Design Verification Procedure Load Case Analysis and Optimization of Car Hood Assembly

Mahesha J[1], Prashanth A S[2], Doddanna A K[3], Venkatesha Reddy[4]
[1, 2, 3, 4] Department of Mechanical Engineering
[1, 2, 3, 4] Dr. Ambedkar Institute of Technology, Bengaluru, India

Abstract- *The automobile industries are going towards the green approach at present time especially, because of marketing purpose. Thus, car industries are beginning to development and design of cars with less environmental effects. This development is done through many different altered ways. The most known and presented is the reduction in consumption of fossil fuels. The different way that could be followed to reduce the environmental impact is to decrease the quantity of material used for each part. This type of reduction of materials brings many different advances and improvements in new car development. First of all it has direct benefits due to less use of raw materials and energy for parts production. At second stage, it has an indirect influence and benefit on fuel consumption due to the lower weight of the car as lesser the weight lesser the consumption of fuel and finally it reduces the costs for the firm too.*

Keywords- Car Hood; Analysis; Optimization; Weight reduction;

I. INTRODUCTION

Car hood assembly or bonnet is the hinged cover above the engine of motor vehicles that permits accessing the engine compartment for maintenance and repairing. Engine hood assembly comprise of various parts such as base hood, lock, hinges, sealing straps and some clips which are used for fixing these clips on Base Hood. In order to apply new light-weight materials in the manufacture of car body components in its place of steel, it is essential to estimate the new required dimension of the same part without loss of safety, strength and stiffness. With rapid development of computer technology and numerical methods, the field of optimization is actively being studied by many researchers. In general, the optimization problem in structural design in the automotive industry can be divided into two processes namely; a determination process of cross-section, shape and configuration and a total optimization process based on the design variables selected. The main parts of Car Hood assembly consist of Hood Outer and Inner Panel, Rear and Front Reinforcement Bracket and Striker Plate. When car come across any accident from the front portion most of the time bonnet system gets damaged and absorbs some part of impact energy resulting from crash.

II. LITERATURE SURVEY

Rupesh Rodke, Dileep Korade[1] discussed about hood design and development for new project considering two durability load cases namely, Torsional stiffness and Cross member bending. The essential information of hood assembly and development has done according to Computer-Aided Engineering (CAE) results. Number of iterations was done to fulfil the acknowledgment criteria of durability tests. Computer Aided Design (CAD) tool utilized for design and development of hood assembly. In the static load analysis part, a Finite Element Analysis (FEA) was done using software Nastran and Hypermesh. This paper has shown design modification for the increased strength, weight reduction and also reduced process time and cost.

N. Bhaskar, P. Rayudu[2] discussed Design and Analysis of Car Bonnet. In this paper they mainly concentrated on reducing the weight of the car hood assembly with respect to base line design by performing some important static structural analysis on existing car bonnet design. For designing the bonnet 3D model CATIA V5 R19 is used then imported the designed cad model in to Hypermesh and pre-processing is done in Hypermesh, solver used is Optistruct FEA which helps to do the analysis for hood assembly. For post processing results visualization Hyperview and HyperGraph is chosen. Static structural analysis like, oil canning, torsional stiffness and lateral stiffness analysis have been done on car hood. Compare to base line design they have achieved highest weight reduction of 36.95%, highest stress reduction of 36.57% and negligible displacement increment in lateral stability analysis.

III. FINITE ELEMENT METHOD

A. Problem Definition

The car engine hood is analysed for its strength assessment for the different Design Verification Procedure (DVP) load cases. The nonlinear static structural analysis is carried out on engine hood to evaluate the stresses, displacement, plastic strain induced in the hood assembly components for the different DVP load cases. In this project work, both material non linearity (i.e. change in stiffness) and contact non linearity (i.e. changes in gap) is considered for the analysis work. The base line model is analysed and from the baseline results, design optimization like thickness, shape topology operation carried out to reduce the overall weight of the hood assembly.

