International Journal of Advanced Information Science and Technology (IJAIST) ISSN: 2319:268
Vol.2, No.5, May 2013 DOI:10.15693/ijaist/2013.v2i5.1-4

Design Of Compressor Mounting Bracket Using Optimization.

Chaitanya Deshpande, Sharad Gawade,
Ajinkya Ghatekar, Aparajit Kharade.
Prof. Amar . P. Pandhare **Mr. Vasudeva Rao Kodati**

Abstract
At present in the competitive automotive world a light weight component is playing a vital role in fuel efficiency and cost of a vehicle. This leads to design a light weight component for the desired safety standards. This paper describes the design of an engine mounting bracket by the application of Altair Optistruct, which is an effective tool to obtain an optimum design. Based on requirements of mounting bracket, topology optimization is used in the design process. For complete parametric study, Hyper study is used. Compared with original design of mounting bracket, the mass on the final design is reduced by 43% and it is strong enough in fatigue and strength with improved stiffness values.

Abbreviations: FEA- Finite Element Analysis, CAD-Computer aided design.

Keywords: Topology Optimization, Mounting Bracket.

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.
Recently, in the course of designing large commercial vehicles, various optimization methods were attempted for optimal structure design, specially the mounting brackets of chassis components such as an air-conditioned (A.C.) compressor, a leaf spring, a radius rod, etc. It is very important that these mounting brackets should have proper stiffness and strength to provide sound durability and to a vehicle, while maintaining light weight.
Therefore, these brackets undergo many design changes throughout the vehicle development process. In addition, the brackets are usually made of cast iron with solid complicated shapes and large densities, but have simple load cases. All these conditions make the brackets preferable candidates for the optimization application, even though in principle the optimization methods can apply to any structural parts.
The engine mount system includes an engine, three to four engine mounts and a foundation (vehicle body). Since the engine is the largest concentrated mass in the vehicle and if it is not properly constrained and isolated, it will cause vibrations in the vehicle body and front end sheet metal. The behavior of the engine mount systems not only depends on the performance of individual mounts, but also on the complete system as well.

The design of an engine mount system involves the following:
(1) location of engine center of gravity (C.G.) and its orientation,
(2) location and orientation of individual mount,
(3) selection of stiffness coefficients of each mount.
The main purpose of this paper is to present an analytical optimization tool developed to assist design engineer in arriving at an engine mounting system with the most potential to decouple the modes and without violating the constraints imposed by packaging and manufacturing.
This program has the following functions:

(1) Optimization of the bracket.
(2) Strength analysis of the bracket.

Process Methodology:
Topology optimization is a method which distributes the density of an initially homogenous volume to achieve a certain objective function while observing the defined constraints. Minimize volume is usually considered as an objective function, while the displacement acts as a constraint and with manufacturing constraint such as casting of the bracket. Topology optimization is often used in the early design process to define the optimum part layout. The optimized models performance in the form of stiffness and strength evaluation is done and parametric study is carried out using HyperStudy according to fulfil the design and testing standard values.
The total optimization process is followed as in below Figure 1.

