Analysis of Light Motor Vehicle Component Using Topology Optimization Method

M. Vijay Kumar Reddy¹, P. Chitti Babu², K. C. K. Bharathi³
¹Department of Mechanical Engineering AITS, ²³Department of MCA, APGCCS, Rajampet, India

Abstract - This work is based on the analysis of engine mounting bracket. There are four mounting brackets to mount engine on vehicle chassis. Analysis is done for only one bracket and the results are considered for other three brackets. Due to the increase in global competitive market, automotive designs are pushed to the limits to decrease the cost and at the same time increase the stiffness to mass ratio and performance. Above all, the biggest constraint is to achieve all these with reduced design time. In order to get better designs, industries are using tools like FEM Optimization; these techniques are useful in designing the new shape and size with reduced design time. New Topology optimization tool is used in this work in designing engine mounting bracket for analysis to get better design with less time. It also helps in the reduction of unwanted material and thereby mass of the component decreases respectively.

Keywords - Analysis, Engine mounting bracket, Topology Optimization, Hypermesh, Optistruct Solver, Hyperview.

I. INTRODUCTION

Automobile production consumes large amounts of iron, steel, aluminum, natural rubber, copper, glass, zinc, leather, plastic, and lead. In order to reduce wastage during manufacturing of automobile components, optimum utilization of material is taken into consideration. An automobile component is analyzed by using Hypermesh tool and applying Topology optimization techniques to reduce the material consumption [2, 10]. An objective function in general will be either to minimize the expense or maximize the benefit for the issue in investigation. For instance, in a manufacturing process, profit might be maximized or cost might be minimized.

Optimization problems are made up of three basic ingredients:

1. An objective function, which the authors want to minimize or maximize. In designing an automobile panel, the authors want to maximize the strength.
2. A set of unknowns or variables, which affect the value of the objective function. In fitting-the-data problem, the unknowns are the parameters that define the model. In the panel design problem, the variables used define the shape and dimensions of the panel.
3. A set of constraints that allow the unknowns to take on certain values and excluding others. In the panel design problem, we would probably want to limit the weight of the product and to constrain its shape [5].

Structural optimization is one application of optimization. Here the purpose is to find the optimal material distribution according to some given demands of a structure [6, 9]. Some common functions to minimize are the mass, displacement or the compliance (strain energy). This problem is most often subject to some constraints, for example constraints on the mass or on the size of the component. Structural optimization can be separated in three different areas: size optimization, shape optimization and topology optimization.

Size optimization is the simplest form of structural optimization. If the shape of the structure is already known then the objective function is to optimize the structure by adjusting sizes of the components. Here the design variables are the sizes of the structural elements. For example the diameter of a rod, the thickness of a beam, a sheet metal, etc., are considered as design variables.

![Figure 1: Sizing optimization](Image)

Shape optimization, as with size optimization, the topology (number of holes, beams, etc.) of the structure is already known [4], but the shape optimization will not result in new holes or split bodies apart. In shape optimization, the design variables can for example be thickness distribution along structural members, diameter of holes, radii of fillets or any other measure. A fundamental difference between shape vs. topology optimization and size optimization is that instead of having one or more design variable for each element, the design variables in shape optimization affect many elements.

211
The most general form of structural optimization is topology optimization. The purpose of topology optimization is to find the optimum distribution of material [3]. With topology optimization the resulting shape or topology is not known, the number of holes, bodies, etc., are not decided upon. The purpose is to find the optimum distribution of material and voids. To solve this problem, it is discredited by using the finite element method (FEM) and dividing the design domain into discrete elements (mesh).

II. METHODOLOGY

The main program used for performing finite element analyses and optimizations is the solver Optistruct10.0 from Altair Engineering. In this work Optistruct is chosen as optimization software which is to be able to set up the problem and review the results by using Hypermesh and Hyperview. Hypermesh is the preprocessor which is used to discretize (mesh) a CAD model, set boundary conditions, properties and options and to set up the problem to be solved (optimization, static analysis, modal analysis etc.)[1]. From Hypermesh a file which completely describes the problem is exported and then processed using Optistruct. The results from Optistruct can then be evaluated using the postprocessor Hyperview. A schematic overview of the work flow can be seen in below figure. HyperMesh, Optistruct and HyperView are all part of the software suite Hyperworks10.0 [2], and as such they are designed to easily integrate with each other.

Step 3: Meshing the geometry model with hexa elements in Hypermesh.
Step 4: After meshing defining material and its properties, and then create constraints and forces.
Step 5: Applying vertical load 1000N in vertical direction and constrain three holes of the bracket.
Step 6: Running the operation in optistruct solver.
Step 7: Optistruct solver executes results; these results can be seen in Hyperview.
Step 8: The obtained results can be taken as input parameters to the topology optimization.
Step 9: Defining objective function in topology optimization, mass and displacement responses.
Step 10: Running the operation in optistruct solver, strain energy results can be viewed in Hyperview.
Step 11: The strain energy results describe the possibility to reduce material from bracket.
Step 12: The material can be removed in Hypermesh, after this again next iteration can be performed by applying load and boundary conditions.
Step 13: After performing several iterations optimum results can be obtained.

Meshing the component - Part of the Altair Hyperworks® software package, Altair HyperMesh® was used as the finite element meshing utility in preparation for the optimization study. The design space previously created in a CAD program was imported as an IGES (Initial Graphics Exchange Specification) file, and the geometry was “cleaned” to prepare for meshing [8]. This means that some of the lines in the imported model were toggled from edge lines to suppressed (or manifold) lines so that they would not represent an artificial edge that would force the finite elements to unnecessarily align themselves.