B. Modelling

The FE model is generated using meshing software Hypermesh V13.0. The 2D shell mesh is used for FE modelling of all hood components. The spot welds between reinforcement brackets and inner panel are created using RBE3-HEXA-RBE3 element configuration. The structural adhesives between inner panel and outer panel is created using RBE3-HEXA-RBE3 element configurations. The bolted joint between inner panel and reinforcement panels are created using RBE2-BEAM-RBE2 element configurations. The FE model and connections are as per the industries guidelines. The reinforcement panels are connected through spot welds and bolt connections. The Shell element types S3 for triangular elements and S4R for quadrilateral elements are used. For Hexa elements C3D8R element types are used. The FE model is created with a global element size of 5mm with capturing all features. Fig.1 and Fig. 2 shows the Finite element modelling of Outer and Inner Hood assemblies respectively.

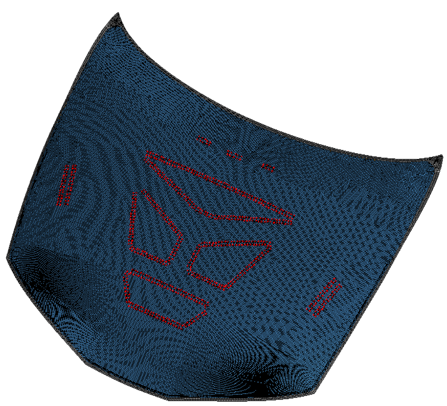

Fig. 1. Outer Hood Assembly FE Model

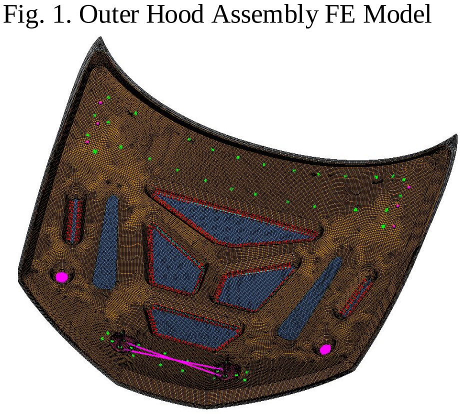

Fig. 2. Inner Hood Assembly FE Model

TABLE I

Hood Components Gauge Thickness, Material Details

Component	Material	Gauge Thickness(mm)
Hood Inner Panel	Al - AA5182 0%	1.0
Hood Outer Panel	Al - AC170PX 0%+PB	1.1
Front Reinforcement	Al - AA5182	1.2
Rear Reinforcement	Al - AA5182	1.5
Striker Plate	Steel	1.5
Spot Welds	Aluminium	6.0
Structural Adhesive	Adhesives	

C. Boundary Conditions

In off body conditions also called as rigid bed conditions, only hood assembly will be considered for the analysis and in place of hinge mounting brackets mock up hinges set will be considered. In this project work, off body load cases were performed to evaluate the strength of hood assembly using mock-up hinges. As per Automotive Industry Standards, several DVP load cases are needs to be performed on hood assembly for both on body and off body conditions. The DVP load cases for off body conditions are

1) Torsional Rigidity Load:

The torsional rigidity analysis is performed to evaluate the torsional stiffness of the hood. In this load case, a load of 180 N is applied at bump stop location. To perform torsional rigidity load case, dummy hinge base is constrained for all DOF and other bump stop is constrained for Uz translation DOF. The boundary conditions are shown in Fig. 3.

