Design of Integrated Super Bracket for Heavy Commercial vehicles

Amar Vhanshetti, Nitin S. Chavan
Dept. of Mechanical Engineering
JSPM's JSCOE, Hadapsar, S. P. P. U., Pune, India

Abstract—*Growing competition in automotive market makes it more and more necessary to reduce the development time and cost of the product development process. One of the most costly phases in the vehicle development process is the field durability test and high expenses for this phase can be attributed to the number of prototypes used and time needed for its execution. Also, multiple iterations of designing, building and testing prototypes are no longer affordable against the time and cost constraints for developing a competitive product. Today, analytical tools in the form of computer simulation have been developed to such a level that they reliably predict performance. Hardware prototypes cannot be made in early design phase, however, today with the use of computer aided engineering tools virtual models can be created to accurately represent physical models and to take right decisions at the right time. It became necessary to come up with innovative solutions which are cost effective and effective and at the same time which will meet the performance requirement, to sustain in growing market. A step towards frugal engineering is to replace number of subassemblies in the vehicle with a single multifunctional invention. A massive trend is coming to unite various subassemblies together and give one integrated solution which will reduce cost of designing, tooling, manufacturing, assembling and overall cost of many subsystems. The work design of integrated multifunctional module for heavy commercial vehicle will replace various subsystems including towing device mountings, suspension mounting systems, cabinet mounting systems with single design. This work involves evaluate various loads coming on the module during different working conditions. Based on these loadings and space available the module will be designed which will meet the structural performance criteria. The main objective of the module is to lower the weight and cost of component, and provide more robust design to customer.*

Keywords—Superbracket, optimization, topology optimization, integrated bracket, durability

I. INTRODUCTION

Cost reduction is the key to the successes of the industry. If different single functional parts get replaced by an integrated multifunctional module along with the comparative weight reduction then it will not only advantage of the different aspects of the cost reduction, but also an added advantage to the vehicle mileage.

I. PROCESS METHODOLOGY

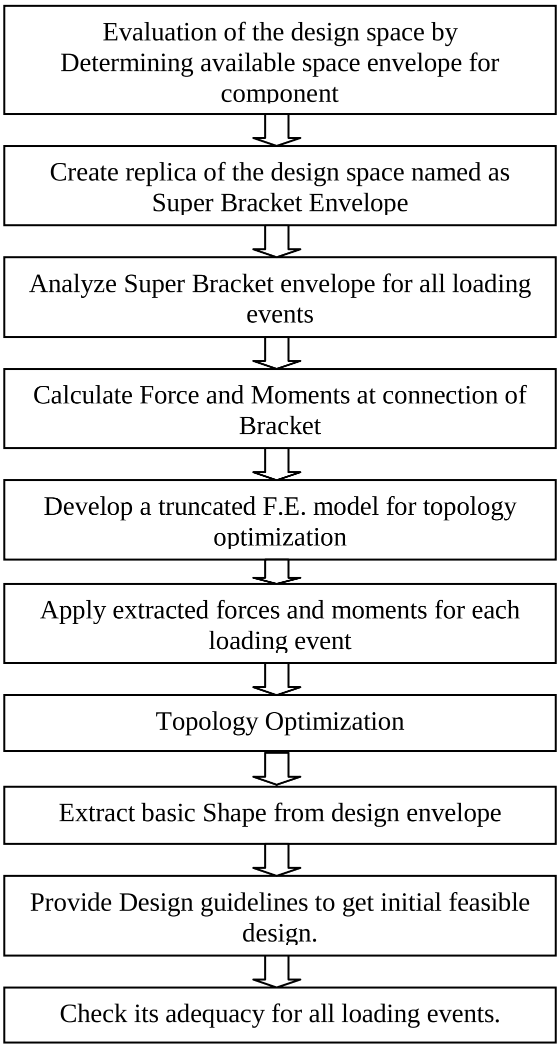


Fig1. Flow chart of Process Methodology

At the same time product must be introduced with minimum lead time, that emphasis on the reduction of the lead time with innovative computational techniques. In this work three different parts; cab mount, suspension mount and tow hook mount are replaced by the single bracket named as Super bracket and final design of this is obtained through the optimization technique. In order to reduce the lead time finite element analysis method of sub structuring is used [3]. In this method, initially a Super bracket is analyzed using the full frame model which includes some millions degree of freedoms, so forces and moments at the joints are extracted. In further analysis this forces are used to simulate actual load path. With this, trimmed finite element analysis model is used to optimize the Super bracket.

