INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH & DEVELOPMENT

ISSN: 2278 – 0211 (Online)

# Application Of CAD And CAE To The Development And Optimization Of Automobile Outer Rear View Mirror Based On The Vibration Study

**Trupti Nirmal**
Bharti Vidyapeeth University, Pune, Maharashtra, India
**Professor V. K. Kurkute**
Bharti Vidyapeeth University, Pune, Maharashtra, India

***Abstract:***
*Study of vibrations and vibration analysis has always been a need for successful design and development of new parts and components belonging to auto industry, aerospace industry, manufacturing industry and in the development machine tools.*
*Finite element model for the automobile rear view mirror was created to predict mirror vibration response based on modal analysis study. The materials used in this FEA model were initially provided by the mirror manufacturer and also some modification was done based on the study. Hypermesh Optistruct solver was used to build the complete FEA ORVM model. Vibration modes were predicted for the mirror assembly with a special focus on the mirror mounting bracket as it's the only part through which the vibrations transfers to mirror assembly in actual vehicle. The natural frequency of the first mirror FEA model was correlated with the first test results and then further iterations were done using this FEA model to predict the changes required in material and geometry to achieve the target frequency value.*
*The current paper, as a part of curricular activity of Masters program, focuses on the basics of vibration, importance of vibration study in a automobile mirror and use of CAD & CAE softwares in the development and optimization of a mirror assembly based on its mechanical behavior.*

## 1.Introduction

Automobile rear view mirrors are engineering components that must meet very stringent vibration criteria, the reflected image must not be distorted even though the mirror fixture to the windscreen or the roof or the mirror fixture to A pillar of the BIW is subjected to some kind of vibration fro. Vehicle and road. There must be no distortion regardless of the material frequencies of the mirror assembly, i.e. Even if they coincide with those of the natural frequencies of the BIW A-pillar on which the Outer Rear View Mirrors (ORVM) are normally mounted. Aspects of the design such as case material, structure and stiffness, glass weight and supports, the behaviour of the ball and socket joint and the connection of the bracket to the automobile are all critical to the vibration behaviour of the mirror. At the moment in mirror design, many tests are required to access the performance of the mirror and this both time consuming and expensive. Computerized design and development aids that would be able to predict the response of the proposed design would be very advantageous. In the work described in this article, the finite element method, using the OPTISTRUCT package, was used to predict the vibration response of the selection of outer rear-view mirrors using mirror geometry imported from CAD files. Initially the 3D CAD of the mirror assembly was created from 2D drawings. The first natural frequency of the first FEA model of ORVM was matched to the first natural frequency value of the mirror assembly received from the test and the first model was thus considered as the base model in the development of computerized design and development aid of the mirror.

## 2.History

The Rear view mirror is a term which encompasses internal and external mirrors of the vehicle. The external mirrors are also called as Rear Vision Mirrors, wing mirrors or side view mirrors. The focus of this study paper is on the external rear view mirror and its mechanical behaviour. However methodologies developed for external mirrors can also be applied to an internal mirrors.
The first known appearance of a rear view mirrors on a car to monitor the external traffic was in a race car in 1911 at the Indiana Pois' 500(war02). Before this time a second person was required to tell the driver about the position of the nearest racing competitor. But the rear view mirror totally replaced this concept. Elmer Berger is usually referred to as the inventor of the rear vision mirror for having contributed to their mass production in 1914. And today every vehicle small or big has this rear vision mirror mounted on it.

**3.Problem Definition**

The existing actual mirror assembly has a first natural frequency of 22 Hz. The target is to suggest and do the modifications in the mirror assembly to bring the frequency value close to 45Hz.

The main hindrance in meeting the functional requirements of a RVM on road, comes from the issues faced due to mirror vibrations. And hence a successful design of a ORVM needs study of its vibration characteristics. A successful design is always verified for or revolves around the First Fundamental Natural Frequency of the ORVM. And so its important to achieve a minimum required First natural Frequency for a Mirror Assembly.

