DYNAMIC ANALYSIS OF COMPRESSOR MOUNTING BRACKET OF AUTOMOBILE AIR CONDITIONING SYSTEM

A dissertation report submitted in partial fulfillment of the requirement for the award of

MASTER OF ENGINEERING

IN

CAD/CAM & ROBOTICS

Submitted by

TARUNDEEP SINGH BRAR

Roll No. 801181025

Under the Guidance of

MR. DALJEET SINGH

Assistant Professor

Department Of Mechanical Engineering

Thapar University, Patiala

DEPARTMENT OF MECHANICAL ENGINEERING

THAPAR UNIVERSITY

PATIALA-147004, INDIA

July – 2013

DECLARATION

I hereby declare that the work in this thesis report entitled **"DYNAMIC ANALYSIS OF COMPRESSOR MOUNTING BRACKET OF AUTOMOBILE AIR CONDITIONING SYSTEM"** is an authentic record of my study carried out as a requirement for the award of degree of **Master of Engineering (CAD/CAM & Robotics)** at **Thapar University, Patiala** under the guidance of **Mr. DALJEET SINGH,** Assistant Professor, Mechanical Engineering Department, Thapar University, Patiala

The matter embodied in this report has not been submitted in part or full to any other university or institute for the award of any other degree.

Tarundeep Singh Brar

(Tarundeep Singh Brar)
Reg. No. 801181025

....

This is to certify that above declaration made by the student concerned is correct to the best of our knowledge and belief.

Mr. DALJEET SINGH
Assistant Professor,
Mechanical Engineering Department,
Thapar University,
Patiala-147004.

....

Countersigned by:

Dr. AJAY BATISH
Professor and Head,
Mechanical Engineering Department,
Thapar University,
Patiala-147004.

Dr. S.K. MOHAPATRA
Dean of Academic Affairs,
Thapar University,
Patiala-147004.

ACKNOWLEDGEMENT

I would like to express a deep sense of gratitude and thank profusely to my guides ***Mr. Daljeet Singh*** *for their sincere & invaluable guidance, suggestions and attitude, which inspired me to submit thesis report in the present form. Their dynamism and diligent enthusiasm have been highly instrumental in keeping my spirits high. Their flawless and forthright suggestions blended with an innate intelligent application have crowned my task with success.*

I am also thankful to ***Dr. Ajay Batish,*** *Professor and Head, Department of Mechanical Engineering for his encouragement and inspiration for execution of the thesis work.*

I am deeply indebted to my parents for their inspiration and ever encouraging moral support, which enabled me to pursue my studies.

I am also very thankful to the entire faculty and staff members of Mechanical Engineering Department for their intellectual support and cooperation.

Tarundeep Singh Brar

Reg. No.: 801181025

ABSTRACT

With the automotive air-conditioning 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 dynamics analysis technique is proposed in the simulation of the automobile compressor mounting bracket for the vibration loads. The mounting bracket has been analyzed using the standard testing conditions. The results revealed that the compressor mounting bracket 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, a new compressor mounting bracket is analyzed and optimized. Also, the importance of certain ribs or stiffness is studies using the proposed methodology.

INDEX

List of Figures

List of Tables

Chapter 1

INTRODUCTION

1.1 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.

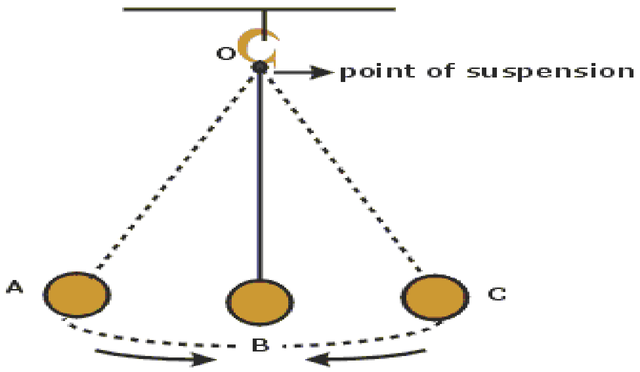


Fig – 1.1 Swinging of simple pendulum

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.

1.1.1 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 then 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 upto 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 red/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.