A. Details of Design Space

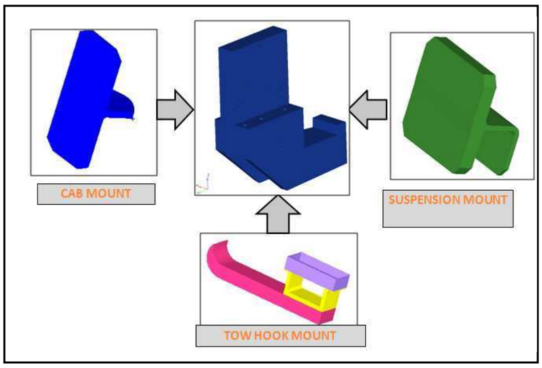


Fig.2. Design envelope formed after replacing the prescribed parts

Figure 2 shows the design space calculated based on the actual room for filling the material. This design space will be acting as boundary for integrated Super bracket. The total mass of the Subsystems to be replaced are 26kg per side, while the Mass of Design space envelope is 113kg per side. So the target is to engrave the final shape of Super bracket from this block which will weight much lesser than 26kg, resulting considerable mass reduction.

B. Analyzing super bracket for all loading events

This is the very first version of the design. At this stage, we have to validate the design envelope against the standard load case. The super bracket in full model is shown in the Figure 3. At this stage super bracket is highly over designed and it is followed by the Optimization.

Results of this analysis are post processed in two steps. In first step stresses in the super bracket are checked against the acceptance criteria and in second step bolt forces at the attachment points of the super bracket are bracket are extracted for the sub structuring. The finite element model of the full frame chassis used in the analysis is shown in Figure 3. Three mounts are replaced by the super bracket. Once finite element modeling was completed, the model connection validity is ensured by carrying out by performing an unconstrained modal analysis. To ensure if both the axles were properly loaded, the forces at the constraints of axle were calculated and compared with the rated capacity of the both the axles.

Analysis setup and acceptance criteria: Following loads were considered for evaluating the performance of the Super bracket and aggregated mounted on it [2].

1) Jounce: 2g Even bump on front axle.
2) Tramp: 2g Uneven bump on front axle.
3) Aggressive Turn: Severe turning scenario.
4) Frame Twist: Cross Twist and Bogie Twist.
5) Peak Inertial Loads: Due to self weight and road accelerations.
6) 3g Vertical loading.
7) 1g Vertical + 0.9g longitudinal loading.
8) 0.3g Lateral loading.
9) Racking: Vehicle turning of multi rear axle vehicle.

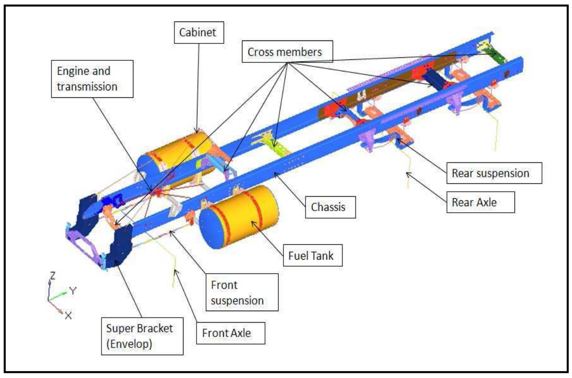


Fig.3. Full frame chassis model with Super bracket

For these loadings, acceptance criteria for design evaluation are as follow: under racking and cornering loads, stresses on the super bracket should be less than endurance limit of the material while for rest of the load cases stresses on the bracket should be less than yield strength of the material.

