Design and Development of Three Wheeler Chassis

#1 Aditya V. Sahasrabudhe, #2 Rajesh V. Patil
1sahasrabudhe.aditya@gmail.com
rvpatil_sits@singhagad.edu
#12 Department of Mechanical Engineering, Sinhgad Institute of Technology and Science, S. P. Pune University, Pune, India

ABSTRACT
This project work aims to optimize the three wheeler chassis to achieve weight and cost reduction by performing stress analysis. Material used for the structure is cold rolled steel IS513. Modeling and Finite Element Analysis is performed using software packages like Creo/ProE and Altair HyperWorks respectively. Firstly the 3D model of existing three wheeler chassis is studied. Values of stress and displacement are determined by performing linear static analysis. Optimization of the chassis is limited to the front part, keeping the rear part of the chassis same. Analysis of the optimized chassis is performed to obtain stress and displacement plots. Computed results for optimized chassis are then compared with the existing chassis where it is found that the optimized chassis is safe and a total of approximately 4.5 kg of weight reduction is achieved with an approximately cost saving of Rs. 300.

Keywords- three wheeler chassis, weight reduction, static analysis, Creo, Hyperworks

I. INTRODUCTION
Chassis is a French term and was initially used to denote the frame parts or Basic Structure of the vehicle. It is the backbone of the vehicle. A vehicle without body is called Chassis. The components of the vehicle like Power plant, Transmission System, Axles, Wheels and Tires, Suspension, Controlling Systems like Braking, Steering etc., and also electrical system parts are mounted on the Chassis frame. It is the main mounting for all the components including the body. So it is also called as Carrying Unit. Chassis is also required to support the various sub systems and components of the vehicle like the engine, gearbox, clutch, frame, propeller shaft, differential, etc.

Various Loads acting on the chassis are like

- Inertia loads – During application of brakes,
- Impact loads – Due to collision of the vehicle,
- Static loads – Due to chassis and body parts,
- Overloads – Design considerations beyond capacity, etc.

Function of the chassis is as follows:

- o withstand stresses caused due to bad road condition
- o withstand forces caused due to sudden braking or acceleration
- o carry loads of passenger or goods
- o support loads of body and other parts like engine, axle, etc.

II. METHODOLOGY
The main objective of the study is to perform comparative analysis of the existing chassis with the optimized chassis. For that purpose finite element analysis is carried out to obtain the values of max. stress and deflection. Optimization of the chassis is done to achieve weight reduction and comparison of the values of stress and deflection of optimized chassis is done with the existing chassis. The overall process of the study is presented in a flowchart as shown in fig. 1:
FINITE ELEMENT ANALYSIS OF EXISTING CHASSIS

Finite Element Analysis consists of obtaining the effect of actions on all or part of the structure in order to check the ultimate limit states. Structural analysis is the determination of the effects of loads on physical structures and their components. A structure refers to a body or system of connected parts used to support a load. It is common practice to use approximate solutions of differential equations as the basis for structural analysis. This is usually done using numerical approximation techniques. The most commonly used numerical approximation in structural analysis is the Finite Element Method.

Fig. 2 shows the 3D model of the three wheeler chassis. Modeling of the chassis is done using modeling software Creo/Pro-E (version 5.0). The members of the chassis have box section.

Fig. 3: Meshed Model of Existing Three Wheeler Chassis

Meshing is nothing but dividing the mechanical model into number of small elements. Meshing of the existing model of chassis is done in hypermesh using HyperWorks software (v12.0). The process starts with creation of the mid-surface for every individual component followed by refining of topology to achieve quality mesh. The element size taken during meshing is 5 and mesh used is of mixed type. Spot welds are simulated by 1D bar elements of 5mm diameter, bolt connector are simulated by beam elements and area connectors are simulated by rod elements.

Inertia relief analysis is to be conducted on the chassis for 100% overload condition. In inertia relief analysis, the structure or the system is assumed to be in a state of static equilibrium. Since rigid body motions are restrained, conventional static analysis can be performed. Loading values are given as follows:

<table>
<thead>
<tr>
<th>Loading Position (Wheel Points)</th>
<th>Vertical Loading</th>
<th>Lateral Loading</th>
<th>Longitudinal Loading</th>
</tr>
</thead>
<tbody>
<tr>
<td>FAW</td>
<td>2578 N</td>
<td>630.5 N</td>
<td>1390 N</td>
</tr>
<tr>
<td>RAW (LH)</td>
<td>4408 N</td>
<td>3810 N</td>
<td>2831 N</td>
</tr>
<tr>
<td>RAW (RH)</td>
<td>3794.5 N</td>
<td>3164 N</td>
<td>2878 N</td>
</tr>
</tbody>
</table>

Results are obtained for the above stated loading conditions.
Figure shows the stress plot for load case of **Vertical 2G**. The maximum stress acting on the chassis is **278 MPa**.