This threshold value of First Normal Mode or natural Frequency is the outcome of vibrations coming from vehicle body and the vibrations due to air drag when the vehicle is moving on road. And is generally decided by the OEMs or Automobile manufacturers based on various factors as mentioned below.

**4.Factors Deciding The Threshold Natural Frequency Of Mirror Assembly**

- Vibrations coming from car engine at different speeds (In current case:  (28-43 Hz))
- Vibrations coming from road conditions (In current case: (0-15 Hz))
- Perception of reflected Image by drivers' eyes (General Finding: (20-25 Hz))
- Vibrations due to wind drag (very unpredictable)

Considering the other uncertain causes of vibrations during the actual vehicle and mirror assembly's life cycle a factor of safety is considered and the threshold frequency close to  45Hz was decided as a target first natural frequency of the mirror assembly.

So the aim of the work is to increase the first natural frequency as close as possible to 45Hz while meeting the constraints as manufacturing feasibility, cost and weight..

**5.Selection Of Pre-Processor And FEA Solver**

There are many FEA pre-processors available in the market. The popular pre-processors are Hypermesh and Ansa. While the popular solvers available in the market are Abaqus, Nastran, Ideas, Ls-Dyna and Optistruct & Radioss that comes along with the Altair Hyperworks.

Every FE-solver software has its own strength. For example Abaqus and Ls-Dyna are preferred non-linear calculations; Ls-Dyna is highly preferred for Vehicle Crash calculations, Nastran is preferred for Vibration analysis. However the Modal analysis could be accurately done in most of the solvers.

For the current case the choice of Pre-processor and FE-solver hardly mattered, as the obvious selection was that Pre-processor which was available in the company. The company has the Altair Hypermesh10.0 and hence Hypermesh is used to do the pre-processing.

The obvious advantage was the use of Optistruct solver which is in-build with Hypermesh. Most companies prefer the Hypermesh as a pre-processor and Nastran or Optistrct as a solver for doing Modal analysis. However other more critical vibration related simulations are done in MSc Nastran.

**6.Complete Mirror**

There are total thirteen to fourteen different important parts in the complete mirror assembly and a decision of how to model these parts in FEA was made based on their geometric characteristics and there functionality. The mirror housing, backplate, mirror plate, mirror mounting bracket cover was modelled using the 2D shell elements. The Mirror housing bracket, mirror base bracket, mirror regulation cover and regulation base were modelled using the 3D tetra elements.  A slightly larger element size was used to mesh the mirror housing case and the mirror plate whereas the mounting bracket and the housing bracket were models with small tetra elements to capture the geometry properly. The reason is that the stiffness  play major role in determining the natural frequency of the assembly and features that provide stiffness of the mounting bracket should be captured properly. Again the importance of the accuracy in the  geometry and the optimum mesh density were noted through various studies.

Some of the connections between the inner parts of the assembly were snap fits and these were modelled in FEA just by capturing the connection areas more precisely. It was realized in the studies that just modelling these areas precisely was not sufficient, but multiple node's rigid connection was necessary in these areas to represent the equivalent stiffness.  The connection between the main housing and the base bracket was done using rigid spider and beam. From various simulations and the deformation behaviour, most precise method of modelling and connection was decided.