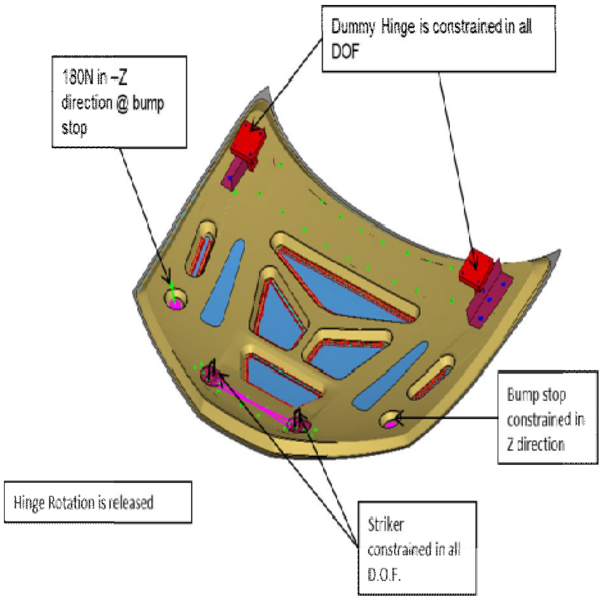

Fig. 3. Boundary Conditions for Torsional Rigidity Analysis

2) Front Corner Stiffness Load:

The front corner analysis is performed to evaluate the stiffness of the hood assembly. In this load case, a load of 150 N is applied at distance of 25mm away from the extreme corner on inner panel near bump stop location and also self-weight of the hood assembly is considered. In this load case, dummy hinge base is constrained for all DOF and bump stop is constrained for Uz translation DOF. The boundary conditions are shown in Fig. 4.

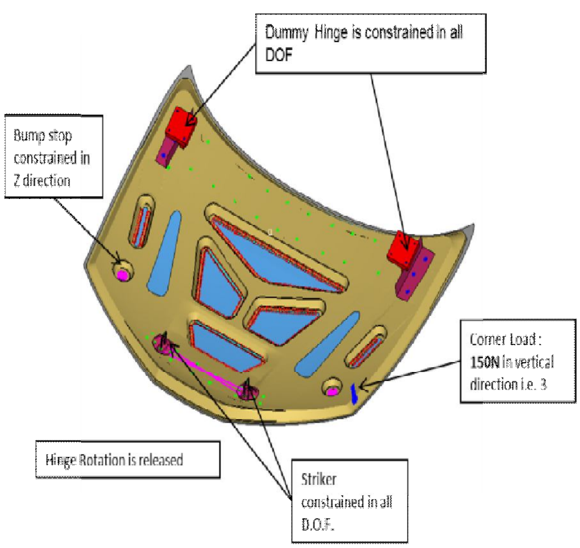

Fig. 4. Boundary Conditions for Front Corner Stiffness Analysis

3) Latch Load:

The latch load analysis is performed to evaluate the lateral stiffness of the hood assembly. In this load case, a load of 810 N is applied latch point and also self-weight of the hood assembly is considered. In this load case, dummy hinge base is constrained for all DOF and bump stop is constrained for Uz translation DOF. The boundary conditions are shown in Fig. 5.

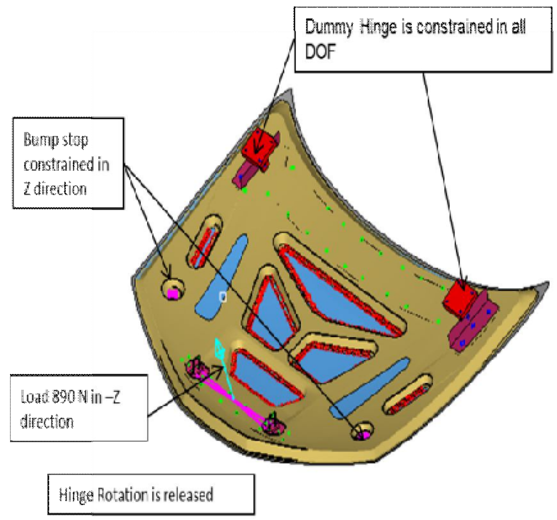

Fig. 5. Boundary Conditions for Latch Load Analysis

D. Solver and Post Processor

The nonlinear static analysis is carried out on Hood assembly for the above load cases. The commercial FE solver ABAQUS 6.131 version is used for numerical solutions. The results are post processed using Hypermesh V13.0.

IV. RESULTS AND DISCUSSIONS

The nonlinear static analysis is performed on hood assembly for rigid bed condition to assess the structural strength for Design verification procedure load cases. The material and contact non linearity effects are considered in this analysis. The induced displacement, permanent set and von-Mises stress in the hood assembly is listed in TABLE II (Refer at the end of the paper).

