



DECLARATION


This is to certify that the thesis entitled "Dynamic Analysis of Radiator Mounting Brackets of Automobile Cooling System" is an authentic record of my own work carried out as per requirements for the award of degree of Master of Engineering in CAD/CAM & Robotics from Thapar University, Patiala, under the guidance of Mr. Daljeet Singh (Assistant Professor, MED) and Mr. Bikramjit Sharma (Assistant Professor, MED) during the period July 2013 to July 2014.

Date: 21/7/2014



Amrinder Singh
(Reg. No: 821181001)


It is certified that the above statement made by the student is correct to the best of our knowledge and belief.


(Mr. Daljeet Singh)
Assistant Professor
Mechanical Engineering Department
Thapar University, Patiala


(Mr. Bikramjit Sharma)
Assistant Professor
Mechanical Engineering Department
Thapar University, Patiala

Countersigned by


(Dr. Ajay Batish)
Head, Mechanical Engineering Department
Thapar University, Patiala


(Dr. S.K. Mohapatra)
Dean of Academic Affairs
Thapar University, Patiala

DYNAMIC ANALYSIS OF RADIATOR MOUNTING BRACKETS OF AUTOMOBILE COOLING SYSTEM

A dissertation submitted in partial fulfillment for the requirement of degree of

**Master of Engineering
in
CAD/CAM & Robotics**

By

AMRINDER SINGH CHEEMA

(Roll No. 821181001)

Under the guidance of

Mr. Daljeet Singh
Assistant Professor
Mechanical Engineering Department

Mr. Bikramjit Sharma
Assistant Professor
Mechanical Engineering Department



MECHANICAL ENGINEERING DEPARTMENT

THAPAR UNIVERSITY

Patiala, Punjab

July 2014

DECLARATION

This is to certify that the thesis entitled “**Dynamic Analysis of Radiator Mounting Brackets of Automobile Cooling System**” is an authentic record of my own work carried out as per requirements for the award of degree of **Master of Engineering in CAD/CAM & Robotics from Thapar University, Patiala**, under the guidance of **Mr. Daljeet Singh (Assistant Professor, MED)** and **Mr. Bikramjit Sharma (Assistant Professor, MED)** during the period July 2013 to July 2014.

Date: _____

Amrinder Singh
(Reg. No: 821181001)

It is certified that the above statement made by the student is correct to the best of our knowledge and belief.

(Mr. Daljeet Singh)
Assistant Professor
Mechanical Engineering Department
Thapar University, Patiala

(Mr. Bikramjit Sharma)
Assistant Professor
Mechanical Engineering Department
Thapar University, Patiala

Countersigned by

(Dr. Ajay Batish)
Head, Mechanical Engineering Department
Thapar University, Patiala

(Dr. S.K. Mohapatra)
Dean of Academic Affairs
Thapar University, Patiala

ACKNOWLEDGEMENTS

I express my sincere gratitude and regards to my supervisor **Mr. Daljeet Singh**, Assistant Professor, Mechanical Engineering Department, Thapar University, Patiala, for his valuable guidance and suggestions. Without his encouragement and guidance, this thesis would not have materialized.

I feel privileged to offer my sincere thanks to my co-supervisor **Mr. Bikramjit Sharma**, Assistant Professor, Mechanical Engineering Department, Thapar University, Patiala, for his guidance and encouragement at every step of my work. He guided me and gave me full time to understand the minute details of each and every step, for the successful completion of my thesis work.

I would like to express my gratitude to **Dr. Ajay Batish**, Head of Mechanical Engineering Department, Thapar University, Patiala for his kind cooperation and encouragement which helped in the completion of the present work.

I would also like to thank M/s Hira Motors, Patiala, M/s Climax Motors, Patiala and M/s B.S Motors for providing me necessary parts for the present work.

The generous support of all the staff members of Mechanical Engineering Department is greatly appreciated.

Above all, I express my gratitude to my family members for playing the most important role of constantly motivating me for the completion of the present work.

Amrinder Singh
(Reg. No. 821181001)

ABSTRACT

With the automotive cooling industry aiming at higher levels of quality, cost effectiveness and a short time to market, the need for simulation is at an all time high. In the present work, the use of dynamic analysis is proposed in the simulation of the automobile radiator mounting brackets for the vibration loads. The mounting brackets has been analysed using the standard testing conditions. The results revealed that the radiator mounting brackets may fail due to resonance in dynamic analysis, but in the static analysis, resonance cannot be predicted under the same magnitude of load. Therefore, dynamic analysis gives a realistic method for its design validation. With the use of the above methodology, new mounting brackets are analysed and optimized.

INDEX

Sr. No.	CONTENTS	Page No.
	Declaration	i
	Acknowledgment	ii
	Abstract	iii
	Index	iv-vi
	List of Figures	vii-viii
	List of Tables	ix
1	Introduction	1-34
1.1	Automotive Cooling Systems	1
	1.1.1 Cooling System Functions	2
	1.1.2 Liquid Cooling System	2
	1.1.3 The cooling system components	3
1.2	Computer-aided engineering (CAE)	8
	1.2.1 CAE in the automotive industry	8
	1.2.2 CAE tools of solving any engineering problem	9
	1.2.3 Classification of Numerical Methods	10
1.3	Introduction to Finite Element Analysis	13
	1.3.1 Procedure for Finite Element Analysis	14
	1.3.2 Discretization of problem	15
	a. Discretization Process	16
	b. Need for Discretization Process	17
	1.3.3 Type of elements	18
	1.3.4 Necessity of Finite Element Analysis	18
	1.3.5 Applications of Finite Element Analysis	19
	a. According to types of analysis	19
	b. According to types of jobs	22
	1.3.6 Advantages of Finite Element Analysis	23
1.4	Introduction to Vibration	24
	1.4.1 Importance of Vibration	26

	1.4.2	Parts of Vibration	28
	1.4.3	Types of Vibration	29
1.5	Dynamic Analysis Process		30
	1.5.1	Dynamic Analysis Types	32
1.6	Softwares to be used		33
	1.6.1	HyperWorks Introduction	33
2 Literature Review			
2.1	Literature Review		35
3 Problem Formulation			
3.1	Gaps in literature		42
3.2	Objectives of the present work		42
3.3	Methodology		43
4 Analysis of Radiator Mounting Bracket			
4.1	Simulation of Radiator Mounting Bracket		44
4.2	Reverse Engineering		44
4.3	Mesh Generation		47
4.4	Dynamic Analysis		49
4.5	Normal Mode Analysis		49
	4.5.1	Normal Mode Analysis of the Mounting Bracket	50
4.6	Modal Frequency Response analysis of Radiator Mounting Bracket		53
	a).	Modal Frequency Response analysis along X-axis	53
	b).	Modal Frequency Response analysis along Y-axis	54
	c).	Modal Frequency Response analysis along Z-axis	54
4.7	Static Structural Analysis of Radiator Mounting Bracket		55
5 Optimization of Radiator Mounting Bracket			
5.1	Meshing		57
5.2	Dynamic analysis		57
	5.2.1	Normal mode analysis of the mounting bracket	57
	5.2.2	Modal frequency response analysis of Mounting Bracket	57
5.3	Modal frequency response analysis on the removal of material		59

	a).	Modal frequency response analysis along X-axis	59
	b).	Modal frequency response analysis along Y-axis	59
	c).	Modal frequency response analysis along Z-axis	60
6	Conclusion and scope for future work		61
6.1	Conclusion		61
6.2	Scope for future work		61

References

62-63

List of Figures

Fig. No.	Title	Page No.
1.1	The cooling system	2
1.2	Liquid cycle in the system	2
1.3	The cooling system components	3
1.4	Working of Radiator	3
1.5	The radiator fan and its working	5
1.6	Position of a thermostat	7
1.7	Discretization Process	16
1.8	Need for discretization Process	17
1.9	Area of circle by different nodes elements.	17
1.10 (a)	One-Dimensional Elements	18
1.10 (b)	Two -Dimensional Elements	18
1.10 (c)	Three-Dimensional Elements	18
1.11	Swinging of simple pendulum	25
1.12	Degrees of freedom	27
1.13	Parts of vibration	29
1.14	Dynamic Analysis Process	31
4.1	Radiator Mounting Brackets	45
4.2 (a)	Detailed Drawing Of Small Left Radiator Mounting Bracket	46
4.2 (b)	Detailed Drawing Of Small Right Radiator Mounting Bracket	46

4.2 (c)	Detailed Drawing Of Radiator Mounting Big Bracket	46
4.3.1	Tetrahedral Meshing of Radiator Mounting Bracket	47
4.3.2	Tetrahedral Meshing of Radiator Mounting Bracket	47
4.3.3	Tetrahedral Meshing of Radiator Mounting Bracket	48
4.3.4	Meshing of Radiator Assembly	49
4.5(a)	Static Structural Analysis	51
4.5(b)	Natural frequencies and mode shapes	51
4.5(c)	Natural frequencies and mode shapes	52
4.5(d)	Natural frequencies and mode shapes	52
4.6(a)	Von Mises Stress in Mounting Bracket along X-axis	53
4.6(b)	Von Mises Stress in Mounting Bracket along Y-axis	54
4.6(c)	Von Mises Stress in Mounting Bracket along Z-axis	54
4.7 (a)	Deformation in Mounting Brackets	55
4.7 (b)	Deformation in Mounting Brackets	56
4.7 (c)	Deformation in Mounting Brackets	56
5.1	Natural Frequencies and Mode Shapes	58
5.2	Natural Frequencies and Mode Shapes	58
5.3	Natural Frequencies and Mode Shapes	59
5.4	Von-Mises Stress in Radiator Mounted Bracket along X-axis	59
5.5	Von-Mises Stress in Radiator Mounted Bracket along Y-axis	60
5.6	Von-Mises Stress in Radiator Mounted Bracket along Z-axis	60

List of Tables

Table No.	Title	Page No.
1.1	Freezing point and Boiling point of Radiator Fluids	6
1.2	Difference between Analytical, Numerical and Experimental methods.	10

Chapter 1

INTRODUCTION

1.1 Automotive Cooling Systems:

Modern automotive internal combustion engines generate a huge amount of heat. This heat is created when the gasoline and air mixture is ignited in the combustion chamber. This explosion causes the piston to be forced down inside the engine, levering the connecting rods, and turning the crankshaft, creating power. Metal temperatures around the combustion chamber can exceed 1000° F. In order to prevent the overheating of the engine oil, cylinder walls, pistons, valves, and other components by these extreme temperatures, it is necessary to effectively dispose of the heat.

It has been stated that a typical average-sized vehicle can generate enough heat to keep a 5-room house comfortably warm during zero degree weather (and I'm not talking about using the exhaust pipe). 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. Modern automotive engines have basically dumped the Air Cooled System for the more effective Liquid Cooled System to handle the job. In a liquid cooled system, heat is carried away by the use of a heat absorbing coolant that circulates through the engine, especially around the combustion chamber in the cylinder head area of the engine block. The coolant is pumped through the engine, then after absorbing the heat of combustion is circulated to the radiator where the heat is transferred to the atmosphere. The cooled liquid is then transferred back into the engine to repeat the process. Excessive cooling system capacity can also be harmful, and may affect engine life and performance. You must understand that coolant temperatures also affect oil temperatures and more engine wear occurs when the engine oil is below 190° F. An effective cooling system controls the engine temperature within a specific range so that the engine stays within peak performance.

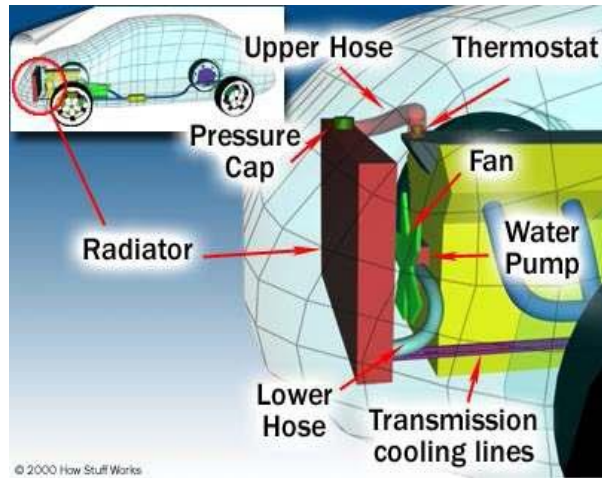


Figure - 1.1: The cooling system.

1.1.1 Cooling System Functions:

Temperatures in the combustion chamber of the engine can reach 4,500° F (2,500° C), so cooling the area around the cylinders is critical. Areas around the exhaust valves are especially crucial, and almost all of the space inside the cylinder head around the valves that is not needed for structure is filled with coolant. If the engine goes without cooling, the metal got hot enough for the piston to weld itself to the cylinder. This usually means the complete destruction of the engine. The cooling system removes enough heat to keep the engine at a safe temperature for best performance. A secondary function of the cooling system is to provide interior cabin heat during cold winter.

1.1.2 Liquid Cooling System:

Figure the cooling system components, and in these sections we'll talk about each part of the system in more detail. *Figure - 1.2*, illustrates the liquid cycle in the system.

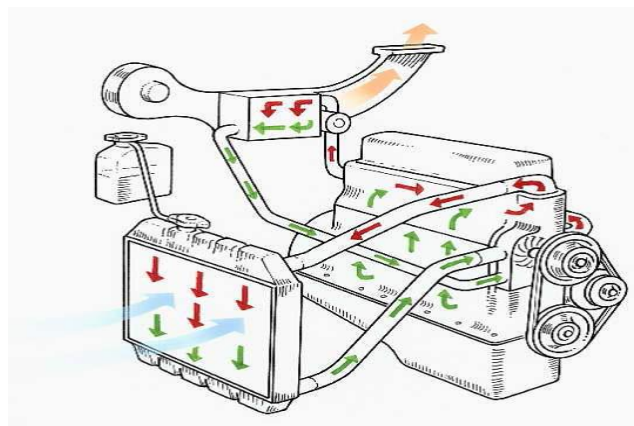


Figure - 1.2: Liquid cycle in the system.

1.1.3 The cooling system components:

Components of the cooling system are shown in *figure - 1.3* and the mechanisms of these components are explained below:-

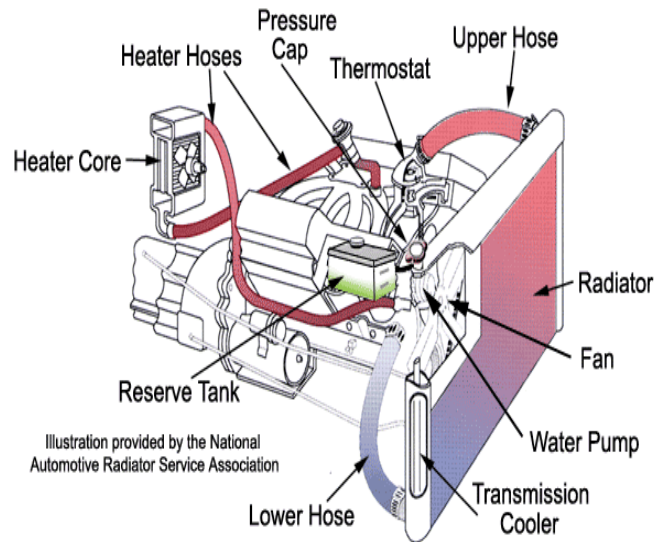


Figure - 1.3: The cooling system components.

1.) Radiator:

A radiator is a type of heat exchanger. It is designed to transfer heat from the hot coolant that flows through it to the air blown through it by the fan. Most modern cars use aluminum radiators. These radiators are made by brazing thin aluminum fins to flattened aluminum tubes. The coolant flows from the inlet to the outlet through many tubes mounted in parallel arrangements. The fins conduct the heat from the tubes and transfer it to the air flowing through the radiator as shown in *figure - 1.4*.

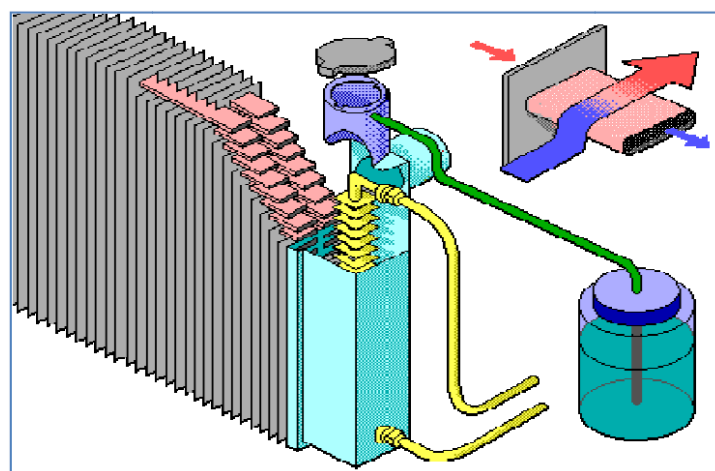


Figure 1.4: Working of Radiator

2.) Pressure cap:

The radiator cap actually increases the boiling point of your coolant by about 45° F (25° C). How does this simple cap do this? The same way a pressure cooker increases the boiling temperature of water. The cap is actually a pressure release valve, and on car it is usually set to 15 psi. The boiling point of water increases when the water is placed under pressure. When the fluid in the cooling system heats up, it expands, causing the pressure to build up. The cap is the only place where these pressures can escape, so the setting of the spring on the cap determines the maximum pressure in the cooling system. When the pressure reaches 15 psi, the pressure pushes the valve open, allowing coolant to escape from the cooling system. This coolant flows through the overflow tube into the bottom of the overflow tank. This arrangement keeps air out of the system. When the radiator cools back down, a vacuum is created in the cooling system that pulls open another spring loaded valve, sucking water back in from the bottom of the overflow tank to replace the water that was expelled.

3.) Radiator Fan:

A radiator fan is used to draw the air towards the radiator and help in the cooling process. The radiator fan has four or more blades that spin rapidly to provide sufficient air that would cool the engine. It is usually mounted between the radiator and the engine so that the air can easily get to the radiator. Some cars have an additional fan in front of the radiator in order to draw more cool air into the engine. Especially when it is so hot and the vehicle isn't moving fast enough, very little cool air reaches the radiator, and thus, the engine is not cooled properly. *Figure – 1.5*, illustrates the radiator fan.