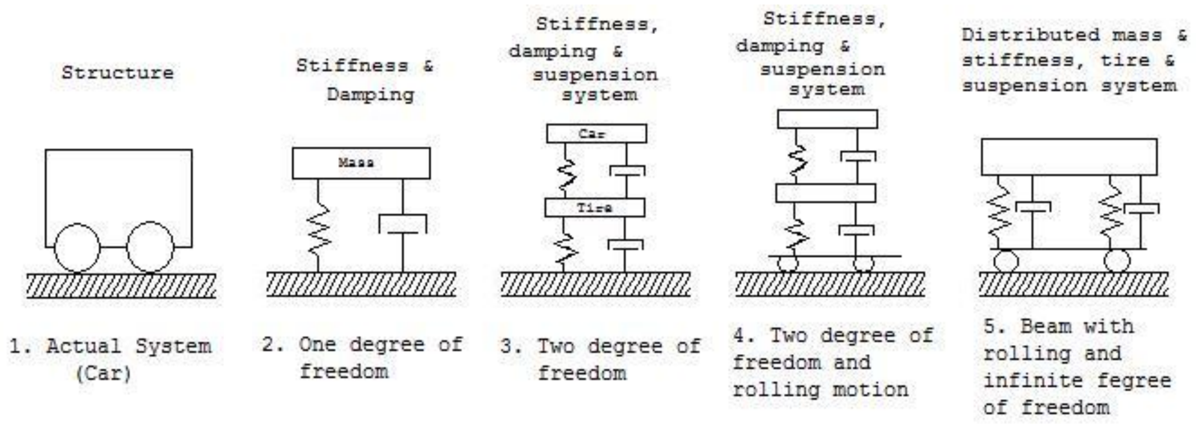


Fig 1.2 Degrees of freedom

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 x1 and x2 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.

1.1.2 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 (1/2 mx^2), in the spring in the form of potential energy (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 Fig 1.3

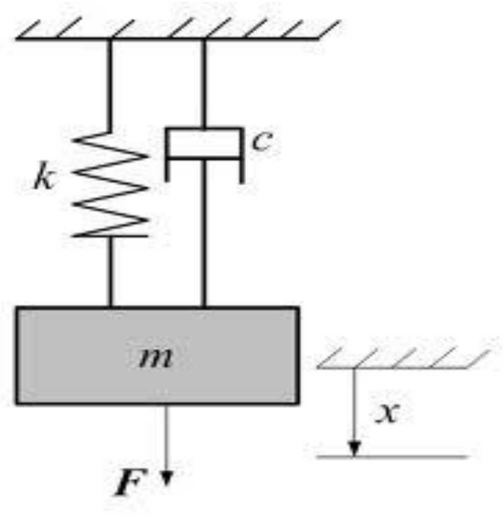


Fig 1.3 Parts of vibration

1.1.3 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".

1.2 Methods 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-mate 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.

1.2.1 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 • Finite Element Method • Boundary Element Method • Finite Volume Method • Finite Difference Method	• Strain gauge • Photo elasticity • Sensors for temperature & pressure • Fatigue test etc.

Table – 1.1: Difference between Analytical, Numerical and Experimental methods.

1.2.2 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 nodded 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 softwares 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.

Steps in Processing:

a) Compute the element stiffness matrix

After the continuum is discretize with the discrete element shapes and the number, and then the element stiffness matrix is formulated. Basically it is matrix formed by using the governing equations which tells how a parameter varies in the matrix. It is a square matrix with its size depending upon the degree of freedom of each node i.e.

[No. of columns] = [No. of rows] = No. of nodes in an element × Degree of freedom.

There are 3 methods for deriving Stiffness Matrix:

1) ***Direct method*** – Easy to understand but difficult to program. It is not used for commercial software code generation.
2) ***Variational method*** – Rayleigh – Ritz method is used, difficult to understand; moderate from code writing point of view.
3) ***Weighted Residual method*** – Galerkin method: difficult to understand but easy from programming point of view. This method is used in most of the commercial software's.

b) Compute the overall stiffness matrix

Previous step we have found the algebraic equation that give the characteristic of the element now all those algebraic equation are combined together to form a complete set of equation that govern the domain or structure. All the element stiffness matrices generated in last step are combined to form a overall stiffness matrix. It is always a symmetric square matrix with size as

[No. of columns] = [No. of rows] = Total No. of nodes in the body × Degree of freedom.

c) Formation of Element load matrix

Load applied on the body is very important parameter in any problem load applied inside the element is transferred at the node and an element load matrix is formed, it is a column matrix with *[No. of rows] = No. of nodes in an element × Degree of freedom.*

d) Formation of the overall load matrix

Like the overall stiffness matrix the element load matrix is assembled to form the overall load matrix. It is a column matrix with

[No. of rows] = (Total No. of nodes in the body × Degree of the freedom).

