IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
ISO 3297:2007 Certified
Vol. 3, Issue 8, August 2016

Design Verification Procedure (DVP) Load Case Analysis of Car Bonnet

Mahesha J[1], Prashanth A S[2]
M.Tech Student, Machine Design, Dr. A.I.T, Bangalore, India[1]
Asst. Professor, Department of Mechanical Engineering, Dr. A.I.T, Bangalore, India[2]

Abstract: The car engine hood or bonnet has been analysed for its strength assessment for the different Design Verification Procedure (DVP) load cases. The nonlinear static structural analysis has been carried out on hood assembly to evaluate the stresses, displacement and plastic strain induced in the hood 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 been considered for the analysis work. The base line model been analysed and from the baseline results, design optimization like thickness and shape topology operation carried out to reduce the overall weight of the hood assembly. The strength assessment of hood assembly is important and it should meet the design standards. The hood assembly is important subsystem in car from safety point of view as well as from appearance aspect. During crash of an automobile, the car hood component absorbs crash kinetic energy and avoids major impact to passengers and drivers. Hence, it is important to assess the strength of hood assembly.

Keywords: Hood assembly, Strength, Stiffness, FEM Analysis.

I. INTRODUCTION

In the competitive business the automobile companies have to update for the aesthetic look of vehicle with its efficiency, service and cost of vehicle. Bonnet is an important component of the front portion of the car which is used for many purposes. Bonnet is used to decorate front portion of the car and to add luxurious look. It is made aerodynamic in shape to reduce air effect.

Generally it is used to cover car engine, radiator and many other parts, therefore bonnet must be designed in such a way that all the maintenance parts should be ease of access and it should give a minimum impact of external disturbance on the engine. 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. So there is a need for analysis of bonnet system. For that research has been done on the existing design with its limitation. This project focuses on Analysis and techniques used for optimization of Bonnet.

II. LITERATURE SURVEY

Rupesh Rodke, Dileep Korade[1] discussed about hood design and development for new project with two durability load cases are considered, 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 has been done to fulfil the acknowledgment criteria of durability tests. This paper focuses on examination and strategies employed for improvement of hood assembly with the assistance of CAD and FEA tools. The design has modified for the increased strength, weight reduction and also reduced process time and cost. Number of CAE iteration has done and as per that development has done on hood assembly.

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. 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 MODELLING

A. FEM Method

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

IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
ISO 3297:2007 Certified
Vol. 3, Issue 8, August 2016

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. Finite element analysis is a vast simulation process and FE modelling is the application tool of this analytical process. This modelling tool is used for accurate calculation of deflection, stresses, natural frequencies and dynamic response of mechanical systems without costly experiments, were most the product cost is incurred. Element types like non-linear contact, solid modelling, beams and shells, lumped mass, springs/dampers, etc. Fig.1 and Fig. 2 shows the Finite element modelling of Outer and Inner Hood assemblies respectively.

Fig. 1. Outer Hood Assembly FE Model

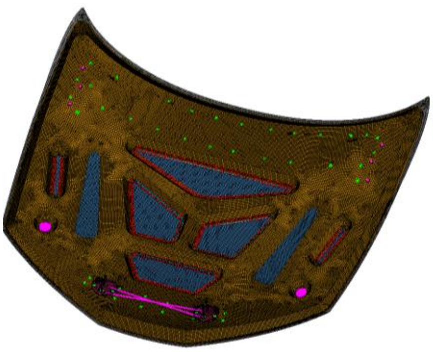

Fig. 2. Inner Hood Assembly FE Model

B. Contact Definition

In hood assembly, the reinforcement brackets overlaps on inner and outer panel. In FE model, these brackets are connected both through bolts and spots at desired locations.