From the results, it is clear that the hood design does meet the design requirements for all DVP load cases. Also observed, there is a scope for design optimization to reduce the weight of the hood assembly. Under this light design iterations are carried out to reduce the weight of the hood assembly.

Design case study1 is performed on hood assembly for same DVP load cases. In this iteration, the inner panel gauge thickness is reduced to 0.9mm from 1.0mm and outer panel thickness is reduced to 1mm from 1.1mm. With these two changes total mass of the hood assembly reduced to

10.19Kg from 11.08 Kg. From the results, it got cleared that the hood design does meet the design requirements for all DVP load cases except latch load and oil canning load cases. Hence change in gauge thickness will not meet the design requirements; hence this case study have been omitted and not suggested for optimization.

Case study 2 is performed in this iteration, the topology optimization i.e. shape optimization is carried out on inner panel and rear reinforcement panel. Based on the previous iterations results, it is observed that stress is concentrated at critical regions and no stress is observed at planar region of the inner panel. Hence the additional cut-outs are made at inner panel which will not affect the strength of the hood and to reduce the weight of the inner panel. Similarly on rear reinforcement bracket cut-out length is extended which will not affect the strength. The geometry changes made on rear bracket and inner panel are shown in Fig. 6 and Fig. 8 respectively.

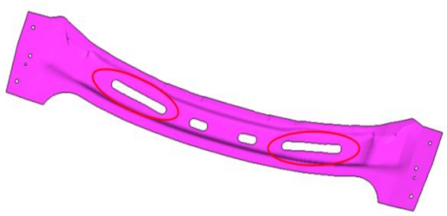

Fig. 6. Case study 2 model of Rear Reinforcment Bracket

The slot dimensions of the rear reinforcement bracket are increased to 190*25mm from 60*25mm as shown in Fig. 7. With these design changes the weight of the reinforcement bracket is reduced to 0.8 kg from 1 kg.

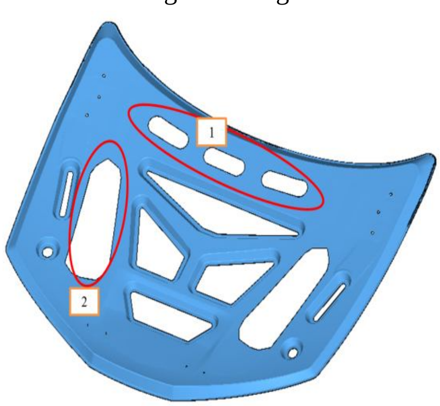

Fig. 7. Case study 2 model of Inner Panel

On hood inner panel majorly two design changes are made,

1. The additional three slots are made at rear mount bracket location of dimensions 190*80mm as shown in Fig. 7.
2. The dimensions of the slot near bump stop mounting location are increased to 4800*160 mm from 290*70 mm as shown in Fig. 7. With these two topology changes the weight of the inner panel is reduced to 3.22kg from 4.2 Kg.

With these changes the total weight of the hood assembly is reduced to 9.9 Kg from 11.08 Kg. The results of case study 2 are discussed below.

A. Latch Load

Latch load is performed on hood for the following two steps

1) Gravity Load + Latch Load (890N):

The maximum displacement induced in the model after latch loading is 19.2mm at latch locations. The induced displacement is less than the allowable limit of 25mm. Hence the model meets the design requirements.