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

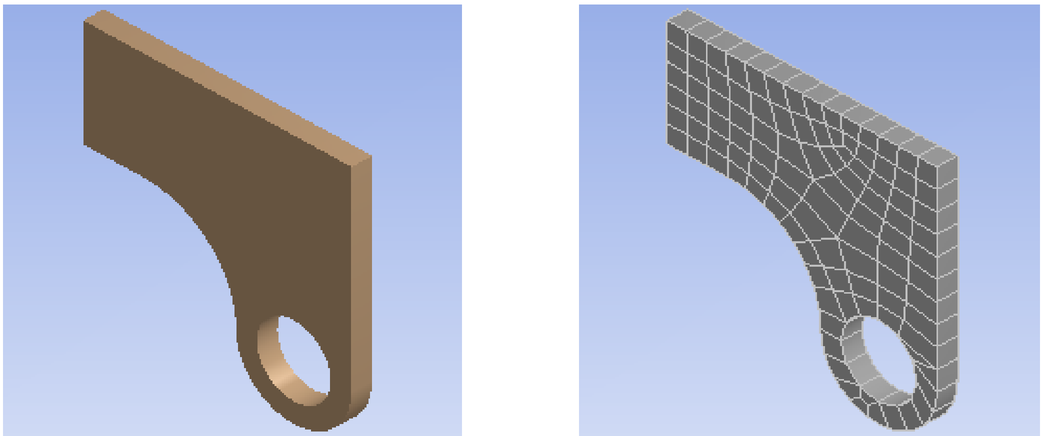


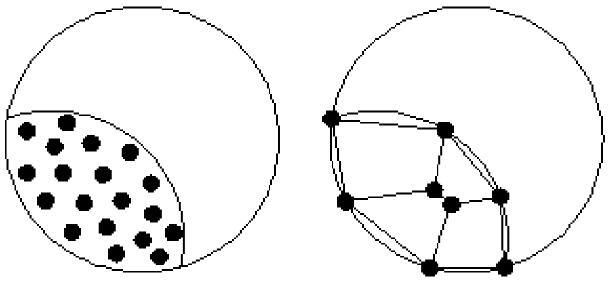
Fig. – 1.4 Discretization Process

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 safe and accuracy increases. An example of discretization process is shown in Fig. – 1.5



No. of Points = ∞	*No. of Nodes = 8*
DOF per Point = 6	*DOF per Point = 6*
Total equations = ∞	*Total equations = 48*

Fig - 1.5 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.

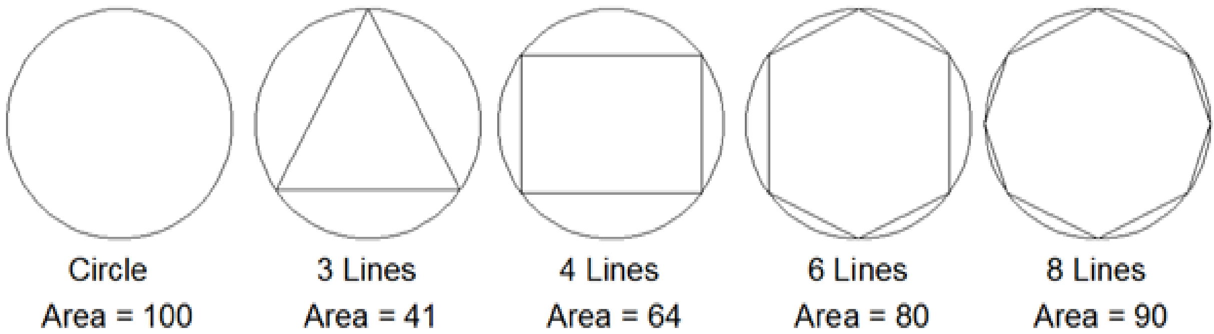


Fig. – 1.6 Area of circle by different nodes elements.

