International Journal of Enhanced Research in Science Technology & Engineering, ISSN: 2319-7463
Vol. 4 Issue 6, June-2015, pp: (473-479), Impact Factor: 1.252, Available online at: www.erpublications.com

Optimization and NVH analysis of Instrument Cluster

Vijay Raghunath Darade[1], Bala Bhaskar Rao Gamini[2], Prof. M.M.Ghadmode[3]

[1,3]Amrutvahini College of Engineering, Sangamner, Pune University, Maharashtra
[2]Lead Analyst, CAE, Visteon Engineering Center India, Pune -18

Abstract: This paper deals with NVH Analysis of an Instrument Cluster. Based on Modal analysis the first natural frequency of base model observed more stress levels as compared with allowable fatigue strength of material. So, it is required to increase first natural frequency of cluster with topology optimization. After implementation of topology optimization the first natural frequency of cluster increased and stress level on components of assembly comparatively below allowable limit of the material i.e. all the components of cluster assembly can withstand for road input excitation.

Keywords: CAE, NVH, Vibration Analysis, Instrument Cluster, Topology Optimization.

INTRODUCTION

Instrument cluster is an Electronics device under driver information category which is used to inform driver and passenger the various condition of vehicle through light emission and pointer gauges. The various information are speed of vehicle, fuel level indicator, temperature of engine coolant, taco-meter, warning symbols such as door ajar, engine check, seat belt etc. The goal of modal analysis in structural mechanics is to determine the natural mode shapes and frequencies of an object or structure during free vibration. Natural frequency is the number of times a system will oscillate between its original position and its displaced position, if there is no outside interference. Resonance is a phenomenon where by large amplitudes of vibration or shaking occurs when the frequency of the external load matches one of the natural frequencies of the structure. It is very hard to control the input frequencies created by road surfaces. So, there are Change the natural frequency of the system is practical ways to control the magnification factor or response of any system. Optimization demonstrates its potential benefits when it is used in the early product design stage.

Fig. 1: Instrument Cluster

OBJECTIVE

- To carryout Modal analysis of an Instrument cluster for natural frequency.
- Topology and Topography Optimization to increase the first natural frequency of an Instrument Cluster to meet target frequency.
- To carryout NVH analysis of an Instrument cluster after Topology Optimization for natural frequency and frequency response to find stresses & displacements.
- Correlate the CAE results with the Physical Test results using Load data.

METHODOLOGY

- Import CAD model of an Instrument Cluster
- Meshing of all the parts which are going through the preprocessor HYPERMESH 12.0.
- NVH analysis by using the solver NASTRAN.
- Find out the results in HYPERVIEW 12.0.
- Compare Natural frequency with Target frequency
- Topology Optimization, NVH analysis and CAE results
- Physical test and correlation with CAE results.

NVH ANALYSIS AND RESULTS

Analysis of the cluster is going through the three steps,

1. Preprocessor,
2. Solver,
3. Post processor.

The preprocessor we used Hypermesh 12.0 for the meshing all the parts. While meshing we have to follow the quality criteria as per Visteon guidelines. Here we have to mesh the Lens, Mask, Mounting Plate, Metal bracket, Rear cover, PWB (Printed Wiring Board).

a. Meshing.
For model making first step is to convert the 3D CAD data into exact mesh model and mesh quality is ensured during this activity.

b. Assigning material and property to components.
Material properties like Poisson's ratio, density, Young modulus and thickness property are assigned to each and every component of the model so as to represent the actual vehicle conditions.

c. Making Connections
A host of connectors available in the software are used to simulate Weld spots / Bolts / CO2welds / Bolt connections to replicate actual vehicle conditions.

d. Assigning constraints / boundary conditions
Loads are defined in terms of their numerical value, direction and assigned to the areas as per the regulation requirements. The constraint boundary conditions are provided on case basis.