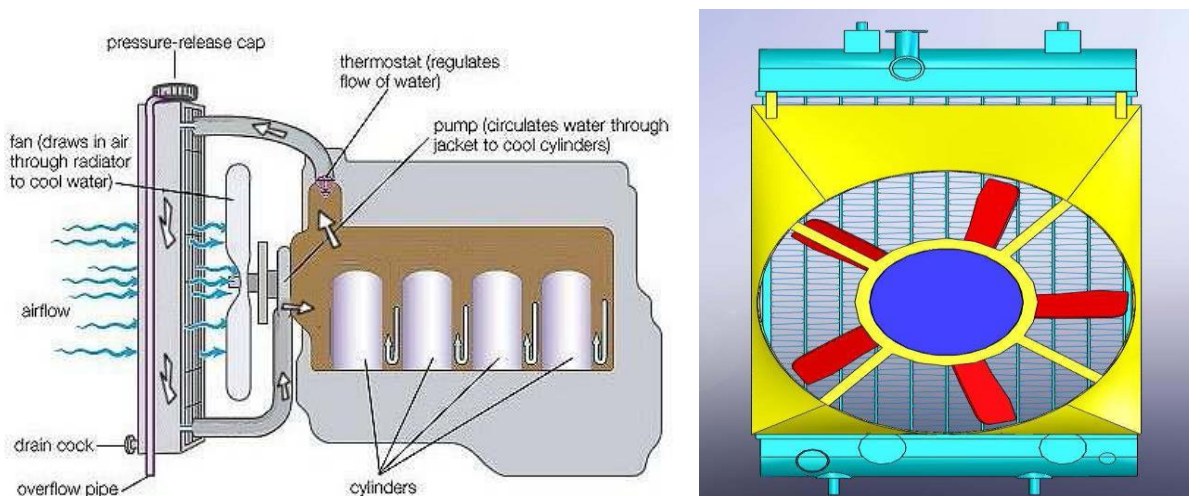


Figure - 1.5: The radiator fan and its working

4.) Water Pump:

The water pump is a simple centrifugal pump driven by a belt connected to the crankshaft of the engine. The pump circulates fluid whenever the engine is running. The water pump uses centrifugal force to send fluid to the outside while it spins, causing fluid to be drawn from the center continuously. The inlet to the pump is located

near the center so that fluid returning from the radiator hits the pump vanes. The pump vanes fling the fluid to the outside of the pump, where it can enter the engine. The fluid leaving the pump flows first through the engine block and cylinder head, then into the radiator and finally back to the pump.

5.) Plumbing:

The cooling system has a lot of plumbing. We'll start at the pump and work our way through the system. The pump sends the fluid into the engine block, where it makes its way through passages in the engine around the cylinders. Then it retunes through the cylinder head of the engine. The thermostat is located where the fluid leaves the engine. The plumbing around the thermostat sends the fluid back to the pump directly if the thermostat is closed. If it is open, the fluid goes through the radiator first and then back to the pump. There is also a separate circuit for the heating system. This circuit takes the fluid from the cylinder head and passes it through a heater core and then back to the pump.

6.) Fluid:

Car operates in a wide variety of temperatures, from well below freezing to well over 100° F (38° C). So whatever fluid is used to cool the engine has to have a very low freezing point, a high boiling point, and it has to have the capacity to hold a lot of heat. Water is one of the most effective fluids for holding heat, but water freezes at too high a temperature to be used in car engines.

Fluids	Pure Water	C ₂ H ₆ O ₂ /Water (50/50)	C ₂ H ₆ O ₂ /Water (70/30)
Freezing Point	0°C / 32°F	-37°C / -35°F	-55°C / -67°F
Boiling Point	100°C / 212°F	106°C / 223° F	113°C / 235°F

Table – 1.1: Freezing point and Boiling point of Radiator Fluids

The fluid that most cars use is a mixture of water and ethylene glycol (C₂H₆O₂), also known as antifreeze. By adding ethylene glycol to water, the boiling and freezing points are improved significantly. The temperature of the coolant can sometimes reach 250° to 275° F (121° to 135° C). Even with ethylene glycol added, these temperatures would boil the coolant, so something additional must be done to raise its boiling point. The cooling system uses pressure to further raise the boiling point of the coolant. Just as the boiling temperature of water is higher in a pressure cooker, the boiling temperature of coolant is higher if you pressurize the system. Most cars have a pressure limit of 14 to 15 pounds per square inch (psi), which raises the boiling point another 45° F (25° C) so the coolant can withstand the high temperatures.

7.) Thermostat:

The thermostat's main job is to allow the engine to heat up quickly, and then to keep the engine at a constant temperature. It does this by regulating the amount of water that goes through the radiator. At low temperatures, the outlet to the radiator is completely blocked -- all of the coolant is re-circulated back through the engine. Once the temperature of the coolant rises to between 180° and 195°F (82° - 91°C), the thermostat starts to open, allowing fluid to flow through the radiator. By the time the coolant reaches 200° to 218°F (93° - 103°C), the thermostat is open all the way.

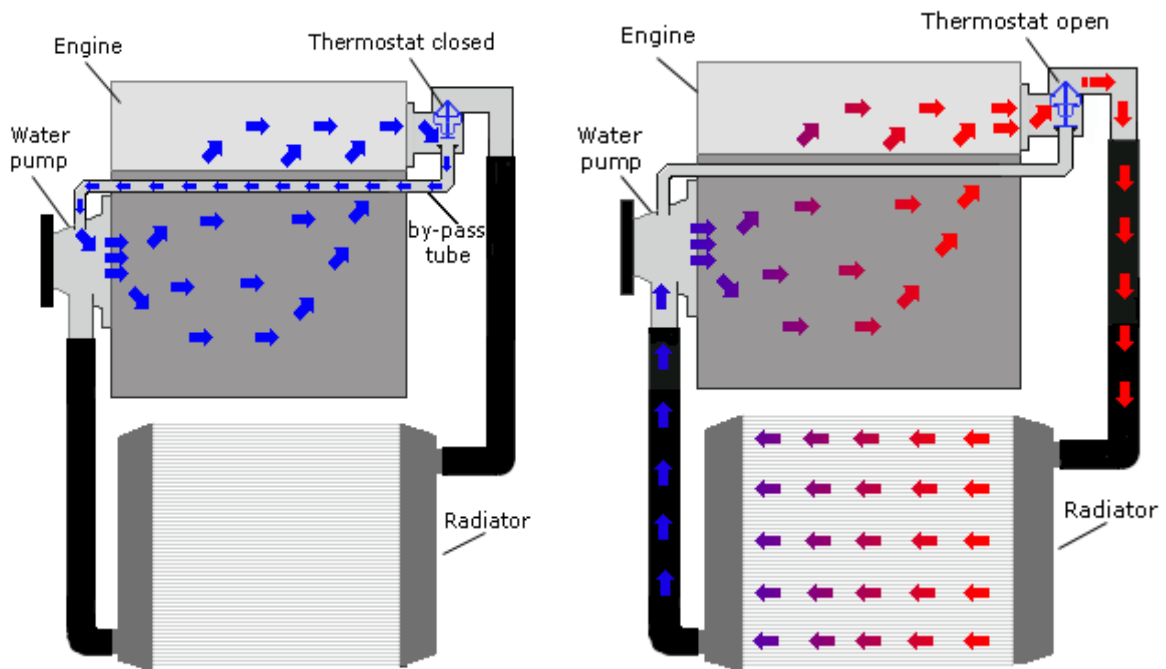


Figure - 1.6(a): Closed positions

Figure - 1.6(b): Open positions

Figure - 1.6: Position of a thermostat

The secret of the thermostat lies in the small cylinder located on the engine-side of the device. This cylinder is filled with a wax that begins to melt at around 180F (different thermostats open at different temperatures, but 180F is a common one). A rod connected to the valve presses into this wax. When the wax melts, it expands significantly, pushing the rod out of the cylinder and opening the valve.

1.2 Computer-aided engineering (CAE)

It is the broad usage of computer software to aid in engineering analysis tasks. It includes Finite Element Analysis (FEA), Computational Fluid Dynamics (CFD), Multi-body dynamics (MBD), and optimization.

Software tools that have been developed to support these activities are considered CAE tools. CAE tools are being used, for example, to analyze the robustness and performance of components and assemblies. The term encompasses simulation, validation, and optimization of products and manufacturing tools.

CAE areas include:

- Stress analysis on components and assemblies using FEA (Finite Element Analysis);
- Thermal and fluid flow analysis, computational fluid dynamics (CFD);
- Multi-body dynamics (MBD) & Kinematics;
- Analysis tools for process simulation for operations such as casting, molding, and die press forming.
- Optimization of the product or process.
- Safety analysis of postulate loss-of-coolant accident in nuclear reactor using realistic thermal-hydraulics code.

1.2.1 CAE in the automotive industry

CAE tools are very widely used in the automotive industry. In fact, their use has enabled the automakers to reduce product development cost and time while improving the safety, comfort, and durability of the vehicles they produce. The predictive capability of CAE tools has progressed to the point where much of the design verification is now done using computer simulations rather than physical prototype testing. CAE dependability is based upon all proper assumptions as inputs and must identify critical inputs. Even though there have been many advances in CAE, and it is widely used in the engineering field, physical testing is still used as a final confirmation for subsystems due to the fact that CAE cannot predict all variables in complex assemblies (i.e. metal stretch, thinning).

1.2.2 CAE tools of solving any engineering problem

There are three methods to solve any engineering problem.

1. Analytical Method
2. Numerical Method
3. Experimental Method

1. Analytical Method

An analytical solution is a mathematical expression that gives the values of the desired unknown quantity at any location in the body, as a consequence it is valid for infinite number of location in the body. Analytical method is a classic approach which gives accurate results. But this method is best suitable for simple problems like find

the deflection of cantilever, simply supported beams etc and also find stresses and strains etc, by using ready-made equations. But it consume more time as compare to Numerical Method.

2. Numerical Method

The use of numerical methods enables the engineer to expand his ability to solve practical design problems. It is not possible to obtain analytical mathematical solutions for many engineering problems. For problems involving complex materials properties and boundary conditions, the engineer's prefer to numerical methods that provide approximate, but acceptable solutions. Numerical method is a mathematical representation which gave approximate results.

3. Experimental Method

Experimental method is an actual measurement method. It physically test the prototype under varies condition. Thus it gives 100% accurate results. But engineers can't prefer because it require expensive set up and more time consuming method as compare with analytical method and numerical method.

Difference between Analytical, Numerical and Experimental methods

Sr. No	Analytical Method	Numerical Method	Experimental Method
1.	Classic approach	Mathematical representation	Actual measurement
2.	Accurate result	Approximate results	100% Accurate results
3.	Requires, mathematical equations	Requires CAD model	Applicable only if physical prototype is available
4.	Applicable only for simple problems	Applicable for complicated problems	
6.	Results depend on mathematical equations	Results cannot be believed blindly & must be verified by calculation for knowing the range of results or analytical or experimental methods	Results cannot be believed blindly & minimum 3 to 5 prototypes must be tested.

7.	Analytical method obtain results from different types of mathematical equations	Types of Numerical Methods <ul style="list-style-type: none"> • Finite Element Method • Boundary Element Method • Finite Volume Method • Finite Difference Method 	<ul style="list-style-type: none"> • Strain gauge • Photo elasticity • Sensors for temperature & pressure • Fatigue test etc.
----	---	---	---

Table - 1.2: Difference between Analytical, Numerical and Experimental methods

1.2.3 Classification of Numerical Methods

Numerical methods are broadly classified into four categories:

1. Finite Element Method (FEM)
2. Boundary Element Method (BEM)
3. Finite Volume Method (FVM).
4. Finite Difference Method (FDM).

1. Finite Element Method (FEM)

Finite element method, sometimes referred to as finite element analysis, is a computational technique used to obtain approximate solutions to boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of domain. Boundary value problems are also sometimes called field problems. The field is domain of interest and most often represents a physical structure. The field variables are the dependent variables of interest governed by differential equation. Depending on the type of physical problem being analyze, the field variable may include displacement, temperature, heat flux and fluid velocity etc. FEM is the most popular numerical method due to its applications.

Applications - Linear, nonlinear, buckling, thermal, dynamic & fatigue analysis etc

2. Boundary Element Method (BEM)

It is a very powerful and efficient technique to solve acoustics or NVH problems. Just like finite element method it also requires nodes and elements but as the name suggest it considers only outer boundary of the domain. So in case if the problem is of a volume only outer surfaces are considered. If the domain is area then only outer periphery is considered. This way it reduces dimensionality of the problem by a degree of one & helps in solving it faster.

3. Finite Volume Method (FVM)

All Computational Fluid Dynamics (CFD) software is based on FVM. Unit volume is considered in Finite Volume Method (similar to element in finite element analysis). Variable properties at nodes are pressure, velocity, area, mass etc. It is based on Navier - Stokes equations (Mass, Momentum and Energy conservation equilibrium equations).

4. Finite Difference Method (FDM)

Finite Element and Finite Difference Method share many common things, General Finite Difference Method is described as a way to solve differential equations. It uses Taylor's series to convert differential equations to algebraic equations. In the conversion process higher order terms are neglected. It is used in combination with BEM or otherwise FVM to solve Thermal and CFD coupled problems:

“Finite Difference Method is discretization of partial differential equations while - Finite Element Method, Boundary Element Method and Finite Volume Method are discretization of integral form of equations”.

It is possible to use all the listed methods (FEA, BEA, FVM, and FDM) to solve similar problem (say cantilever problem). But the difference is in accuracy achieved, programming ease & time required to obtain the solution.

When internal details are required (such as stresses inside a 3-d object) BEM will lead to poor results (as it considers only outer boundary), while FEM or FDM or FVM are preferable. FVM has been used for solving stress problems but it is well suited for computational fluid dynamics problems where conservation & equilibrium is quite natural. FDM has limitations with complicated geometry, assembly of different material components and combination of various types of elements (1-D, 2-D & 3-D). For these type of problems FEM is far ahead of its competitors.

Numerical methods like FEM are based on Discretization of integral form of equation. Basic theme of all numerical methods is to make calculations at only limited number of points & then interpolate the results for entire domain (surface or volume). Even before getting the solution we assume how the unknown is going to vary over a domain. Say for example, when meshing is carried out using linear quadrilateral elements, assumption made is linear variation of displacement over the domain and for 8 noded quadrilateral elements, assumption is parabolic variation. This may or may not be the case in real life & hence all numerical methods are based on an initial hypothetical assumption. After getting the results there are several ways to check numerical as well as practical or field result correlation accuracy & minimization of errors.

“All the numerical methods including FEM are approximate & one should not believe the results blindly”.

1.3 Introduction to Finite Element Analysis

The finite element analysis is numerical analysis technique for obtaining approximate solutions to a wide variety of engineering problems. Because of its diversity and flexibility as an analysis tool, it is receiving much attention in almost every industry. In more and more engineering situations today, we find that it is necessary to obtain approximate solutions to problem rather than exact closed form solution. It consists of a computer model of a material or design that is loaded and analyzed for specific results. It is used in new product design and existing product refinement. Basic theme is to make calculations at only limited (Finite) number of points and then interpolate the results for entire domain (surface or volume).

Finite - 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).

Element - All the calculations are made at limited number of points known as nodes. Entity joining nodes and forming a specific shape such as quadrilateral or triangular etc. is known as Element. To get value of variable (say displacement) anywhere in between the calculations point interpolation function (as per the shape of element) is used.

Analysis - FEA has become a powerful tool for numerical method of wide range of engineering problems. Application range from deformation and stress analysis of automotive, aircraft, building and bridge structures to field analysis of heat flux, fluid flow, magnetic flux and other flow problems. With the advances in computer technology and CAD systems, complex problems can be modelled with relative ease. Several alternative configurations can be tried out on a computer before the first prototype is built. In this method of analysis, a complex region defining a continuum is discretize into simple geometric shapes called finite elements and expressed in terms of unknown values at element corners. An assembly process, duly considering the loading and constraints, results in set of equations. Solution of these equations gives us the approximate behaviour of the continuum.

1.3.1 Procedure for Finite Element Analysis

Certain steps in formulating a finite element analysis of a physical problem are common to all such analyses, whether structural, heat transfer, fluid flow, or some other problem. The steps are described as follows:

1. Pre-processing
2. Processing or Solution
3. Post processing

1. Preprocessing

The preprocessing steps are described as follows:

- Define the geometric domain of the problem.
- Define the element type(s) to be used.
- Define the material properties of the elements.
- Define the geometric properties of the elements (length, area, and the like).
- Define the element connectivity (mesh the model).
- Define the physical constraints (boundary conditions).
- Define the loadings.

There are specialized software's available for CAD modeling, meshing and analysis. CAD model & meshing consumes most of the time. For example- typical time for a single person to model (CAD) a 4 cylinder engine block is 6 weeks & for brick meshing 7 weeks.

Boundary conditions consume least time but it is the most important step. Three months of hard work of meshing & CAD data preparation of an engine block can be undone in just one day if boundary conditions are not applied properly.

2. Processing or Solution

During the solution phase, finite element software assembles the governing algebraic equations in matrix form and computes the unknown values of the primary field variables. The computed values are then used by back substitution to compute additional, derived variables, such as reaction forces, element stresses, and heat flow. As it is not uncommon for a finite element model to be represented by tens of thousands of equations, special solution techniques are used to reduce data storage requirements and computation time. For static, linear problems, a wave front solver, based on Gauss elimination is commonly used.

During pre-processing user has to work hard while solution step is the turn of computer to do the job. User has to just click on the solve icon. Internally software carries out matrix formation, inversion, multiplication and solution for unknown e.g. displacement and then find strain & stress for analysis.

3. Post-processing

Analysis and evaluation of the solution results is referred to as post-processing. Postprocessor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution. Examples of operations that can be accomplished include:

- Sort element stresses in order of magnitude.
- Check equilibrium.
- Calculate factors of safety.
- Plot deformed structural shape.
- Animate dynamic model behavior.
- Produce color-coded temperature plots.

Post processing is viewing results, verifications, and conclusions and thinking about what steps could be taken to improve the design. While, solution data can be manipulated many ways in post-processing, the most important objective is to apply sound engineering judgment in determining whether the solution results are physically reasonable.

1.3.2 Discretization of problem:

All real life objects are continuous means there is no physical gap between any two consecutive particles. As per material science, any object is made up of small particles, particles of molecules, molecules of atoms and so on and they are bonded together by force of attraction.

Solving a real life problem with continuous material approach is difficult and basic of all numerical methods is to simplify the problem by discretizing (discontinuation) it. In simple words nodes work like atoms and with gap in between -filled by an entity called as element. Calculations are made at nodes and results are interpolated for elements.