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.*

Fig – 1.7 (a) One-Dimensional Elements

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

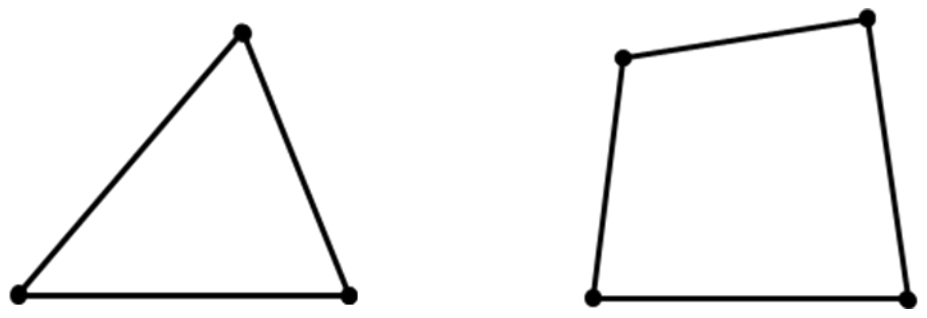


Fig – 1.7 (b) Two -Dimensional Elements

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

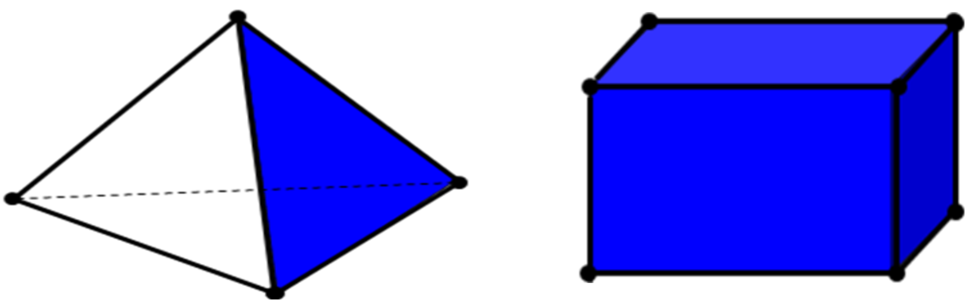


Fig - 1.7 (c) Three-Dimensional Elements

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. **σ = εE** 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, ∑ ***Force = 0,*** ∑ ***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, Anys, 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

1.3.7 FEA Software packages

Some of the commercially available and popular FEA packages are:

1. Ansys
2. Abacus
3. Nastran
4. I-deas
5. Radioss
6. Hyper Mesh
7. Patran etc.

Chapter 2

LITERATURE REVIEW

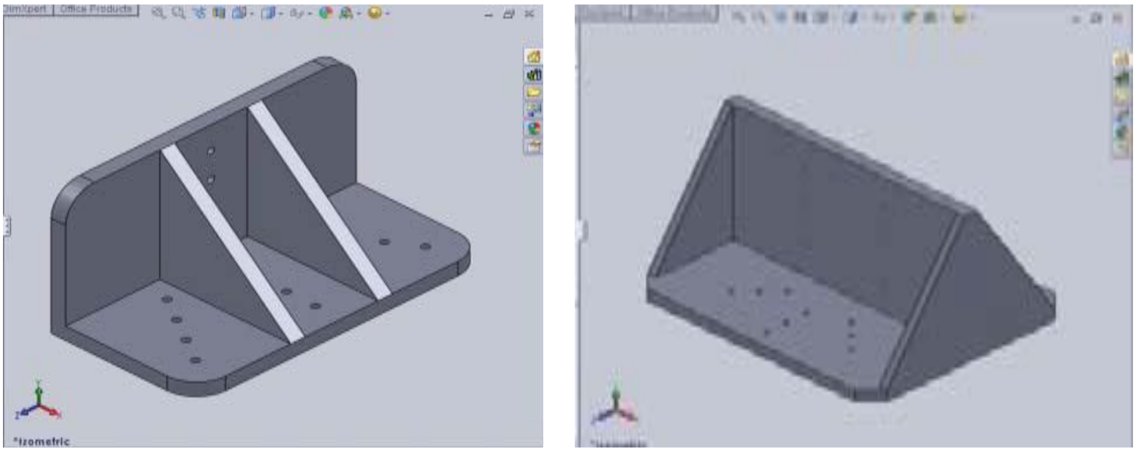
2.1 Literature Review

The compressor plays a very important role in the automotive air conditioning system. The unbalanced forces produced from the engine and compressor causes the structure vibrations. To reduce these vibratory forces, the compressor is supported by the engine mounting called compressor mounting bracket. A review of the literature related to the design and analysis with a focus on vibration analysis of these mounting brackets is presented here.

G. Phani Sowjanya *et al* [1] performed a study on vibration parameters to test the avionic equipment. Vibration is the most important mode of failure in avionic equipment. The avionic equipment fitted into the aircraft has to withstand high vibrations.

As vibration equipment cannot be placed directly on the shaker table, a mechanic at structure called fixture was placed between the equipment and the machine. The working conditions of the aircraft as per military standards range from 20Hz to 2000Hz frequency levels. The fixture was designed such that its natural frequency does not fall under the frequency range of 20 – 2000Hz. If the natural frequency of the fixture lies in the test range, resonance occurs resulting in amplification of the input, which was undesirable. To ensure proper transmission of input to the avionic equipment, an effective design of the fixture was required. After selecting the suitable design of vibration fixture as per the requirement, it was analyzed by using finite element analysis.

