Abstract: The purpose of the car wheel rim is to provide a firm base on which the tire could be fitted. 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. The model is created in Catia and then it is imported in Ansys workbench through IGES file. The finite element idealization of this model was then produced using the tetrahedron solid element.

The given car rim is made up of Al-6061 which is having precipitation aluminum alloy containing magnesium and silicon as its major alloying elements and this is subjected to different boundary conditions. In this project our aim is to carry out the different analysis such as static structural analysis, modal analysis and harmonic analysis. In static structural analysis we found out equivalent von mises stresses, deformation and maximum principal stresses and in the modal analysis we checked the modes shapes of the car rim and further we found out the harmonics stresses, acceleration response, deformation response etc. Later we compared the induced stresses are less than the allowable stresses of the material.

Keywords: Finite element analysis, Ansys Workbench, Modal, Harmonic

1. INTRODUCTION
Wheel is generally composed of rim and disc. Rim is a part where the tire is installed. Disc is a part of the rim where it is fixed to the axle hub. The rim of a wheel is the outer circular design of the metal on which the inside edge of the tire is mounted on vehicles such as automobiles. Wheels rim should not fail during its working. From design point of view, the strength and fatigue life of rim are critical issues. In order to reduce costs, design for light-weight, wheel rim made with materials like,
- aluminum alloy
- steel alloy
- forged steel
- magnesium alloy

The given car rim is made up of aluminum alloy and it is subjected to forces like the pressure of air inside the tire, weight of the car acting through the center. here project aim is to carry out the different analysis such as static structural analysis, modal analysis and harmonic analysis. Static structural analysis to find equivalent von mises stresses, deformation and maximum principal stresses and in the modal analysis to check the modes shapes of the car rim and the natural frequencies, further harmonic stresses, acceleration response, deformation response etc. are to analyze. The whole project formed by using design and simulation software’s along with finite element analysis.

Properties of Material
Car rim volume is 0.003789 m³
Car rim mass is 10.765 kg

<table>
<thead>
<tr>
<th>Material</th>
<th>Density (ρ)</th>
<th>Young’s modulus (E)</th>
<th>Tensile strength (σt)</th>
<th>Elongation (ε) at break</th>
<th>Poisson’s ratio (ν)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A6061</td>
<td>2.70 g/cm³</td>
<td>68.9 GPa (9,990 ksi)</td>
<td>124–290 MPa (18.0–42.1 ksi)</td>
<td>12–25%</td>
<td>0.33</td>
</tr>
</tbody>
</table>

Table 1: Material Properties
MatWeb is a online material information resource Over 1700 Aluminum alloy entries are listed in MatWeb

2. LITERATURE SURVEY

K. Venkateswara Rao, Dr. T. Dharmaraju [1]

The author carried out the analysis on wheel rim for different alloys such as steel alloy, aluminum alloy, magnesium alloy and forged steels and found out steel alloy is having higher strength and sustaining more number of cycle and Further he spokes about the optimization of wheel rim by reducing the thickness of a wheel rim.

the stress distribution and fatigue wheel life of wheel rim was summarized on performing more number of iterations on all four different materials. Observed that the aluminium alloy shows the better result in case of number of cycles to failure, compared to all other three materials and the steel alloy is the poor material respectively. Concluded that on optimizing the material thickness, by reducing the material consumption the material life to be extended.

Karan Valetava, Param Jethava [2]

Here the author focused on structural behavior and the fatigue life of the rim. The analysis is made for two different materials that is aluminium alloy (A356.2) and carbon fiber. The model is created in CATIA and analysis is carried out in ANSYS. Observed the material behavior in static condition, on application of the load, the deformation, alternative stresses and principal stresses were noted.in fatigue analysis based on the life, safety factor and damage of spoke wheel were analyzed. Concluded with comparative statement that carbon fiber is suggested for the better design in both the cases.