Material Properties

Table 01: Material Properties

Sr. No.	Part Name	Material	Thickness range (mm)	Density (kg/m3)	Young's Modulus (MPa)	Poisson's ratio
1	LENS	PMMA	1 to 2	1180	3300	0.35
2	MASK	ABS	1.5 to 4.0	1050	2400	0.36
3	MOUNTING PLATE	PPT40	1 to 5	1180	4100	0.35
4	METAL BRACKET	STEEL	0.5 to 0.8	7900	210000	0.30
5	PWB	FR4	1.57	3100	18000	0.17
6	REAR COVER	ABS	1.5 to 4	1050	2400	0.36

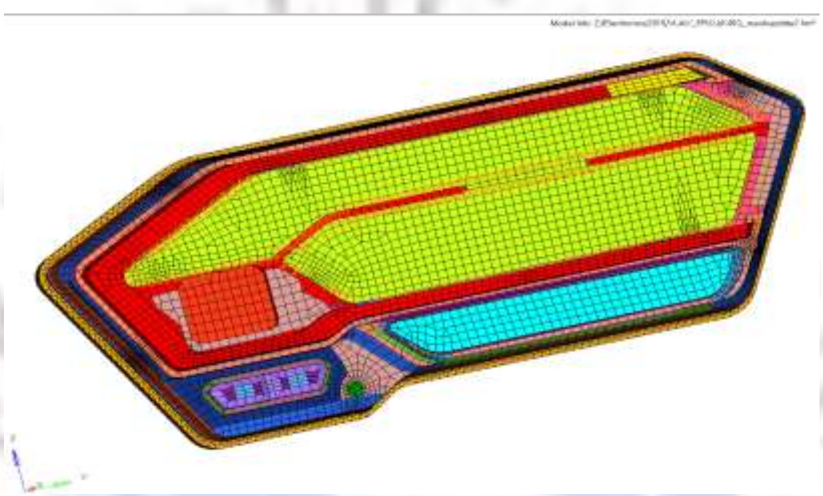

Fig. 02: Meshed model of Instrument Cluster

Modal analysis of base model and results

First natural frequency is observed in results of Modal Analysis of base model of an Instrument Cluster 157.7 Hz which is below than target frequency i.e.200Hz.

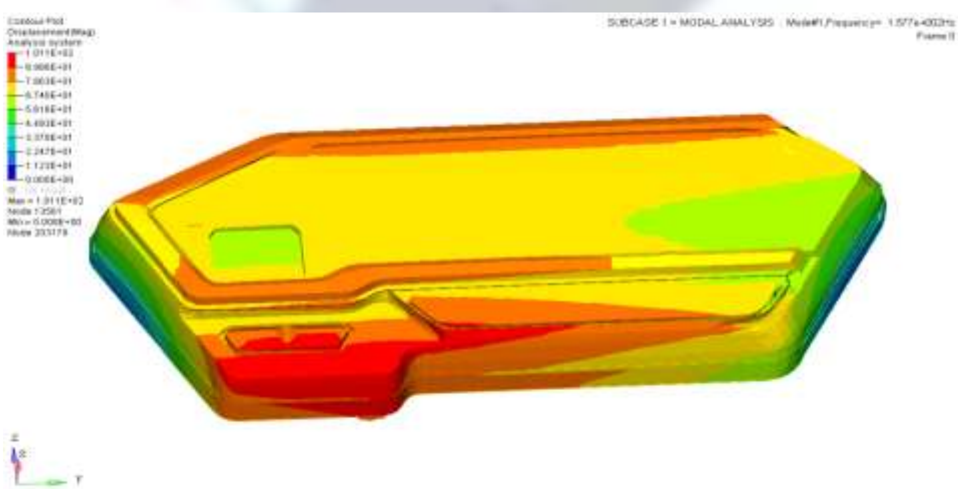

Fig. 03: Modal analysis for first natural frequency of base model 157.7Hz

Frequency Response Analysis

Frequency response analysis is used to calculate the response of a structure about steady state oscillatory excitation. The analysis is to compute the response of the structure under certain input loading due to Road surface input loads. We give the input excitation acceleration of 10g with frequency range of 50 to 2000Hz, and we measure the frequency response as displacement and stress in X, Y and Z directions. Measuring frequency response, Point 1is placed to mounting point where input excitation is given by 10g, Point2 is placed at some distance by experience from mounting point to measure displacement and stress where it can be seen more at that place.

OPTIMIZATION

Different optimization techniques that are adopted for optimization can be classified into three main categories as follows:

1. Load Path Optimization (Topology)
Topology can be defined as a tool for optimal Load path identification i.e. material distribution should be only on places where it is required and hence elimination of surplus material

2. Bead Placement Optimization (Topography)
Topography can be defined as optimization of bead patterns reinforcement to satisfy the input conditions like rigidity, panel deformation or natural frequency response of the part. For example natural frequency of the given part can be increased by changing the bead pattern of the part.