For designing the vibration fixture various sizes and shapes were considered. Geometrical configuration of different vibration fixtures is shown in Fig 2.1 (a) & (b).



a.) L-shape fixture *b.) Inverted T-shaped fixture*

Fig. 2.1 (a) & (b): Geometrical configuration of vibration fixture

Modal analyses of both fixtures were done, with material aluminum alloy and magnesium alloy. The natural frequencies of L shaped fixture were greater than the natural frequencies of inverted T shaped fixture. Thus the range of natural frequencies obtained was beyond the requirement of 2000Hz as required.

Nouby M. Ghazaly *et al* [2], was concerned with the FEA and modal testing of a commercial disc brake assembly. The goal was to study squeal noise prediction at early stage of design development using a more realistic FE model. The FE model was corrected using modal testing data. The modal testing was divided into two stages. The first stage was to obtain dynamic characteristics of the individual disc brake components with free-free boundary conditions. The second stage was to perform dynamic characteristics of the complete assembly. The results showed that good agreement between the FE model using solid elements and measured natural frequencies. The simulation result showed that with an increase in the friction coefficient, there was an accompanying increase in the instability of the system. Also the friction coefficient had been investigated to assist the understanding of the mode-coupling mechanism which leads to identify brake squeal occurrence.

Pavan B. Chaudhari *et al.* [3], 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.

Sahil Naghate *et al.* [4] 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.

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.

Asheber Techane [6] conducted the dynamic analysis of a bus body structure. The solid model of the structure was developed in Solid Works and the generation of the FE model and the dynamic analysis were performed using ANSYS. Modal analysis which reveals the natural behavior of the structure and PSD (power spectral density) analysis are included in the dynamic analysis of the structure and it was concluded that the roof of the bus structure was prone to higher deflection, and as a recommendation it was forwarded that there be carried out complete evaluation of the vehicle to prove robustness of the structure in terms of its mechanical properties.

Jeong Kim *et al* [7], performed a study to considered four methods of modelling a bolted joint in finite element form. These included a solid bolt preloaded thermally, a beam element coupled to nodes on the gripped bodies preloaded by an initial strain on the beam element, a beam element connected to the gripped bodies with 3D element spiders also preloaded by an initial strain on the beam element, and finally a preload pressure applied directly to the contacted bodies with no bolt represented. It was found that the solid bolt model gave the best correlation to experimental results. However, the coupled bolt and spider bolt model significantly improved the computational efficiency of the model.