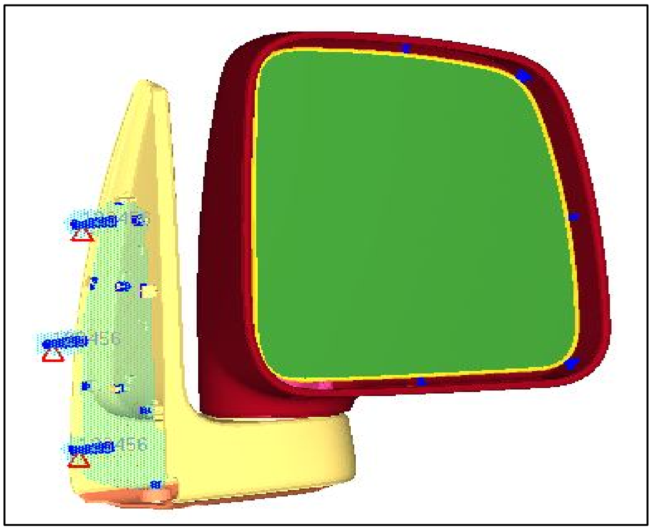

*Figure 1: FEA Model Of Complete Mirror Assembly*

**7.Modal Analysis In FEA**

3D cad data were generated from the 2D drawings of the mirror assembly and meshing and model preparation for the modal analysis was done in Hypermesh using the Optistruct user profile. Load asce setup for the modal analysis and simulation was also done in Optistruct. Many iterations were done with material changes, basic rib thickness changes, with minor geometric modifications to observe how the natural frequency is affected with these changes and finally after getting the initial approach of where to add stiffness and where the mass could be reduced without affecting the functionality of various parts in the assembly, major geometric modifications were suggested and carried out with the help of cad softwares. These models were then submitted for modal analysis and the results were observed. The table below shows some of the important modifications and the resulting frequency.

| Model | Changes | First Natural Frequency Target (Hz) | First Natural Frequency Achieved(Hz) |
|---|---|---|---|
| Base line CAD model | | ~45 | 22 |
| Proposal-1 | Material of the base bracket changed to aluminium diecast | ~45 | 35.6 |
| Proposal-2 | Proposal 1 + a snap is added at the end of base and cover plate | ~45 | 35.9 |
| Proposal-3 | Proposal2 + base bracket ribs extended | ~45 | 38.4 |
| Proposal-4 | Proposal3 + small ribs added in the housing bracket | ~45 | 44.4 |
| Proposal-5 | Proposal4 + cylindrical portion of housing bracket extended | ~45 | 45.6 |
| Proposal-6 | Proposal5 + extended cylindrical portion and upper closed section of cylindrical portion removed due to manufacturing constraints and instead splines are added on original cylindrical portion to maintain stiffness | ~45 | 44.4 |

*Table 1*

The proposal 6 was tested and the frequency value obtained in the test was found matching with the FEA.

**8.Summary & Conclusion**

Finite element models for resonant frequency and mode shape prediction of mirror assembly was developed and was found matching with the test. Some of the important points noted were as follows. capturing the geometry of the model was very important although small fillets and very small features which will not affect the stiffness and mass of the assembly anyway was done. This way the meshing efforts required for all subsequent iterations were reduced to some extent. For the dynamic analysis the important requirement for meshing is that the mesh is kept as uniform as possible with greater mesh density in the region of more complexity so that the stiffness and mass can be distributed as accurately as possible. Modeling of the connection details is of significant importance. Trial and error iterative approach was one of the fundamental approach to do various combinations of mass and stiffness changes and hence change the natural frequency of mirror assembly. initially contact modeling was tried to simulate some connections in the snap fit areas, but it was imposing additional constraint in the FEA model and due to this the local stiffness was altered. And hence contact was removed from the FEA model and was replaced by other suitable connections.

**9.References**

1. Design and Optimizing Side View Mirrors. Martin Olsson. Dept. of Applied Mechanics. Vehicle Engineering and Automotive Systems, Chalmers Univer sity of Technology, Sweden.
2. Optimization and Flowchart Control in the Industry Process using Abaqus, CATIA, Moldflow with Insight Integration. Mariana Batista, Ruben A. G., Guardia.
3. Practical Finite Element Analysis by Nitin Ghokale and Sanjay Deshpande.
4. Time and Frequency Domain Instantaneous Center Locator Methods With Applications. Nathan Mouw and Chuck Van Karsen. Mechanical Engineering. Michigan Technological University.