Figure 2: Shape optimization

Figure 3 Topology optimization

Figure 4 Methodology of the work flow in Hyperworks

Step 1: Importing CAD geometry model in Hypermesh
Step 2: Performing geometry cleanup and mounting bracket, it gives accurate mesh to the component.

Step 3: Meshing the geometry model with hexa elements in Hypermesh.
Step 4: After meshing defining material and its properties, and then create constraints and forces.
Step 5: Applying vertical load 1000N in vertical direction and constrain three holes of the bracket.
Step 6: Running the operation in optistruct solver.
Step 7: Optistruct solver executes results; these results can be seen in Hyperview.
Step 8: The obtained results can be taken as input parameters to the topology optimization.
Step 9: Defining objective function in topology optimization, mass and displacement responses.
Step 10: Running the operation in optistruct solver, strain energy results can be viewed in Hyperview.
Step 11: The strain energy results describe the possibility to reduce material from bracket.
Step 12: The material can be removed in Hypermesh, after this again next iteration can be performed by applying load and boundary conditions.
Step 13: After performing several iterations optimum results can be obtained.

Meshing the component - Part of the Altair Hyperworks® software package, Altair HyperMesh® was used as the finite element meshing utility in preparation for the optimization study. The design space previously created in a CAD program was imported as an IGES (Initial Graphics Exchange Specification) file, and the geometry was “cleaned” to prepare for meshing [8]. This means that some of the lines in the imported model were toggled from edge lines to suppressed (or manifold) lines so that they would not represent an artificial edge that would force the finite elements to unnecessarily align themselves.

Figure 2: Shape optimization

Figure 3 Topology optimization

Figure 4 Methodology of the work flow in Hyperworks

Step 1: Importing CAD geometry model in Hypermesh
Step 2: Performing geometry cleanup and mounting bracket, it gives accurate mesh to the component.
Material definition - the last input parameter required before implementation of the topology optimization study is the material identification. In the case of mounting bracket, the material was specified as mild steel; as a result, the material was defined and the values for Young’s modulus and Poisson’s ratio were entered. The values used for the study are 210GPa and 0.33, respectively.

Hexameshing - once the geometry was cleaned, the design space volume is filled with hexa elements using the auto-mesh features of HyperMesh [6]. This was done with a volume-hexaelement, and curvature and proximity adaptation enabled to refine the mesh in the regions of more complex geometry [8]. The resulting mesh that is used as the design space for the topology optimization study [11] can be seen in below Figure.6.

Boundary Conditions - As in any finite element analysis, proper boundary conditions (BC’s) are crucial if the results are to be of any significance, and this is especially true for topology optimization studies since these BC’s will be the basis for the resulting distribution of material. An example of this lies in the treatment of the boundary condition around the region of a bolted joint.

Constraints and loads - in this work, engine mounting bracket consists of three holes, these three holes are considered to be constraints. In this work, point load acting on each mounting bracket in downward direction is 1000N, as shown in below Figure.7.

After the boundary conditions and loads, the operation is run by using Optistruct solver. In the optistruct solver displacement and maximum principle stress are taken as result type. The results can be seen in Hyperview. Figure 7 shows the displacement occur in the mounting bracket and Figure 8 shows the maximum principle stresses occur in the mounting bracket before optimizing the component.

Post processing - After applying mass and displacement responses in optimization then the operation can be carried out by using optistruct solver and results can be seen in Hyperview, Figure 10 shows the strain energy of the mounting bracket after optimization. In this figure, blue colored portion represents that there is a possibility to remove the material, hence it can withstand under same loading condition [7], and this can be seen in Figure 10.
The figure 11 shows the possibility to remove the material in determined portion, by using Hypermesh the material can be removed as seen in below figure 12.

### III. RESULTS AND DISCUSSIONS

**Results before optimization:** From the structural analysis, displacement in the mounting bracket takes place before optimization by applying loads and boundary conditions, the maximum displacement of the bracket is 0.306mm which has indicated in red color as shown in below figure 13.

Maximum principle stresses in the mounting bracket before optimization by applying loads and boundary conditions, the maximum principle stress of the bracket is 61.34 N/mm² which is indicated in red color as shown in below figure 14.

**Results after optimization:** If weight is reduced to 0.22kgs displacement of the mounting bracket after optimization by applying loads and boundary conditions, the maximum displacement of the bracket is 0.357mm this has indicated in red color, in this case the material is saved and also the component can be withstand as shown in below figure 15.

From these results, the optimized mounting bracket is more suitable because it is able to withstand under applied loads and boundary conditions.
Results obtained after topology optimizations are: 61.34 N/mm² Stress and 0.357 mm Displacement which satisfy the properties of actual model before optimization. From all these circumstances the proposed model said to be an optimized model.

IV. CONCLUSIONS

This work is covered two different phases. Better design and design time within a mean time. This is very important parameter because the product market life is reducing. There is a need to get more new and better designs with less time to decrease the cost and at the same time increase the stiffness to mass ratio and performance. In this work, the mass is reduced by 0.22kgs per component.

Mass and cost saving - Substantial cost saving can be seen with the reduction in weight. Assuming a material cost of 0.8 kgs and a annual production of 200,000 pieces, a 35,200 saving can be realized in material cost alone.

Annual Material Saving:

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass reduction</td>
<td>0.8kgs per four pieces</td>
</tr>
<tr>
<td>Annual production volume</td>
<td>2,00,000 units</td>
</tr>
<tr>
<td>Total material saving</td>
<td>1, 60,000 Kgs</td>
</tr>
<tr>
<td>Nominal material cost</td>
<td>30Rs/- per kg</td>
</tr>
<tr>
<td>Cost of total material saving</td>
<td>48, 00,000Rs/-</td>
</tr>
</tbody>
</table>

REFERENCES