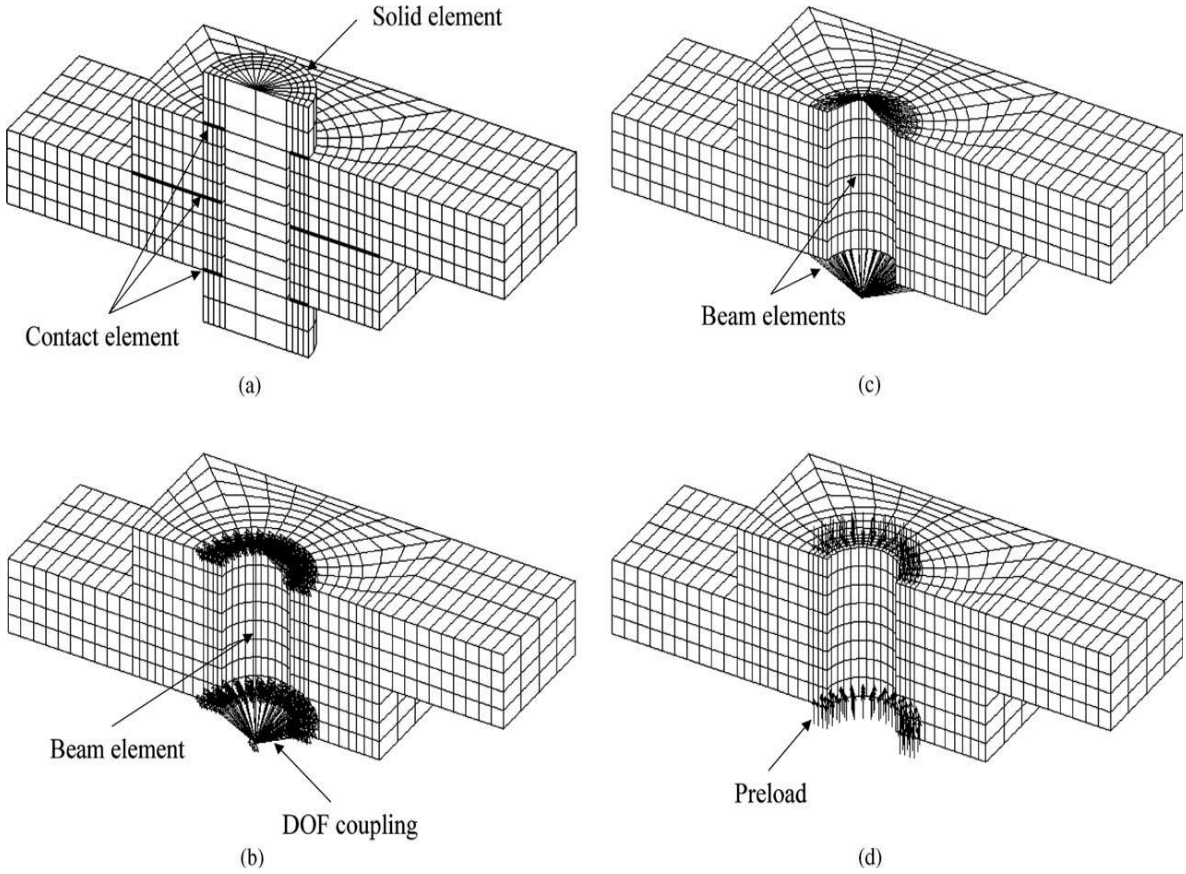


Fig 2.2 Finite element models for the structure with a bolted joint. (a) Solid bolt model, (b) Coupled bolt model, (c) Spider bolt model and (d) No-bolt model

Doo-Ho Lee *et al.* [8], 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 was 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.

Sheldon Imaoka [9] provides some information on differences between the two automated methods of generating constraint equations. CERIG and RBE3 provide useful tools in automatically generating complex constraint equations for a variety of purposes such as tying together dissimilar meshes or distributing loads through bolt connection. It concluded that the constraint equations in ANSYS are linear, so these are not valid for large-rotation analysis. Hence the use of the beta element, RIGID184, is useful for generating rigid connections/regions in nonlinear problems, valid for large rotation problems.

Erke Wang *et al* [10] presents some analytical results and some test results for different mechanical problems, which were then simulated using finite element analysis with tetrahedral and hexahedral shaped elements. The comparison was done for linear static problems, modal analysis and nonlinear analysis. According to results, it concluded that quadratic tetrahedron element are good and can always be used over hexahedral elements.

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 engine of an automobile, by taking the case of a compressor 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 mounting bracket by using the proposed dynamic analysis procedure will be done.

3.3 Methodology

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

1. CAD modeling of the bracket, compressor and fixture in CAD software.
2. 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).
3. Solution for normal modes and frequency response analysis.
4. Post-processing – viewing displacements, stresses and interpreting the results.
5. For optimization, remove the material from mounting bracket where stresses are negligible.

CHAPTER 4

ANALYSIS OF COMPRESSOR MOUNTING BRACKET

4.1 Simulation of Compressor Mounting Bracket

This section discusses the methodology of analysis of the compressor mounting bracket. Following steps were performed for the simulation of the bracket:

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

4.2 Reverse Engineering

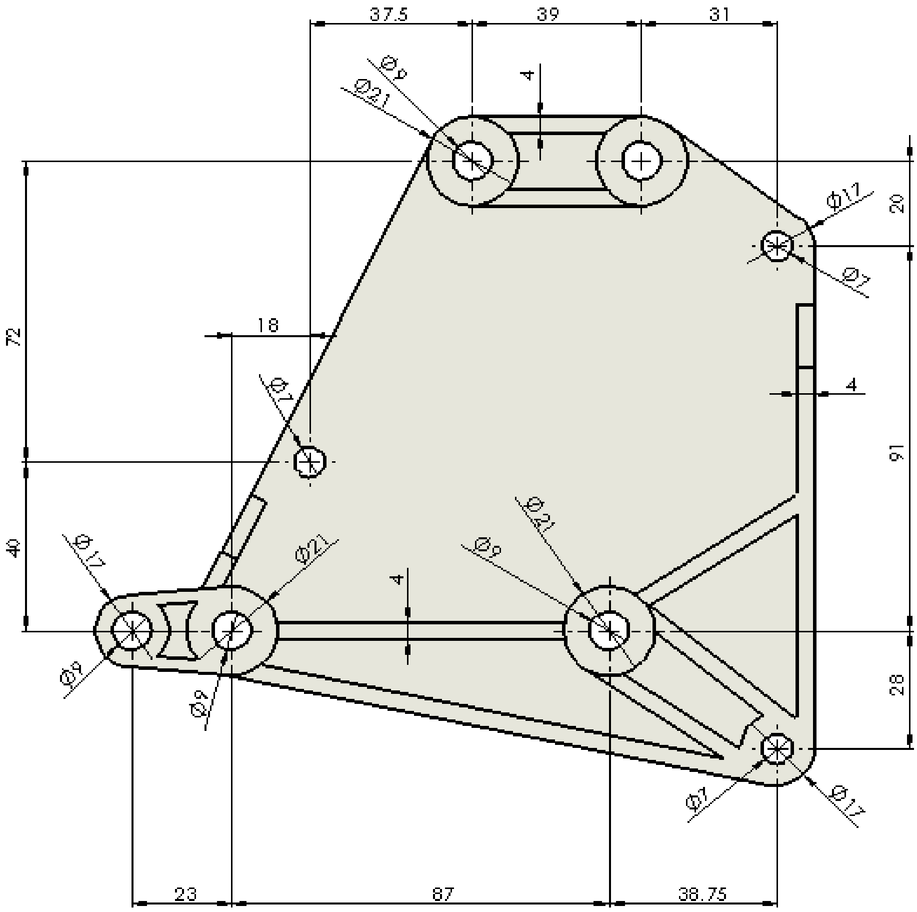
As computer-aided design (CAD) has become more popular, reverse engineering has become a viable method to create a 3D virtual model of an existing physical part for use in 3D CAD, CAM, CAE or other software. The reverse-engineering process involves measuring an object and then reconstructing it as a 3D model. The physical object can be measured by using:

- **3D scanning technologies** like Coordinate-measuring machine (CMMs), laser scanners, structured light digitizers, or industrial CT Scanning (computed tomography).
- **Manual measuring devices** such as flat surface plate or table, level meter, scriber, vernier caliper, square, divider, scale, depth gauge etc.

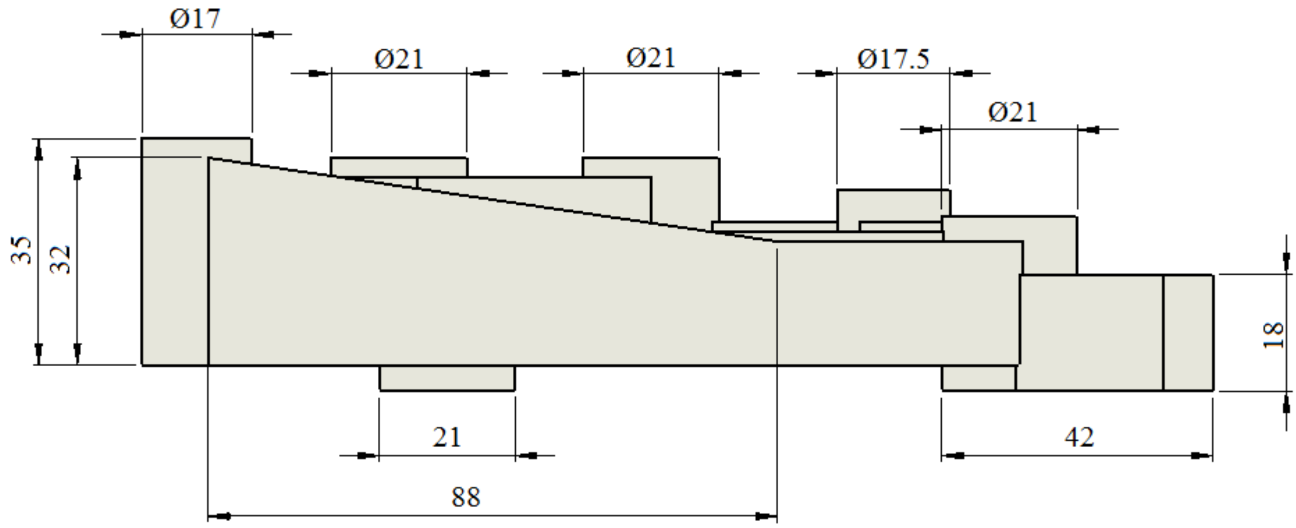
Reverse engineering of the compressor mounting bracket, shown in Fig. 4.1 (a) & (b), is done by using manual measuring devices. The detailed drawings are shown in the Fig. 4.2.