3. Gauge (Thickness) Optimization
Thickness plays a very important role in optimizing the weight. Region specific thickness requirements can be achieved by this tool as it gives output of thickness variations in given thickness fringe. In this case, we optimize base model of an Instrument cluster with following techniques for topology and topography optimization.

1. Adding number of ribs and increasing thickness of ribs in mounting point areas.
2. Adding connection between case inner, mounting plate and PWB.
3. Webbing to metal shield and avoid sharp corners.

Modal analysis of optimized model and results

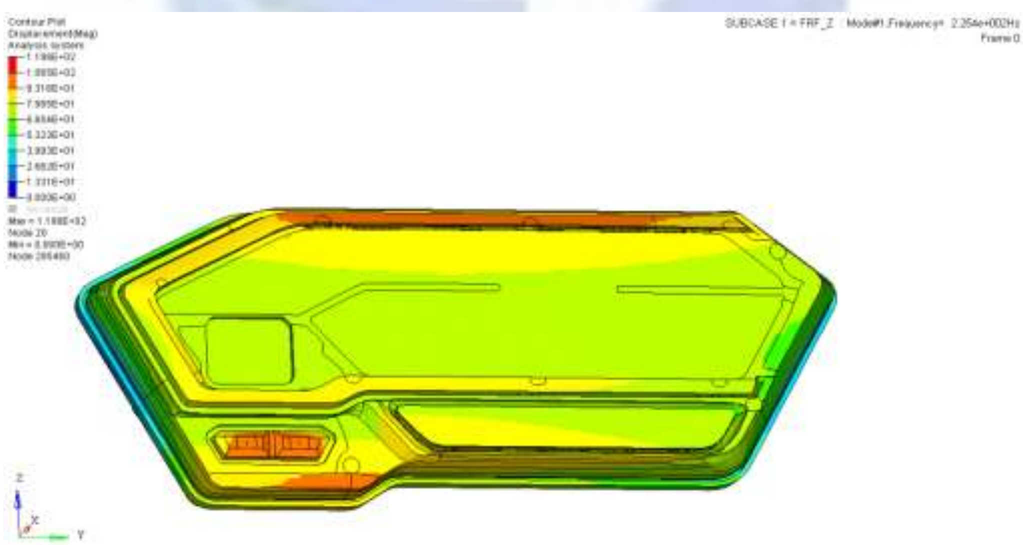

Fig.04 Modal analysis for first natural frequency of optimized model 225.4Hz

First natural frequency is observed in results of Modal Analysis of optimized model of an Instrument Cluster 225.4 Hz which is above target frequency i.e.200Hz.

RESULTS AND DISCUSSION

Based on Modal analysis the first natural frequency of base model (157.7Hz) observed more stress levels as compared with allowable fatigue strength of material so it is required to increase first natural frequency of cluster above 200 Hz with topology optimization. After implementation of topology optimization the first natural frequency of cluster increased to 225.4Hz and stress level on components of assembly comparatively below allowable limit of the material i.e. all the components of cluster assembly can withstand for given input excitation of 10g load.

Table 02: Compare results for Frequency Response Analysis of Base Model and Optimized Model of Instrument Cluster

Comparison Between Maximum Stress Results for Frequency Response Analysis of Base Model And Optimized Model of Instrument Cluster						
Direction	Part Name	Material	Max. Stress of Base Model (Mpa)	Max. Stress of Optimized Model(Mpa)	Fatigue Strength (Mpa)	Yield Strength (Mpa)
X-axis	Mask	ABS	8	9.6	~10	44
	lens	PMMA	2.7	2	14	75
	Metal bracket	Steel	236	28.9	90	120-180
	PCB	FR4	41.2	14.5		
	Rear cover	ABS	10.7	4.6	~10	44
	Mounting plate	PP-T40	18.7	10.4	20.9	31
Y-axis	Mask	ABS	11.7	15.1	~10	44
	lens	PMMA	4	2.1	14	75
	Metal bracket	Steel	193	53.1	90	120-180
	PCB	FR4	31.4	21.5		
	Rear cover	ABS	18.3	7.8	~10	44
	Mounting plate	PP-T40	26.3	18.7	20.9	31
Z-axis	Mask	ABS	11.8	11.8	~10	44
	lens	PMMA	3.5	2	14	75
	Metal bracket	Steel	164	94.5	90	120-180
	PCB	FR4	42.4	33.7		
	Rear cover	ABS	43.8	18	~10	44
	Mounting plate	PP-T40	26.8	18	20.9	31

PHYSICAL VALIDATION

The objective of the test is to determine performance & durability of cluster assembly. Test was conducted in Sine sweep vibration test as per OEMs test specifications. As shown in Figure, Instrument Cluster is mounted to shaker table (vibration machine) through fixture.

