STRUCTURAL ANALYSIS OF TRUCK CHASSIS USING FINITE ELEMENT METHOD

Vinayak R. Tayade*, Avinash V. Patil

*Pursuing M.E Department of Mechanical Engineering, S S G B COE&T Bhusawal, India.
Head of Department Mechanical Engineering, S S G B COE&T Bhusawal, India

ABSTRACT
This paper aims to redesign a modified chassis for truck by keeping the material and dimension similar and using ‘I’ cross section area instead of ‘C’ resulted in more safer stresses than ‘C’ and the material used is mild steel. Automotive chassis is an important part of an automobile. The chassis serves as a frame work for supporting the body and different parts of the automobile. Also, it should be rigid enough to withstand the shock, twist, vibration and other stresses. Along with strength, an important consideration in chassis design is to have adequate bending stiffness for better handling characteristics. So, maximum stress, maximum equilateral stress and deflection are important criteria for the design of the chassis. FEA is done on the modeled chassis using the ANSYS Workbench11; the FEA is carryout for static, dynamic and shape optimization.

KEYWORDS: Stress Analysis, Finite Element analysis, truck chassis

INTRODUCTION
Truck chassis is a major component in a vehicle system. This work involves static and dynamics analysis to determine the key characteristics of a chassis. The static characteristics include identifying location of high stress area and determining the torsion stiffness of the chassis. The dynamic characteristics of chassis such as the natural frequency and mode shape were determined by using finite element (FE) method. [1,2] Automotive chassis is an important part of an automobile. The chassis serves as a frame work for supporting the body and different parts of the automobile. Also, it should be rigid enough to withstand the shock, twist, vibration and other stresses. Along with strength, an important consideration in chassis design is to have adequate bending stiffness for better handling characteristics. So, maximum stress, maximum equilateral stress and deflection are important criteria for the design of the chassis.[2] The major challenge in today’s ground vehicle industry is to overcome the increasing demands for higher performance, lower weight, and longer life of components, all this at a reasonable cost and in a short period of time. The chassis of trucks is the backbone of vehicles and integrates the main truck component systems such as the axles, suspension, power train, cab and trailer. Since the truck chassis is a major component in the vehicle system, it is often identified for refinement. [3]
STRUCTURAL ANALYSIS OF CHASSIS

What Is Structural Analysis?
Structural analysis is the prediction of the performance of a given structure under prescribed loads and/or other external effects, such as support movements and temperature changes. The performance characteristics commonly of interest in the design of structures are:

1. Stresses or stress resultants, such as axial forces, shear forces, and bending moments
2. Deflections; and

ANSYS as Tool for Structural Analysis
The ANSYS structural analysis software suite is trusted by organizations around the world to rapidly solve complex structural engineering problems with ease. FEA analysis (finite element) tools from ANSYS provide the ability to simulate every structural aspect of a product:

- Linear static analysis that simply provides stresses or deformations
- Modal analysis that determines vibration characteristics
- Advanced transient nonlinear phenomena involving dynamic effects and complex behaviors

All users, from designers to advanced experts, can benefit from ANSYS structural analysis software. The fidelity of the results is achieved through the wide variety of material models available, the quality of the elements library, the robustness of the solution algorithms, and the ability to model every product — from single parts to very complex assemblies with hundreds of components interacting through contacts or relative motions.

ANSYS FEA tools also offer unparalleled ease of use to help product developers focus on the most important part of the simulation process: understanding the results and the impact of design variations on the model. We have performed structural analysis of Chassis with the help of ANSYS Workbench 11.0 software which is FEM tool.

FINITE ELEMENT ANALYSIS OF CHASSIS

Analysis of existing ‘C’ cross section chassis using ANSYS tool
The existing chassis geometry is generated in ANSYS workbench 11 by selecting toolbox where various commands like draw, dimensioning, constraints, extrude, generate, rotate etc. are used. Then mesh is generated on the model and after that load points are defined and load values are given. Then the results are generated automatically for stresses and deformation in solution phase. Figure shows the ANSYS results for Mesh Generated, Total Deformation, Equivalent (von misses) Stress, Shear Stress, Normal Stress, Biaxiality Indication, Factor of Safety.

Fig. Force Diagram
Fig. Shear Stresses

Fig. Normal Stresses

Fig. Biaxiliary indication
Analysis of modified ‘I’ cross section chassis using ANSYS tool

Fig. Factor of Safety

Fig. Force Diagram

Fig. Mesh Diagram
Fig. Total Deformation

Fig. Equivalent (Von-Mises) Stresses

Fig. Shear Stress

http://www.ijesrt.com

© International Journal of Engineering Sciences & Research Technology

[726]
RESULTS AND DISCUSSION

Table 1. Comparison between Existing Chassis and Proposed Chassis Using ANSYS

<table>
<thead>
<tr>
<th>SR NO</th>
<th>FACTORS</th>
<th>EXISTING ‘C’ SECTION</th>
<th>SUGGESTED ‘I’ SECTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Equivalent (Von Mises) Stress</td>
<td>108.11 MPa</td>
<td>88.642 MPa</td>
</tr>
<tr>
<td>2</td>
<td>Total Deformation</td>
<td>2.5090 m</td>
<td>1.7199 m</td>
</tr>
<tr>
<td>3</td>
<td>Shear Stress</td>
<td>37.66 MPa</td>
<td>17.424 MPa</td>
</tr>
<tr>
<td>4</td>
<td>Normal Stress</td>
<td>65.518 MPa</td>
<td>40.575 MPa</td>
</tr>
<tr>
<td>5</td>
<td>Biaxiality Indication</td>
<td>0.93667</td>
<td>0.96354</td>
</tr>
<tr>
<td>6</td>
<td>Factor Of Safety</td>
<td>0.7973 to 5</td>
<td>0.9724 to 10</td>
</tr>
</tbody>
</table>

CONCLUSION
1. More Safer stresses are obtained in new suggested design.
2. Increase in Factor of Safety obtained in new suggested design.
3. The newly designed ‘I’ section Chassis reduces of weight as compared to the existing ‘C’ section Chassis As raw material required is reduced, reduction in cost is achieved.

ACKNOWLEDGEMENTS
We feel great pleasure in expressing our deepest sense of gratitude and sincere thanks our HOD Prof.A.V.Patil for his valuable guidance during the research work, without which it would have been very difficult task. We have no words to express our sincere thanks for valuable guidance, extreme assistance and co-operation extended to all the Staff Members of our department.

REFERENCES