ABSTRACT

In the current scenario automobile engineering and transportation vehicles require safety and energy efficiency. Articulated vehicle is vehicle which has a permanent or semi-permanent pivoting joint in its construction allowing the vehicle to turn more sharply. These vehicles are always used for transportation of heavy load, so bear this heavy load chassis of vehicle plays important role. The purpose of this work is to simulate and forecast the structural response of the trailer of articulated vehicle, in terms of stress and displacement, under constraining conditions, which aim at reflecting the actual duty cycle of the vehicle. This study aims to investigate the critical points of stresses that lead to or induce failure. To analyze, finite element method (FEM) is used for modelling and stress analysis. Finite element analysis of trailer frame revealed the stress distribution on trailer frame. Modifications have been made to current trailer frame which leads to optimization of trailer. These modifications or optimization leads to optimization of weight of trailer along with reduction in stress values leading to safe design.

Keywords — articulated vehicle, finite element analysis (FEM), optimization, trailer frame.

INTRODUCTION

Trailers are widely used for transporting, building construction material, industrial equipment and agriculture product from one place to another. The power required to pull the trailer is provided by human, animals and machines. In trailer many varieties are available but use of particular trailer depends upon the application. The main requirements of trailer manufacturing are high performance, easy to maintains, longer working life and robust construction. Nowadays, transportation industry plays a major role in the economy of modern industrialized and developing countries. The goods and materials carried through heavy trucks are dramatically increasing. There are many aspects to consider when designing a heavy trucks chassis trailer frame, including component packaging, material selection, strength, stiffness and weight. Chassis of trailer frame is one of the important parts that used in automotive industry. This structure is the bigger component in the vehicle as shown in Figure 1.

Fig.1 Chassis of heavy duty truck[1].

The main function of the chassis trailer frame is not only support the components but also payload mounted upon it. The trailer frame is subjected to stress, bending moment and vibrations due to road roughness, weather and components that mounted on it, when the truck travels along the road,
stress that acting on chassis is varies with the displacement and each part on the chassis. The major challenge in today’s ground vehicle industry is to overcome the increasing demands for higher performance, lower weight in order to satisfy fuel economy requirements, and longer life of components, all this at a reasonable cost and in a short period of time also give new safety requirements.

I. BASIC CALCULATION
Length of trailer=8000mm.
Width of trailer=3000mm.
Young’s modulus=2.1×10^5 N/mm^2.
Poisson’s ratio=0.3
Payload = 20ton= 20000×9.81=196200 N.

![Fig.2 Wire frame CAD model of trailer with dimension.](image)

II. MOELLING & MESHING OF TRAILER FRAME.
For carrying out FE analysis of frame the CAD model is prepared and meshing is done in HYPERMESH software. HYPERMESH software from altair is used to mesh the solid model. To mesh the geometry use the surface first and then solid. While meshing parameters like minimum element size, aspect ratio, number of elements has been maintained.

![Fig.3 Solid CAD model of trailer.](image)

Element type used: Hexahedron; Number of elements generated: 60784; Number of nodes generated: 77390.

![Fig.4 Meshing of trailer frame.](image)

III. FEA RESULTS
Load is applied in the form of pressure which is shown in above figure 4 in which load is distributed on number of elements.

![Fig.5 Load applied on trailer frame.](image)

IV. RESULTS WITHOUT MODIFICATION
By performing analysis maximum deflection of trailer frame is 4.388 mm and von mises stress is 21.833 Mpa. Results are shown in fig.6 and fig.7

![Fig.6 Deformation of trailer frame.](image)

![Fig.7 Von mises stress of trailer.](image)

VI. MODIFIED CAD MODEL OF TRAILER FRAME.
Different practises are available for optimization by trailer frame modification. Here suitable changes are made in design by reducing the thickness of trailer frame and some
extra ribs or stiffeners are introduced in bottom region of trailer to get some support while loading. Modified CAD model is shown in figure 8 and figure 9.

VII. MODIFIED MESHED MODEL OF TRAILER.
As thickness of trailer is reduced for optimization, so dense meshing is done in the region where load is applied and it is checked by doing analysis. Number of elements generated after medication: 40184; Number of nodes generated after optimization: 58408.

VIII. FEA RESULTS AFTER MODIFICATIONS
By performing analysis maximum deflection of trailer frame is reduced from 4.388 mm to 0.931 mm and von mises stress is reduced from 21.833 Mpa to 20.52 Mpa. Results are shown in fig.12 and fig.13

VIII. COMPARISON OF RESULTS FOR WEIGHT OPTIMIZATION
Results of trailer frame before modification and after modification are as shown in table I. Here result shows that by applying suitable modifications in trailer frame weight of trailer frame is reduced from 77558 kg to 50311 kg which is desirable.

<table>
<thead>
<tr>
<th>Sr.No.</th>
<th>Title</th>
<th>No. of elements</th>
<th>No. of nodes</th>
<th>Total mass in kg</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Before</td>
<td>60784</td>
<td>7739</td>
<td>77558</td>
</tr>
</tbody>
</table>
After optimization

<table>
<thead>
<tr>
<th></th>
<th>optimization</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>After</td>
<td>40184</td>
</tr>
</tbody>
</table>

IX CONCLUSION

Finally static structural analysis before modification and after modification has been done in this work. Aim of research has been successfully completed by doing suitable modification in the trailer. Modified trailer frame is desirable as its von misses stresses and deflections are under control which satisfies the design.

ACKNOWLEDGMENT

I am very thankful to my project guide Prof. Dinesh G. Joshi, Assistant professor, Mechanical Engineering Department, RMD Sinhgad School of Engineering, Warje Pune, for his continuous support and encouragement in completing this work.

REFERENCES