Material Details of Super bracket are as follows:

1) Austempered Ductile Iron (ADI)
2) Yield Strength= 550 MPa
3) Endurance Strength = 317 MPa

Vehicle specifications are as follow:

1) Front Axle Weight Rating (FAWR): 71.26 kN
2) Rear Axle Weight Rating (RAWR): 204.87 kN
3) Gross Vehicle Weight Rating (GVWR): 276.13 kN

The results of the FE analysis are checked against the acceptance criteria and it is observed that the design envelope is structurally adequate and there is further scope for the mass reduction. The One another objective of this analysis was to evaluate the forces and moments generated on all bolting location of the Super bracket for all load cases. So, these forces and moments can be directly used to analyze the truncated model of frame with only super bracket. This method will reduce the time required for analysis. During this analysis force and moments are calculated and based on that force and moment matrix is developed [5]. For example the force and moment matrix for Jounce load case is given below in Table I.

After analyzing the results of the full frame static analysis with super bracket, the next objective is to engrave the shape of the super bracket from design envelope. For this optimization tool of Hyper Works is used.

TABLE I. FORCE AND MOMENT MATRIX FOR JOUNCE LOAD CASE

Bolt No.	Fx(N)	Fy(N)	Fz(N)	Mx(N.mm)	My(N.mm)	Mz(N.mm)
B1	-2.21E+01	6.43E+03	-2.60E+03	-8.44E+05	2.39E+04	3.50E+04
B2	-2.56E+02	6.42E+03	2.19E+03	-3.97E+03	1.43E+03	-1.90E+01
B3	-1.09E+03	3.06E+03	4.86E+02	-4.86E+03	-1.80E+03	1.44E+03
B4	2.12E+02	3.15E+03	5.00E+03	-2.76E+03	5.13E+03	4.38E+02
B5	-2.58E+02	1.38E+03	6.01E+03	-2.13E+03	5.12E+03	4.74E+02
B6	-1.67E+03	-2.14E+03	1.24E+03	-1.37E+03	3.91E+03	2.96E+03
B7	5.48E+02	-8.83E+02	1.81E+03	-1.12E+03	-5.54E+03	1.13E+03
B8	1.99E+03	-2.21E+02	2.75E+03	-4.14E+02	-1.17E+04	1.11E+03
B9	5.19E+02	-9.13E+02	-1.18E+04	-1.46E+02	2.16E+04	-5.17E+03
B10	-7.36E+02	1.12E+03	-8.03E+02	-2.13E+03	-1.32E+03	-4.01E+03
B11	-2.71E+02	-8.97E+02	5.21E+02	1.55E+03	-1.26E+03	-3.20E+03
B12	3.96E+02	-4.75E+01	4.13E+02	3.23E+02	-1.06E+03	2.46E+02

II. TOPOLOGY OPTIMIZATION

Topology optimization can be defined as a mathematical approach that optimizes material layout within a given design space, and for a given set of loads and boundary conditions such that the resulting structure meets a prescribed set of performance targets. Using topology optimization, engineers can find the best concept design that meets the design requirements [4].

This replaces time consuming and costly design iterations and hence reduces design development time and overall cost while improving design performance. As discussed earlier the full frame model will take a longer time to give solution, this will cause higher computational cost. This also increases the design lead time and limits the number of iterations carried out so full frame model is truncated suitably. The truncated FE model used in optimization is shown in Figure 4. Based on the loads extracted in earlier step, loads and boundary conditions are applied to the truncated model. The extracted forces were applied at their respective bolting location while the truncated frame was constrained in all direction. Now this shorter model will take less time for solving. Figure 4 shows the details of load and boundary conditions considered for topology optimization. For Topology optimization the most important inputs requires are design space, design constraints and objective function. Figure 5 shows the details of the design space and non-design space. All the mounting locations of the super bracket, where bracket and its bolts with frame rails were considered as non design space. However, the rest of the area of super bracket was termed as design space, from where material will be removed. For extracting feasible shape of Super bracket various iterations were performed by changing numerous parameters. Amongst this iteration one of the following iteration described below has provided most feasible solution:

The objective of optimization was minimizing volume up to 20% of original design space volume. For this optimization all forces coming during all loading events were considered. The design constraint was, the stresses induced in the final design shape under all loading condition should be less than 250 MPa i.e. 50% of yield strength of Austempered ductile Iron (material of Super bracket).