There are two approaches to solve any problem

1. Continuous Approach (all real life components are Continuous)
2. Discrete Approach (Equivalent Mathematical modeling)

From mechanical engineering point of view any component or system could be represented by three basic elements:-

1. Mass 'm'.
2. Spring 'k'.
3. Damper 'c'.

All the numerical methods including Finite Element follow discrete approach. Meshing (nodes & elements) is nothing but Discretization of a continuous system with infinite degree of freedoms to finite degree of freedoms.

a.) Discretization Process

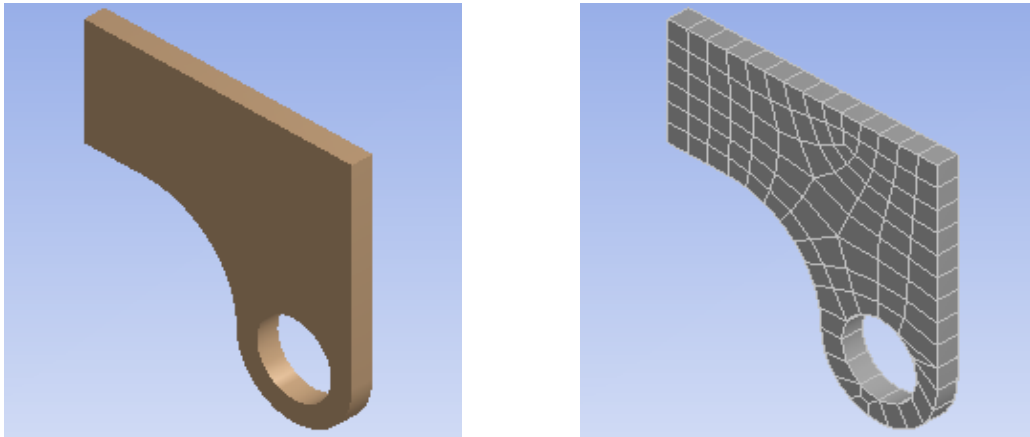


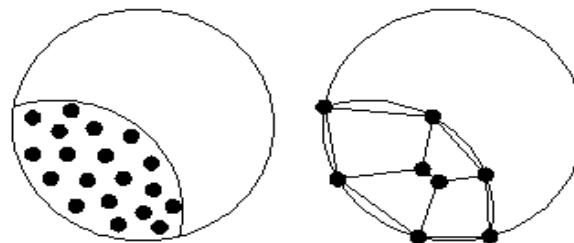
Figure - 1.7: Discretization Process

Continuous elastic structure (geometric structure) divided into small (but finite), well-defined substructures, called *elements*.

- Elements are connected together at a point called *nodes*.
- Discretization process known as *meshing*.

b.) Need for Discretization Process

Discretization process is most important process in finite element analysis. Its main reason is to generate finite number of nodes and form these nodes we get finite no. of equation. Thus, more time can save and accuracy increases. An example of discretization process is shown in *Figure – 1.8*



<i>No. of Points = ∞</i>	<i>No. of Nodes = 8</i>
<i>DOF per Point = 6</i>	<i>DOF per Point = 6</i>
<i>Total equations = ∞</i>	<i>Total equations = 48</i>

Figure - 1.8: Need for Discretization Process

Accuracy is dependent on nodes. If number of calculation points (nodes & elements) increase then accuracy also increases. It is proved with the following example:

Example: - Area of circle is compare by 3, 4, 6, 8 nodes elements.

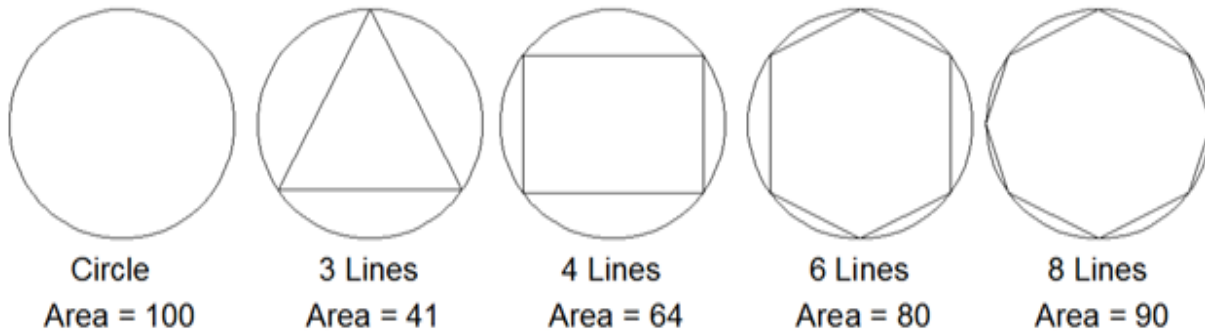


Figure - 1.9: Area of circle by different nodes elements.

Thus, if higher number of nodes and elements leads to higher accuracy then why not, always create a very fine mesh with maximum possible nodes and elements because the reason is solution time is directly proportional to (d.o.f.)ⁿ.

Where, **n = 1 to 4**, depending on type of analyses and solver.

1.3.3 Type of elements

1.) **One-Dimensional Elements** - *Rods, Beams, Trusses.*



Figure - 1.10 (a): One-Dimensional Elements

2.) **Two-Dimensional Elements** - *Triangular, Quadrilateral, Plates, Shells.*



Figure - 1.10 (b): Two -Dimensional Elements

3.) **Three-Dimensional Elements** - *Tetrahedral, Rectangular Prism (Brick)*

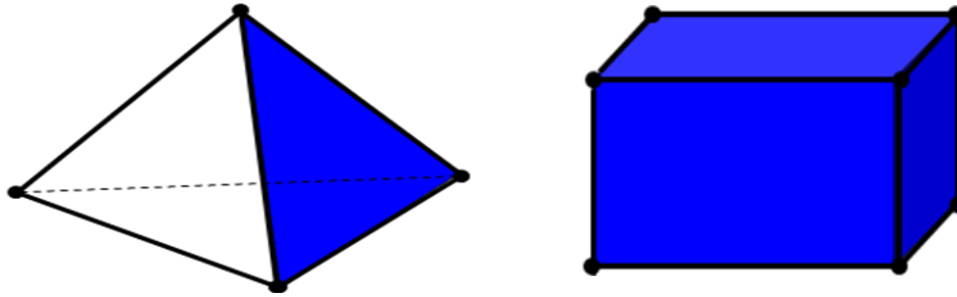


Figure - 1.10 (c): Three-Dimensional Elements

1.3.4 Necessity of Finite Element Analysis

a. Finite Element Analysis for Design Engineer

The FEA offers many important advantages to the design engineer:-

1. Easily applied to complex, irregular-shaped objects and having complex boundary conditions.
2. Applicable to problems like steady-state, time dependent etc.
3. Applicable to linear and nonlinear problems.
4. One method can solve a wide variety of problems, including problems in solid mechanics, fluid mechanics, chemical reactions, electromagnetic, biomechanics and heat transfer etc.
5. The FEA can be coupled to CAD programs to facilitate solid modeling and mesh generation.

b. Finite Element Analysis for Design Organization

Simulation using the FEA also offers important advantages to the design organization:

1. Reduced testing and redesign costs thereby shortening the product development time.
2. Identify issues in designs before tooling is committed.
3. Refine components before dependencies to other components prohibit changes.
4. Optimize performance before prototyping.
5. Discover design problems before litigation.

1.3.5 Applications of Finite Element Method

a. According to types of Analysis

The various types of analysis, which can be done with FEM, are -:

- i. Linear static analysis
- ii. Dynamic analysis
- iii. Buckling analysis

- iv. Thermal analysis
- v. Fatigue analysis
- vi. Optimization
- vii. CFD analysis
- viii. Crash analysis

i. Linear Static Analysis

It is the simplest and most commonly used type of analysis. Linear means straight line. $\sigma = \epsilon E$ is an equation of straight line ($y = mx$) passing through origin. “E” Elastic Modulus is slope of the curve and is a constant. In real life after crossing yield point material follows non linear curve but software follows same straight line.

There are two conditions for static analysis:

- a) No variation of force with respect to time (dead weight), $dF / dt = 0$
- b) Equilibrium condition, $\sum Force = 0, \sum Moments = 0$

Practical applications: All Aerospace, Automobile, Offshore and civil engineering industries perform linear static analysis.

Commonly used softwares: Nastran, Ansys, Abaqus, I-deas, Radioss, Cosmos, UG, Pro-Mechanica and Catia etc.

ii. Dynamic Analysis

Static analysis does not take in to account variation of load with respect to time. Output in the form of stress, displacement etc. with respect to time could be predicted by dynamic analysis.

Practical applications: Dynamic behavior of components subjected to dynamic loads.

Commonly used software: Nastran, Ansys, Abaqus, Matlab, I-deas NX, Radioss etc

iii. Linear Buckling Analysis

Linear buckling analysis is applicable for only compressive load. It is used to analyze the slender beams and sheet metal parts. Output of analysis is Critical value of load.

Practical applications: Commonly used for civil engineering applications. Mechanical engineering applications- vacuum vessel, long gear shifted rod analysis etc.

Commonly used software's: Nastran, Ansys, and Abaqus etc

iv. Thermal Analysis

Thermal analysis is used to predict the thermal response of structures. Adequate knowledge of temperature distribution in structures, thermal flux and structural response to thermal gradients is critical to successful designs.

Practical applications: Engine, radiator, exhaust system, heat exchanger, power plants, satellite design etc.

Commonly used software's: Ansys, Nastran, Abaqus, I-deas NX etc.

v. Fatigue Analysis

Fatigue analysis is used to calculate the life of the structure when subjected to repetitive load. S-N curve (alternating stress vs. cycles) or ϵ -N (alternating strain vs. reversals) is the base for fatigue calculation (like σ - ϵ diagram for static analysis).

Practical applications: Applicable to all components subjected to dynamic loading i.e. all automobile components. Fatigue accounts for 90% of failure in the real life.

Commonly used software's: MSC Fatigue, FEMFAT, FE SAFE, LMS etc

vi. Optimization

Optimization analysis is used to optimize the geometric parameters and shapes of over or under designed components.

Optimization for geometry parameters, work well at individual component level rather than complicated assemblies. Software is not useful to add or remove the geometry but it works only within specific limits.

Shape optimization is usually restricted to linear static or normal mode of dynamics. It is good tool for innovation kind of product (when initial shape is not known or fixed) Software can help for addition or removal of geometry.

Practical applications: Applicable to any component which is over or under designed.

Commonly used Software: Opti-struct, Tosca, Nastran, Ansys etc

vii. CFD Analysis

CFD is the branch of the fluid mechanics which use the numerical method to analyse the fluid dynamic problems. It is based on the Navier- Stroke equations (Mass, Momentum and Energy conservation equilibrium equations).

Practical application: Drag prediction and stream lining of a car, combustion chamber design to check an optimum fuel – air mixing, Aeroplane design etc.

Commonly used Software: Fluent, Star CD, CFX, CFD Expert etc

viii. Crash Analysis

Crash analysis is performed to find deformation, stress and energy absorbing capacity of various structural components of a vehicle hitting a stationary or moving object. Crash analysis can also be done to find the effects of crash on human body and making the ride safe for driver as well as passengers.

Commonly used software's: LS-Dyna, Pamcarsh, Radioss, Abaqus-Explicit, Madymo etc

b. According to types of Jobs

CAE group responsible for FEA related activities, receive following types of job orders -

- New design
- Optimization or cost cutting projects
- Failure analysis

i. New Design:

New or innovative kind of design is a real challenge for design engineer. In automobile industry, when new version of existing vehicle is launched (upgraded version), most of the components are quite similar to the existing one (scaled proportionately). Innovative kind of components is usually not more than 15 %.

At least initial run of this category of job is easy for CAE engineer. Sit with design & test engineer to decide boundary conditions and then run the analysis. Real work starts only when the prototype is prepared and test & FEA result correlation process is initialized. After achieving correlation various permutations and combinations could be carried out to make the product better and optimum from cost as well design point of view.

ii. Cost cutting or optimization projects:

At the moment Indian Auto sector is experiencing a boom but from 1995 to 2003 there was a slack. During the period most of industries were busy with cost cutting measures for their survival.

In Indian market till late 80's, same kind of vehicles were running on the road with out any change (do you remember old designs of Indian cars and heavy, bulky & noisy-trucks). These designs were transferred to India; companies in- 50's & 60's from their overseas collaborators (mainly American & Europeans). Design philosophy was different at that time i.e. design for infinite life. But slack in the market and emergence of new

tools like CAD/CAM/CAE, new cost efficient manufacturing techniques and availability of low cost materials forced auto manufacturers to adapt to the changing circumstances via optimization of design.

Suppose selling price of the product is Rs. 100 & actual manufacturing cost is Rs. 60. Reduction of cost even by say Rs 1/- by using CAD / CAM / CAE (reduction in thickness, change in material etc.) will result in lot of profit for the company.

Earlier days design philosophy was, “*Design for infinite life*”

- Survival for years
- Heavy & oversized components
- Noisy & rough operations
- High cost

Now a day’s design philosophy is, “*Design for warranty life*”

- Use & throw concept
- Life just greater than warranty offered by company
- Additional source of income, after sales services
- Light weight components
- low cost

iii. Failure Analysis

Warranty: Every company offers warranty on its product. Company is under legal binding to replace the component failing within warranty period, free of cost. It is not only additional cost which is incurred but also bad name to the product and company.

Probable reasons of failure

- Improper process
- Manufacturing defects
- Faulty material
- Environmental Conditions
- Weather
- Road Condition
- Genuine Design problem

1.3.6 Advantages of FEA

The advantages of FEA are as follows:

1. Easy to model irregular shapes
2. Possible to evaluate different materials
3. Can apply general load conditions
4. Large numbers and kinds of boundary conditions are possible in FEA
5. Different sizes of elements can be used where necessary
6. Dynamic effects, nonlinear behaviours and nonlinear materials can be examined
7. Reduce the number of prototypes required in the design process
8. Increase the visualization of the product
9. Reduce the Design Cycle time of a product
10. Testing on prototype also decreases
11. Optimum design

1.4 Introduction to Vibration

Vibration is the motion of a particle or a body or system of connected bodies displaced from a position of equilibrium. Most vibrations are undesirable in machines and structures because they produce increased stresses, energy losses, cause added wear, increase bearing loads, induce fatigue, create passenger discomfort in vehicles, and absorb energy from the system. Rotating machine parts need careful balancing in order to prevent damage from vibrations.

Vibration occurs when a system is displaced from a position of stable equilibrium. The system tends to return to this equilibrium position under the action of restoring forces (such as the elastic forces, as for a mass attached to a spring, or gravitational forces, as for a simple pendulum). The system keeps moving back and forth across its position of equilibrium. A system is a combination of elements intended to act together to accomplish an objective. For example, an automobile is a system whose elements are the wheels, suspension, car body, and so forth. A static element is one whose output at any given time depends only on the input at that time while a dynamic element is one whose present output depends on past dynamic. In the same way we also speak of static and dynamic systems. A static system contains all elements while a dynamic system contains at least one dynamic element.

A physical system undergoing a time-varying interchange or dissipation of energy among or within its elementary storage or dissipative devices is said to be in a dynamic system. All of the elements in general are called passive, i.e., they are incapable of generating net energy. A dynamic system composed of a finite number of storage elements is said to be lumped & discrete, while a system containing elements, which are dense in physical space, is called continuous system. The analytical description of the dynamics of the discrete case is a set of ordinary differential equations, while for the continuous case it is a set of partial differential equations. The analytical formation of a dynamic system depends upon the kinematic or geometric constraints and the physical laws governing the behavior of the system.

With the discovery of musical instruments like drums, the vibration became a point of interest for scientists and since then there has been much investigation in the field of vibration. The mass is inherent of the body and elasticity causes relative motion among its parts. When body particles are displaced by the application of external force, the internal forces in the form of elastic energy are present in the body. These forces try to bring the body to its original position. At equilibrium position, the whole of the elastic energy is converted into kinetic energy and body continues to move in the opposite direction because of it. The whole of the kinetic energy is again converted into elastic or strain energy due to which the body again returns to the equilibrium position.

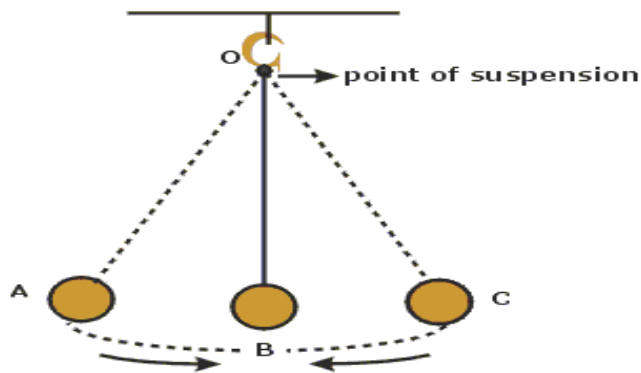


Figure - 1.11: Swinging of simple pendulum

In this way, vibratory motion is repeated indefinitely and exchange of energy takes place. Thus, any motion which repeats itself after an interval of time is called vibration or oscillation. The swinging of simple pendulum as shown in Fig. 1.1 is an example of vibration or oscillation as the motion of ball is to and fro from its mean position repeatedly.

The main reasons of vibration are as follows:

1. Unbalanced centrifugal force in the system. This is caused because of non-uniform material distribution in a rotating machine element.
2. Elastic nature of the system.
3. External excitation applied on the system.
4. Winds may cause vibrations of certain systems such as electricity and telephone lines, etc.

1.4.1 Importance of Vibration study in Engineering

The structures designed to support the high speed engines and turbines are subjected to vibration. Due to faulty design and poor manufacture there is unbalance in the engines which causes excessive stresses in the rotating system because of vibration. The vibration causes rapid wear of machine parts such as bearings and gears. Unwanted vibrations may cause loosening of parts from the wheels of locomotive can leave the track due to excessive vibration which results in accident or heavy loss. Many buildings, structures and bridges fall because of vibration. If the frequency of excitation coincides with one of the natural frequencies of the system, a condition of resonance is reached, and dangerously large oscillations may occur which may result in the mechanical failure of the system.

Sometimes because of heavy vibrations proper readings of instruments cannot be taken. Excessive vibration is dangerous for human beings. Thus keeping in view all these devastating effects, the study of vibration is essential for a mechanical engineer to minimize the vibration effects over mechanical components by designing them suitably.

