"Structural Analysis of Engine Cooling System for Passenger Car Vehicle"

Gaikwad Vikrant C.*, Dr. Sawant Suresh M. **, Mr. K. Parmeshwar***

*Research Scholar, Department of Mechanical Engineering, RIT Sakharale 415414, Sangli, Maharashtra, India

**Professor, Department of Mechanical Engineering, RIT Sakharale 415414, Sangli, Maharashtra, India

***Engine Cooling Head, Behr India limited, Pune-410501, Maharastra, India

ABSTRACT

Engine cooling systems for vehicles are used for cooling the engine fluids. The cooling system normally consists of following components: radiator, expansion tank, cooling fan, fan drive, and shroud. The mounting structure for this system must be designed to withstand the loads that will be imposed by the vehicle operation which consists of stresses such as those caused by linear static and dynamic loading. Automotive industries perform various tests on vehicles in the end-user environment to reduce failures; these investigations are carried out on the design using finite element method (FEM). Finite element methods are being used routinely to analyze for structural behaviour. Modelling is done CATIA software, meshing is carried out with HYPERMESH software and solution is acquired using NASTRAN solver. For linear static analysis, Force was applied on the assembly using 'g' criterion, where g is the acceleration due to gravity and for linear dynamic analysis with forced vibration have used the frequency response along with enforced motion. The different analysis of vehicle condition it is observed that design is safe for linear static and linear dynamic analysis. But for linear static even if the design is safe that might failure during linear dynamic analysis.

Keyword- **Finite element analysis (FEA), Mounting points, and Structural analysis.**

1. INTRODUCTION

Modern automotive internal combustion engines generate a huge amount of heat. Approximately 1/3 of the heat in combustion is converted into power to drive the vehicle and its accessories. Another 1/3 of the heat is carried off into the atmosphere through the exhaust system. The remaining 1/3 must be removed from the engine by the cooling system. Engine cooling system helps in dissipating this heat to the surrounding and keeps the temperature of the body under acceptable valve. Radiator, charge air cooler, fan, shroud are essential parts of an engine cooling system.

Structural analysis is mainly concerned with finding out the behaviour of a structure when subjected to some action. This action can be in the form of load due to the weight of things such as due to the mass of engine cooling component etc. or some other kind of excitation such as a base excitation etc. In essence all these loads are dynamic, including the self-weight of the structure because at some point in time these loads were not there. The distinction is made between the dynamic and the static analysis on the basis of whether the applied action has enough acceleration in comparison to the structure's natural frequency.

2. THEORETICAL BACKGROUND

2.1 Structural analysis

Structural analysis mainly consists of two types

2.1.1. Static Analysis

If a load is applied sufficiently slowly, the inertia forces (Newton's second law of motion) can be ignored and the analysis can be simplified as static analysis. Statics deals with the effect of forces on bodies at rest.

2.1.2. Dynamic analysis

Structural dynamics, therefore, is a type of structural analysis which covers the behaviour of structures subjected to dynamic (actions having high acceleration) loading. A dynamic analysis is also related to the inertia forces developed by a structure when it is excited by means of dynamic loads applied suddenly (e.g., wind blasts, explosion, earthquake, base excitation).

A dynamic load is one which changes with time fairly quickly in comparison to the structure's natural frequency .If it varies quickly (relative to the structure's ability to respond), the response must be determined with a dynamic analysis. First, dynamic loads are applied as a function of time. Second, this time-varying load application induces time-varying response (forces and stresses).

Dynamic Analysis is basically two types

2.1.2.1. Free Vibration

If a system, after an initial disturbance, is left to vibrate on its own, the ensuring vibration is known as free vibration. It is also called as real eigenvalue analysis or modal analysis (undamped free vibrations). Modal analysis is used to determine the basic dynamic characteristics of a structure. The results of a modal analysis indicate the frequencies and shapes at which a structure naturally tends to vibrate. Although the results of a modal analysis are not based on a specific loading, they can be used to predict the effects of applying various dynamic loads.