N. Satyanarayana [3]

The author has carried out the analysis on aluminum wheel under the radial load conditions and found out the total deformation of wheel is 0.2833mm. The stresses developed due to the different boundary conditions are 163MPa and 0.038MPa. which is lower than the yield strength of aluminium alloy, the maximum and minimum life of the wheel to be observed at the cross sectional area of the wheel by means of fatigue test that is 1.7667x106 cycles. Concluded with the safe design of rim on comparing with the strength of material considered.

Siva Prasad [4]

Siva Prasad does stress and harmonic analysis of car wheel rim by using ANSYS. This paper explains the structural behavior of the car rim under the several operating loading conditions. 21.3KPa is applied on the circumference of the wheel rim, harmonic response to be observed in between 170Hz to 420Hz.

Concluded with the better results as the design is safe with all aspects and suggested the aluminium alloy as the suitable material for the rim for good results.

Sourav Das [5]

This paper is carried out by giving more importance to the optimization of the rim mass. The detail dimensional parameters are to be considered for the better results, on varying the mass of rim, conducted so many iterations to obtain the better life to the rim. Observed with two different loads along the axis of the bolt, 1) radial load at $0^\circ$ from the bolt axis and 2) radial load at $36^\circ$ from the bolt. Concluded that on application of load inclined to the rim the deformation will be reduced.

Fig.1: Wheel Rim Dimensions
3. CAD MODEL OF A CAR RIM

Fig. 2: Finite Element representation

Fig. 3: The detailed 2D drawing of a car wheel rim
The given model is discretized into 127004 nodes and 68880 elements in ANSYS workbench. The tetrahedral and hex elements are used in a meshing. To get results at any point in the model meshing is required. If the number of elements increases, the time required for solving will also increase.

**BOUNDARY CONDITIONS**

- The total mass of vehicle on a rim is 2000 kg which is distributed over four wheels
- Each wheel carrying the load = Total mass / 4 = 500 kg.
- Total force on each wheel = 500 * 9.81 = 4905 N.
- The maximum pressure allowed in a car rim = 40 psi = 40 * 0.00689 = 0.275 MPa
RESULTS AND DISCUSSIONS

4. RESULTS AND DISCUSSIONS

Static structural analysis

a) Equivalence von mises stress

The maximum equivalent stress induced at the outer circumference of the rim with the magnitude of 23.094 MPa for the given boundary condition. The yield stress of Al 6061 is 240 MPa. The induced stress is very less as compared to yield stress of the material. The model can withstand the given load condition.
b) **Maximum Principal Stress**

![Maximum Principal Stress Diagram]

Fig. 9: Max principal stress of 23.152 MPa is observed which is a singular stress @ Bolt Location

The maximum principal stress induced at the outer circumference of the rim with the magnitude of 23.152 MPa for the given boundary condition. The yield stress of Al 6061 is 240 MPa. The induced stress is very less as compared to yield stress of the material. The model can withstand the given load condition.

c) **Minimum Principal Stress**

![Minimum Principal Stress Diagram]

Fig 10: Minimum principal stress

The minimum principal stress induced at the outer circumference of the rim with the magnitude of -23.511 MPa for the given boundary condition. The yield stress of Al 6061 is 240 MPa. The induced stress is less as compared to yield stress of the material. The model can withstand the given load condition.

d) **Displacement**

![Displacement Diagram]

Fig. 11: Total deformation observed in the rim is 0.14mm
The maximum deformation induced at the outer circumference of a rim is with the magnitude of 0.143mm. The percentage of elongation is 12-25% which is very much greater than the induced percentage of elongation. So the model will remain safe with induced deformation of the rim.

e) Directional deformation

The maximum deformation induced in the x direction is with the magnitude of 0.05748mm. The percentage of elongation is 12-25% which is very much greater than the induced percentage of elongation. So the model will remain safe with induced deformation of the rim.

