ABSTRACT
The requirements for improved stiffness, reliability, fatigue life and increased efficiency involves challenges of developing innovative design solutions. The present work mainly focus on the design of car alloy wheel, where the analytical and FEM analysis approach was implemented to analyze baseline design. Initially static analysis was performed to obtain total deformation, strain and the stress of car alloy wheel. Three Dimensional model was created using CATIA while it was discretized using ANSYS to perform post processing analysis for obtaining expected solution. The results were obtained through linear static analysis in terms of Total deformation, Minimum principal stress, Max Principal stress on 4 arms wheel and later dynamic analysis (modal analysis) was been done to obtain different modes with different frequency for 4 arms wheel.

KEYWORDS: Alloy wheel, Principal stress, Deformation & dynamic analysis.

INTRODUCTION
Wheels have been made, using various casting techniques such as sand casting, gravity die casting, centrifugal casting, low and low pressure die casting. Sand and gravity casting are less controllable operations and having problem with blowholes and shrinkages. Aluminum wheels should not fail during service as their strength and fatigue life are critical. In order to reduce costs, design for light-weight, and limited life is increasing being used for all vehicle component. In the actual, product development automobile wheels have complicated geometry and must satisfy manifold design criteria, such as style, weight, manufacturability, and performance. In addition to a fascinating wheel style, wheel design also needs to accomplish a lot of engineering objectives including some required performance and durability requirements. The present research work is focused on the automobile sector, specifically on the car alloy wheel rim design to improve the quality of the wheel by evaluating the fatigue life, structural integrity, over speed & burst speed margin and to reduce cost and weight reduction & ease of replacing. Design modifications of the existing alloy wheel rim include conversion of elliptical cross section in to an rectangular cross section for a good overall outlook and style. The linear and bilinear static structural analysis is also imparted for higher efficiency and longer life. The pressure distribution about the rim surface is to be maintained at 32psi and load consideration on the rim when this pressure decreases below 32psi will be more on the bolt and bolt holes. Over speed & burst speed margin is the limit where in the rim should withstand the stress & strain on the rim which is under operational condition and tough road condition. The tendency of a material is to break, under repeated cyclic loading at a stress considered less than the tensile strength in a static test. Fatigue cracks can terminate the usefulness of a structure or component by more ways than just fracture.
GEOMETRY OF 4 ARM ALLOY WHEEL

2D Drawing   Front View                                           Side view

Figure 1: 2D Diagram of the 4 arms wheel.

3D MODEL                                          MODEL WITH MESH

Figure 2: 3D Diagram of the 4 arms wheel and with mesh.

The model of 4 arms wheel as shown in fig 1 was designed in unigraphics and later loaded in to ansys 14 to perform linear static structural analysis. The meshing was been done for the 4 arms wheel.

Linear stress, strain and deformation of a 4 arms wheel by linear static structural analysis

Equivalent stress                                          Maximum principal stress

Figure 3: Equivalent stress and Maximum principal stress for 4 arms wheel in linear stress analysis
LINEAR STATIC STRUCTURAL ANALYSIS FOR 4 ARMS WHEEL

The model of 4 arms wheel as shown in fig 1 was done in unigraphics and later loaded in to ansys 14 to perform linear static structural analysis. The meshing was done for the 4 arms wheel as shown in figure 2. The boundary conditions were applied on the rim surface of the aluminium wheel with pressure of 0.25 Mpa, rotational velocity of 222.2 rds or 2122 rpm and bolt pretension of 23340 N. After applying the boundary conditions following results were obtained:
1. Von-mises stress= 67.814 Mpa
2. Maximum principal stress= 101.79 Mpa
3. Minimum principal stress=31.913 Mpa
4. Total Deformation = 0.146868 mm

Bi-linear results of stress, strain and deformation of a 4 arms alloy wheel by linear static structural analysis.

Figure 4: minimum principal stress and Total deformation for 4 arms wheel in linear stress analysis.

Figure 5: Equivalent stress and Maximum principal stress for 4 arms wheel in linear stress analysis

Minimum principal stress

Total deformation
BI-LINEAR STATIC STRUCTURAL ANALYSIS FOR 4 ARMS WHEEL

Nonlinear stress analysis calculates the stresses and deformations of products under the most general loading and material conditions for:
1. Dynamic (time dependent) loads.
2. Large component deformations.
3. Nonlinear materials, such as rubber or metals, beyond their yield point.