2.1.2.2 Forced Vibration

If a system is subjected to an external force (often, a repeating type of force), the resulting vibration is knows as forced vibration.
Forced vibration analysis again is of two types [1].

- **Frequency response**

Frequency response analysis is an efficient method for finding the steady-state response to sinusoidal excitation. In frequency response analysis, the loading is a sine wave for which the frequency is specified. Linear frequency response analysis is steady-state response of linear structures to loads that vary as a function of frequency.

- **Transient response**

Transient response analysis is the most general method of computing the response to time-varying loads. The loading in a transient analysis can be of an arbitrary nature, but is explicitly defined (i.e., known) at every point in time. The time-varying (transient) loading can also include nonlinear effects that are a function of displacement or velocity. Linear transient response analysis is response of linear structures to loads that vary as a function of time.

2.2 Vehicle Dynamics

Vibration is an avoidable phenomenon in vehicle dynamics. Most of the optimization methods for vehicle suspensions and vehicle vibrating components are based on frequency responses [2].

2.2.1. Predominant Sources of Vehicle Vibration

2.2.1.1 Road Roughness

Depending upon cause, the vibration may be free or forced, the free vibration may occur when the vehicle passes over an isolated irregularities in the road surface, on the other the forced vibration may result when disturbances occur persistently such as passing over obstacles on a proving road.

The road roughness, causes vertical acceleration of vehicle because of which passenger gets the proving and this adds to their discomfort, when a vehicle is being drive over the road, the oscillations of its spring have frequencies which has not only dependant on the frequency at which road impulses or bumps are encountered but also on the relation between the spring stiffness and the mass of the spring part of vehicle, the real description of road is random in nature.

2.2.1.2 Vibration Due To Engine Unbalance

The engine is one of the main sources of vibration transmitted to the vehicle chassis, which is attributed to its unbalance forces transmitted to the vehicle. Without isolation, these forces could cause rapid fatigue of vehicle component and discomfort for the occupant. The reciprocating parts of the engine may cause vibration of an automobile due to the periodic disturbances.

2.2.1.3 Vibration Due To Fan

The reciprocating parts may cause vibration of an automobile due to the periodic disturbances.

2.3 Frequency Response of Vibrating Systems

Frequency response is the steady-state solution of equations of motion, when the system is harmonically excited. Steady-state response refers to a constant amplitude oscillation, after the effect of initial conditions dies out. A harmonic excitation is any combination of sinusoidal functions that applies on a vibrating system. If the system is linear, then a harmonic excitation generates a harmonic response with frequency-dependent amplitude. In frequency response analysis, we are looking for the steady-state amplitude of oscillation as a function of the excitation frequency. In frequency response analysis the excitation is explicitly defined in the frequency domain. All of the applied forces are known at each forcing frequency. Forces can be in the form of applied forces and/or enforced motions (displacements, velocities, or accelerations) [3][5]. Enforced motion specifies the displacements, velocities, and/or accelerations at a set of grid points for frequency and transient response. Enforced motion is used when motion is specified instead of or in conjunction with applied loads.

There are four types of one-DOF harmonically excited systems are Base excitation, Eccentric excitation, Eccentric base excitation and Forced excitation.

Base excitation is the most practical model for vertical vibration of vehicles. Eccentric excitation is a model for every type of rotary motor on a suspension, such as engine on engine mounts. Eccentric base excitation is a model for vibration of any equipment mounted on an engine or vehicle. Forced excitation, has almost no practical application, however, it is the simplest model for forced vibrations, with good pedagogical use.

3 FINITE ELEMENT ANALYSIS (FEA)

The finite element method is used in a wide variety of disciplines and engineering applications. In the beginning, the use of these techniques was customary only in the aerospace and nuclear fields. Subsequently, the use has spread to a variety of products, physical situations, and manufacturing processes. Some of the interesting features of FEA are structural analysis, Noise Vibration and Harshness (NVH), Fatigue, crash and Optimization.

In order to obtain better design in FEA, following procedure (Fig.1) is applied so that stresses can be easily modelled in the system.