Vibration can be used for useful purposes such as vibration testing equipments, vibratory conveyors, hoppers, sieves and compactors. Vibration is found very fruitful in mechanical workshops such as in improving the efficiency of machining, casting, forging and welding techniques, musical instruments and earthquakes for geological research. It is useful for the propagation of sound.

Thus undesirable vibrations should be eliminated or reduced up to certain extent by the following methods:

1. Removing external excitation, if possible.
2. Using shock absorbers.
3. Dynamic absorbers.
4. Resting the system on proper vibration isolators.

Important terms

Natural frequency - When no external force acts on the system after giving it an initial displacement, the body vibrates. These vibrations are called free vibrations and their frequency as natural frequency. It is expressed in rad/sec or Hertz.

Fundamental Mode of Vibration - The fundamental mode of vibration of a system is the mode having the lowest natural frequency.

Degree of freedom - The minimum number of independent coordinates required to specify the motion of a system at any instant is known as degrees of freedom of the system. A cantilever beam has infinite degree of freedom.

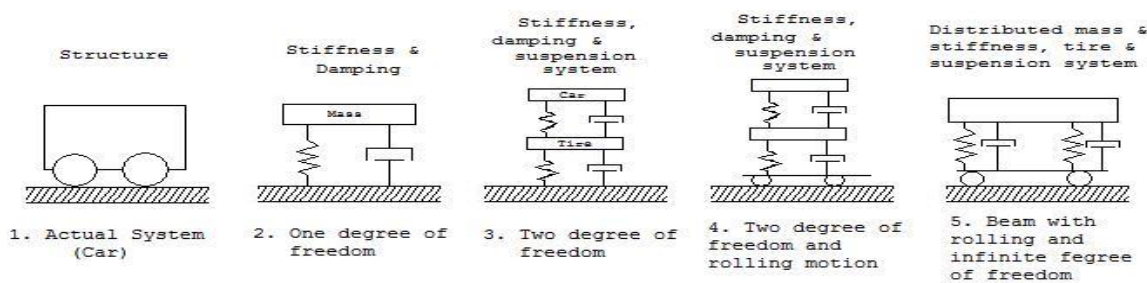


Figure - 1.12: Degrees of freedom

Simple Harmonic Motion - The motion of a body to and fro about a fixed point is called simple harmonic motion. The motion is periodic and its acceleration is always directed towards the mean position and is proportional to its distance from mean position.

Damping - It is the resistance to the motion of a vibrating body. The vibrations associated with this resistance are known as damped vibrations.

Phase difference - Suppose there are two vectors x_1 and x_2 having frequencies ω rad/sec each. The vibrating motions can be expressed as

$$x_1 = A_1 \sin \omega t$$

$$x_2 = A_2 \sin(\omega t + \phi)$$

In the above equation the term ϕ is known as the phase difference.

Resonance - When the frequency of external excitation is equal to the natural frequency of a vibrating body, the amplitude of vibration becomes excessively large. This concept is known as resonance.

Mechanical systems - The systems consisting of mass, stiffness and damping are known as mechanical systems.

Continuous and Discrete Systems - Most of the mechanical systems include elastic members which have infinite number of degree of freedom. Such systems are called continuous systems. Continuous systems are also known as distributed systems. Cantilever, simply supported beam etc. are the examples of such systems.

System with finite number of degree of freedom is called discrete or lumped systems.

1.4.2 Parts of Vibration

A vibratory system basically consists of three elements, namely the mass, the spring and damper. In a vibrating body there is exchange of energy from one form to another. Energy is stored by mass in the form of kinetic energy ($\frac{1}{2} m\dot{x}^2$), in the spring in the form of potential energy ($\frac{1}{2} kx^2$) and dissipated in damper in the form of heat energy which oppose the motion of the system. Energy enters the system with the application of external force known as excitation. It disturbs the mass from its mean position. The kinetic energy is converted into potential energy and potential energy into kinetic energy. This sequence goes on repeating and the system continues to vibrate. At the same time damping force acts on the mass and oppose its motion. Thus some energy is dissipated in each cycle of vibration due to damping. The free vibration dies out and the system remains at its static equilibrium position. A basic vibratory system shown in *Figure - 1.13*

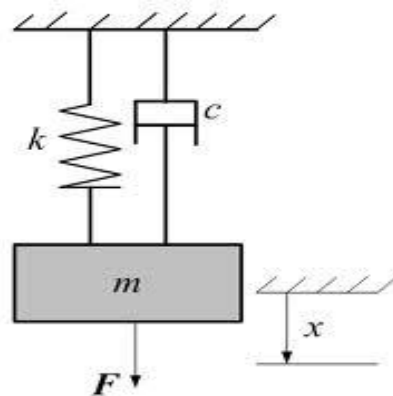


Figure - 1.13: Parts of vibration

1.4.3 Types of Vibration

Some of the important types of vibration are as follows:

1. Free and Forced Vibration

After disturbing the system the external excitation is removed, and then the system vibrates on its own. This type of vibration is known as “free vibration”. Simple pendulum is one of the examples.

The vibration which is under the influence of external force is called “forced vibration”. Machine tools, electric bells etc. are the suitable examples.

2. Linear and Non-linear Vibration

If in a vibration system mass, spring and damper behave in a linear manner, the vibrations caused are known as “linear” in nature. Linear vibration is governed by linear differential equations. They follow the law of superposition. Mathematically speaking, if a_1 and a_2 are the solutions of equations, then (a_1+a_2) will be the solution of equation

$$mx+cx+kx=F_1(t)$$

$$mx+cx+kx=F_2(t)$$

$$mx+cx+kx=F_1(t)+F_2(t)$$

On the other hand, if any of the basic components of a vibratory system behaves non-linearly, the vibration is called “non-linear”. Linear vibration becomes non-linear for very large amplitude of vibration. It does not follow the law of superposition.

3. Damped and Undamped vibration

If the vibratory system has a damper, the motion of the system will be opposed by it and the energy of the system will be dissipated in friction. This type of vibration is known as “damped vibration”.

On the contrary, the system having no damper is known as “undamped vibration”.

4. Deterministic and Random Vibration

If in the vibratory system the amount of external excitation is known in magnitude, it causes “deterministic vibration”. Contrary to it the non-deterministic vibrations are known as “random vibrations”.

5. Longitudinal, Transverse and Torsional Vibrations

A body of mass m carried on one end of a weightless spindle, the other end being fixed. If the mass m moves up and down parallel to the spindle axis, it is said to “longitudinal vibrations”.

When the particles of the body or shaft move approximately perpendicular to the axis of the shaft the vibrations so caused are known as “transverse”.

If the spindle gets alternately twisted and untwisted on account of vibratory motion of the suspended disc, it is called to be undergoing “torsional vibrations”.

6. Transient Vibration

In ideal systems the free vibrations continue indefinitely as there is no damping. The amplitude of vibration decays continuously because of damping (in a real system) and vanishes ultimately. Such vibration in a real system is called “transient vibration”.

1.5 Dynamic Analysis Process

Before conducting a dynamic analysis, first define the goal of the analysis prior to the formulation of the finite element model. Consider the dynamic analysis process shown in *Figure - 1.14* and evaluate the finite element model in terms of the type of dynamic loading to be applied to the structure. This dynamic load is known as the “dynamic environment”. The dynamic environment governs the solution approach (i.e., normal modes, transient response, frequency response, etc.). This environment also indicates the dominant behavior that must be included in the analysis (i.e., contact, large displacements, etc.). Proper assessment of the dynamic environment leads to the creation of a more refined finite element model and more meaningful results.

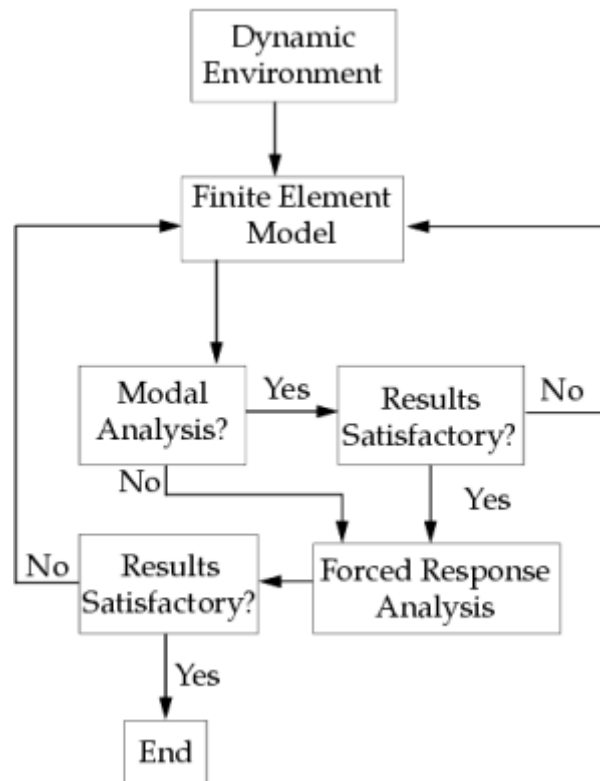


Figure – 1.14: Dynamic Analysis Process

An overall system design is formulated by considering the dynamic environment. As part of the evaluation process, a finite element model is created. This model should take into account the characteristics of the system design and, just as importantly, the nature of the dynamic loading (type and frequency) and any interacting

media (fluids, adjacent structures, etc.). At this point, the first step in many dynamic analyses is a “modal analysis” to determine the structure’s natural frequencies and mode shapes.

In many cases the natural frequencies and mode shapes of a structure provide enough information to make design decisions. For example, in designing the supporting structure for a rotating fan, the design requirements may require that the natural frequency of the supporting structure have a natural frequency either less than 85% or greater than 110% of the operating speed of the fan. Specific knowledge of quantities such as displacements and stresses are not required to evaluate the design.

Forced response is the next step in the dynamic evaluation process. The solution process reflects the nature of the applied dynamic loading. A structure can be subjected to a number of different dynamic loads with each dictating a particular solution approach. The results of a forced response analysis are evaluated in terms of the system design. Necessary modifications are made to the system design. These changes are then applied to the model and analysis parameters to perform iteration on the design. The process is repeated until an acceptable design is determined, which completes the design process. The primary steps in performing a dynamic analysis are summarized as follows:

- Define the dynamic environment (loading).
- Formulate the proper finite element model.
- Select and apply the appropriate analysis approaches to determine the behavior of the structure.
- Evaluate the results.

1.5.1 Dynamic Analysis Types

The following types of basic dynamic analysis are:

1. Modal Analysis (Real eigenvalue analysis).
2. Frequency response analysis (steady-state response of linear structures to loads that vary as a function of frequency).
3. Transient response analysis (response of linear structures to loads that vary as a function of time).

1. Modal Analysis (Real eigenvalue analysis)

Modal analysis is used to determine the basic dynamic characteristics of a structure. The results of an analysis indicate the natural frequencies and shapes at which a structure naturally tends to vibrate. Although

the results of an analysis are not based on a specific loading, they can be used to predict the effects of applying various dynamic loads.

2. Frequency Response Analysis

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, amplitude, and phase are specified. Frequency response analysis is limited to linear elastic structures.

3. Transient Response analysis

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. Transient response analysis is most commonly applied to structures with linear elastic behavior.

1.6 Softwares to be used

1.6.1 HyperWorks Introduction

HyperWorks is an enterprise simulation solution for rapid design exploration and decision-making. As one of the most comprehensive CAE solutions in the industry, HyperWorks provides a tightly integrated suite of best-in-class tools for modeling, analysis, optimization, visualization, reporting, and performance data management. Leveraging a revolutionary “pay-for-use” token-based business model, HyperWorks delivers increased value and flexibility over other software licensing models:

- HyperMesh
- OptiStruct
- HyperView

1. HyperMesh

Altair HyperMesh is a high-performance finite element pre-processor that provides a highly interactive and visual environment to analyze product design performance. With the broadest set of direct interfaces to commercial CAD and CAE systems, HyperMesh provides a proven, consistent analysis platform for the entire enterprise. With a focus on engineering productivity, HyperMesh is the user-preferred environment for:

Solid Geometry Modeling

Shell Meshing

Model Morphing

Detailed Model Setup

Surface Geometry Modeling

Solid Mesh Generation

Automatic mid-surface generation

Batch Meshing

2. OptiStruct

OptiStruct is a finite element solver for linear problems. OptiStruct is a part of the HyperWorks toolkit, described as a finite element and multi-body dynamics software which can be used to design and optimize structures and mechanical systems.

3. HyperView

HyperView is a complete post-processing and visualization environment for finite element analysis (FEA), CFD, multi-body system simulation, digital video, and engineering data. HyperView enables to visualize data interactively as well as capture and standardize your post-processing activities using process automation features. HyperView combines advanced animation and XY plotting features with window synching to enhance results visualization.

Chapter 2

LITERATURE REVIEW

2.1 Literature Review

A review of the literature related to the design and analysis with a focus on vibration analysis of the radiator mounting brackets is presented here.

Doo-Ho Lee *et al.* [1], optimized the shape of the air-conditioner compressor mounting bracket of a passenger car by using finite element software package, and the optimized results were verified by tests. An objective function was to reduce the weight of the bracket. Two design methods were used, first were resonant frequency of the compressor assembly and second were to make a model of compressor assembly & analyze it by using finite element analysis. The bracket was modeled by solid elements and the compressor was represented by rigid masses. For simulation of the dynamics stresses in the durability test, the lumped mass method was used. Optimal shapes of the bracket were obtained by using MSC/NASTRAN. The verification tests were conducted on the workbench and in a vehicle. The optimized bracket verification tests were fulfilled. Test results showed that the developed optimization procedure of the bracket was valid in the complex real world.

E.S. Palma *et al.* [2], analyzed the fatigue behavior of an automobile body part, according to the standards of performance. The methodology is based on experiments performed on a rear trailer tow hook pin of a passenger automobile vehicle. Experiments were performed simulating the actual conditions in the customer environment. Stress and strain were experimentally measured by using strain gages, bonded on assembly critical points. Besides, stress analysis was also performed using a finite element program. Fatigue analysis is used to assess and to compare the fatigue damage imposed during laboratory experiments.

G. Fournalis *et al.* [3], observed that the use of high strength steels (HSS) in automotive components is steadily increasing as automotive designers use modern steel grades to improve structural performance, reduce vehicle weight and enhance crash performance. Weight reduction can be achieved by substituting mild steel with a thinner gauge HSS, however, it must be ensured that no deterioration in performance including fatigue capability occurs.

In this study, tests have been carried out to determine the effects that gauge and material strength have on the fatigue performance of a fusion welded automotive suspension arm. Current finite element (FE) modeling and fatigue prediction techniques have been evaluated to determine their reliability when used for thin strip steels.

Results have shown the fatigue performance of welded components to be independent of the strength of the parent material for the steel grades studied, with material thickness and joining process the key features determining the fatigue performance.

This study also indicates that with the application of modern technologies, such as tailor-welded blanks (TWB), significant weight savings can be achieved. This is demonstrated by a 19% weight reduction with no detrimental effect on the fatigue performance.

Gaikwad Vikrant C *et al.* [4], studied the 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 behavior. Modeling is done CATIA software, meshing is carried out with HYPERMESH software and solution is acquired using NASTRAN solver.

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.

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.

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.

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 for a thickness of 3 mm. In iteration 2, we have increased the thickness to 4mm. The stress value i.e.107.11 N/mm² is still greater than the yield stress i.e. 80 N/mm². In spite of this, the failure occurs. Iteration 3, we have increased the thickness 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.

Jeong Woo Chang et al. [5], optimize the topology at the concept design stage where structural analysis methodology of compressor bracket was verified on the static and dynamic loading condition. Objective function was to minimize the natural frequency. New bracket shape was based on the topology optimize results which compared with the traditional concept model. It was analyzed that a new bracket would not fail during a vibration testing and these results were verified with a fabricated real sample under the durability condition.

Pavan B. Chaudhari et al. [5], optimized the natural frequency of engine mount bracket by using three different lightweight materials by using finite element analysis. Selected materials were Aluminium (Al), Magnesium (Mg) and Cast Iron (CI). Evaluation of proposed model of engine mount bracket was performed using finite element analysis (FEA) and modal analysis techniques.

In FEA, all possible combination of three materials Al, Mg and CI for different mesh sizes of 2, 3, 5, 10 and 15 mm had been done and graph of stress verses natural frequency and stress verses deformation was plotted by Ansys software. It conclude that Mg alloy had higher natural frequency followed by Al alloy. Where, in modal analysis the engine mounting bracket was tested on shaker table. Two excitation method, traditional impulse and periodic impulse method were used to measure the natural frequency. Average natural frequency values for Al, Mg and CI were 258.2 Hz, 257.9 Hz and 200.3 Hz respectively.

Both FEA and modal analysis results were compared and it concludes that CI gives less frequency thus rejected. They suggest that Mg and Al both were preferred material for engine mounting brackets.

Saharash Khare *et al.* [7], the customers of the vehicles reported high noise and vibration in the engine at an early stage of service lifetime. An analysis of the various components of the internal combustion engine was carried out. Subsurface cracks and pit marks were seen in the crank pin, roller bearings and big end surfaces of the connecting rod. It was found that high wear at the interface of these components was the main culprit. A laboratory test set-up was developed to correlate and reproduce the field failures. The loads and boundary conditions obtained from the experiments were used in the finite element model of the connecting rod assembly. Results show high interfacial pressure and stresses near the junction of web and flange of the connecting rod. The modified design of the connecting rod shows significant reduction in the extreme pressure in FEM resulting in the significant enhancement of durability life in laboratory test. A discussion of the spalling problem has been provided leading to the connection of pick pressure and spalling phenomena.

Sahil Naghate *et al.* [8], used finite element analysis tool to analyze the engine mounting bracket using FEA Package. Materials selected were aluminum alloy and magnesium alloy. The results obtained from the static and dynamic analysis have shown that the magnesium is better than aluminum. The main advantage of the magnesium engine mounting bracket was its light weight, which can increase fuel efficiency. The main problem of using magnesium instead of aluminum was its higher cost. Thus, it can be concluded that magnesium and aluminum both preferred as a material for an engine mounting bracket.