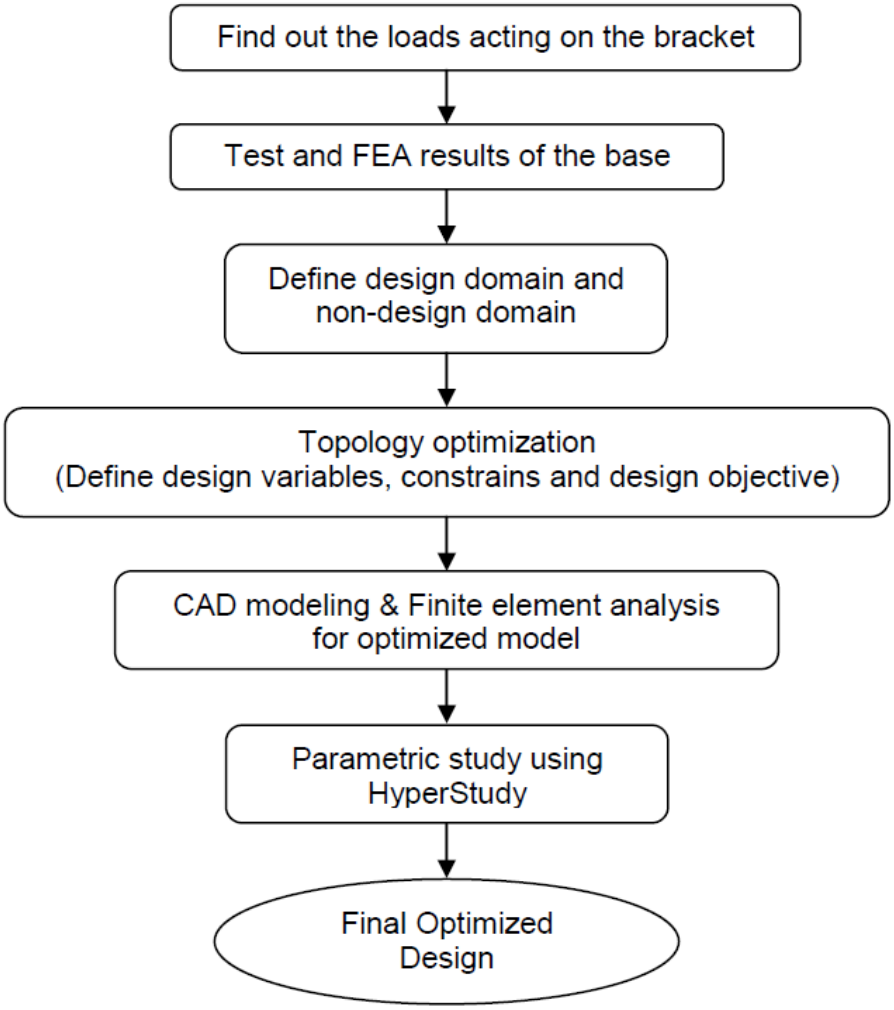

Figure 1. Flow chart for Optimization

Initially we need to collect the information regarding different loads acting on the bracket and the packaging data for fixing design space. The base bracket results from testing and finite element analysis (FEA) point of view for evaluating final optimized design.

Topology Optimization:
The topology optimization consists of the following sequence of steps.
• Define the design space
• Define optimization parameters
• Material removal process and detail design

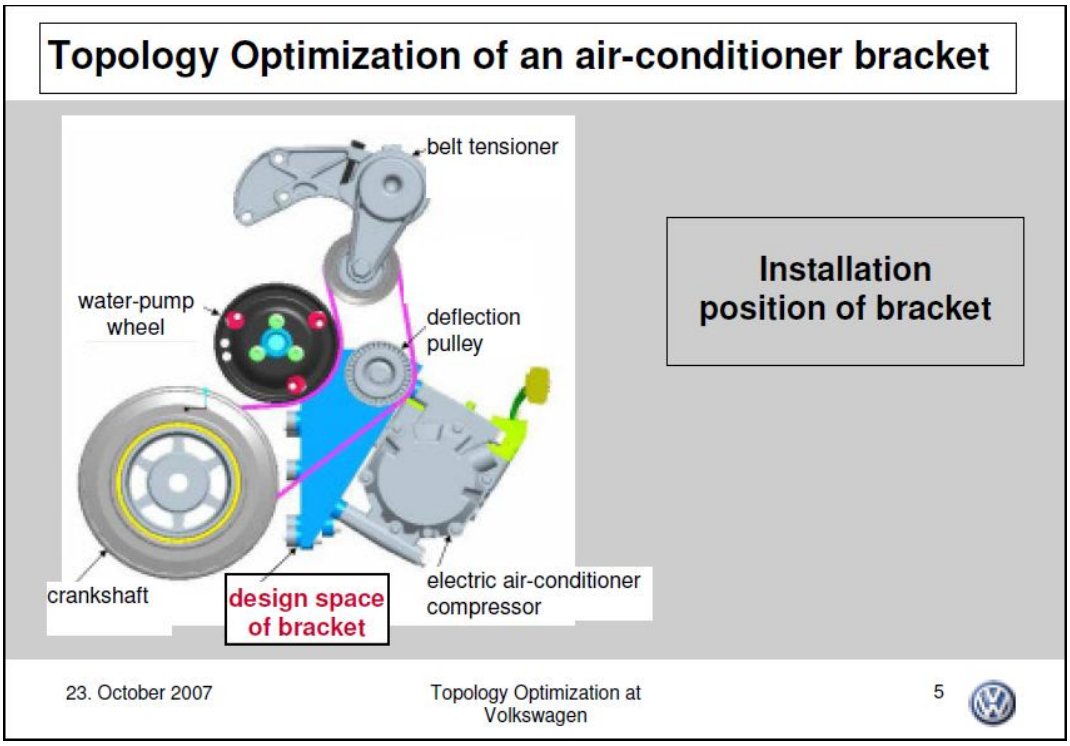

Figure 2.