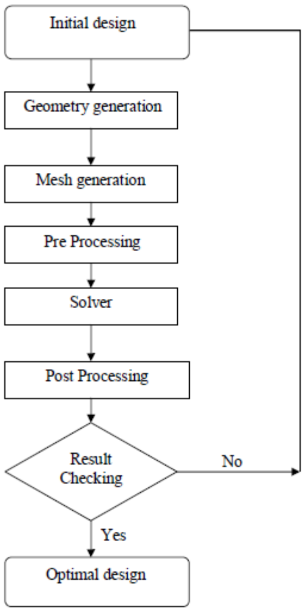

Fig.1. Flow chart of procedure FEA

3.1 Initial design and geometry generation

Initial design of the model is a planning decision and the geometry is generated depending on these initial design considerations, using modelling tools. This is the first stage for any design problem where all the properties and consideration regarding the design of the problem are discussed. After completing the initial design the model is generated using different modelling software.

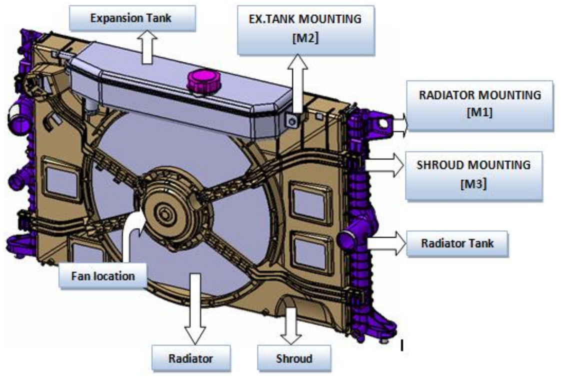

Fig.2. Engine cooling system model showing all the mountings

Fig.2 shows the engine cooling system model of various mountings. In engine cooling system model, different types of mounting's are available at different location i.e. radiator mounting [M1] and assembly i.e. Expansion tank mounting [M2] and Fan shroud mounting [M3].In this project have analyzed the above three mountings with linear static and linear dynamic analysis.

3.2 Mesh Generation

Basic theme of FEA is to make calculations at only limited (Finite) number of points & interpolate the results for entire domain (surface or volume). Any continuous object has infinite degrees of freedom & it's just not possible to solve the problem in this format. Finite Element Method reduces degrees of freedom from Infinite to Finite with the help of discretization i.e., meshing (nodes & elements).

HYPERMESH is a powerful tool that lets designers and analysts of component create high-quality meshes, while preserving the underlying geometry. HyperMesh is a high-performance finite-element pre-processor that provides a highly interactive and visual environment to analyze product design performance.

For meshing, Element type selection is based on Geometry size and shape, type of analysis, time allotted for project and hardware configuration. In this process we have used 2 D meshing used with shell element.

The assumptions for FEA,

1. Weight of the radiator with coolant
2. Weight of the fan.

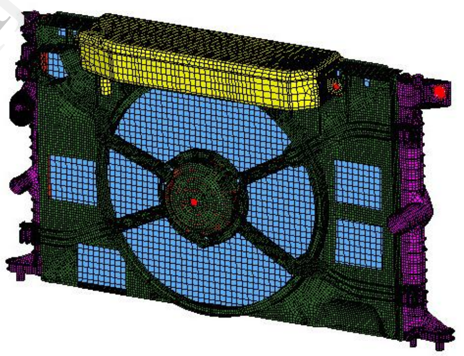

Fig.3. 2D meshing of engine cooling system model

Fig.3 2D meshing of engine cooling system model gives total number of elements are generated is 54368 in number.

3.2.1 Material and Property Information

After meshing is completed, material (e.g. Young's Modulus) and property information (e.g. thickness values) are assigned to the elements.

In Model, basically Expansion tank, Radiator tank, shroud all parts are made up of plastic and radiator is made up of aluminium. The material and their properties are shown in table 1.

Table 1 - Material of Component

Material	Young's modulus(E) N/mm^2	Passions ratio (μ)	Density (tonne/ mm^3)
Plastic	6100	0.35	1360×10^{-9}
Al	70,000	0.33	2.8×10^{-9}

3.3 Pre-Processing

The operation pre-processing provides the solver with all necessary information for the calculation. Pre-processing involves creating the FE model and applying the necessary loads and boundary conditions.