Senthilnathan Subbiah *et al.* [9], automotive industries perform durability tests on vehicles in the end-user environment to reduce failures and warranty costs in the end-user hands. In this paper we present the failure analysis of muffler mounting brackets of three-wheeler vehicles observed during the durability test. Cracks at the weld location between the engine cradle and brackets were observed in all the vehicles at an average distance of 10,000 km. Many possible causes of the failures are identified using fishbone diagram. The fishbone diagram is a graphical analysis tool that provides a systematic way of looking at effects and the causes that contribute to those effects. Investigations were carried out on the design using finite element method (FEM). A FEM model was developed for the engine cradle assembly in which engine and muffler were modeled as point mass. Vertical forces were applied on the assembly using '4g' criterion, where 'g' is the acceleration due to

gravity. The applied force accounts for the high impact forces that act on the structure during durability testing. Results show high magnitude of stresses and strain energy at the weld location. Analysis of the design suggests that bracket was acting as a cantilever beam with one-plane welding mounted on the engine cradle. Modified design, though eliminated the above failure, shifted the failure mode to the bush-bracket region. Various design modifications were carried out and its effect on durability has been discussed. FEM analysis on the final design shows significant reduction in stresses at the critical locations. The new design of the bush-bracket system passed the durability target of 1,00,000 km.

Umesh S. Ghorpade *et al.* [10], the need for light weight structural materials in automotive applications is increasing as the pressure for improvement in emissions and fuel economy increases. The most effective way of increasing automobile mileage is to reduce vehicle weight. The incorporation of aluminium and magnesium alloys into automotive structures has steadily increased to meet all these requirements. Engine bracket has been designed as a framework to support engine. Vibration and fatigue of engine bracket has been continuously a concern which may lead to structural failure. It is a significant study to understand the structural characteristics and its dynamic behavior. This paper presents and focuses on some Finite Element (FE) analysis of a typical engine bracket of a car will be carried out and natural frequency will be determined.

Vibration plays a critical role in Engine components, especially in the supporting bracket. Gray Cast Iron is essentially a brittle material and this is evident in the results that the low natural frequency will prove as a hindrance in vibration characteristic of the bracket.

In terms of analysis, Al alloy and Mg alloy are showing almost same value of natural frequency and indicate that any one of them would be a better choice than Gray Cast Iron.

However, in terms of FEA there is a caveat, which being that in Modal FEA, the effect of Damping is not considered. In Practical terms, Mg alloy exhibits better damping characteristics than Al alloy. Hence as far as the recommendation goes, Mg alloy will be preferred.

V. Velso *et al.* [11], the failure of longitudinal stringer of vehicle was reported. Plastic deformation and cracks were observed during durability tests of prototype vehicle. Stress analysis was performed using finite element method. 6 different reinforced models were proposed and a combination of higher strength and cost were the reason of choice of best solution. The finite element analysis of several different models leads to a reduction of physical and expensive tests. So, it is not necessary the production of several prototype. Compression test were performed to compare with numerical results. Then three vehicles were produced with the reinforced stringer according to selected model. These vehicles were submitted to durability test and cracks and other damages were not longer observed. Thus the proposed solution was approved.

CHAPTER 3

PROBLEM FORMULATION

3.1 Gaps in literature

From the literature review it is found that a lot of work has been done in the area of vibration analysis for the different parts of an automobile. But, most of the work has mainly focused on modal analysis, to find the natural frequency of the component under study and static analysis to evaluate the stresses.

So, some of the gaps that have been identified are as follows:

- In the studied literature only modal analysis has been done.
- Frequency response analysis has not been done to find the displacements and stresses in the brackets, which is required in a dynamic analysis.
- The methodology for doing the frequency response analysis has not been discussed in the studied literature.

3.2 Objectives of the present work

- i. To propose the dynamic analysis procedure to analyze the vibration characteristics of different parts mounted on the body of an automobile (on the engine side), by taking the case of a radiator mounting bracket.
- ii. To carry out the modal analysis for finding the natural frequency & mode shapes and frequency response analysis to analyze the response of the structure in terms of displacements and stresses.
- iii. Also, the proposed analysis procedure will be used to study the effects of different materials and features like ribs in the bracket for improvements like increasing the natural frequency and lowering the stress levels.
- iv. Mass optimization of the radiator mounting bracket by using the proposed dynamic analysis procedure will be done.

3.3 Methodology

The analysis for the radiator mounting bracket will be done as follows:

CAD modeling of the bracket, radiator and fixture in CAD software.

- Pre-processing in a CAE software
- Discretization of the geometry, using solid elements.
- Application of the boundary conditions (taken from standard testing conditions in the automotive industry).
- Solution for normal modes and frequency response analysis.
- Post-processing – viewing displacements, stresses and interpreting the results.
- For optimization, remove the material from radiator mounting bracket where stresses are negligible.

CHAPTER 4

ANALYSIS OF RADIATOR MOUNTING BRACKET

4.1 Simulation of Radiator Mounting Bracket

This section discusses the methodology of analysis of the radiator mounting bracket.

Following steps were followed for the analysis of the bracket.

- CAD model was generated with the help of reverse engineering of the radiator mounting bracket (physical part).
- Mesh was generated for the analysis.
- Dynamic analysis (normal modes and modal frequency response) was performed.
- Static analysis was done and compared with dynamic analysis.

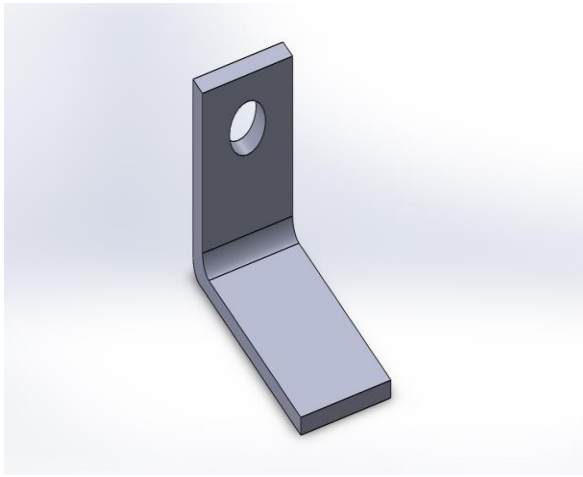
4.2 Reverse Engineering

As computer aided design has become more popular, reverse engineering has become a viable method to create a 3D virtual model of an existing physical part for use in 3D CAD, CAM, CAE or other software. The physical object can be measured by using:

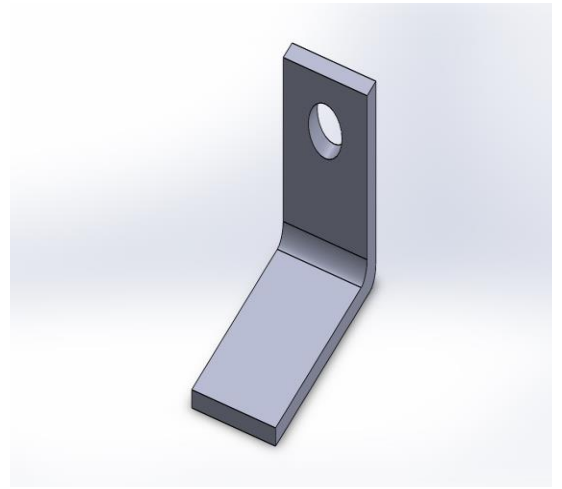
3D scanning technologies like Coordinate measuring machine, laser scanners, structured light digitizers, or industrial CT Scanning (computed tomography).

Manual measuring devices such as flat surface plate or table, level meter, scribe, vernier caliper, square, divider, scale, depth gauge etc.

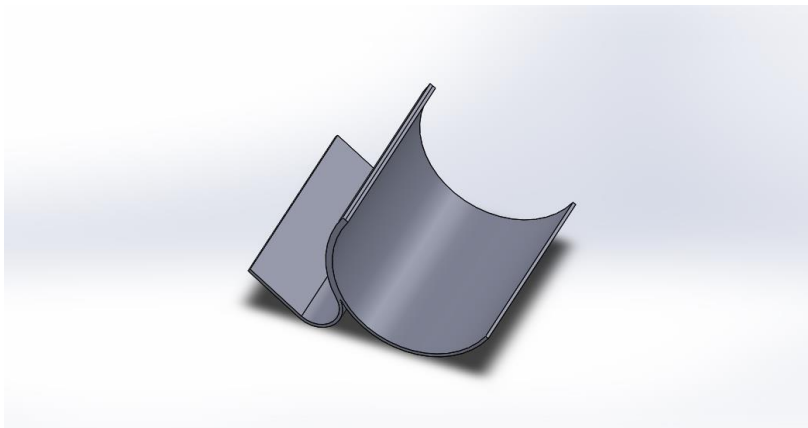
Reverse engineering of the radiator mounting bracket is shown in fig 4.1 using manual measuring devices.



a)
Small Left Bracket



b)
Small Right Bracket



c)
Big Bracket

Fig 4.1: Radiator Mounting Brackets

CAD Model

From the reverse engineering data, the component is modeled in Solid Works software.

The generated CAD model is shown in fig. 4.2

Solid Works has .SLDPRT extension which cannot be directly read by the CAE software due to data losses. Thus the file (.SLDPRT) was converted into a neutral file format for the data transfer (like IGES, Para-solid and Step etc).

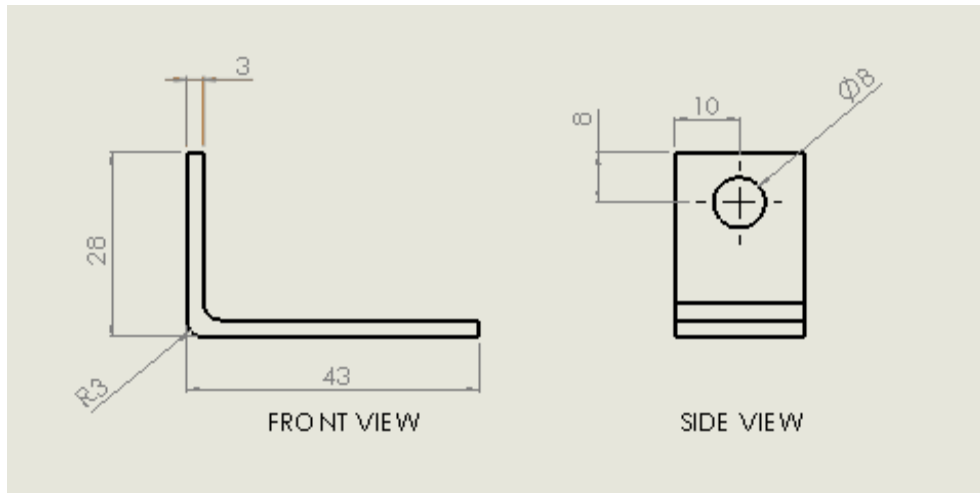


Fig 4.2(a): Detailed Drawing of Small Left Radiator Mounting Bracket

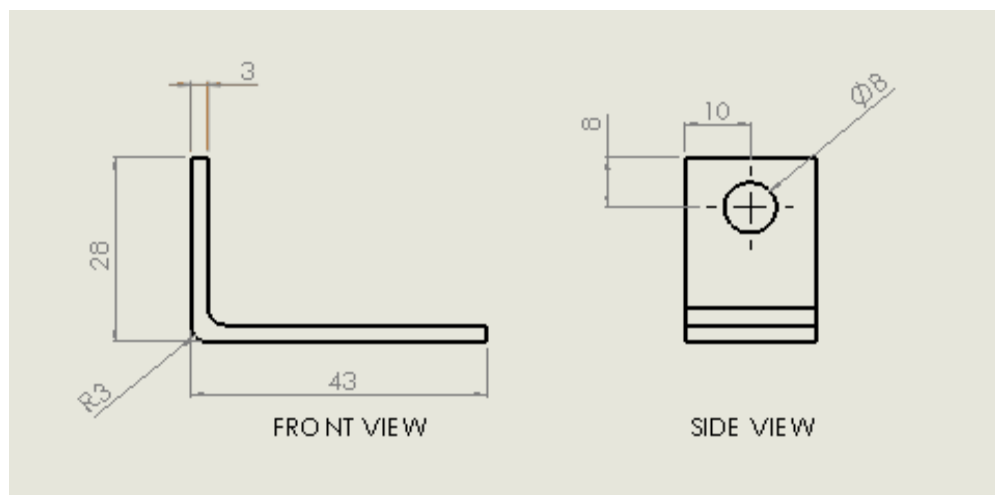


Fig 4.2(b): Detailed Drawing of Small Right Radiator Mounting Bracket

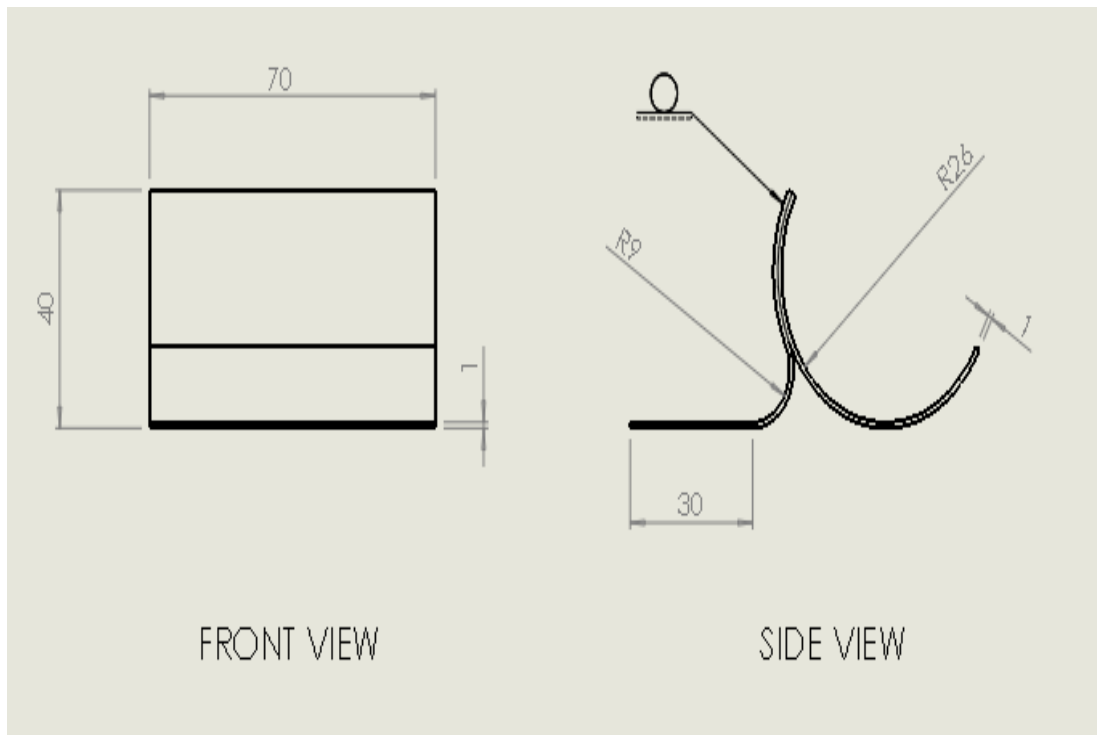


Fig 4.2(c): Detailed Drawing of Radiator Mounting Big Bracket

4.3 Mesh Generation

To import the CAD model in CAE software, it was converted into the step file format. For meshing tetrahedral elements were used. Meshing of the radiator mounting bracket was done according to the geometry.

