of spokes from 5 to 4 for the existing model. Based on simulation work, a better material for alloy wheels may be analyzed from the results obtained and validated.

KEYWORDS: Alloy Wheel, CATIA, ANSYS, Static and Fatigue analysis

INTRODUCTION

A wheel is a circular device that is capable of rotating on its axis, facilitating movement or transportation while supporting a load (mass), or performing labour in machines. Safety and economy are particularly of major concerns when designing a mechanical structure so that the people could use them safely and economically. Style, weight, manufacturability and performance are the four major technical issues related to the design of a new wheel and/or its optimization mainly for Aluminum wheels according to governmental regulations and industry standards [1-3]. In the real service conditions, the determination of mechanical behaviour of the wheel is important, but the testing and inspection of the wheels during their development process is time consuming and costly. For economic reasons, it is important to reduce the time spent during the development and testing phase of a new wheel. Finite element analysis (FEA) was carried out by simulating the test conditions to analyze the stress distribution and fatigue life of alloy wheels. The analytical results using FEA to predict the wheel fatigue life agreed well with the experimental results [4]. A mathematical model was developed to predict the residual stress distribution of an A356 alloy wheel, taking into account the residual stress evolution during the T6 quench process and redistribution of residual stress due to the material removal at the machining stage. The fatigue life of an A356 wheel was predicted by integrating the residual stress into the in-service loading and wheel casting defects (pores). The residual stress showed a moderate influence on the fatigue life of the wheel, which was more sensitive to casting pore size and service stress due to applied loads [6]. By improved Smith formula, finite element analysis of stress values as the basic parameters for wheel fatigue life prediction [5]. ABAQUS software to build the static load finite element model of Aluminum wheels for simulating the rotary fatigue test [7]. The equivalent stress amplitude was calculated based on the nominal stress method by considering the effects of mean load, size, and fatigue notch, surface finish and scatter factors. The fatigue life of Aluminum wheels was predicted by using the equivalent stress amplitude and Aluminum alloy wheel S-N curve. 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 1.0×10^5 and satisfied the design...
requirement. A mathematical model was developed to predict the residual stress distribution of an A356 alloy rim, taking into account the residual stress evolution during the T6 quench process [9]. Static and fatigue analysis of Aluminum alloy wheel A356 by finite element idealization modal using the 10 node tetrahedron solid element in static condition and the wheel was designed using CATIA [8], total deformation, alternative stress and shear stress is simulated by using FEA software.

This paper starts by modelling of the alloy wheel in a two-wheeler racing bike using the Pro/Engineer Software for five different materials viz. LM 25, LM25TB7, LM 25TE, LM25TF and AM60A and conducting the tests: Static and Fatigue analysis using the CATIA software by reducing the number of spokes from 6 to 5 and then 5 to 4 for the existing model. Based on simulation work, a better material for alloy wheels may be analyzed from the results obtained and validated.

MODELING IN PRO-E

Pro/ENGINEER Wildfire is the standard in 3D product design, featuring industry-leading productivity tools that promote best practices in design while ensuring compliance with industry and company standards. Figure 1 shows the sketch of alloy wheel.

CATIA Works

CATIA is useful software for design analysis in mechanical engineering. CATIA is a design analysis automation application fully integrated with Solid Works. This software uses the Finite Element Method (FEM) to simulate the working conditions of your designs and predict their behaviour. FEM requires the solution of large systems of equations. Powered by fast solvers, CATIA makes it possible for designers to quickly check the integrity of their designs and search for the optimum solution. A product development cycle typically includes the following steps:

1. Build your model in the Solid Works CAD system.
2. Prototype the design.
3. Test the prototype in the field.
4. Evaluate the results of the field tests.
Table- 1 Material Properties