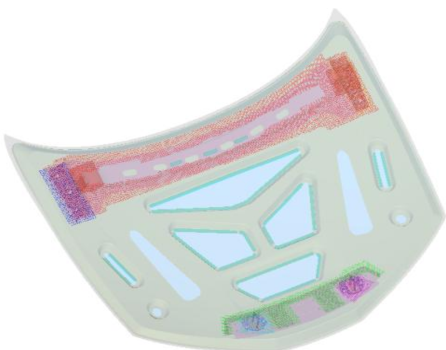

Fig. 3. Frictional Contact at inner panel

TABLE I THE HOOD COMPONENTS GAUGE THICKNESS, MATERIAL DETAILS

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

C. Load and 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. 4.

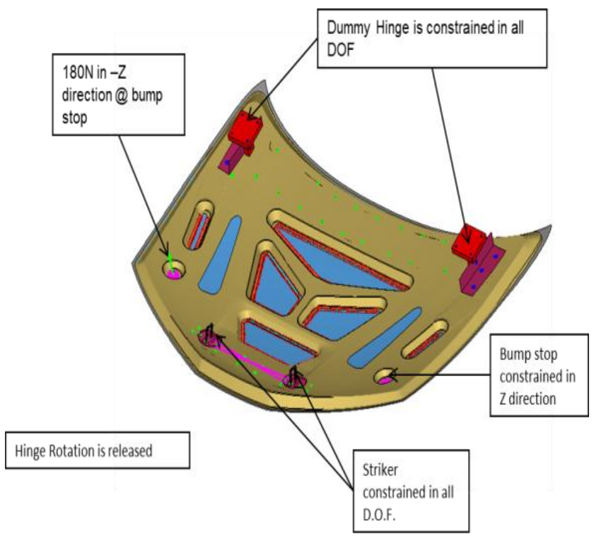

Fig. 4. Boundary Conditions for Torsional Rigidity Analysis

IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
ISO 3297:2007 Certified
Vol. 3, Issue 8, August 2016

Measurable targets are,
1. The displacement should be less than 35mm
2. Stress should be within allowable yield stress
3. Permanent set should be less than 0.8mm

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. 5
Measurable targets,
1. The displacement should be less than 4mm
2. Stress should be within allowable yield stress
3. Permanent set should be less than 0.8mm

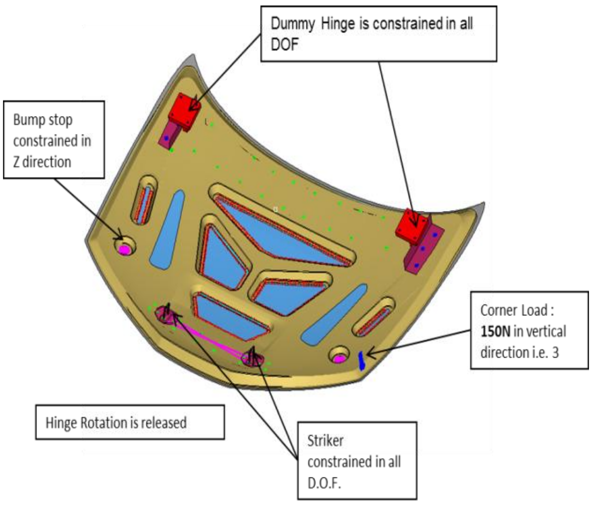

Fig 5 Boundary Conditions for Front Corner Stiffness Analysis

3) Latch Load:
The latch load analysis is performed to evaluate the lateral stiffness of the hood assembly.

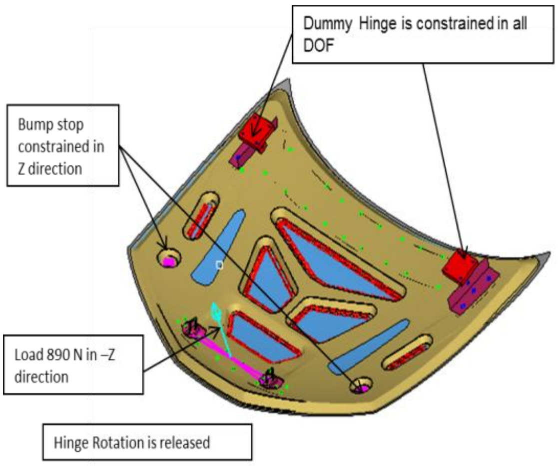

Fig. 6. Boundary Conditions for Latch Load Analysis

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. 6.
The Latch Load Measurable targets are as follows,
1. The displacement should be less than 25mm
2. Stress should be within allowable yield stress
3. Permanent set should be less than 1.2 mm.