f) Equivalent elastic strain

The equivalent elastic strain induced in the given model is $33.48 \times 10^{-5}$ for the given loading condition. It is observed that elastic strain is more at the center of an outer circumferences compared to the both sides.

g) Shear stress

The maximum shear stress induced at the middle part of outer circumference of the rim with the magnitude of 9.03MPa for the given boundary condition. The model will remain safe under the given loading condition.
Modal analysis

1) **Mode-1 System level Twisting mode along Z axis @ 291.59 Hz**

![Image](image1.png)

Fig. 15: System level Twisting mode along Z axis @ 291.59 Hz

The maximum deformation produced at the outer circumference of the rim with the magnitude of 30.967 for the given boundary condition at 291.59 Hz frequency.

2) **Mode 2 System level Bending mode along Z axis @ 291.78 Hz**

![Image](image2.png)

Fig. 16: System level Bending mode along Z axis @ 291.78 Hz

The maximum deformation produced at the outer circumference of the rim with the magnitude of 32.005 mm for the given boundary condition at 291.78 Hz frequency.

3) **Mode-3 System level Bending mode along Z axis @ 442.15 Hz**

![Image](image3.png)

Fig. 17: System level Bending mode along Z axis @ 442.15 Hz
The maximum deformation produced at the outer circumference of the rim with the magnitude of 23.63mm for the given boundary condition at 442.7Hz frequency.

4) **Mode -4 System level Bending mode along Z axis @ 442.72 Hz**

![Mode-4 System level Bending mode along Z axis @ 442.72 Hz](Fig.18: System level Bending mode along Z axis @ 442.72 Hz)

The maximum deformation produced at the outer circumference of the rim with the magnitude of 23.63mm for the given boundary condition at 442.7Hz frequency.

**Harmonic Response Analysis**

- Load is applied using Base Excitation method
- 3G load (29410mm/s$^2$) is applied at fixed support
- Structural damping of 2% is applied
- Analysis sweeps between 0-150Hz

![Harmonic Response](Fig.19: Bolts location is constrained in all DOF 3G load is applied at constrained location.)

- Acceleration levels at a different frequency
The Acceleration for initial certain iteration was nearly linear for the step of frequency from 0 to $400\text{mm/s}^2$ and suddenly raised to $409.86\text{ Hz}$ between the range of 400 to 500 Hz and later on it got reduced to low acceleration.

- deformation response
The Deformation for initial certain iteration was nearby linear for the step of frequency from 0 to 400 mm/s² and suddenly raised to 442.9 Hz between the range of 400 to 500 Hz and later on it got reduced to low deformation.

$$\text{Table 3: Frequency vs Deformation Response}$$

<table>
<thead>
<tr>
<th>Frequency [Hz]</th>
<th>Amplitude [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>2.347e-007</td>
</tr>
<tr>
<td>271.55</td>
<td>2.9918e-007</td>
</tr>
<tr>
<td>289.99</td>
<td>3.059e-007</td>
</tr>
<tr>
<td>291.44</td>
<td>3.0287e-007</td>
</tr>
<tr>
<td>291.56</td>
<td>3.0259e-007</td>
</tr>
<tr>
<td>291.68</td>
<td>3.0231e-007</td>
</tr>
<tr>
<td>291.69</td>
<td>3.0229e-007</td>
</tr>
<tr>
<td>291.69</td>
<td>3.0227e-007</td>
</tr>
<tr>
<td>291.81</td>
<td>3.0199e-007</td>
</tr>
<tr>
<td>291.93</td>
<td>3.0171e-007</td>
</tr>
<tr>
<td>293.39</td>
<td>2.9841e-007</td>
</tr>
<tr>
<td>313.31</td>
<td>2.9918e-007</td>
</tr>
<tr>
<td>366.98</td>
<td>1.8617e-007</td>
</tr>
<tr>
<td>411.81</td>
<td>2.1889e-006</td>
</tr>
<tr>
<td>439.76</td>
<td>4.6692e-005</td>
</tr>
<tr>
<td>441.97</td>
<td>5.2311e-005</td>
</tr>
<tr>
<td>442.15</td>
<td>5.2517e-005</td>
</tr>
<tr>
<td>442.33</td>
<td>5.2682e-005</td>
</tr>
<tr>
<td>442.44</td>
<td>5.2762e-005</td>
</tr>
<tr>
<td>442.54</td>
<td>5.2827e-005</td>
</tr>
<tr>
<td>442.72</td>
<td>5.2898e-005</td>
</tr>
<tr>
<td>442.9</td>
<td>5.2927e-005</td>
</tr>
<tr>
<td>445.12</td>
<td>4.9971e-005</td>
</tr>
<tr>
<td>475.34</td>
<td>5.2579e-006</td>
</tr>
<tr>
<td>481.57</td>
<td>4.0276e-006</td>
</tr>
<tr>
<td>484.71</td>
<td>3.5791e-006</td>
</tr>
<tr>
<td>517.61</td>
<td>1.4947e-006</td>
</tr>
</tbody>
</table>
HARMONIC STRESSES