TOP **BOTTOM**

Fig. 05: Front case vibration measurement

Frequency response is the measure of any system's output spectrum in response to an input signal. The analysis is to compute the response of the structure under certain input loading due to Road surface input loads. The loads can be forces, displacements, velocity, and acceleration. They are dependent on the excitation frequency. The results from a frequency response analysis are displacements, velocities, accelerations, forces, stresses, and strains. The responses are usually complex numbers that are either given as magnitude and phase angle or as real and imaginary part.

Correlation

By physical test, the frequency response of optimized model of an Instrument cluster at X, Y and Z directions are closed to CAE results at the frequency 474.4Hz at X-axis, 401.3Hz at Y-axis and 225.4Hz at Z- axis. So from the above experiment, we find that the natural frequency from CAE result is approximately same to the frequency found out by physical test method.

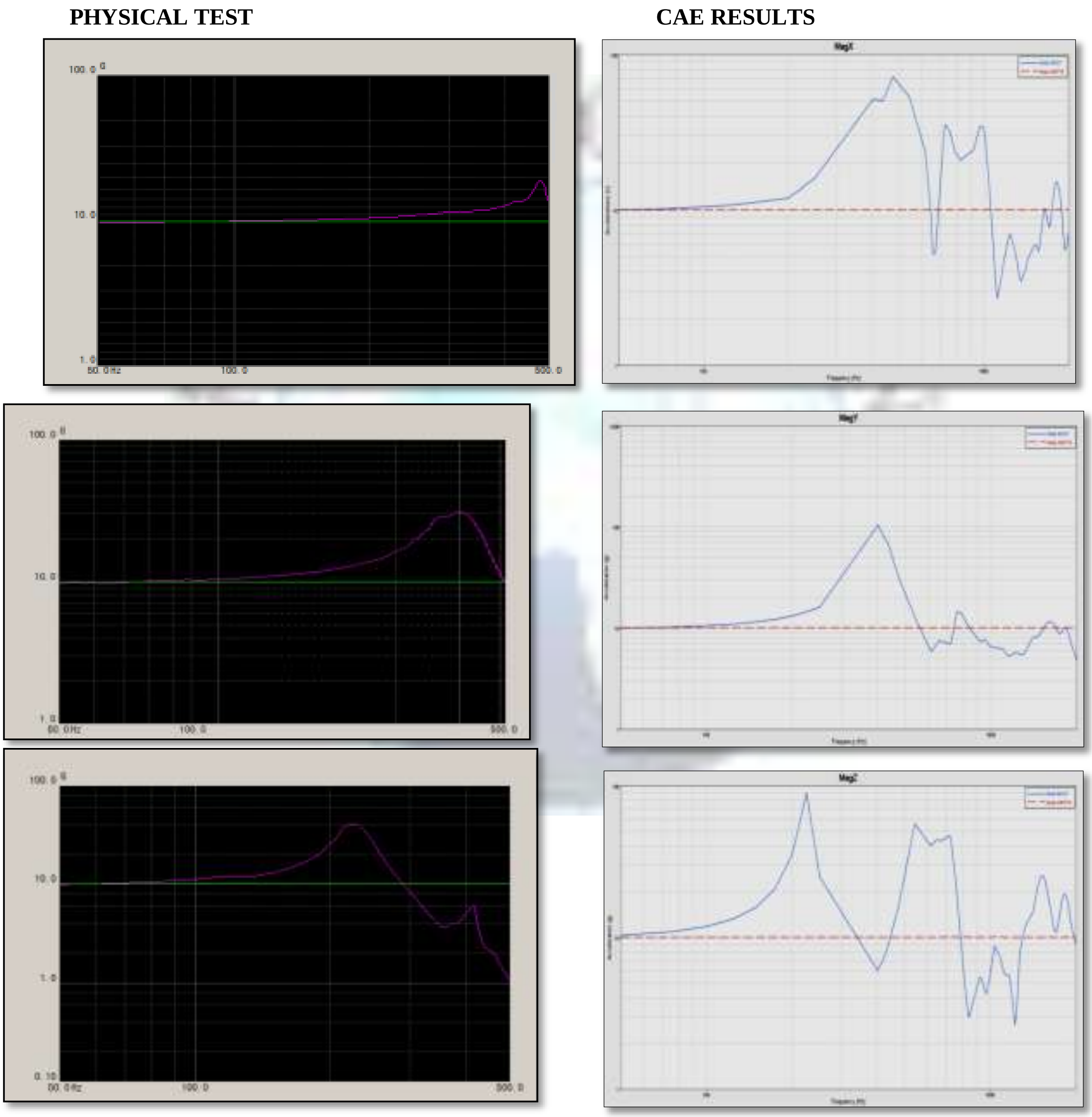

Fig. 06: Correlation shows between Frequency Response analysis by physical tests and Frequency response in X, Y and Z direction at selected two nodes at natural frequencies 474.4Hz, 401.3Hz and 225.4 Hz respectively.

CONCLUSION

Optimization method used in this report, increasing the first natural frequency of an instrument cluster from 157.7 Hz to 225.4 Hz which is above the target frequency 200Hz. Frequency Response Analysis of the optimized model of an Instrument cluster results in sustainability of instrument cluster assembly for given input excitation of 10g. By physical test on air cooled vibration test system (model: i240 /SA3M), the frequency response of optimized model of an Instrument cluster at X, Y and Z directions are closed to CAE results at the different natural frequencies.

This optimization technique gives flexibility to the designer to choose the concept as per the requirement. This method has offered considerable saving in terms of evaluating various concepts without actually building any prototype. Further it reduced the time required to arrive at the best design thus shortening the product design cycle time. We can generate designable and non-designable space as per our requirement.

ACKNOWLEDGEMENT