4) Modal Analysis:
The Modal Analysis is performed to evaluate the natural frequency of the hood and its mode shape. The natural frequencies of the hood should not fall between the operating frequency ranges to avoid the structure failure due to resonance. To performing model analysis dummy hinge base is fixed for all DOF and striker is fixed for all DOF and bump stop is fixed for Uz. The boundary conditions are shown in Fig. 7.
Measurable targets,
1. Natural frequency of the model should be above the operating frequency range

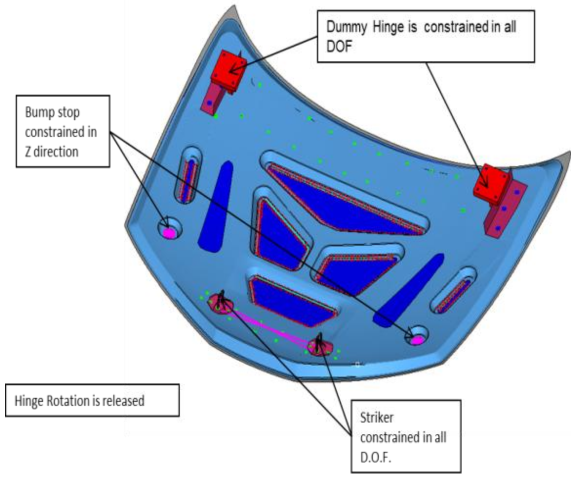

Fig. 7. Boundary Conditions for Modal Analysis

5) Oil Canning Analysis:
Oil canning analysis is performed on outer panel of hood assembly to determine the local stiffness of the panel at the position where load is applied. The load is applied through Indentor of circular shape having diameter of 100m and thickness of 5mm in the direction of normal to Indentor. The loads are applied at weak locations; these weak locations are identified through modal analysis. The oil canning analysis is carried out for location having maximum displacement in modal analysis results. A load of 100 N and 220 N is applied weak location through Indentor. The boundary condition for modal analysis is shown in Fig. 8.
Measurable targets are,
1. No oil canning should be there in the model
2. Maximum displacement should be less than 5mm and no Permanent set exist in the model
3. Stresses should be within the yield limit of the material
4. Permanent set should be less than 0.5mm

IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
ISO 3297:2007 Certified
Vol. 3, Issue 8, August 2016

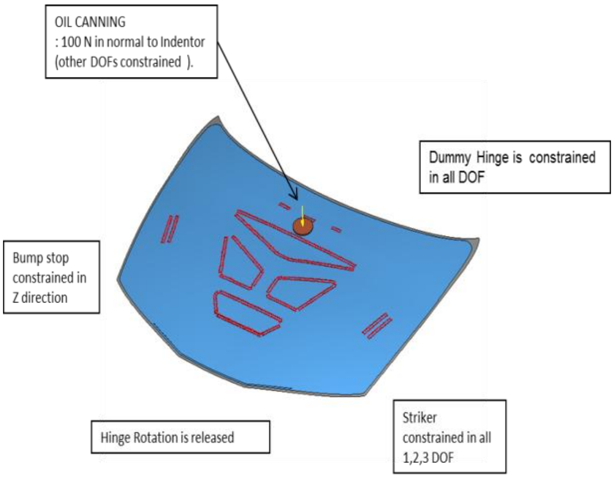

Fig8 Boundary Conditions for Oil canning Analysis

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.

A. Latch Load

Latch load is performed on hood for the following three steps

1) Gravity Load:

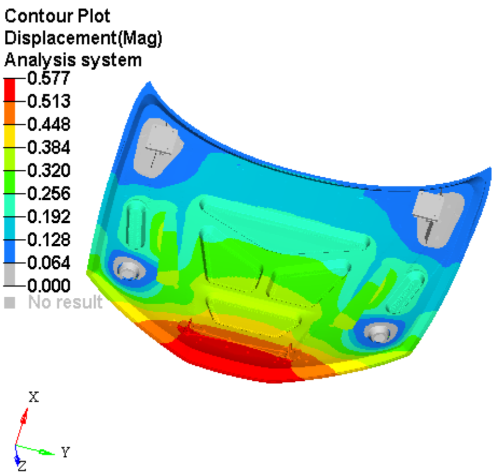

Fig. 9. Displacement Plot for Gravity load

The maximum displacement induced in the model at the end of gravity load step is 0.58mm and is shown in Fig. 9. The maximum displacement is observed near latch locations.