<table>
<thead>
<tr>
<th>S. No</th>
<th>PROPERTY</th>
<th>Al Alloy 201.0-T43 Insulated Mold Casting (SS)</th>
<th>Mg Alloy ZK60*</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Elastic Modulus(GPa)</td>
<td>71</td>
<td>45</td>
</tr>
<tr>
<td>2</td>
<td>Poisson's Ration</td>
<td>0.33</td>
<td>0.35</td>
</tr>
<tr>
<td>3</td>
<td>Mass Density (kg/m3)</td>
<td>2800</td>
<td>1700</td>
</tr>
<tr>
<td>4</td>
<td>Tensile Strength (MPa)</td>
<td>273</td>
<td>425</td>
</tr>
<tr>
<td>5</td>
<td>Yield Strength (MPa)</td>
<td>225</td>
<td>382</td>
</tr>
<tr>
<td>6</td>
<td>Thermal Expansion Coefficient(/K)</td>
<td>1.90E-05</td>
<td>1.90E-05</td>
</tr>
<tr>
<td>7</td>
<td>Thermal Conductivity W/(m. K)</td>
<td>121</td>
<td>160</td>
</tr>
<tr>
<td>8</td>
<td>Specific Heat J/(kg.K)</td>
<td>963</td>
<td>1000</td>
</tr>
</tbody>
</table>

Table- 2 Mesh Information and details are represented

<table>
<thead>
<tr>
<th>TYPE OF WHEEL MODEL</th>
<th>With 6 Spokes</th>
<th>With 5 Spokes</th>
<th>With 4 Spokes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element Size</td>
<td>6 mm</td>
<td>6 mm</td>
<td>6 mm</td>
</tr>
<tr>
<td>Tolerance</td>
<td>0.3 mm</td>
<td>0.3 mm</td>
<td>0.3 mm</td>
</tr>
<tr>
<td>Mesh Quality</td>
<td>High</td>
<td>High</td>
<td>High</td>
</tr>
<tr>
<td>Total Nodes</td>
<td>138283</td>
<td>129933</td>
<td>121024</td>
</tr>
<tr>
<td>Total Elements</td>
<td>77485</td>
<td>72121</td>
<td>66289</td>
</tr>
<tr>
<td>Maximum Aspect Ratio</td>
<td>27.471</td>
<td>27.339</td>
<td>27.337</td>
</tr>
<tr>
<td>% of elements with Aspect Ratio &lt; 3</td>
<td>76.2</td>
<td>74.2</td>
<td>72.8</td>
</tr>
<tr>
<td>% of elements with Aspect Ratio &gt; 10</td>
<td>0.246</td>
<td>0.326</td>
<td>0.291</td>
</tr>
<tr>
<td>% of distorted elements(Jacobian)</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Time to complete mesh*(hh:mm:ss):</td>
<td>00:02:00</td>
<td>00:01:59</td>
<td>00:01:56</td>
</tr>
<tr>
<td>TABLE 3 : SIMULATION RESULT DETAILS</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>-------------------------------------</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Meshed Models</th>
<th>With 6 Spokes Al alloy</th>
<th>With 5 Spokes Al alloy</th>
<th>With 4 Spokes Mg alloy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fig 1</td>
<td>Fig 8</td>
<td>Fig 15</td>
<td></td>
</tr>
<tr>
<td>Stress Analysis</td>
<td>Fig 2</td>
<td>Fig 9</td>
<td>Fig 16</td>
</tr>
<tr>
<td>Displacement Analysis</td>
<td>Fig 3</td>
<td>Fig 10</td>
<td>Fig 17</td>
</tr>
</tbody>
</table>

STRESS ANALYSIS FOR BREAKING TORQUE

Fig 4
Fig 11
Fig 18

DISPLACEMENT ANALYSIS FOR BREAKING TORQUE

Fig 5
Fig 12
Fig 19

FATIGUE LIFE ANALYSIS

Fig 6
Fig 13
Fig 20
**ISOMETRIC VIEWS OF MODELS**

| Fig 7 | Fig 14 | Fig 21 |

---

**For Different Loads Stress on Each Rim** –

**Applied Loads**

- **Load 0** = weight of Bike (143 vehicle +20 extra kg)
- **Load 1** = (163+65) kg
- **Load 2** = (163+65X2) kg
- **Load 3** = (163+65X3) kg
- **Load 4** = (163+65X4) kg
- **Load 5** = (163+65X5) kg
- **Load 6** = (163+65X6) kg

**Analysis for strength needed:**

- **Mass of Bike, Dead Weight of Bike** = 143 kg
- **Other Loads** = 20 Kg
- **Total Gross Weight** = 143 + 20 = 163 Kg
  = 163X 9.81 N

Tires and Suspension system reduced by 30% of Loads

\[ W_{net} = 163 \times 9.81 \times 0.7 \text{ N} = 1119.32 \text{ N} \]