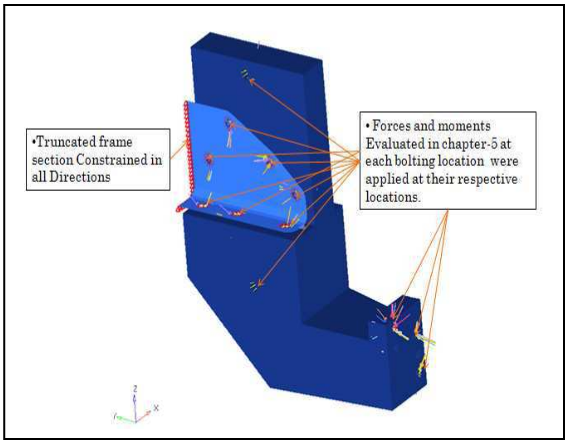


Fig.4. The truncated FE model used for optimization

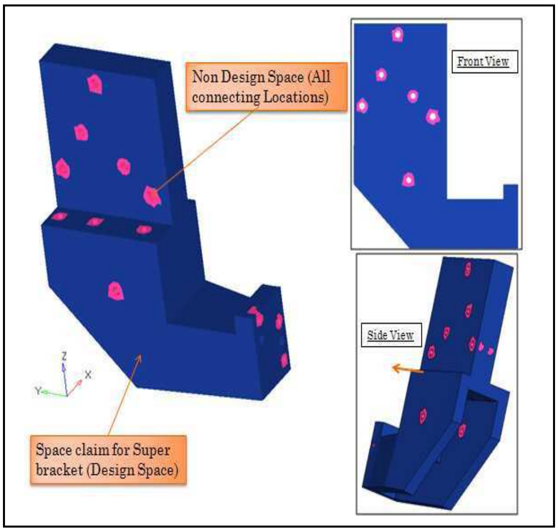


Fig.5. Design and non-design space definition

Along with this, following manufacturing conditions were also applied which includes:

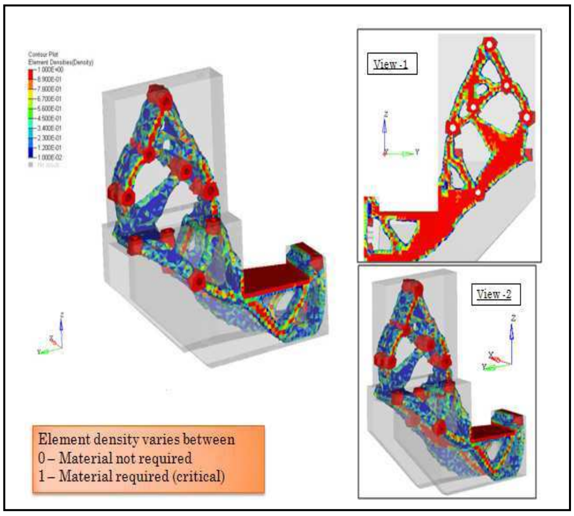
1) The design should be symmetrical about YZ plane
2) The casting should be single drawn and draw direction should be along X-axis
3) Minimum wall thickness of suggested design should be 15mm, to ensure good casting.

ISSN: 2349-6495

The figure 6 shows the optimization set up.

A. Result Discussion of the Topology Optimization

Result of optimization study was post processed in the form of element density using Altair Hyper View. For elements having density zero was removed as it indicates no material required at that location, while elements having density 1, shows that material is must [1]. Accordingly all the elements in design space were assigned a contour ranging from 0 to 1. As a standard practice followed by many experts in computer aided engineering industry and based on literature survey, the threshold value to remove the material was considered as 0.3, it meant the elements having density less than 0.3 was to be removed to get final shape. The figure 7 shows the results for the topology optimization.