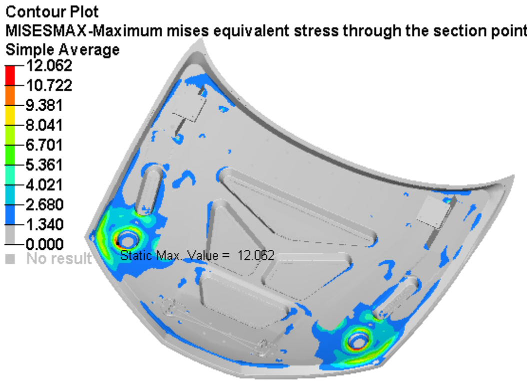

Fig. 10. Von-Mises stress Plot for Gravity load

The maximum von-Mises stress induced in the model after gravity step is 12 MPa at bottom fillet of the bump stop location is shown in Fig. 10.

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

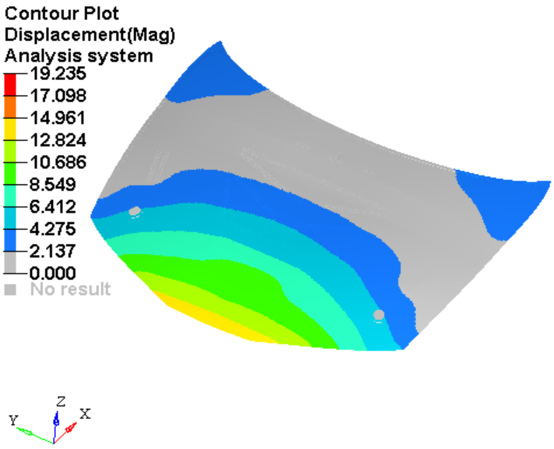

Fig. 11. Displacement Plot for Gravity + latch load

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.

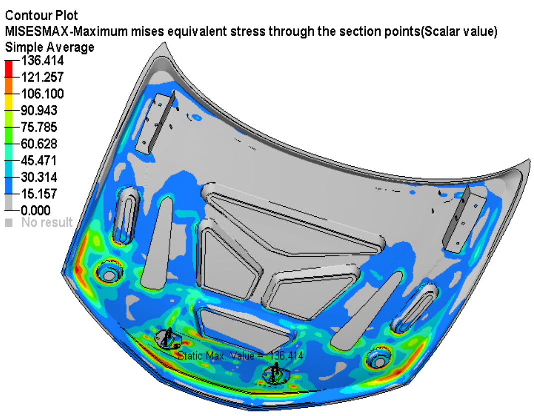

Fig. 12. Von-Mises stress Plot for Gravity load+ Latch Load

IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
ISO 3297:2007 Certified
Vol. 3, Issue 8, August 2016

The induced von-Mises stress in the model is 136MPa 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.

3) Un Loading:
The permanent set induced in the model after unloading is 0.88mm and is less than the Pset target of 1.2mm

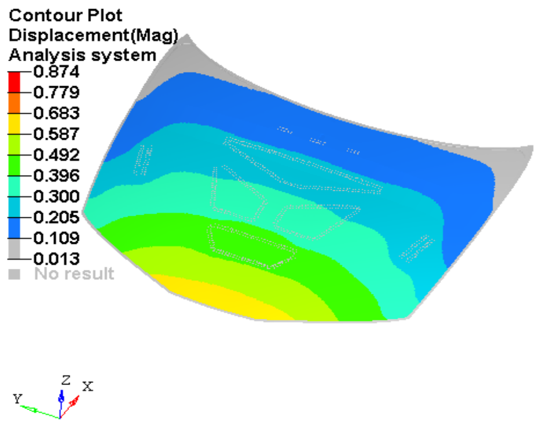

Fig. 13. Displacement Plot for Unloading step

B. Torsional Rigidity Analysis
The torsional rigidity analysis is performed on hood for the following 2 steps

1) Loading Step:

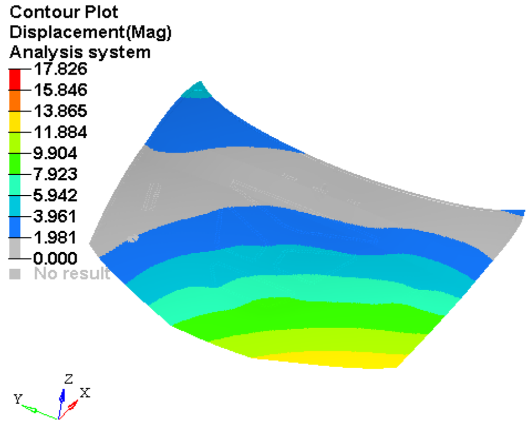

Fig. 14: Displacement Plot for loading step

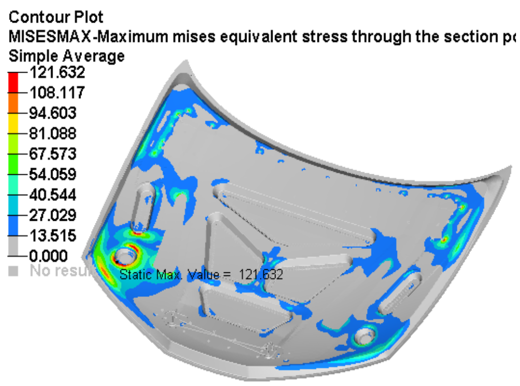

Fig 15 Von-Mises stress Plot Loading

The maximum displacement induced in the model is 17.83mm at bump stop location. The induced displacement is less than the target value of 35mm and hence design is safe.

Induced maximum von-Mises stress in the model is 122 MPa which is less than the yield stress of the material. The static factor of safety is 1.3.

2) Unloading Step:

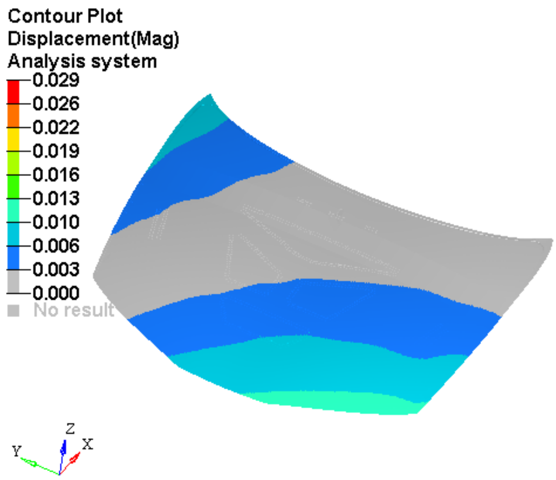

Fig 16 Displacement Plot for Unloading step

The induced permanent set in the hood assembly is 0.029mm 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:
Fig. 17 shows the Gravity Load step for Front Corner Stiffness analysis. The maximum displacement after the Gravity Load step falls below the allowable displacement so the design is safe under this load condition.

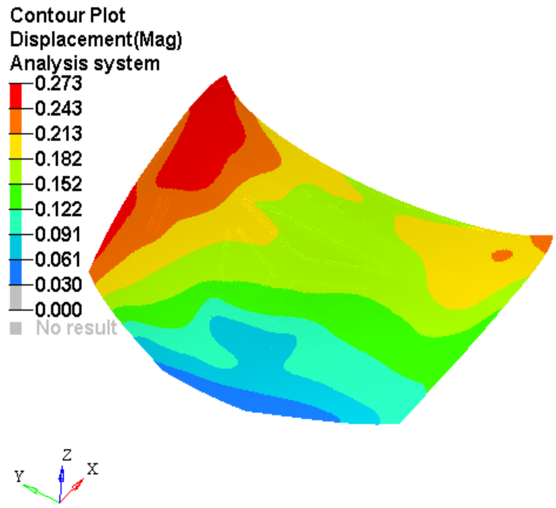

Fig. 17. Displacement Plot for gravity step

DOI 10.17148/IARJSET.2016.3816

IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
ISO 3297:2007 Certified
Vol. 3, Issue 8, August 2016

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

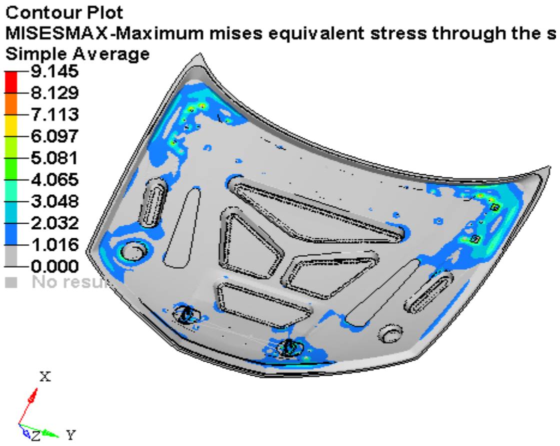

Fig. 18. Von-Mises stress Plot

The von-Mises stress induced in the model after gravity step is 9MPa.