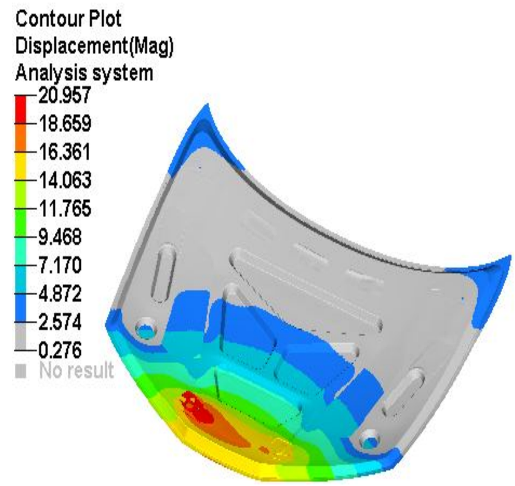

Fig. 8. Displacement Plot for Gravity + latch load

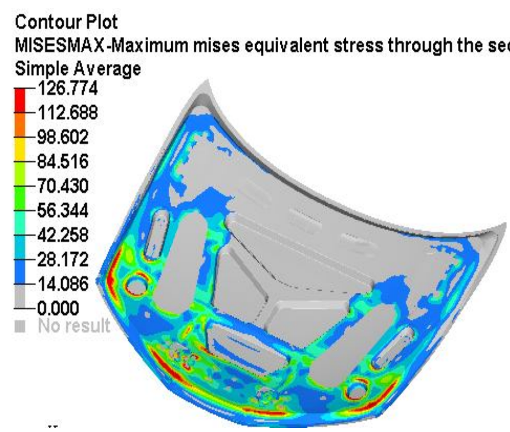

Fig. 9. Von-Mises stress Plot for Gravity load+ Latch Load

The induced von-Mises stress in the model is 127 MPa at the striker mount plate. The induced stress levels are less that the allowable yield limit of the material. The factor of safety is 1.1 and design meets the requirements.

2) Unloading:

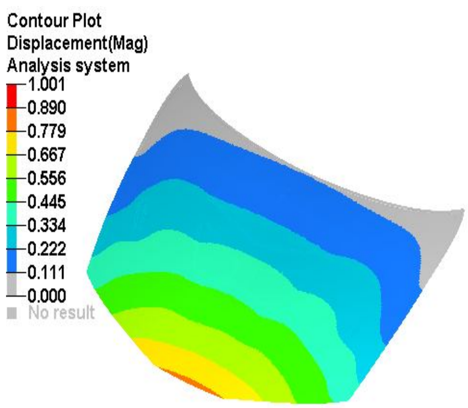

Fig. 10. Displacement Plot for Unloading step

The permanent set induced in the model after unloading is 1.0mm and is less than the Pset target of 1.2mm.

B. Torsional Rigidity Analysis

1) Loading 180N:

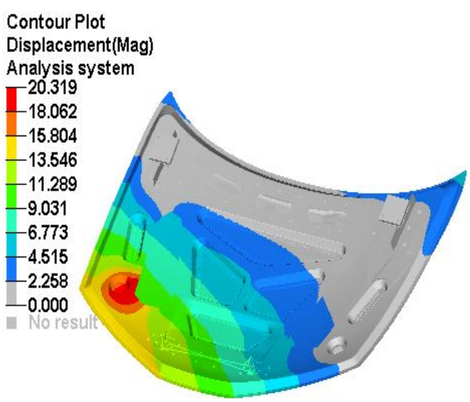

Fig. 11. Displacement Plot for loading step

The maximum displacement induced in the model is 20.39mm at bump stop location. The induced displacement is less than the target value of 35mm and hence design is safe.

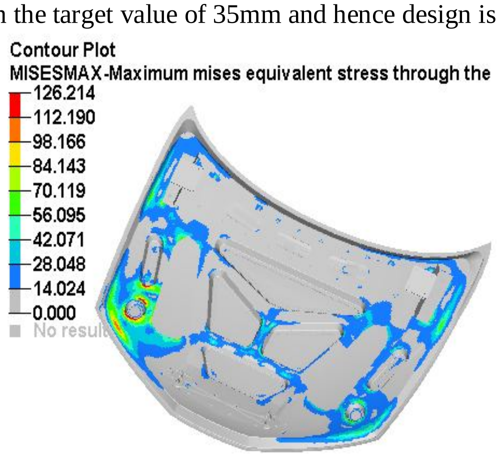

Fig. 12. Von-Mises stress Plot Loading

The maximum von-Mises stress induced in the model is 126.2 MPa which is less than the yield stress of the material. The static factor of safety is 1.3.