In order to convert the shape obtained from the topology optimization analysis in the feasible shape, few minor changes were suggested. These changes helped design team to create appropriate CAD data. The recommended design changes includes, adding filet at sharp edges, adding material which keep the flow of material continuous, add ribbing net at top as suggested by solver, providing tow hook mounting pocket at front region. Figure 8 shows the design recommendations suggested on result of optimized shape [7].

Based on the recommendation and interaction with design team the initial design of the Super bracket was developed. Mass of the initial design of Super bracket was 21.14kg which is very much less than the mass of the subsystems to be removed i.e. 26kg. Additional optimization iterations were performed to reach to the most optimal design. The Mass of the final design of super bracket was 19.26kg which is 26% less than that of other subsystems. This final design of the Super bracket is further validated for all the load cases by using Radioss linear solver and solver and it is found that the design of the Super bracket is structurally adequate considered loading scenario [6].

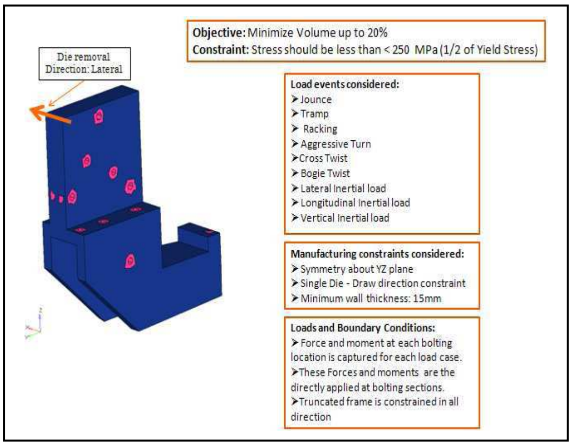


Fig.6. Optimization set up

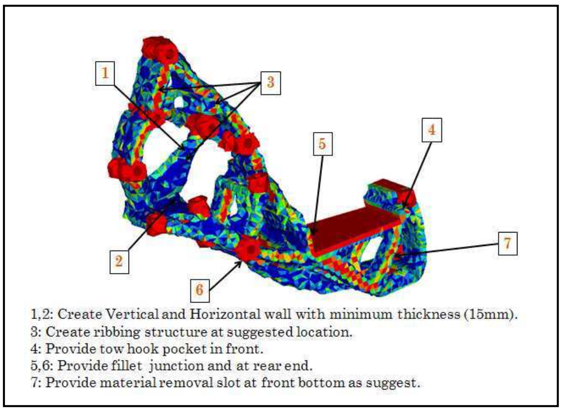


Fig.8. The design recommendations suggested on result of optimized shape.

REFERENCES

[1] Caner Demirdogen.Weight “Optimized design of a front suspension component for commercial heavy truck” Technical report, SAE Documents2004-01-2709.

[2] Basavaraju Raju. “Numerical and experimental verification of optimum design obtained from topology optimization” Technical report, SAE Documents 2003-01-1333.

[3] Masatoshi Shimoda “Non-parametric shape optimization method for rigidity design of automotive sheet metal structures” Technical report, SAE Documents 2006-01-0584.

[4] L.S. Rajaram, “Optimization of Caliper housing using FEM” Technical report, SAE Documents 2005-26-060.

[5] Danial Blower “Analysis of Truck- light weight vehicle crash data for truck aggressivity reduction” Technical report, SAE Documents 2001-01-2726.

[6] Ccline Adalian “Improvement of car-to-truck compatibility in head-on collisions” Technical report, SAE Documents 98-S4-O-12.
[7] Lance Proctor “Shape optimization in MSC/NASTRAN and MSC/PATRAN” Sr. Technical Representative The MacNeal-Schwendler Corporation Grapevine, Texas.