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

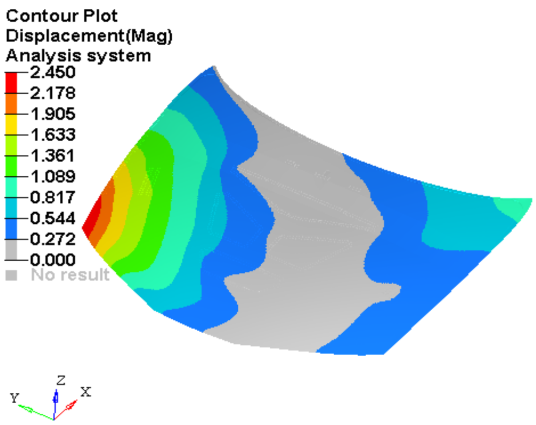

Fig. 19. Displacement Plot for gravity + loading step

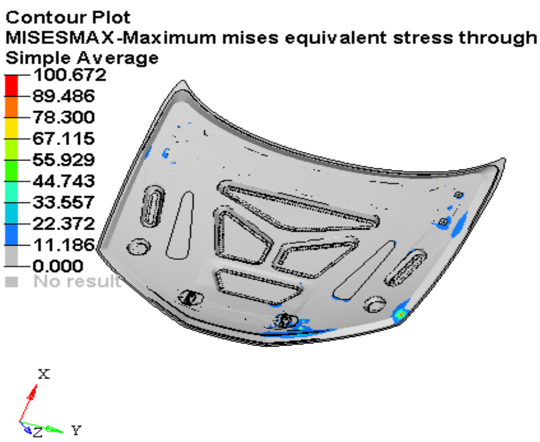

Fig. 20. Von-Mises stress Plot

The maximum displacement induced in the hood model after loading step is 2.45mm and is shown in Fig. 19. The induced maximum Displacement is less than the target value of 4mm. The von-Mises stress in the model after loading step is 101 MPa at loading location and is shown in Fig. 20. The induced stress is less than the yield strength of the material and static factor of safety is 1.5.

3) Unloading:

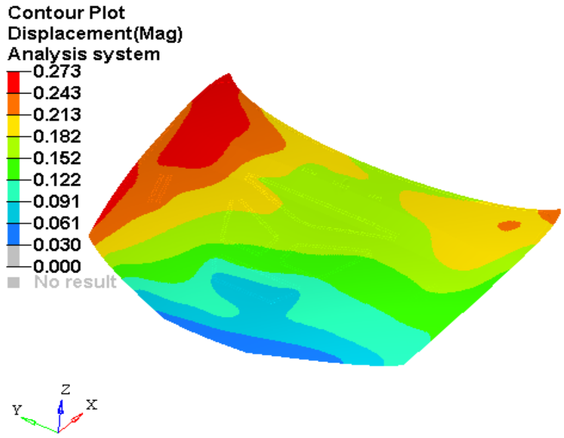

Fig. 21. Displacement Plot for unloading step

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. 21.

D. Modal Analysis

The modal analysis is carried out on hood assembly for rigid bed condition. The induced natural frequency is shown in TABLE II

TABLE II MODAL ANALYSIS FREQUENCY LIST

Mode No	Natural Freq. (HZ)	Mode No	Natural Freq. (HZ)	Mode No	Natural Freq. (Hz)
1	37.737	6	72.289	11	115.22
2	43.836	7	92.308	12	124.65
3	56.777	8	103.21	13	125.95
4	63.788	9	104.46	14	126.55
5	71.276	10	113.59	15	133.13

The first natural frequency is observed at 37.73Hz and mode shape is shown in Fig. 22 to Fig. 25.