There are different boundary condition for analyzing forces, traction, torque, Pressure and Vacuum, Temperature loading, Gravity loading, Centrifuga1 load.

In this process, we have used following boundary conditions. In linear static analysis only '3g' values (General rules) have been used for full vehicle analysis like vertical acceleration (pothole), Lateral acceleration (cornering), axial acceleration (Braking) [1] [4]. In frequency response analysis we consider z axis excitation i.e. enhanced motion ('3 g' acceleration) [1] [4].

3.4 Processing (Solver)

The FEM model (consisting of nodes, elements, material properties, loads and constraints) is then exported from within the pre-processor Hypermesh. The exported FEM model, typically called solver input deck, is an ASCII file based on the specific syntax of the NASTRAN solver. Solution phase solve the deck prepared in pre-processing using NASTRAN solver.

HyperMesh supports a host of different solver formats for both import and export. The following solvers which are support hypermesh software's like RADIOSS, NASTRAN, MATLAB, and ANSYS.

In process, after applying the boundary condition in the pre-processor the model is solved in the NASTRAN-Solver.

3.5. Post-Processing

Post-processing provides the FEA professional with easy-to-use powerful result visualization features for structured. It provides an in depth view of data with visualization tools such as stresses, strain, displacement, normal mode, dynamic behavior of structure of various loading conditions.

After the run is complete, the easiest way to access the results is by using the Hyper View software. The open architecture of Hyper View allows for loading and viewing result files obtained from several sources. Based on the solver type of the files and the results you would like to visualize and analyze, there are different ways to load the input deck and their corresponding results into Hyperview software.

Various results that have got after running the NASTRAN solver are as follows:

3.5.1 Linear Static Analysis

In linear static analysis we have taken the worst possible scenario of failure i.e. a combination of pothole, cornering and braking.

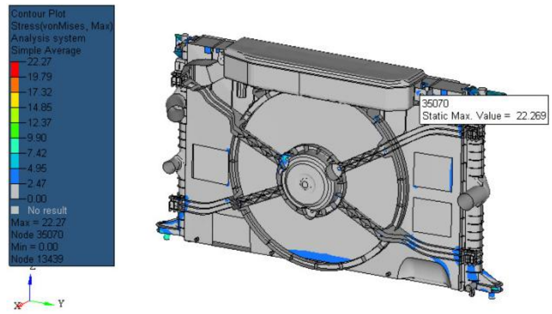

Fig.4. Combined

In linear static analysis, according to standard data available for the plastic material the value of the yield strength is 120 N/mm2. The factor of safety that we have considered is 1.5 for the plastic material, so yield stress for the material is 80 N/mm2.

3.5.2 Linear Dynamic Analysis

It is analysis is carried out linearly at dynamic condition. The linear dynamic analysis carried out under two categories which are explained below.

3.5.2.1 Modal Analysis

Modal analysis determines the vibration characteristics of a structural or a particular component in the form of natural frequency and mode shape of the natural vibration modes. The natural frequency and mode shape are important in the design of a structure for dynamic loading condition.