a) Von-Mises Stress

Fig. 20: Max equivalent stress of 22.411 MPa is observed.

The maximum harmonic stresses induced in a given car rim is 22.41 MPa. The location of a maximum equivalent stress is near the center of a rim. The material yield stress limit is very much greater than induced stresses. The stresses we found out at primary mode frequency @ 442Hz.

b) Maximum Principal Stress

Fig. 21: Max principal stress of 18.494 MPa is observed.

The maximum harmonic stresses induced in a given car rim is 18.494 MPa. The location of a maximum equivalent stress is near the center of a rim. The material yield stress limit is very much greater than induced stresses. The stresses we found out at primary mode frequency @ 442Hz.

c) Maximum shear stress

Fig. 22: Max shear stress of 11.497 MPa is observed.
The maximum harmonic stresses induced in a given car rim is 11.497 MPa. The location of a maximum equivalent stress is near the center of a rim. The material yield stress limit is very much greater than induced stresses. The stresses we found out at primary mode frequency @ 442Hz.

RESULT SUMMARY

<table>
<thead>
<tr>
<th>SL.No</th>
<th>Type of stress</th>
<th>Structural</th>
<th>Harmonic</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Equivalent von mises stresses</td>
<td>23.094MPa</td>
<td>22.411MPa</td>
</tr>
<tr>
<td>2.</td>
<td>Maximum principal stress</td>
<td>23.069MPa</td>
<td>18.494MPa</td>
</tr>
<tr>
<td>3.</td>
<td>Minimum shear stresses</td>
<td>9.03MPa</td>
<td>11.49MPa</td>
</tr>
</tbody>
</table>

It is clearly observed from the above results all the induced stresses in static structural and harmonic analysis in given loading conditions are within the yield stress of Al6061 (car rim material). The yield stress of the Al6061 is 240MPa.

Table 3: static and harmonic results

<table>
<thead>
<tr>
<th>Mode</th>
<th>Natural frequency(Hz)</th>
<th>Max. deformation(mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>291.59</td>
<td>30.96</td>
</tr>
<tr>
<td>2</td>
<td>291.78</td>
<td>32.005</td>
</tr>
<tr>
<td>3</td>
<td>442.15</td>
<td>23.65</td>
</tr>
<tr>
<td>4</td>
<td>442.78</td>
<td>23.63</td>
</tr>
</tbody>
</table>

Table 4: Modal analysis results

5. CONCLUSION

In static structural analysis we found out the equivalent von mises stresses, maximum principal stresses and maximum shear stresses and those all stresses are well beyond the yield stress of the material. In modal analysis we found out the all modes shape of the given model. In the harmonic analysis we studied the dynamic characteristics like acceleration response and deformation response and found out the harmonic stresses which are well beyond the strength of the material. From above all results it is clear that the given car rim is safe under the loading conditions.

REFERENCES