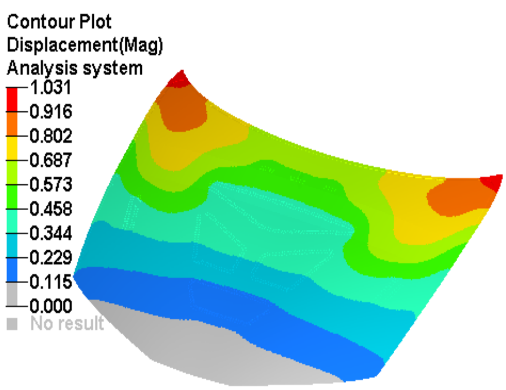

Fig. 22. Mode 1 at 38Hz

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

ISO 3297:2007 Certified

Vol. 3, Issue 8, August 2016

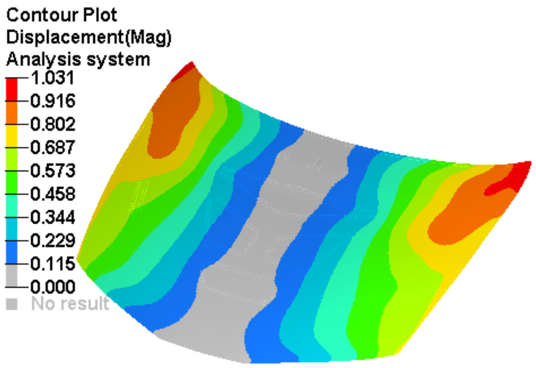

Fig. 23. Mode 2 at 44 Hz

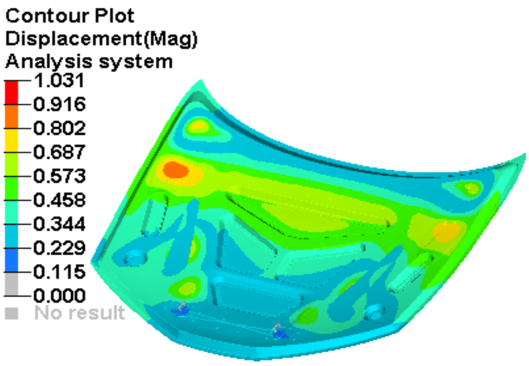

Fig. 24. Mode 3 at 57Hz

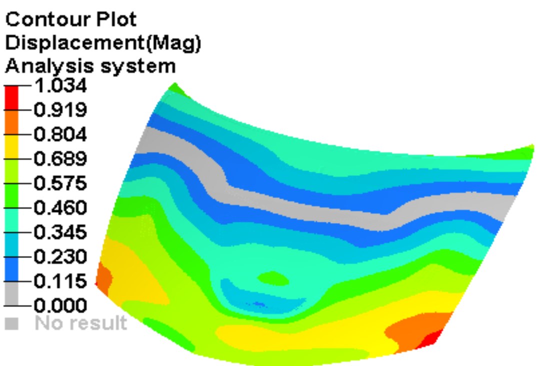

Fig. 25. Mode 4 at 64 Hz

E. Oil canning Analysis

Oil canning analysis is carried out on hood assembly for the following

1) 100 N Load:

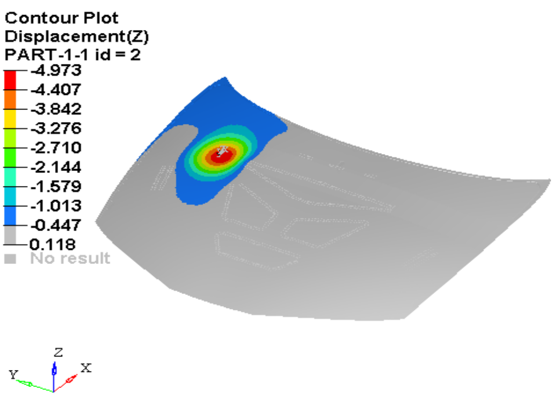

Fig. 26. Displacement Plot for 100N at loc1

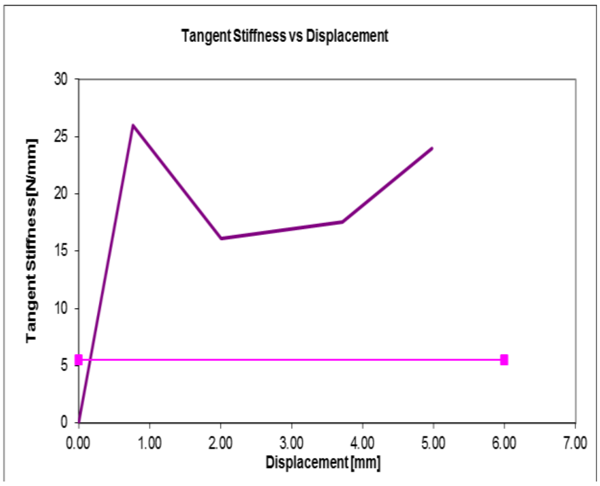

Fig. 27. Tangent stiffness vs displacement at loc1

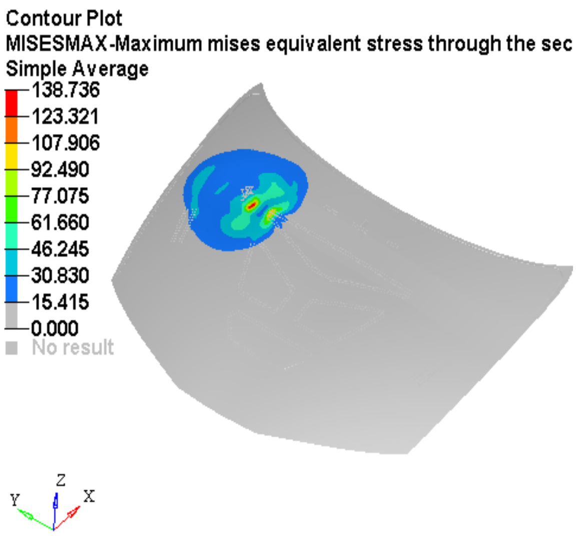

Fig. 28. Von-Mises stress Plot at Loc1

2) 220N at Loc2:

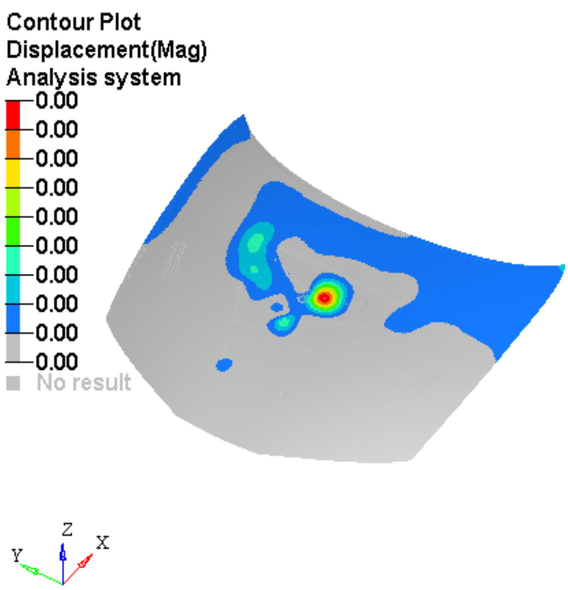

Fig. 29. Pset Plot for 220N at loc2

IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
ISO 3297:2007 Certified
Vol. 3, Issue 8, August 2016

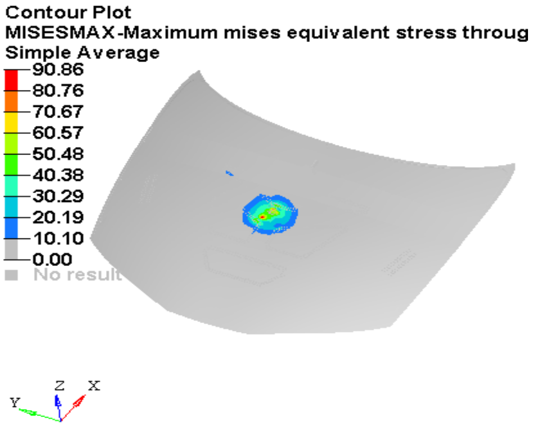

Fig. 30. Von-Mises stress Plot at Loc2

F. Summary
The induced displacement, permanent set and von-Mises stress in the hood assembly are listed in Table III (Refer Table below).

V. CONCLUSIONS

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 to be carried out to reduce the weight of the hood assembly.

TABLE III 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
Oil Canning	< 6	4.9	<150	139	< 0.5	0.00

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