Design Optimization of Tractor Fuel Tank Using OptiStruct Technology

Pawan Singh  
(Manager)  
Eicher Tractors  
Mandideep- 462046  
pawan1@tmtl.co.in

Javed Qureshi  
(Jr. Manager)  
Eicher Tractors  
Mandideep - 462046  
jqureshi@tmtl.co.in


Keywords: Topography, Gauge Optimization, Fuel Tank Assembly.

Abstract

Present off-highway vehicle market demands low cost and light weight component to meet the NVH, Durability and need of cost effective vehicle with fuel efficient. This in turn gives the rise to more effective use of materials for vehicle parts which can reduce the overall weight with enhanced utility of vehicle for various applications. This paper briefs about how topology optimization technique helped to achieve this challenging objective.

In this paper, we present a methodology for designing a Plastic Fuel Tank Assy. Using OptiStruct module of HyperWorks. Compared with baseline design of FT assembly, the mass on the final design is reduced by 20%.

1. Introduction

During design of vehicle structures, it is always challenging to achieve higher stiffness and strength and simultaneously reduce weight, that is to say, to optimize the structures. There have been various types of the optimization methods that were developed and have successfully been used in the vehicle structure design.

The Fuel tank design requires high strength and stiffness. Initial analysis of bench mark design i.e. Modal and static analysis carried out on the complete assembly to extract the stress and displacement in the system. Then New Design Optimized based on

a) Topography & Gauge Optimization - Optimize for strength & weight of the plastic fuel tank with Bead pattern and optimum thickness.

b) Topology & Gauge Optimization - Optimize for weight & strength of the fuel tank mountings.

Fuel Tank Assembly 50 L Tank
Statement of problem:-

Objective:-

a) To design a 100 litre Plastic Fuel tank from available design at concept level, meeting the performance criteria that would be derived by benchmarking an existing 50L plastic fuel tank for certain load cases as listed below:

a) Internal pressure & vacuum

b) Hydrostatic pressure

c) Modal analysis

d) g loads

2. Process Methodology:-
Different Parameters for Optimization:-

Objective
To minimize the compliance index

Constraints:
- Minimum frequency requirement = 30 Hz
- Max allowable stress = 20 Mpa
- Maximum allowable strain = 0.198 mm/mm

Topography Optimization :
- Minimum bead width = 30 mm
- Draw angle = 60 deg
- Draw height= 12 mm
- Planar symmetry is used
- Bead is crated in both the direction

Results & Discussions:-

- Natural Frequency of baseline design in 1st mode increased from 14 Hz to 25 Hz by Topography Bead Optimization.
- Displacement of FT reduced by 35 % in static load cases.
- FRF stresses and displacement are under acceptable limits.
- 20% of Wight Reduction in the Final Design.

Benefits Summary :-

There were many benefits from the optimization simulation carried out. Many concepts were created based on the optimization study;

- Gives the flexibility to the designer to choose the concept as per the requirements of the assembly.
- This has offered considerable saving in terms of evaluating various concepts without actually building any prototype of all the concepts.
- Saves prototype build-up cost, testing time and cost.
- Further, it reduces the time required to arrive at the final design, thus shortening the product design cycle.

Challenges :-

Dynamic Assessment of Plastic
Correlation with measurement
Non linear material behavior
Reducing Design Cycle Time, More Reliable & Quality Products, Improved Decision making
Current Challenges for Automotive Industry Shorter developmental cycles, Higher Quality & Lower costs

**Future Plans**

Prototyping the optimized model is the preliminary task which is in process. Experimental test will need to carry out so as to compare the results with FEA tool results. To carry out above process for family of similar products so as to obtain considerable amount of material saving. The optimization based design helped in reducing the design cycle time significantly. The similar methodology will be implemented for other projects as well.

**Conclusions**

This paper presents finite element based simulation techniques for the topology optimization to meet the vibration Characteristics and evaluation of durability performance of front axle chassis mounting brackets. This simulation helps to reduce the burden on product development cost and time. A number of iterations were performed in CAE world before concluding the final design. This significantly replaced the proto part making and testing cost and time.

**ACKNOWLEDGEMENTS**

The authors would like to acknowledge our Design, testing teams for extending their kind support. The authors would also like to thank the R&D Chief Mr. Hemant Shrikhande, Testing Head Mr. Suresh Gopal, Divisional Manager Design Mr. R.S. Kaushik and Designer Mr. Sandip Dongarkadekar of Eicher Tractors for providing this opportunity for publishing the work.

**REFERENCES**