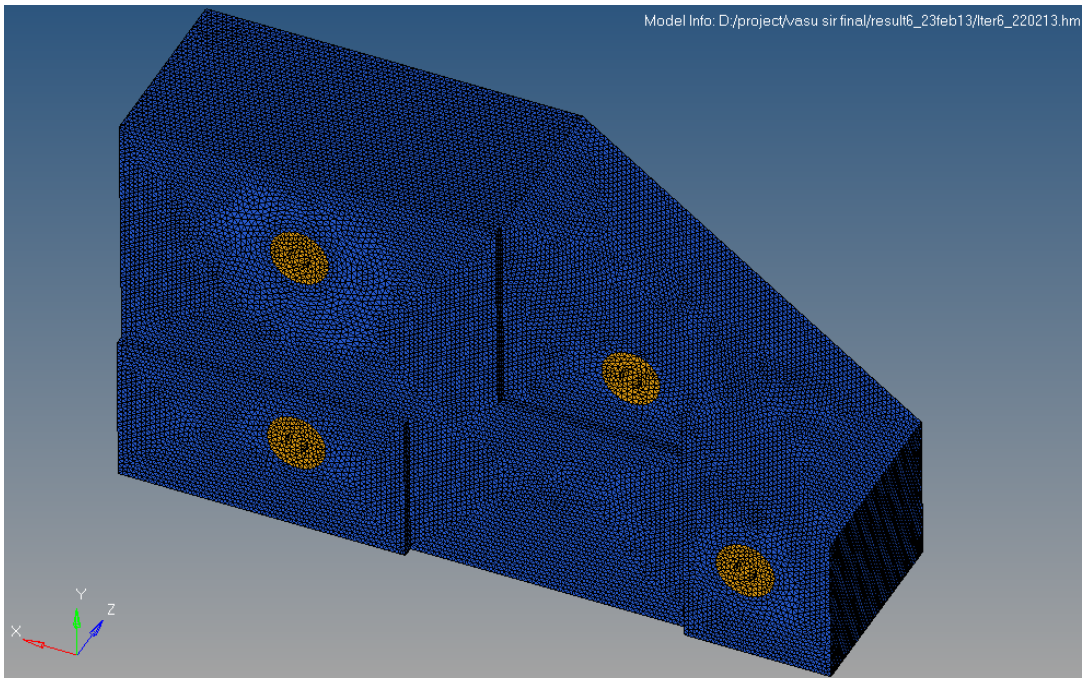

Figure 3. DESIGN SPACE

The design space of the mounting bracket is the available space which does not interfere with any surrounding components when being held in steady condition or in working condition.

Figure 2 shows design space of the mounting racket and interference check in power train mounting system. The design domain of the bracket is given in Figure 3. It is a rough conceptual design from a box model by taking into account bolting points and the areas interfered with surrounding parts.

Defining the Optimization Parameters:

The aim of topology optimization in this project is to minimize the volume without affecting the bracket stiffness and strength compared to base bracket, so the design objective is taken as to minimize the volume. Following parameter is defined as a constraint:

1. Maximum allowable displacement of the bracket.

Material Removal Process and Detail Design

The optimization process took 30 iterations to remove the unnecessary material from the design space.

Output from the optimization process is shown in Figure 4.