Nonlinear analysis is a more complex approach, but results in a more accurate solution than linear analysis, if the basic assumptions of a linear analysis are violated. If the linear analysis assumptions are not violated, then the results of a linear and nonlinear analysis will be the same.

While performing the nonlinear analysis, the time component is important both in controlling the loading (individual load components can be active at different times) and in capturing the response to an impulse load of impact. SOLIDWORKS Simulation provides either an automatic or a manual time control method with a force, displacement, or arc length convergence control. You get power and flexibility to solve challenging and complex simulation problems simply in a straightforward manner.

The model of 4 arms wheel as shown in fig 6 was done in unigraphics and later loaded in to ansys 14 for Bi-linear static structural analysis for further process. The boundary conditions were applied on the rim surface of the aluminium wheel with pressure of 0.25 Mpa, rotational velocity of 222.2 rds or 2122 rpm and bolt pretension of 23340 N. After applying the boundary conditions following results were obtained:
1. Von-mises stress= 305.73 Mpa
2. Maximum principal stress= 317.36 Mpa
3. Minimum principal stress=35.415 Mpa
4. Total Deformation = 0.45735 mm

DYNAMIC ANALYSIS OR MODEL ANALYSIS

<table>
<thead>
<tr>
<th>SL No</th>
<th>mode</th>
<th>Frequency (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.</td>
<td>252.4</td>
</tr>
<tr>
<td>2</td>
<td>2.</td>
<td>252.63</td>
</tr>
<tr>
<td>3</td>
<td>3.</td>
<td>394.5</td>
</tr>
<tr>
<td>4</td>
<td>4.</td>
<td>415.87</td>
</tr>
<tr>
<td>5</td>
<td>5.</td>
<td>449.42</td>
</tr>
<tr>
<td>6</td>
<td>6.</td>
<td>522.66</td>
</tr>
</tbody>
</table>

Mode 1

Mode 2
Dynamic analysis is the testing and evaluation of a program by executing data in real time as its main objective is to find errors in a program while it is running, rather than by repeatedly examining the code offline.

The modal analysis was carried out on the 4 arm rim wheel under different modes and frequency as shown in above figures. The maximum deformation was observed to be 27.553 mm for mode 1 with frequency 252.4 Hz. Similarly for mode 2 deformation 27.554 mm with frequency 252.63 Hz, for mode 3 deformation 35.86 mm with frequency 394.5 Hz, for mode 4 deformation 31.45 mm with frequency 449.42 Hz, for mode 5 deformation 25.86 mm with frequency 415.87 Hz, and for mode 6 deformation 22.55 mm with frequency 522.66 Hz.
frequency 394.5 Hz, for mode 4 deformation 35.069mm with frequency 415.87 Hz, for deformation 18.799mm with frequency 449.42 Hz were obtained.

RESULT AND DISCUSSION
COMPARISON BETWEEN LINEAR AND BI LINEAR VALUES OF THE 4 ARMS WHEEL

Table 2: comparison between linear and bi linear values of the 4 arms wheel.

<table>
<thead>
<tr>
<th></th>
<th>Von mises stress in Mpa</th>
<th>Maximum principal stress in Mpa</th>
<th>Total deformation in mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear analysis</td>
<td>67.814</td>
<td>101.79</td>
<td>0.14686</td>
</tr>
<tr>
<td>Bi linear analysis</td>
<td>305.73</td>
<td>317.36</td>
<td>0.45735</td>
</tr>
</tbody>
</table>

It was observed that the von mises stress obtained from linear analysis for 4 arms wheel was less compared with the results obtained from bilinear analysis. The reason for high von mises stress in bi linear is because of high pressure and high RPM. It was also observed that the stress obtained in linear analysis were within the yield stress for aluminum. Hence the design is safe.

Graph 1: linear and nonlinear stress values

CONCLUSION
1. Static linear analysis was carried out and maximum equivalent stress is 67.814 Mpa which was less than the yield stress of a given material and for applied load, hence the Design is safe.
2. From the above graph it was observed that the max yield stress obtained was within the limit 260 Mpa above which it will yield for failure.
3. Dynamic model analysis was carried out to find six initial modes and natural frequency of the car alloy wheel.

REFERENCES