2) Unloading:

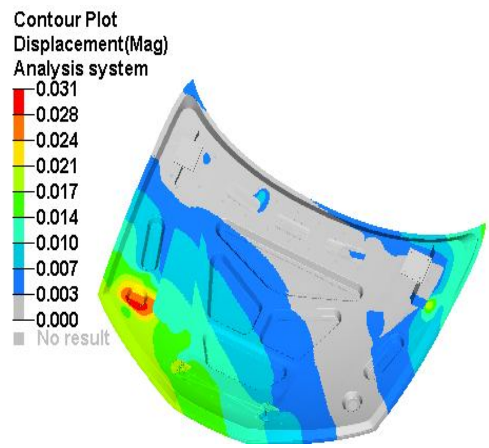

Fig. 13. Displacement Plot for Unloading step

The induced permanent set in the hood assembly is 0.031mm which meets the design target of 0.8mm Pset.

C. Front Corner Stiffness Analysis

Front corner stiffness analysis is carried out on hood assembly for the following

1) Gravity Load step:

The displacement induced in the model after gravity step is 0.27mm

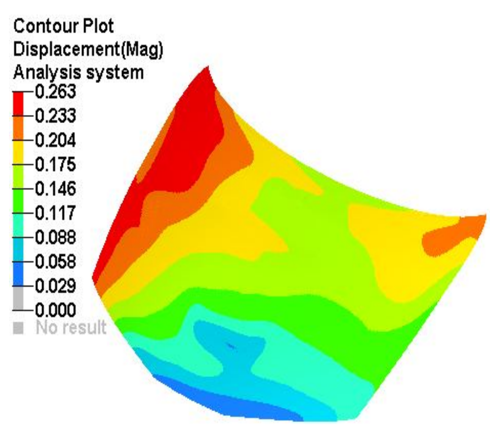

Fig. 14. Displacement Plot for gravity step

2) Gravity + Corner Load (150N) Step:

The maximum displacement induced in the hood model after loading step is 2.708mm as shown in Fig. 15. The induced maximum Displacement is less than the target value of 4mm.

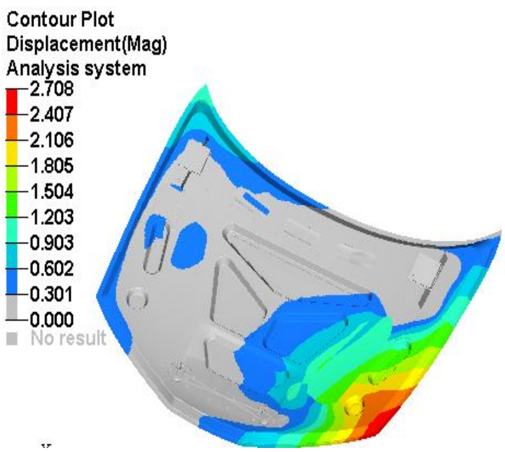

Fig. 15. Displacement Plot for gravity + loading step

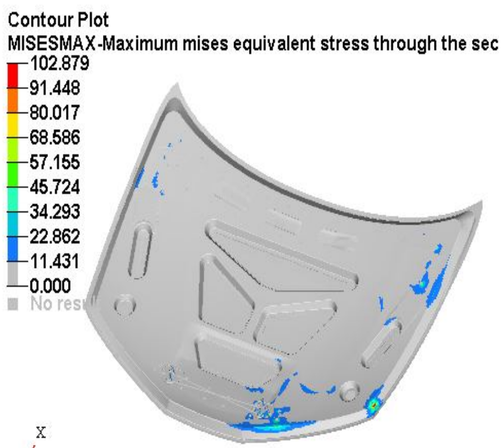

Fig. 16. Von-Mises stress Plot

The von-Mises stress in the model after loading step is 102 MPa at loading location as shown in Fig. 16.