3.5.2.1(A) Mode 1

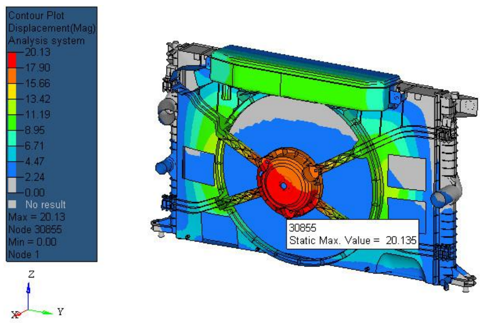

Fig.5. Mode 1

ISSN: 2278-0181
Vol. 2 Issue 7, July - 2013

3.5.2.1(B) Mode 2

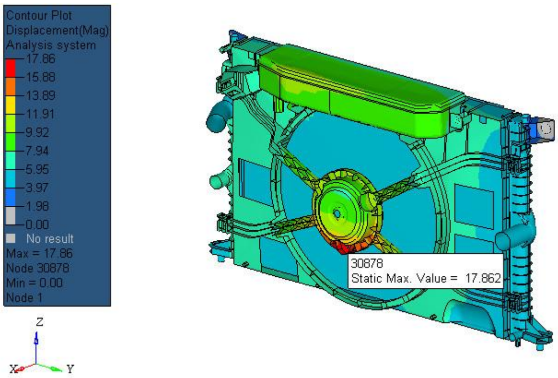

Fig.6. Mode 2

Table 2 - Result summery of model analysis

Mode Shape	Natural Frequency (Hz)	Nature
1	33.22	X-Direction Mode on Shroud
2	36.56	Combined X and Z Direction Mode on Shroud

Different of nature frequency for different mode shapes shows in table 2. In which mode shape I at frequency 33.22 Hz. shows in X-Direction on shroud and mode shape II shows X and Z direction at frequency 36.56 Hz. The speeds of fan i.e. 2700 rpm calculate the fundamental frequency is 45 Hz.

3.5.2.2 Frequency Response Analysis

In this frequency response analysis we consider z axis excitation i.e. enhanced motion.

3.5.2.2(A) Iteration I

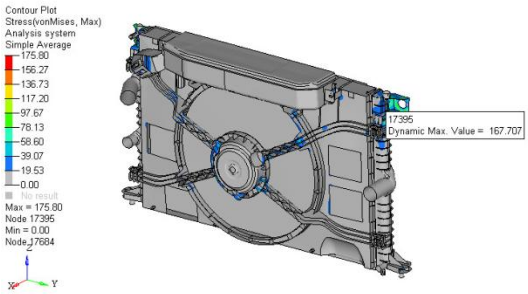

Fig.7. Iteration I

The above fig.7 represents the first iteration while carrying out the frequency response analysis. After solving model for solution it is observed that the available value of maximum stress more than the yield stress and the failure occurs at the mounting [M1] for a thickness of 3 mm. Also, from the contour plot data, it is analyzed that no failure occurs at mounting points [M2] and [M3].

3.5.2.2(B) Iteration II

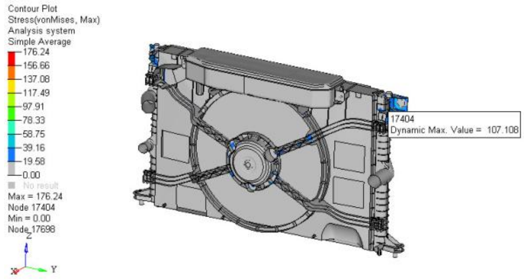

Fig.8. Iteration II

In iteration 2, value of stress is 107 N/mm2 for thickness of the mounting [M1] to 4mm which shows design is not safe.

3.5.2.2(C) Iteration III

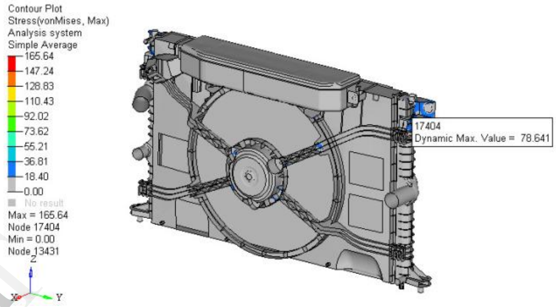

Fig.9. Iteration III

In iteration 3, for thickness of the mounting to 5mm value of dynamic stress is 78.64 N/mm2 which provided with stiffener makes design safe.

4. CONCLUSION

In this paper following software like Catia for modeling, Hypermesh for meshing, Nastran as a solver and Hyperview for visualized result are used. One of the most important advantages of using this software is that it reduces production time and the number of prototype. Also, the result obtained from software is instantaneous.

The Concluding remarks for different types of analysis at the mounting point.