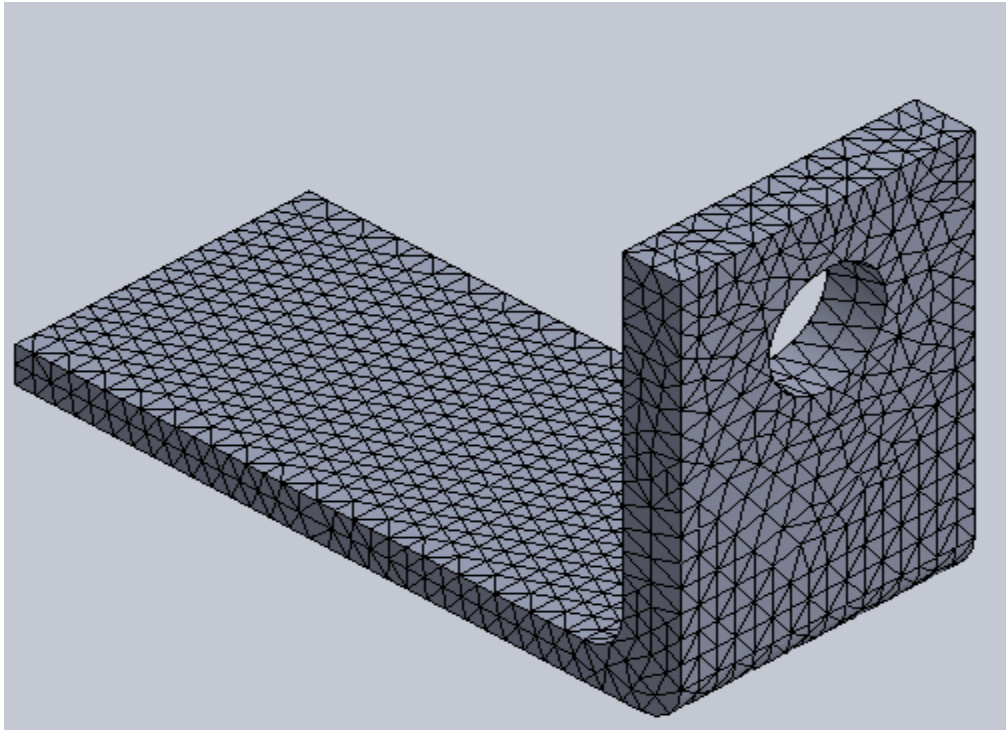


Fig 4.3.1: Tetrahedral Meshing of Small Left Radiator Mounting Bracket

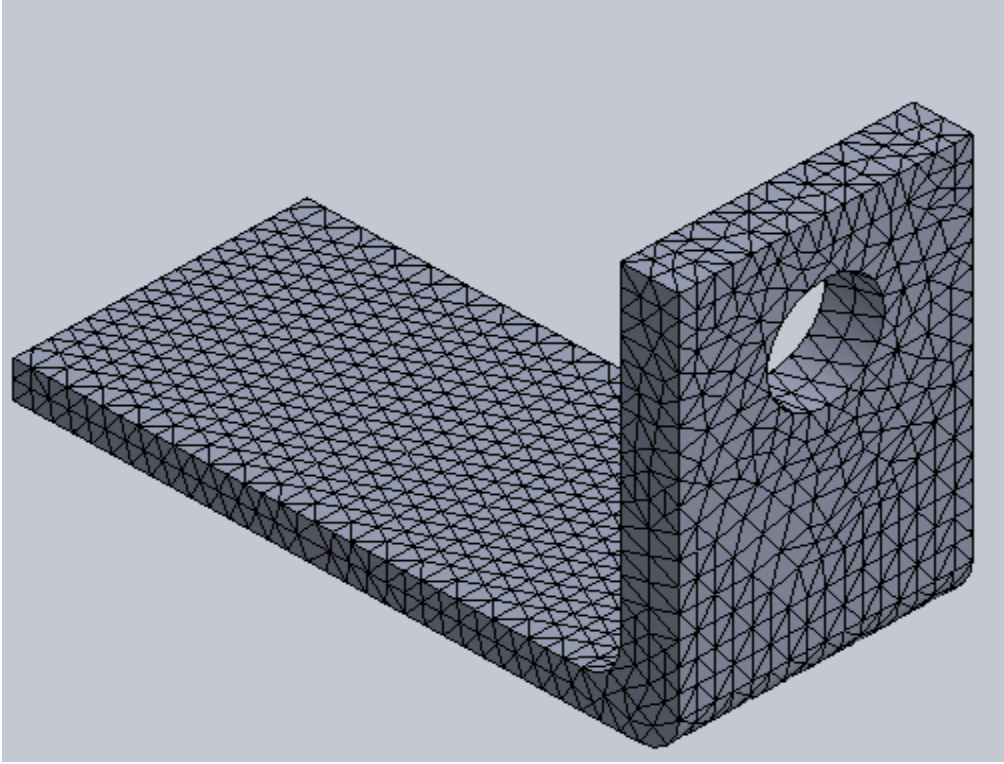


Fig 4.3.2: Tetrahedral Meshing of Small Right Radiator Mounting Bracket

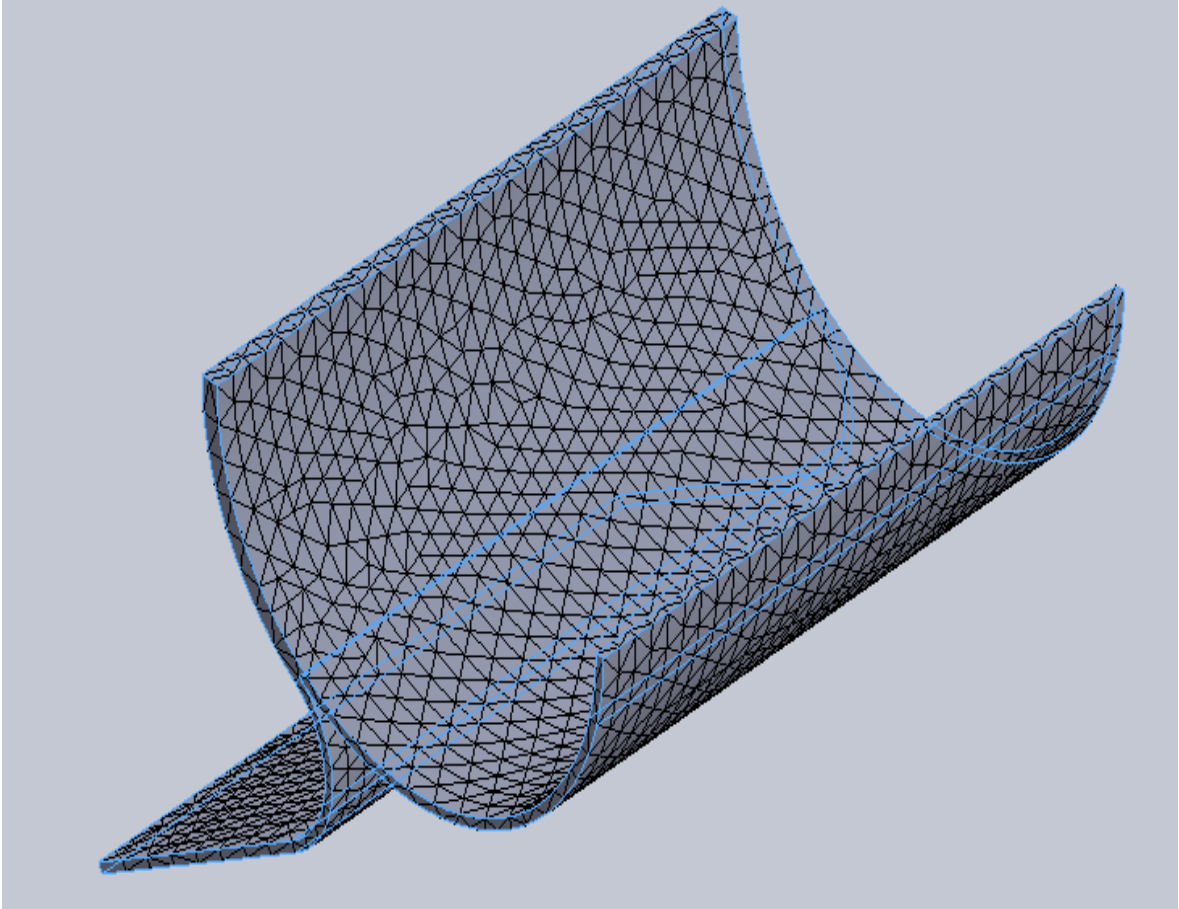


Fig 4.3.3: Tetrahedral Meshing of Big Radiator Mounting Bracket

Quality Checks for Tetra Meshing

The ideal shape for a tetrahedron element is an equilateral tetrahedron. The various quality parameters are

- Tetra Collapse
- Skewness
- Volumetric Skewness
- Aspect ratio
- Jacobian

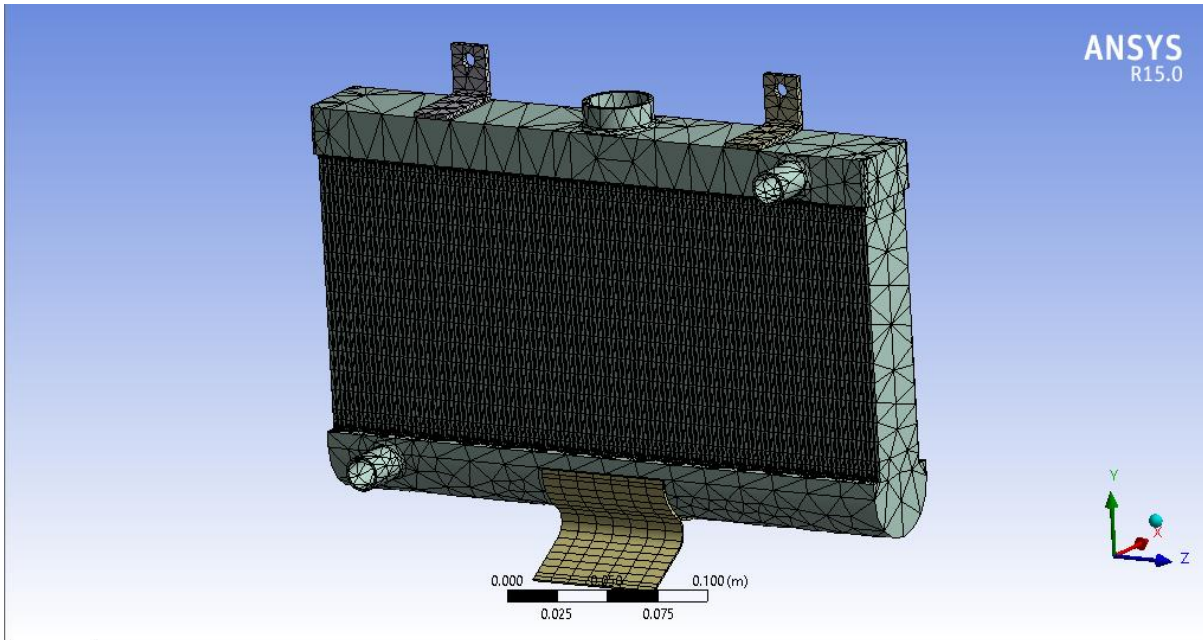


Fig 4.3.4 Meshing of the radiator assembly

4.4 Dynamic Analysis

Dynamic analysis for the radiator mounting bracket is done which is divided in two steps:

- Normal modes analysis.
- Modal frequency response analysis.

These two steps are related to each other. Output parameters of the first are used as input parameters of the second step.

4.5 Normal Modes Analysis

This analysis was done to find out the natural frequencies and mode shapes of the bracket.

- Need of normal mode Analysis. It is the basis design property. For specific components like engine mounting bracket or chassis mounting bracket, it is one of the important design approval criteria.
- Resonance: It is produced when external frequency becomes equal to natural frequency. The components may fail due to resonance as very high amplitude of vibration builds up.
- To control the noise and the vibration in components.

- **Conditions for the normal modes Analysis**

Following conditions are applied for the normal modes analysis:

- No external force as per the requirement of natural frequency external force is not applied.
- Constraints: Natural frequency analysis is carried out as per actual constraint.
- Damping: It is neglected for the natural frequency calculations.
- Output from analysis: The results obtained are magnitude of the frequency and the mode shape.

4.5.1 Normal Modes Analysis of the Mounting Bracket

The modeling and meshing of the components is explained in the previous sections. The boundary conditions applied and the result obtained are presented in the section.

Boundary Conditions: The boundary conditions are applied to the meshed model. The different steps are given below:

- Defining the Material: To define a specific material, the material collectors are used in the software. The material used is "steel" with the following properties: linear elastic, isotropic and temperature independent. The value of different properties are:
 - Modulus of elasticity, $E = 6.6 \times 10^4, (\text{N/m}^2)$
 - Poisson ratio, ν (Nu) = 0.3
 - Density, $\rho = 7.85 \text{ g/cm}^3$
 - Geometry Creation
 - Import of Geometry
 - Mesh Generation
 - Loading Condition
 - Results
-

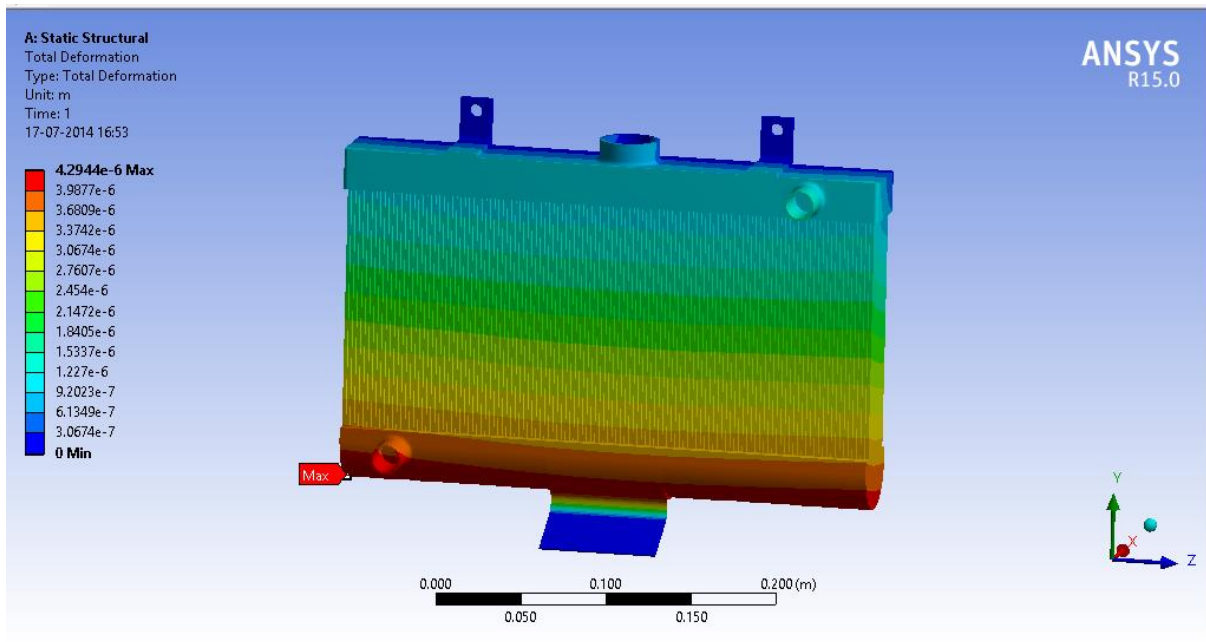


Fig 4.5 (a) Deformation in Mounting Brackets

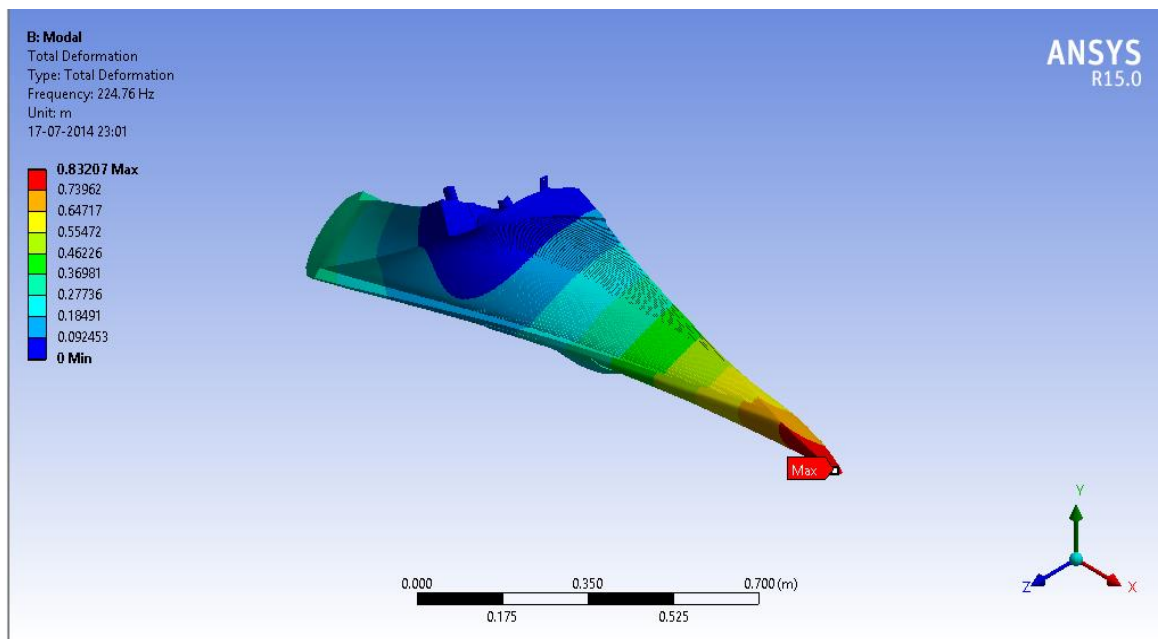


Fig 4.5 (b) Mode 1- 126.17 Hz (Natural Frequencies & Mode Shapes)

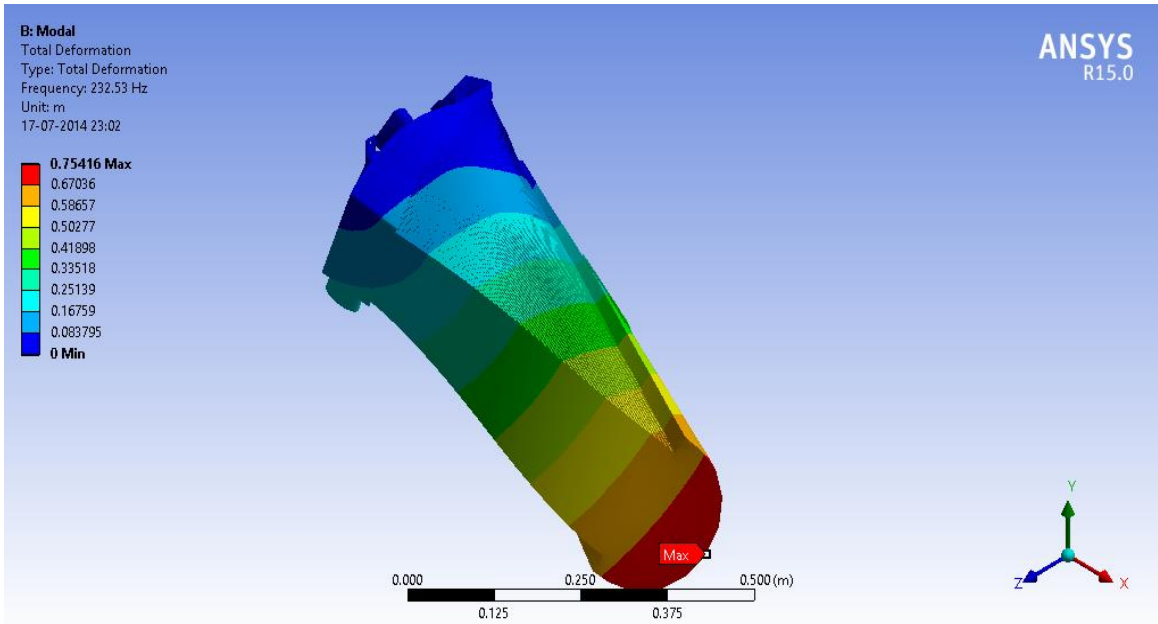


Fig 4.5 (c) Mode 2 – 220.64 Hz (Natural Frequencies & Mode Shapes)

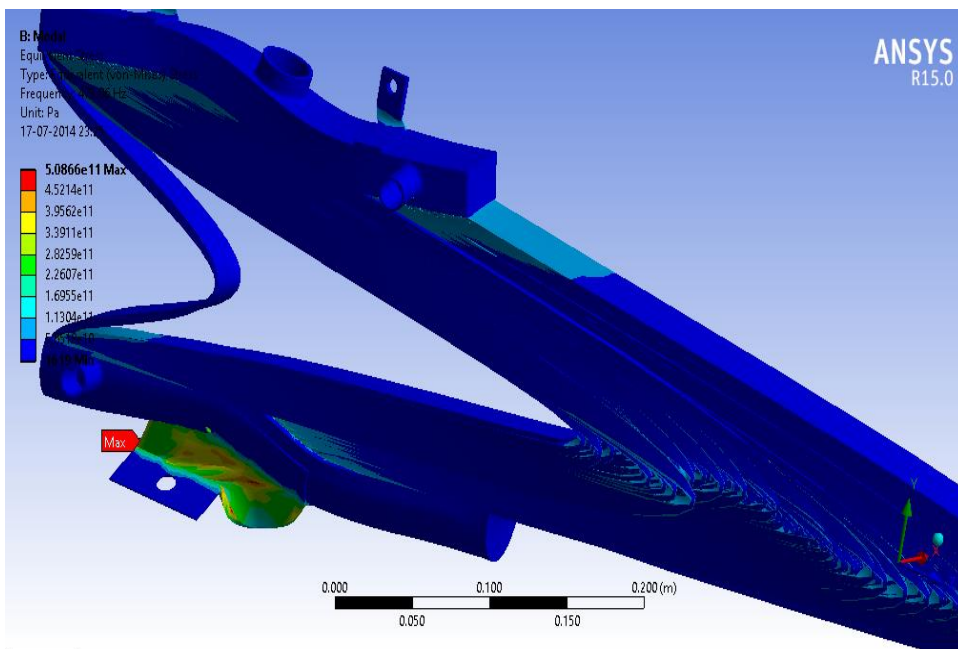


Fig 4.5 (d) Mode 3 – 489.20 Hz (Natural Frequencies & Mode Shapes)