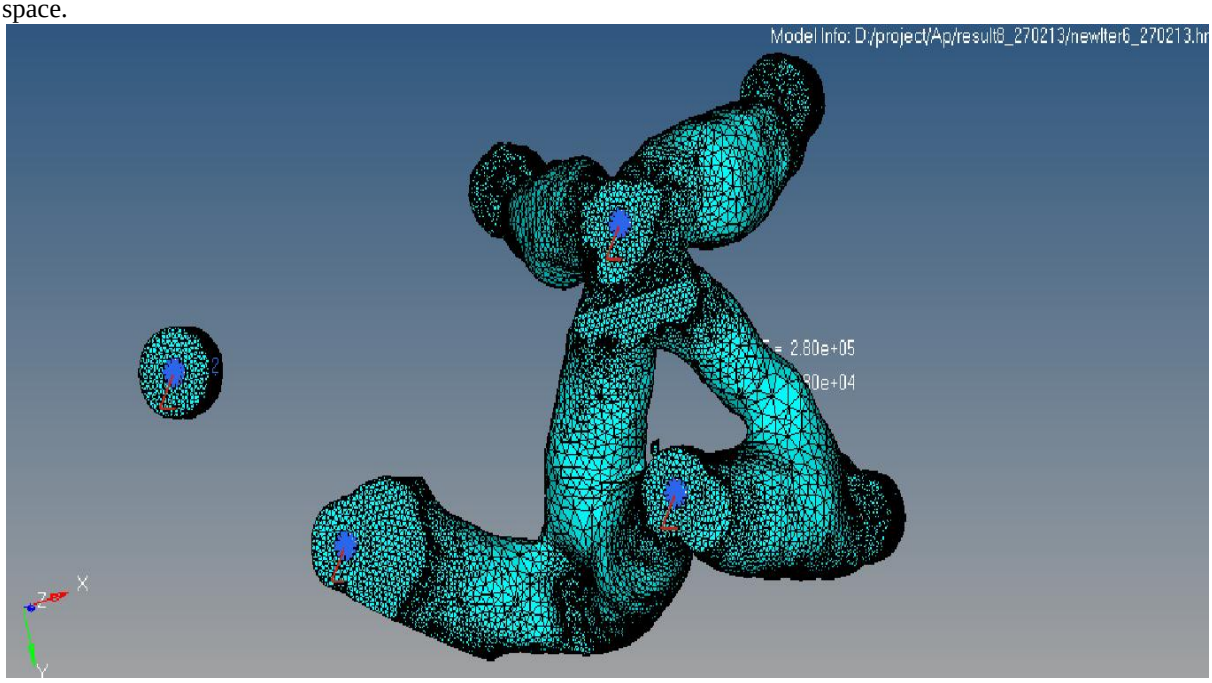

Figure 4. OPTIMIZED MATERIAL DISTRIBUTION.

The output of the topology optimization, an intermediate model which may be called a topology-based based model, is constructed by removing unnecessary materials from the rough conceptual model, as presented in Figure 5.

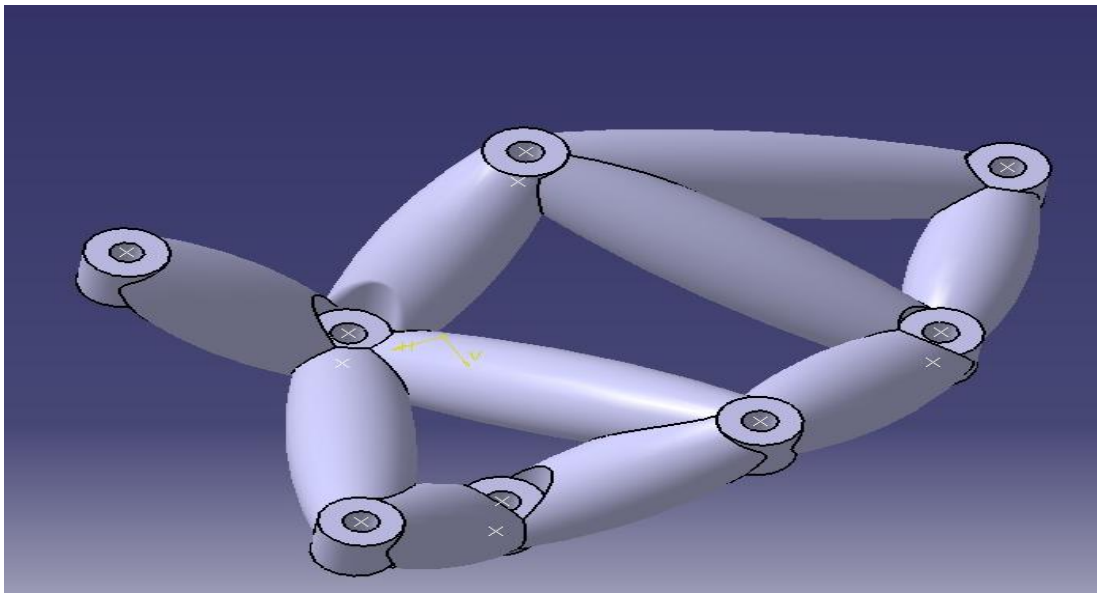

Figure 5 OPTIMIZED CAD MODEL

Analysis and Complete Parametric Study

On topology optimized model, finite element analysis has been carried out using HyperMesh, and strength analysis and complete parametric study has carried out by using Optistruct.

Conclusions:

In this paper, topology optimization approach is presented to create an innovative design of an engine mount bracket. Final comparison in terms of weight and component performance illustrates that structural optimization techniques are effective to produce higher quality products at a lower cost.