The authors would like to acknowledge the continual help and support of CAE team, Visteon Engineering Center India, Pune, especially Mr. Sandeep Sangale (Manager, CAE) and Prof. M. M. Ghadmode (Production Engineering Department AVCOE, Sangamner). The authors would also like to thank Prof. V. B. Shinde (Head of Production Engineering Department) and Prof. B. R. Borkar (M.E. Coordinator, AVCOE, Sangamner) for their constant encouragement.

REFERENCES

[1]. Training document of Instrument Cluster, (2015) Visteon Engineering Centre (India) Pvt. Ltd.
[2]. Kunal Deshmukh, Karan Singh, (2013),"Frequency Optimization of IP (Instrumental panel) &MFD Bracket Using OptiStruct,"Altair Technology Conference, India.
[3]. Altair University, (2014),"Practical Aspects of Finite Element Simulation-A Study Guide" Academic Program
[4]. P. Seshu, (2012),"Text Book of Finite Element Analysis," PHI Learning Pvt. Ltd. New Delhi.
[5]. Abhishek Dwivedi, Soundararajan S, (2010)," Optimization of Exhaust Hanger Location for NVH Performance," HTC Conference, India.
[6]. Nitin S. Gokhale, Sanjay S. Deshpande, Sanjeev V. Bedekar (2008),"Practical Finite Element Analysis", Finite to Infinite, India.
[7]. Andy O. Fox, Raghu Echempati, Ph.D., (2011)," Modeling, Forming, And Modal Analysis of Sheet Metal Parts Using CAE Tools," Kettering University Mechanical Engineering Flint, Michigan, USA.
[8]. S. Vinay Seeba, S. Srikari, V. K. Banthia ,(2010), "Design And Analysis Of A Plastic Door Module For Car Body Application", SASTECH, India.
[9]. Soji Yamakawa, Charles Shaw, Kenji Shimada. (2006)," Layered tetrahedral meshing of thin-walled solids for plastic injection molding FEM", Department of Mechanical Engineering, Carnegie Mellon University, 5000 Forbes Ave., Pittsburgh, PA 15213, USA
[10]. M. Vishnuvarthanan, Rajesh Panda and S. Ilangovan,(2013)," Optimization of Injection Molding Cycle Time Using Moldflow Analysis", Advanced Research School for Technology and Product Simulation, CIPET- Chennai, India.
[11]. Hariyanto Gunawan and Willyanto Anggono," Improving quality of injection mold using mold flow software simulation .Case study: new design plastic cup", (2006), Mechanical Engineering Department, Faculty of Industrial Technology Petra Christian University, Product Innovation and Development Centre Petra Christian University. Siwalankerto 121-131, Surabaya