4.6 Modal Frequency Response analysis of Radiator Mounting Bracket

In the modal frequency response analysis an external harmonic excitation is given by the external force. This force is in the form of $f \sin(\omega t)$, where ω is the frequency of the excitation. When it becomes equal to natural frequency of the system, high value vibrations are produced due to resonance. Modal frequency response analysis of radiator mounting bracket along all three axes is done.

a) Modal frequency response analysis along X-axis

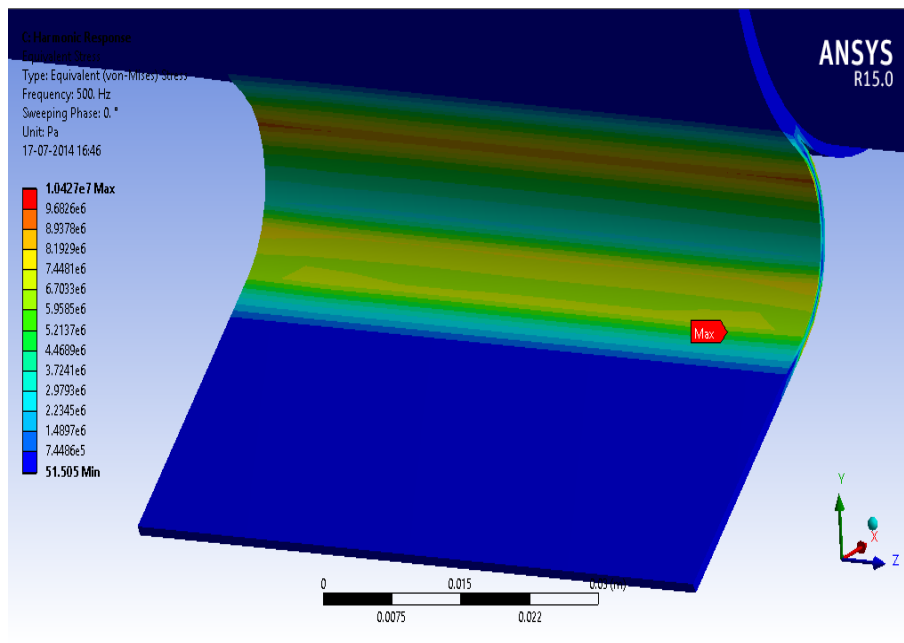


Fig 4.6 (a) Von Mises Stress in Mounting Bracket along X-axis

b) Modal frequency response analysis along Y-axis

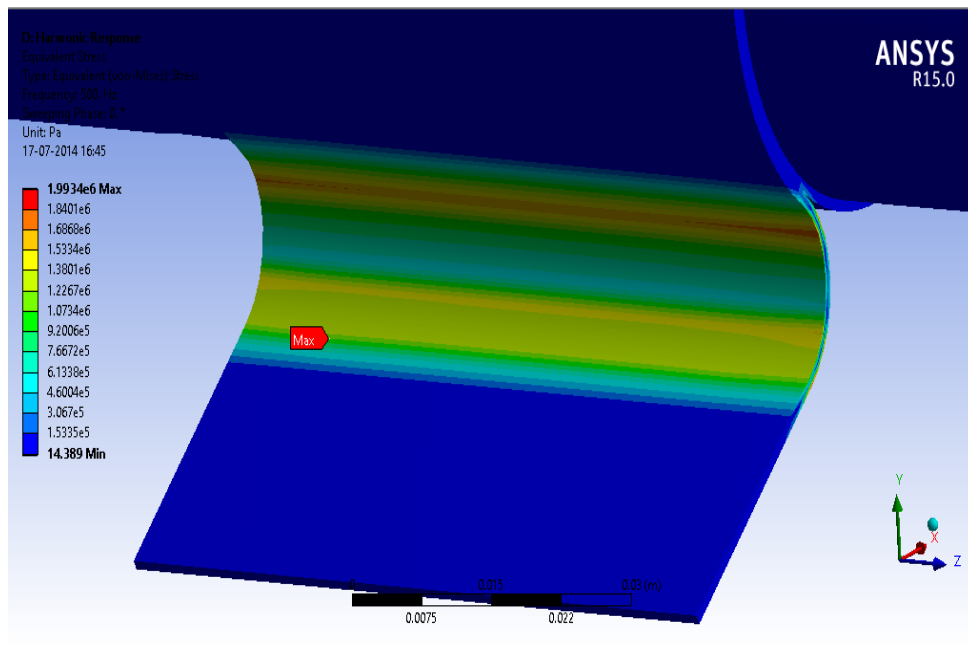


Fig 4.6 (b) Von Mises Stress in Mounting Bracket along Y-axis

c) Modal frequency response analysis along Z-axis

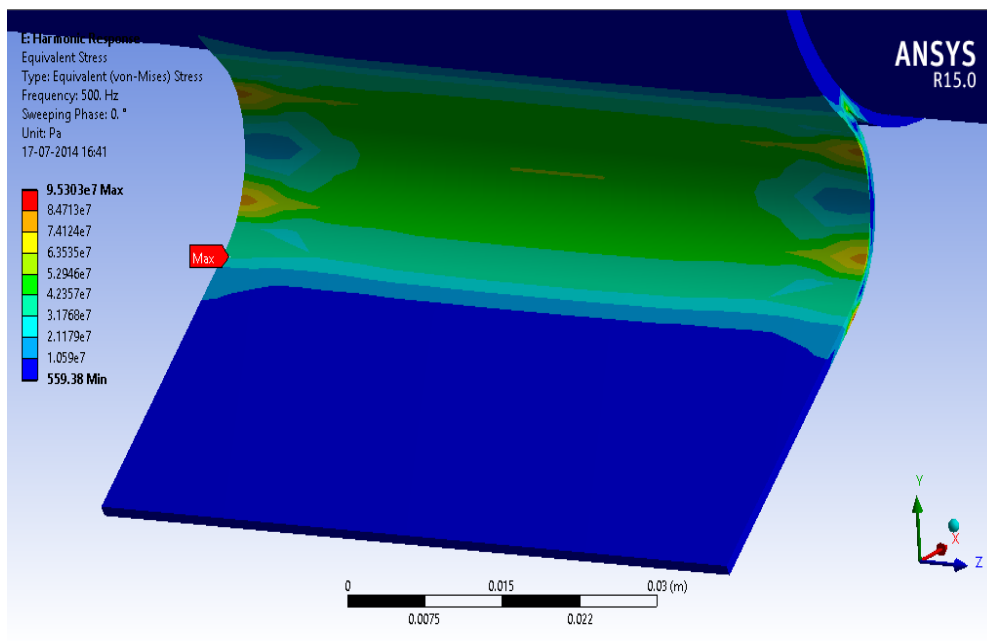


Fig 4.6 (c) Von Mises Stress in Mounting Bracket along Z-axis

4.7 Static Structural Analysis of the Radiator Mounting Brackets

Static analysis is used to determine the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects.

Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time.

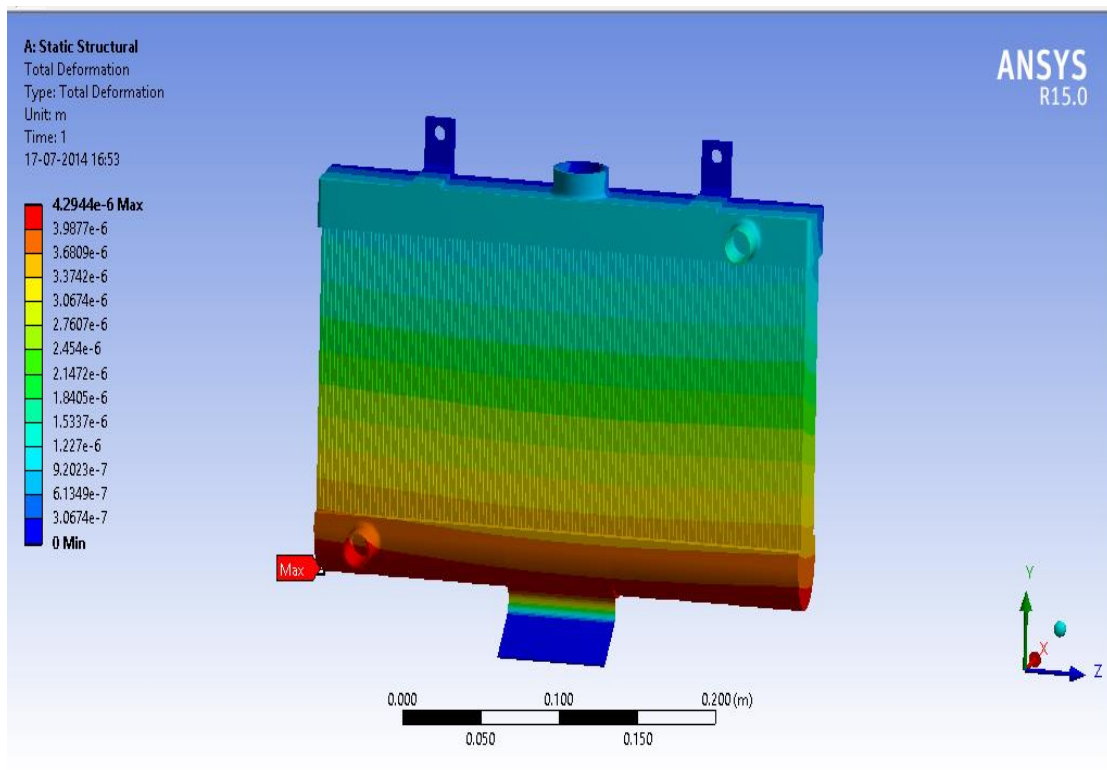


Fig 4.7 (a) Deformation in Mounting Brackets

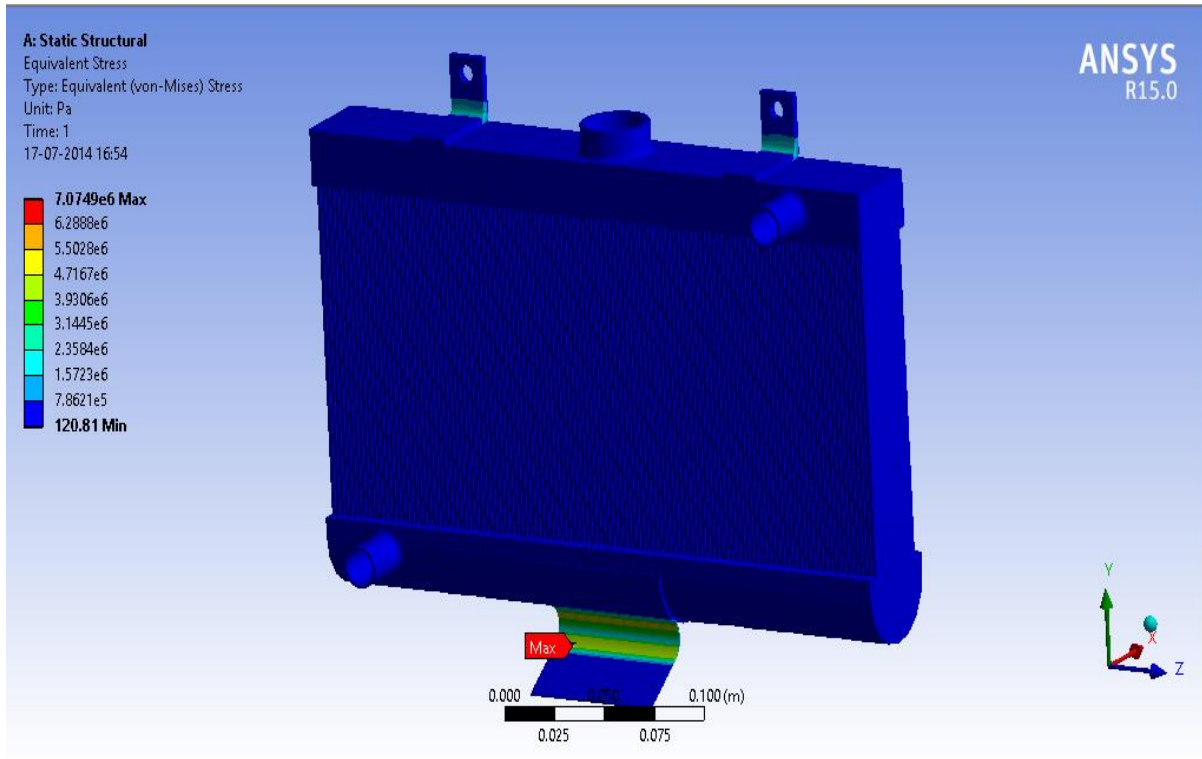


Fig 4.7 (b) Von Mises Stress in Mounting Bracket

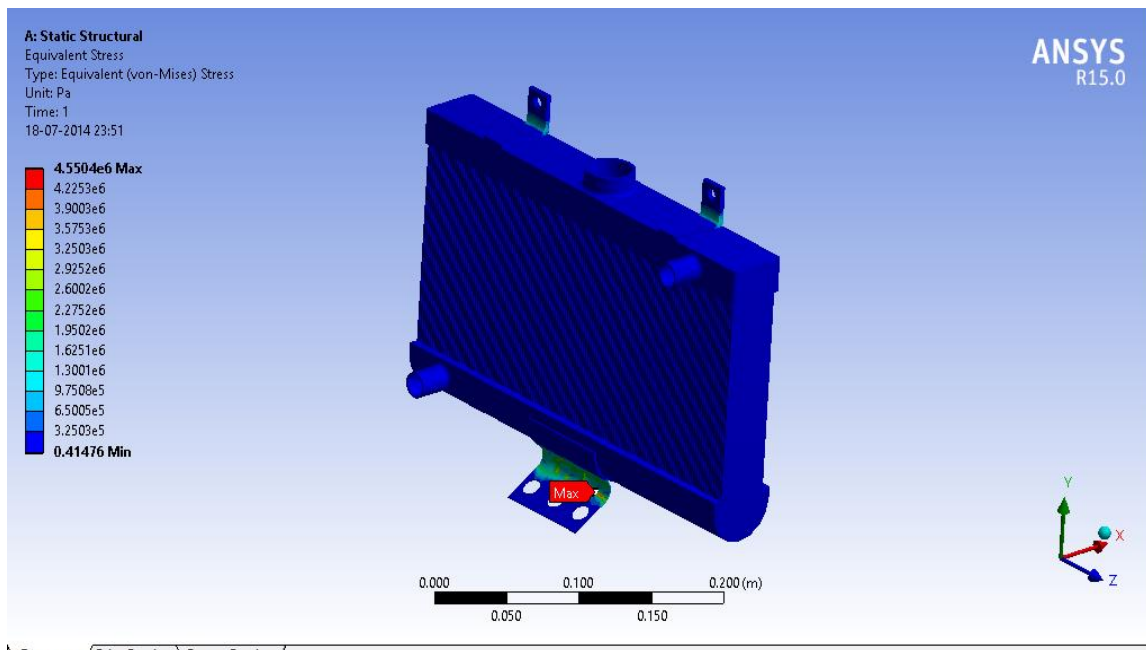


Fig 4.7(c) Von Mises Stress in Mounting Bracket

CHAPTER - 5

OPTIMIZATION OF RADIATOR MOUNTING BRACKET

The material was removed from the low stress region of the bracket. CAD model was meshed and the methodology proposed for the dynamics analysis was used on the modified bracket.

5.1 Meshing

The procedure of meshing is similar as discuss in previous chapter.

5.2 Dynamic Analysis

Dynamic analysis for the radiator mounting bracket is done which is divided in two steps:

- Normal modes analysis
- Modal frequency response analysis.

These two steps are related to each other. Output parameters of the first are used as input parameters of the second step.

5.2.1. Normal Modes Analysis of the Mounting Bracket

The procedure of normal modes analysis of the modified mounting bracket is same as explained earlier.