1. Linear static analysis- In linear static analysis, stress is analyzed for different vehicle conditions. The value of maximum stress i.e. worst condition is 22.27 N/mm2 much below than the yield stress i.e. 80 N/mm2 for all mounting points and hence the design is safe.

2. Modal Analysis- In modal analysis, the fundamental frequency calculated from the fan rpm i.e. 45 Hz. and the mode/natural frequencies i.e. 33.22 Hz and 36.56 Hz are do not match, hence no resonance is created. Also, according to standard data available from Nastran manual, the mode/natural frequency i.e.36.56 Hz should be less than 38.25 (85% of 45 Hz) frequency of the fan. Hence, the condition is satisfied and goes to further analysis.

3. Frequency Response Analysis - In frequency response analysis, first two iterations are not safe and third iteration is safe.

In iteration 1, after solving model for solution we can observe that the available value of maximum stress is 167.70 N/mm2 more than the yield stress i.e. 80 N/mm2 and the failure occurs at the mounting M1 for a thickness of 3 mm. Also, from the contour plot data we can observe that no failure occurs at mounting points [M2] and [M3].

In iteration 2, we have increased the thickness of the mounting [M1] to 4mm. The stress value i.e.107.11 N/mm2 is still greater than the yield stress i.e. 80 N/mm2. Inspite of this, the failure occurs at mounting [M1].

Iteration 3, we have increased the thickness of the mounting [M1] to 5mm and also have added a stiffener and corner radius. After making this changes we can observed that the value of the stress value is less than the yield stress and the design is safe.

So, the design is safe for both the linear static and linear dynamic analysis.

Also, it is concluded that do not depend linear static analysis result because it might failure in linear dynamic analysis.

Acknowledgement:

I wish to thank Behr India limited, Pune-410501, Maharastra, India for giving me an opportunity to work in this field. The guidance, cooperation, practical approach & inspiration given by company especially by Mr. S Prakash (Manager), Mr. Razique Quazi (Assistant Manager) and Mr. Abasaheb Gaikwad (Manager at Tata Motor's Limited, Pune) provided me the much needed impetus to hard work.

REFERENCES

1. Nitin S Gokhale, Sanjay Deshpande, Sanjeeve V. Badekar, Anand N. Thithe,"Practical Finite Element Analysis", 1st Edition, ISBN978-81-906195-0-9.

2. Reza N. Jazar, "Vehicle dynamic-Theory and Application", 2nd Edition, ISBN-978-0-387-7243-4.

3. Singiresu S. Rao,"Mechanical Vibration", 4th Edition, ISBN-978-81-775-8874-3.

4. Japanese Industrial Standard (JIS) - JIS D 1601.

5. Users Reference Manual For the MYSTRAN (NASTRAN) General Purpose Finite Element Structural Analysis Computer Program (Nov 2011).

6. Fengyan Yi a, Meizhi Xie, "Objective evaluation of engine mounting isolation", *International Journal of AASRI Procedia, 3,* 2012, 49 -53.

7 Senthilnathan Subbiah,O.P. Singh, Srikanth K. Mohan, Arockia P. Jeyaraj, "Effect of muffler mounting bracket designs on durability", *International Journal Engineering Failure Analysis, 18,* 2011,1094-1107.

8.Jie Gao and Ke Chen, "Frequency-Domain Simulation and Analysis of Vehicle Ride Comfort based on Virtual Proving Ground", *International Journal of Intelligent Engineering and Systems, 4, 2011*, 1-8.

9. Seyed Ahmad Mirgeei, Seyed Saeid Mohtasebi, Mahmod omid, Reza Alimardani, "Modeling, Dynamic analysis and Optimization of Budsan Truck Engine mount", *Journal of Applied Science, 8,* 2008,2369-2377.

10. J. Melcer, "Vehicle–Road Interaction, Analysis in a Frequency Domain", *Slovak journal of civil engineering,* 2006, 48-52.

11. Shaolin Mao, Changrui Cheng, Xianchang Li c, Efstathios E. Michaelides, "Thermal/structural analysis of radiators for heavy-duty trucks", *Applied Thermal Engineering, 30,*2010, 1438-1446.