**Reaction Forces On Bike** = \( N_r = 1119.32 \text{ N} \)

**Number of Wheels**: 2

But by considering total Reaction Force on only one wheel \( F_r = 1119.32 \text{ N} \)

**Rim surface area which is having 6 spokes**: \( A_6 = 48299.69 \text{ mm}^2 \)

(This can be obtained from selecting faces on rim by using measuring tool in solid works)

**Stress on the each Rim** = \( \sigma_s = 0.02321 \text{ N/mm}^2 \)

So pressure on the each rim for load 0 = 0.02321 N/mm²

It is similarly for different Loads Stress on Each Rim with Loads

**Pressure by Load 1** = 0.0324 N/mm²

**Pressure by Load 2** = 0.0417 N/mm²

**Pressure by Load 3** = 0.0509 N/mm²

**Pressure by Load 4** = 0.0601 N/mm²

**Pressure by Load 5** = 0.0694 N/mm²

**Pressure by Load 6** = 0.0786 N/mm²

**Applying Pressures:**

Apply 0.011945MPa pressure simulations normal to the faces as shown in the figure

Again it is similarly for rims with spokes 5 & 4. The simulation results are as shown in figures.

**Applying Braking Torque:**

In general Acceleration of the street motorcycle: \( a = (v_f - v_i) / t \)

http://www.ijesrt.com
\( v_f \) final velocity = max of 60 miles in 3.5 sec
\( v_i \) initial velocity = 0 miles,
\( a = \text{acceleration} = \frac{7.6636 \text{m/s}^2}{3.5 \text{sec}} \)
Brake force is required to estimate the load on the wheel hub. Now Total force acting on the vehicle:
Mass of the vehicle including rider and other five more persons M = 163 + 65 X 6
\[ F_{\text{total}} = M X a = 4237.9 \text{ N} \]

**RESULTS AND DISCUSSION**
Stress analysis values for 6, 5-Spokes Al-alloy and 4 spoke Mg-alloy in the following table.

<table>
<thead>
<tr>
<th>S.NO</th>
<th>Description</th>
<th>load (N)</th>
<th>Stresses In the Alloy Wheel in MPa</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>With 6 Spoke Al</td>
</tr>
<tr>
<td>1</td>
<td>Motorcycle Load</td>
<td>1119.321</td>
<td>2.182</td>
</tr>
<tr>
<td>2</td>
<td>with 1 Man</td>
<td>1565.676</td>
<td>3.044</td>
</tr>
<tr>
<td>3</td>
<td>with 3 Men</td>
<td>2012.031</td>
<td>3.925</td>
</tr>
<tr>
<td>4</td>
<td>with 4 Men</td>
<td>2458.386</td>
<td>5.054</td>
</tr>
<tr>
<td>5</td>
<td>with 5 Men</td>
<td>2904.741</td>
<td>5.655</td>
</tr>
<tr>
<td>6</td>
<td>with 6 Men</td>
<td>3351.096</td>
<td>6.532</td>
</tr>
<tr>
<td>7</td>
<td>with 7 Men</td>
<td>3797.451</td>
<td>7.398</td>
</tr>
</tbody>
</table>

The Stresses induced in the 4-Spokes Mg Alloy wheel 7.686 MPa is less as compared with the Stresses induced in the 5-Spokes Al alloy (AM60A), and also nearer to Al-alloys with 6 spokes. So in the 4 spoke model can substitute to the 6 or 5 spoke wheels safely.

**CONCLUSION**
The objective was to reduce the weight of the alloy wheel has been achieved. The current design is 60% lighter than the original design. What more can be done to reduce the weight. In this work 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. And also having higher FOS in the four spoke model design.

**SCOPE FOR FUTURE WORK:**
1) Further to do optimization of material thickness to reduce the material consumption.
2) Further to improve life of component by using advanced fatigue strain life approach.
REFERENCES


[5]. Wei-Chan Chang (2008), —Modeling For Impact Test Of Aluminum Wheelsl, Department Of Mechanical Engineering National Central University Jhong-Li 32001, Taiwan, R.O.C


[8]. Mohd Zul Hazml Bin Mhd Fauzy (2008), —Finite Element Analysis Of Car Alloy Wheell, Thesis In Universiti


[12]. Alloy Rim Cracking And Failure Department Of Natural Resources And Mines, Safety Alert No. 287 | 25 May 2012.