3) Unloading:

The permanent set induced in the model is 0.27mm after unloading which is less than the target value of 0.8mm. The deformation in the hood assembly is shown in Fig. 17.

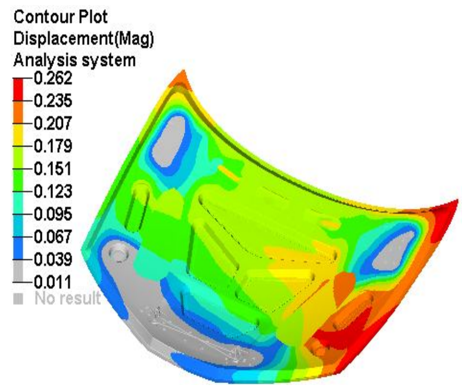

Fig. 17. Displacement Plot for unloading step

D. Summary

The induced displacement, permanent set and von-Mises stress in the hood assembly is listed in TABLE III. From the results it is observed that, the topology changes made on the hood inner panel and rear reinforcement brackets results are meeting the design requirements. The induced stress level is less than the allowable yield limit of the material for all load cases. Also the maximum displacement and permanent set after unloading cases is meeting target for all load cases. The total weight of the hood assembly is reduced to 9.9 Kg from 11.08Kg. Hence the optimized design is recommended for the further analysis.

V. CONCLUSIONS

The reduction in gauge thickness helps in reducing the total weight of the hood assembly but, it increases induced stress levels, Pset in the hood. Hence the reduction in gauge thickness can't be used. From results, it is clear that design Case Study2 does meet target for all load cases hence the design changes made on hood components can be implemented. The total weight of hood assembly of Case Study2 is reduced by ~10% compare to baseline design. The displacement induced in the hood assembly is increased due to increase in flexibility due to removal of material but it does meet the target. From FEA results, it concluded that design Case Study2 performs better than design Case Study1. Hence Case Study2 changes can be recommended.

TABLE II
DVP LOAD CASES SUMMARY

Load cases	Displacement (mm)		Von-Mises Stress (MPa)		Permanent Set (mm)	
	Target	Measured	Target	Measured	Target	Measured
Latch Load	< 25	19.2	< 150	136	< 1.2	0.87
Torsional Rigidity	< 35	17.8	< 150	122	< 0.8	0.03
Front Corner Stiffness	< 4	2.5	<150	101	< 0.8	0.27

TABLE III
SUMMARY OF CASE STUDY2 RESULTS

Load cases	Displacement (mm)		Von-Mises Stress (MPa)		Permanent Set (mm)	
	Target	Measured	Target	Measured	Target	Measured
Latch Load	< 25	21.0	< 150	127	< 1.2	1.00
Torsional Rigidity	< 35	20.3	< 150	126	< 0.8	0.03
Front Corner Stiffness	< 4	2.7	<150	103	< 0.8	0.26

REFERENCES

[1] Rupesh Rodke, Dileep Korade “Design and Development of Passenger Car Hood using FEA” in IERJ, Special Issue 2, Page 2028-2033, 2015, ISNN 2395-1621

[2] Rupesh Y. Bhagat, Amol P. More, “A Review: Analysis and Optimization of Car Bonnet” in IJIRD, Vol 3, Issue 1, January 2014

[3] Anand Vikram Singh, Jyothi Prasad Gooda, “Static and Impact Analysis of a Composite Engine Hood Assembly for Improved Characteristics” in Altair Technology Conference, 2015

[4] N. Bhaskar, P. Rayudu, “Design and Analysis of a Car Bonnet” in International Journal of Current Engineering and Technology, Vol.5, No.5, Oct 2015

[5] D. Costi, E. Torricelli, L. Splendi and M. Pettazzoni, “Optimization Methodology for an Automotive Hood Substructure (Inner Panel)” in Proceedings of the World Congress on Engineering, Vol III, WCE 2011

[6] Kiran Kausadikar, Pankaj Bhirud and Chetan Khadsare, “Optimization An Effective Tool in Bonnet Design cycle” in Altair Technology Conference, 2013