5.2.2 Modal Frequency Response analysis of Radiator Mounting Bracket

The procedure of modal frequency response analysis of optimize the modified mounting bracket is same as explained earlier. The results obtained are:

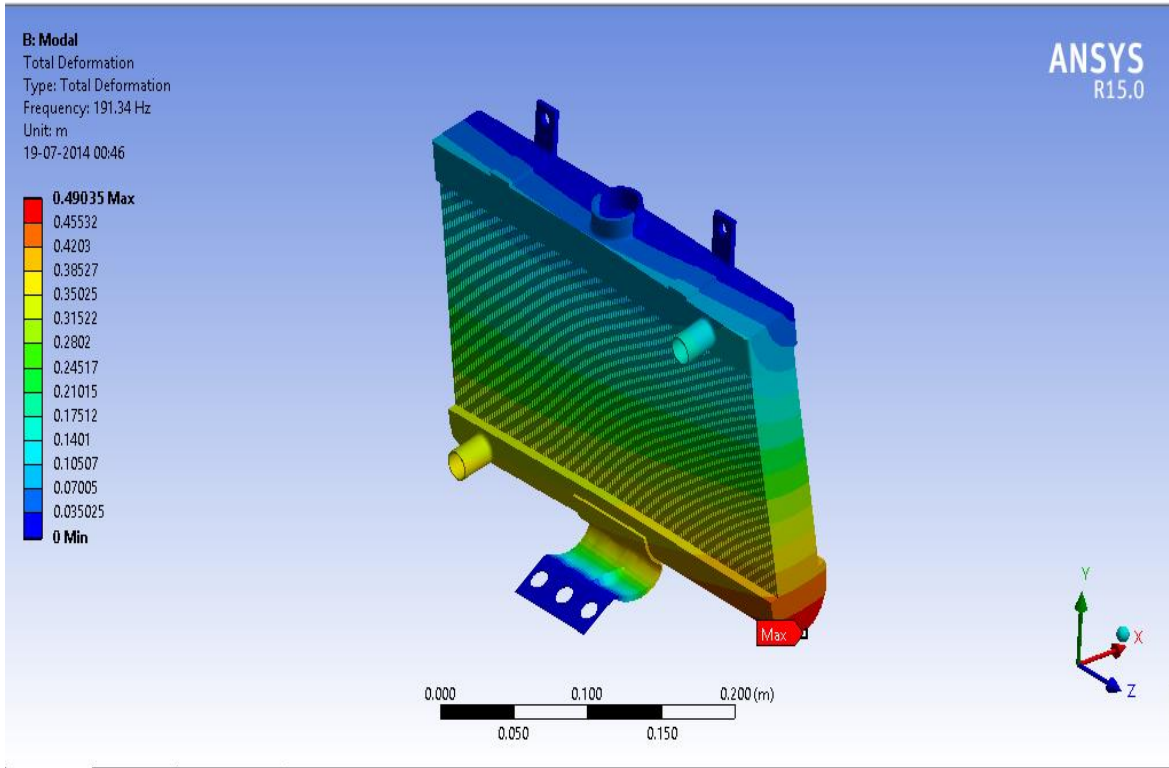


Fig 5.1: Mode 1- 191.34 Hz (Natural Frequencies & Mode Shapes)

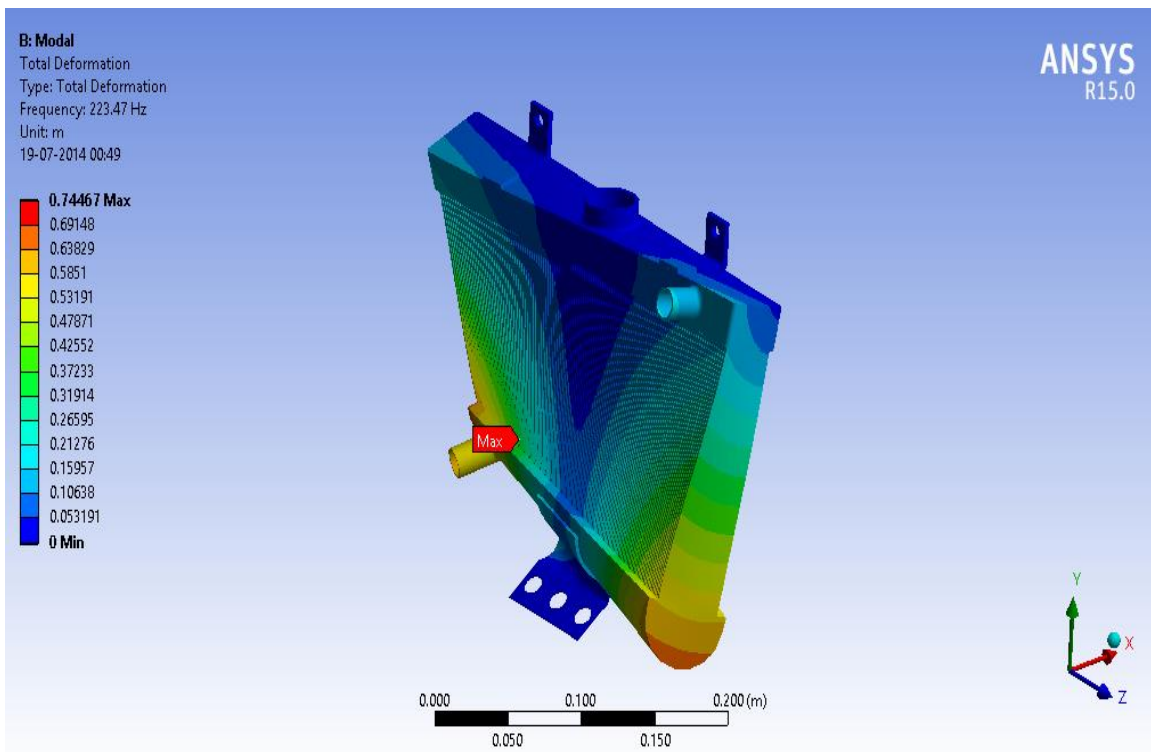


Fig 5.2: Mode 2- 223.47Hz (Natural Frequencies & Mode Shapes)

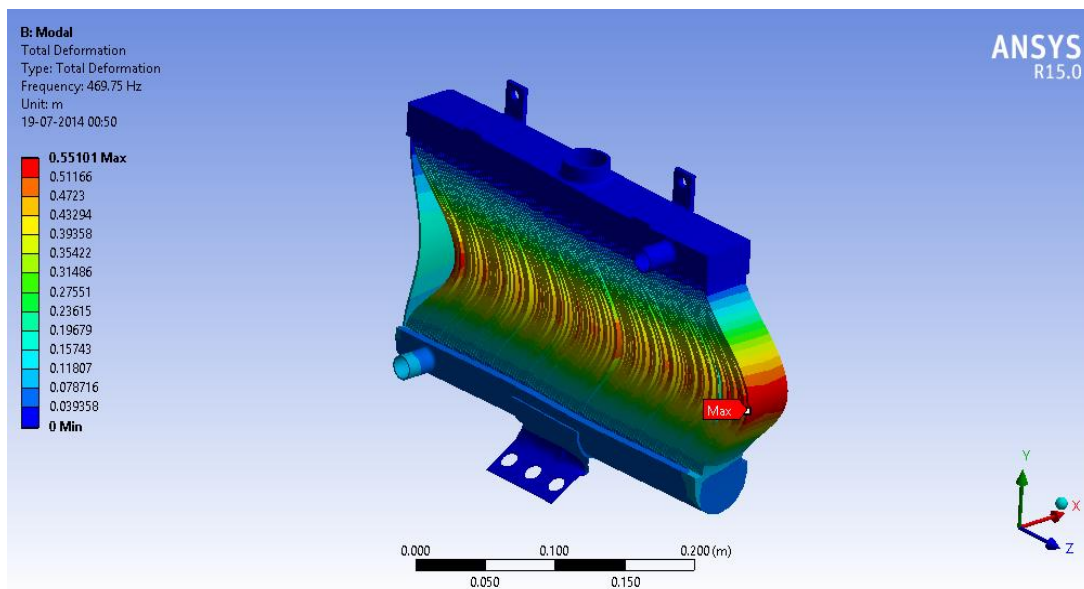


Fig 5.3: Mode 3- 469.75Hz (Natural Frequencies & Mode Shapes)

5.3 Modal frequency response analysis on the removal of material

a) Modal frequency response analysis along X-axis

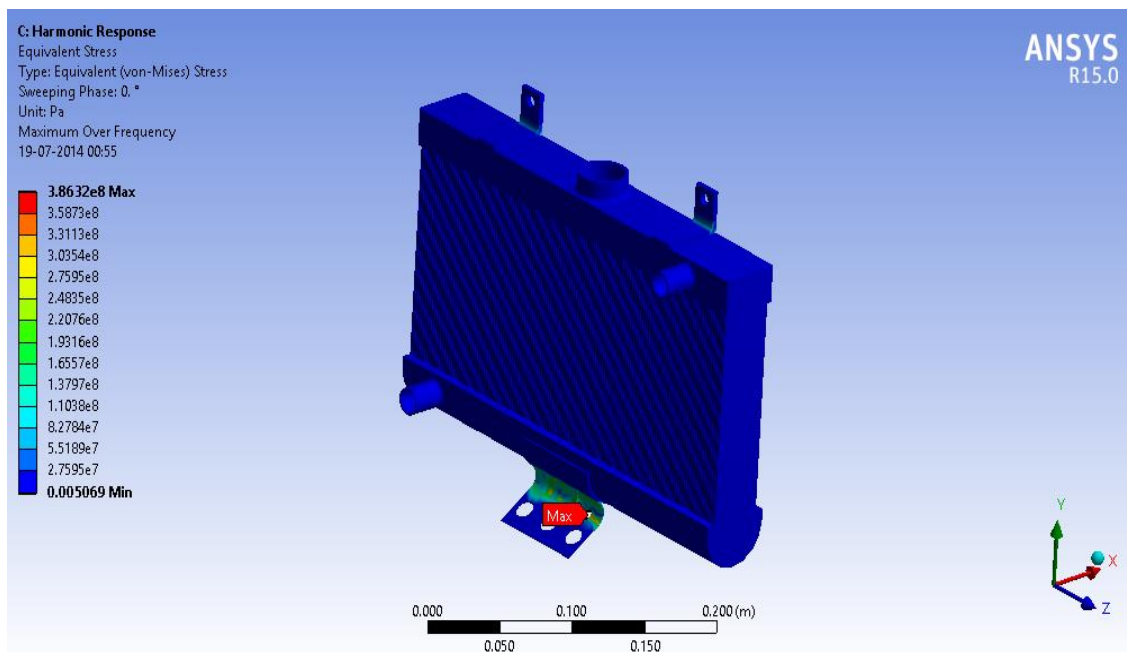


Fig 5.4: Von Mises Stress in Radiator Mounting Bracket along X-axis

b) Modal frequency response analysis along Y-axis

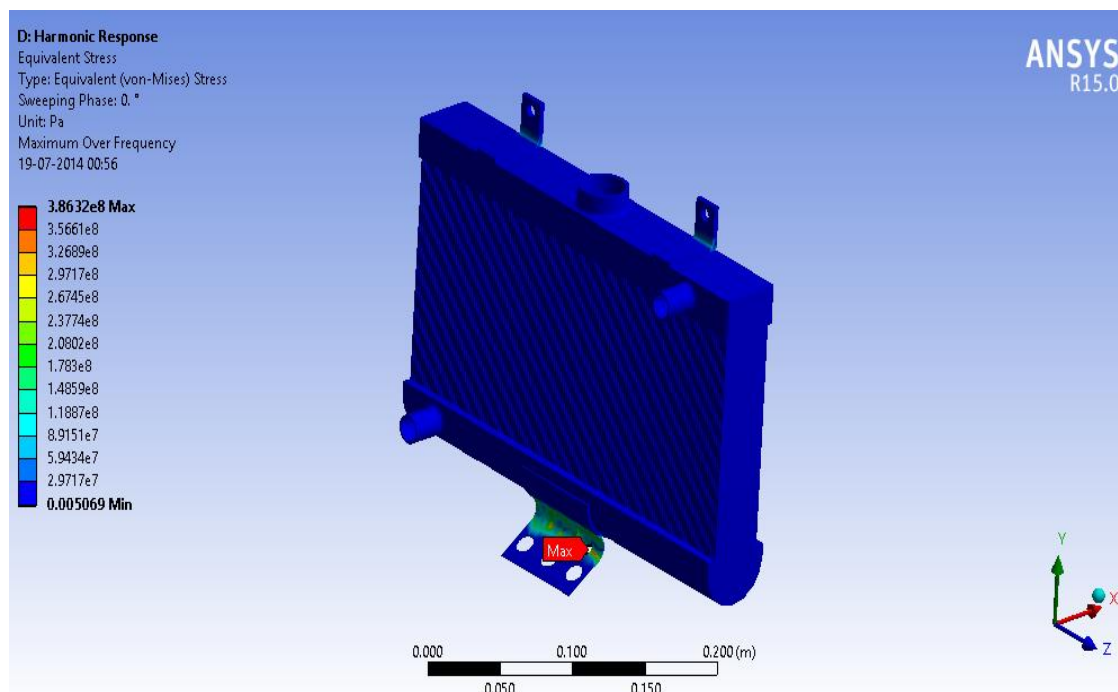


Fig 5.5: Von Mises Stress in Radiator Mounting Bracket along Y-axis

c) Modal frequency response analysis along Z-axis

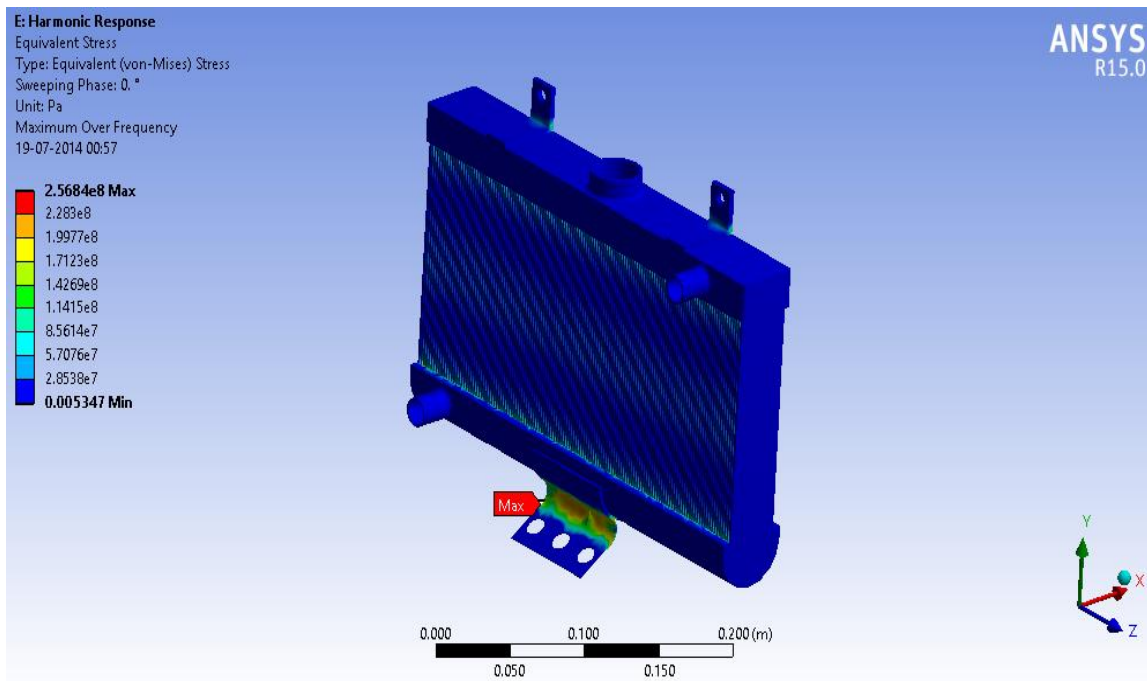


Fig 5.6: Von Mises Stress in Radiator Mounting Bracket along Z-axis

CHAPTER 6

CONCLUSION AND SCOPE FOR FUTURE WORK

6.1 CONCLUSION

Dynamic analysis (normal modes and frequency response analysis) is done on automobile radiator mounting brackets using the standard testing conditions. The results obtained revealed that the high values of stresses are produced in z and y directions on the radiator mounting bracket. It is seen that the stresses increase as the excitation with the natural frequency for the same magnitude of applied load.

The optimization is done for brackets reducing the mass of the same by about 6.4%. Thus, the use of CAE tools leads to an easy visualization and comparison of data thereby helping in the detection of problems early in the design cycle, reduced number of physical prototypes resulting in significant saving of time and cost and last but not the least, more design iterations by incorporating simulation techniques.

6.2 SCOPE FOR FURTHER WORK

The present work can be extended in the following ways:

- a. Optimization and analysis by this method can be planned for other mounting parts in automobiles.
- b. Optimization can be done using the automated CAE software.
- c. The methodology can be done using the practically validated for the value of stresses by using strain gauges, on an electro-dynamic shaker.
- d. For the radiator mounting bracket modal transient response analysis can be done to see the effect of vibration (resonance) with time.

REFERENCES

- [1] **Doo-Ho Lee, Jeong-Woo Chang and Chan-Mook Kim**, “Optimal Shape Design of an Air-Conditioner Compressor Mounting Bracket in a Passenger Car”, Vol-01, 1667–1672, 2003.
- [2] **E.S. Palma a, C.L. Petracconi b, S.E. Ferreira**, “Fatigue behaviour analysis of a rear tow hook pin of a passenger vehicle”, Engineering Failure Analysis, pp. 2408–2416, 2009.
- [3] **G. Fournalis, R. Ellwood, T.B. Jones**, “The reliability of test results from simple test samples in predicting the fatigue performance of automotive components”, Materials and Design, vol.28, pp. 1198–1210, 2006.
- [4] **Gaikwad Vikrant C., Dr. Sawant Suresh M., Mr. K. Parmeshwar**, “Structural Analysis of Engine Cooling System for Passenger Car Vehicle”, International Journal of Engineering Research & Technology, Vol. 2, 2013.
- [5] **Jeong Woo Chang and Young Shin Lee**, “Topology optimization of compressor bracket”, Journal of Mechanical Science and Technology, Vol. 22, 1668-1676, 2008.
- [6] **Pavan B. Chaudhari and Dr. D. R. Panchagade**, “Comparison of Magnesium, Aluminium and Cast Iron to obtain Optimum Frequency for Engine Bracket using Finite Element Analysis”, International Journal of Engineering Research and Applications (IJERA) , Vol. 2, 1016-1020, 2012.
- [7] **Saharash Khare, O.P. Singh, K. Bapanna Dora, C. Sasun**, “Spalling investigation of connecting rod”, Engineering Failure Analysis, vol.19, pp.77-86, 2011.
- [8] **Sahil Naghate and Sandeep Patil (2012)**, “Modal Analysis of Engine Mounting Bracket Using FEA”, International Journal of Engineering Research and Applications (IJERA), Vol. 2, 1973-1979.

- [9] **Senthilnathan Subbiah, O.P. Singh, Srikanth K. Mohan, Arockia P. Jeyaraj**, “Effect of muffler mounting bracket designs on durability”, *Engineering Failure Analysis* 18, pp. 1094–1107, 2011.
- [10] **Umesh S. Ghorpade, Prof. D.S.Chavan, Prof. M.V.Kavade**, “Static Structural And Modal Analysis Of Engine Bracket Using Finite Element Analysis”, *International Journal of Engineering Research & Technology*, Vol. 1, 2012.
- [11] **V. Veloso a, H.S. Magalhaes a, G.I. Bicalho a, E.S. Palma**, “Failure investigation and stress analysis of a longitudinal stringer of an automobile chassis”, *Engineering Failure Analysis* 16, pp. 1696–1702, 2008.
- [12] **Yasuo O., Nagashima H. and Tanahashi**, "Vehicle technology handbook tests and evaluation", Society of Automotive engineers of Japan.