Optimization of Bracket to Meet the Pull Load Criteria Using Optistruct

Prasad Tirodkar¹, M. N. Shaikh²
¹Student, Second Year M.Tech, College of Engineering, Pune, India
²Assistant Professor, College of Engineering, Pune, India

Abstract— The objective of this paper is to investigate how and when structural optimization should be applied in the design process. The used tools are HyperMesh, Optistruct and HyperView which are parts of the software suite HyperWorks from Altair Engineering. Experience and knowledge in using structural optimization have been obtained by an initial literature study combined with evaluation of multiple trial cases of different parameter. The most common task has been to meet the recovery load criteria & mechanical properties as a constraint. This has been used to develop a sensible methodology together with guidelines for practical matters such as parameter values and recommended options.

Applying topology optimization in the concept level design stage a close cooperation between the designer and the analysis engineer to create the design domain. Further a proper assumption of loads and boundary conditions to get feasible solution. Interpreting the results from the topology optimization is a difficult task and requires experience and knowledge of other aspects such as manufacturability & also meets the physical testing requirements. From the interpretations of Topology & Size optimization, one conservative design is proposed. Mega bracket Structure is optimized to minimize the mass and the stress requirements.

Keywords— Topology optimization, Shape optimization, Optistruct Methodology

I. INTRODUCTION

The rapid pace of innovation in design technologies and information-transfer technologies has changed the nature of competition in the automobile & manufacturing industry worldwide. Product and process development have resulted in a market where new-product-development time has need to be cut dramatically if companies are to remain competition. Customers demand that manufacturers deliver products at affordable prices with a multiple specification. In this environment, automakers who can successfully streamline their product-development and manufacturing operations to trim excess cost and introduce innovative products will be the most likely to capture market share.

The flow of the study is shown in Figure 1.1 below. First, we will grasp the basic structure characteristics. The basic structure of Toe hook support bracket will be determined with a conventional design method, and strength analysis will be conducted. Next, conditions for optimization will be set. Characteristics to be improved will be set as objective function. And parameters which are considered to greatly contribute to the objective function will be determined as design variables. Optimization study using the simplified model will be conducted and the obtained result will be studied. Last, the advantage of the structure and validity of the study method will be verified vehicle model analysis on the structure identified.

Fig 1.1 Flow of Problem Study

Verification of Optimized Structure Characteristics
- Verification by Test
- Verification by Full Vehicle Model Analysis

Optimization study
- Setting of Condition for Optimization
- Simplified Modeling
- Study of optimization result

II. ANALYSIS METHODOLOGY

The optimization analysis is the process of finding best possible design by best utilization of the material required to manufacture a component. The FE modeling of the complete geometry will be carried out in HyperMesh v13.0 software, while the solver used for static analysis purpose will be Altair OptiStruct. Design validation & FEA analysis will be carried out using Abaqus software.
Validation of FE results will be carried out by bench test of recovery load requirements in laboratory. Plastic strain on the critical location identified in FE analysis is measured using strain gauge data acquisition. Data extracted from the testing is compared with the FE analysis results & test correlation method developed under this project.

Fig 1.2 Shows flowchart of the simulation process developed for analysis of Mega bracket.

III. DESIGN DESCRIPTION

To prevent using an unnecessary large design space, the allowed connection surfaces on the structure should be decided first. Input from departments such as Chassis Design and safety should be heavily considered. For recovery loading it is important that the attachment surfaces used are not of such nature that vibrations are transmitted into the Frame rail and affect the truck durability. For safety reasons it is important that components that is designed to have specific deformation behavior during recovery loading & plastic strain observed below the target value.

IV. OPTIMIZATION METHODOLOGY

A. The Topology Optimization Setup Parameters

Firstly, the bracket is finely meshed as in case of the topology optimization the solver cannot move the grid points in the model but can assign the different densities to the model.

- Design variable- Geometry
- Constraints: Mass of design space < 30 %
- Objective function (Design function)- Induced stress below the permissible Stress.

V. ANALYSIS SETUP

Figure 3.1 Proposed design Space of bracket for optimization.

Figure 4.1 Optimization methodology
B. Design & Non Design Space Representation

To prevent using an unnecessary large design space, the allowed connection surfaces on the structure should be decided first. Input from departments such as Chassis Design and safety should be heavily considered. For recovery loading it is important that the attachment surfaces used are not of such nature that vibrations are transmitted into the Frame rail and effect the truck durability. For safety reasons it is important that components that is designed to have specific deformation behaviour during recovery loading & plastic strain observed below the target value. The design space consider for the optimization of bracket is shown in Figure 4.1

![Figure 5.1 Design & non design space representation](image)

VI. CONCLUSION

Attempt is made to perform Optimization of a Bracket under load cases as per industry standard using topology optimization methodology. Thesis includes literature review that is deemed to be significant for this analysis, complete modelling of Suspension Shackle Bracket structure; optimization setup for topology optimization. Following is the conclusions drawn from analysis.

1. All the steps of the optimization methodology are described clearly. This methodology can be used to optimize any structure. The bracket is optimized to show application of this methodology.
2. The topology optimization is advanced function of many analysis software this is the functionality to empower design/analysis specialist with these currently available in the market useful tools to optimize any geometry.
3. We have saved the material 60 % by using this methodology
4. The optimized Bracket is manufacturable and the shape does not interfere with the operation of other nearby components.

Acknowledgement

I would like to extend my gratitude to acknowledge those guiding lights imbibed in me the right ingredient and helped me to accomplish this task. I sincerely, acknowledge with deep sense of gratitude to my respected guide Prof. M.N. Shaikh, College of Engineering, Pune for his guidance and encouragement

REFERENCES