Fig. 5: Results for Lateral Load Case

Figure shows the stress plot for **Lateral 1G** load case and the value of maximum stress acting is **253 MPa**.

Fig. 6: Results for Longitudinal Load Case

Figure shows the stress plot for **Longitudinal 1G** load case and the value of maximum stress acting is **447 MPa**.

Fig. 7: Results for LH Twist Load Case

Figure shows the stress plot for **LH Twist 2G** load case and the value of maximum stress acting is **509 MPa**.

Fig. 8: Results for RH Twist Load Case

Figure shows the stress plot for **RH Twist 2G** load case and the value of maximum stress acting is **444 MPa**.

**IV. FINITE ELEMENT ANALYSIS OF OPTIMIZED CHASSIS**

Front part of the chassis is optimized while keeping the suspension system and the rear part of the chassis as per existing model. At first, front part of the chassis is optimized for weight reduction. The optimization exercise is performed based on the theory of thin shells. Finite element analysis of the optimized chassis is to be conducted to check for the stresses acting on the chassis and also on the optimized parts at individual level. If at some locations the stresses are found to be high localized reinforcements will be added and the stress values will be reduced.

Fig. 9: 3D Model of Optimized Three Wheeler Chassis

Modelling is done in Creo/ProE modelling software package as same as existing chassis. Meshing is done in hypermesh with all the parameters of meshing as same as it is done for existing chassis. Mixed type of meshing (i.e. quad and trias elements) with an element size of 5 and spot welding is done by hex element. Bolt connectors are modelled by 1D beam elements.
Linear and isotropic material of cold rolled steel (IS 513) has been taken for the analysis purpose. The material properties used for the entire project work are same and are as follows:

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young's Modulus</td>
<td>$2.1 \times 10^5$ MPa</td>
</tr>
<tr>
<td>Poisson's Ratio</td>
<td>0.3</td>
</tr>
<tr>
<td>Density</td>
<td>7950 Kg/m$^3$</td>
</tr>
<tr>
<td>Yield Strength</td>
<td>240 MPa</td>
</tr>
</tbody>
</table>

V. RESULTS & DISCUSSION

Finite Element Analysis for three wheeler chassis has been performed for both existing and modified models using Altair HyperWorks software package. Inertia relief analysis approach is used for the analysis of three wheeler chassis and comparative analysis is done in which the values of maximum stress and displacement for all five load cases are compared. Comparative analysis is done as linear static analysis is performed on the three wheeler chassis. During
analysis, the software follows the path of straight line. In real life after crossing the yield point material follows nonlinear curve but software follows same straight line.

Hence, a comparative analysis is done and the maximum values of stress and displacement for the optimized model are compared with the existing model.

TABLE III
COMPARATIVE ANALYSIS FOR THREE WHEELER CHASSIS

<table>
<thead>
<tr>
<th>Sr. No.</th>
<th>Load Cases</th>
<th>Max. Stress</th>
<th>Max. Displacement</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Existing Model</td>
<td>Optimized Model</td>
<td>Existing Model</td>
</tr>
<tr>
<td>1</td>
<td>Vertical 2G</td>
<td>278 MPa</td>
<td>311 MPa</td>
</tr>
<tr>
<td>2</td>
<td>Lateral 1G</td>
<td>253 MPa</td>
<td>182 MPa</td>
</tr>
<tr>
<td>3</td>
<td>Longitudinal 1G</td>
<td>447 MPa</td>
<td>334 MPa</td>
</tr>
<tr>
<td>4</td>
<td>LH Twist 2G</td>
<td>509 MPa</td>
<td>494 MPa</td>
</tr>
<tr>
<td>5</td>
<td>RH Twist 2G</td>
<td>444 MPa</td>
<td>433 MPa</td>
</tr>
</tbody>
</table>

In the comparative analysis it is observed that the maximum stress value for vertical 2G load case is increased by 12% and for remaining four load cases the maximum value of stress on an average is reduced by 15%.

VI. CONCLUSION

Initial study of the existing chassis of the three wheeler vehicle is done to understand the chassis design. As the chassis is made of sheet metal, theory of thin shells is studied to understand the load carrying mechanism. Linear static analysis is performed on the existing three wheeler chassis to obtain the maximum stress plot and maximum value of displacement for comparative analysis to be done later.

A comparative study of the optimized model is done with the existing model at the component level to assess the strength by comparing the maximum value of stresses and to check the stiffness by comparing the maximum displacement. For some components analysis showed comparatively high values of stresses which are reduced by addition of reinforcements at the location of high stress.

Hence, it is concluded that
- In the comparative analysis it is observed that the maximum value of stress on an average is reduced by 15%.
- Total weight reduction achieved for the modified model is of 4.5 Kg (approx.)
- Total cost reduction for the modified model is of Rs. 300.

REFERENCES

[18] Pro-E Wildfire 5.0 User Manual Help