a.) Front View b.) Rear View

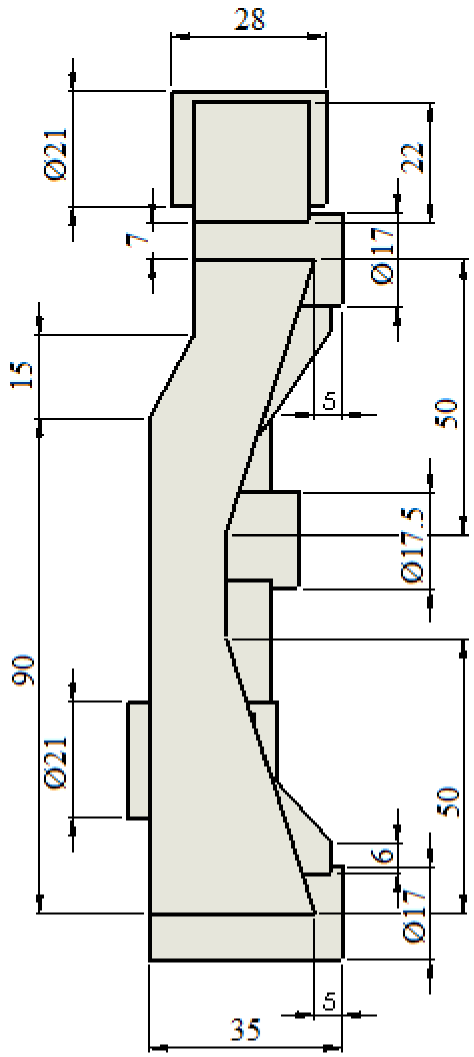
Fig 4.1 (a) & (b): Compressor mounting bracket



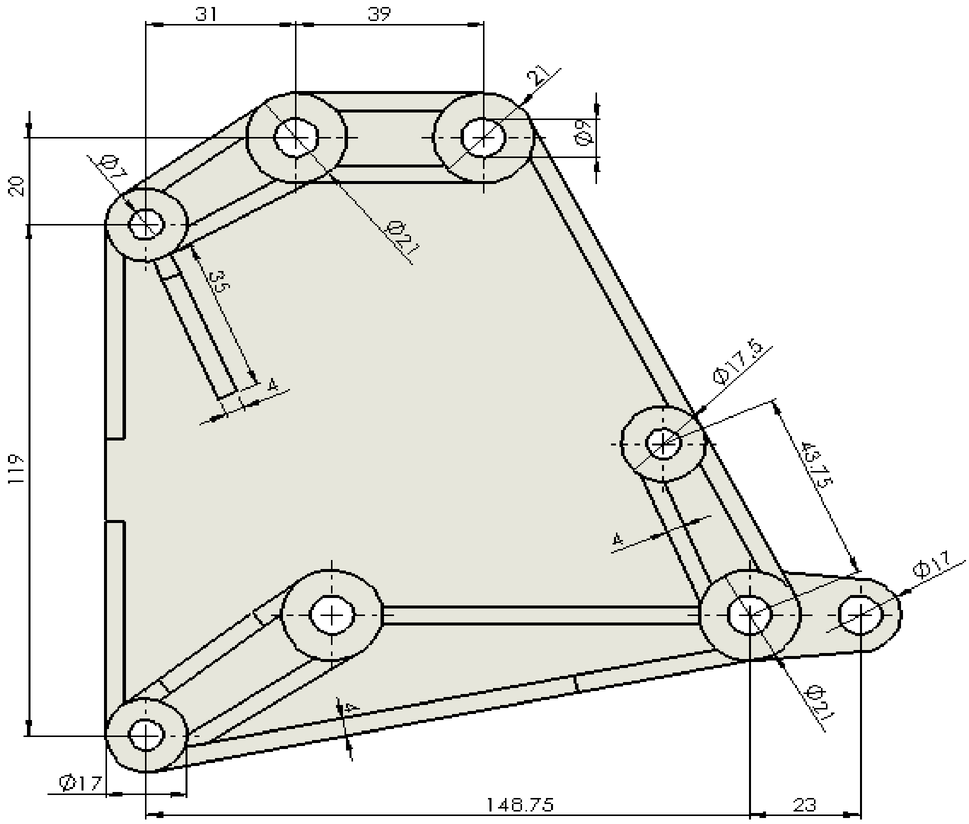
a) Front View



b) Top View



c)Right Side View



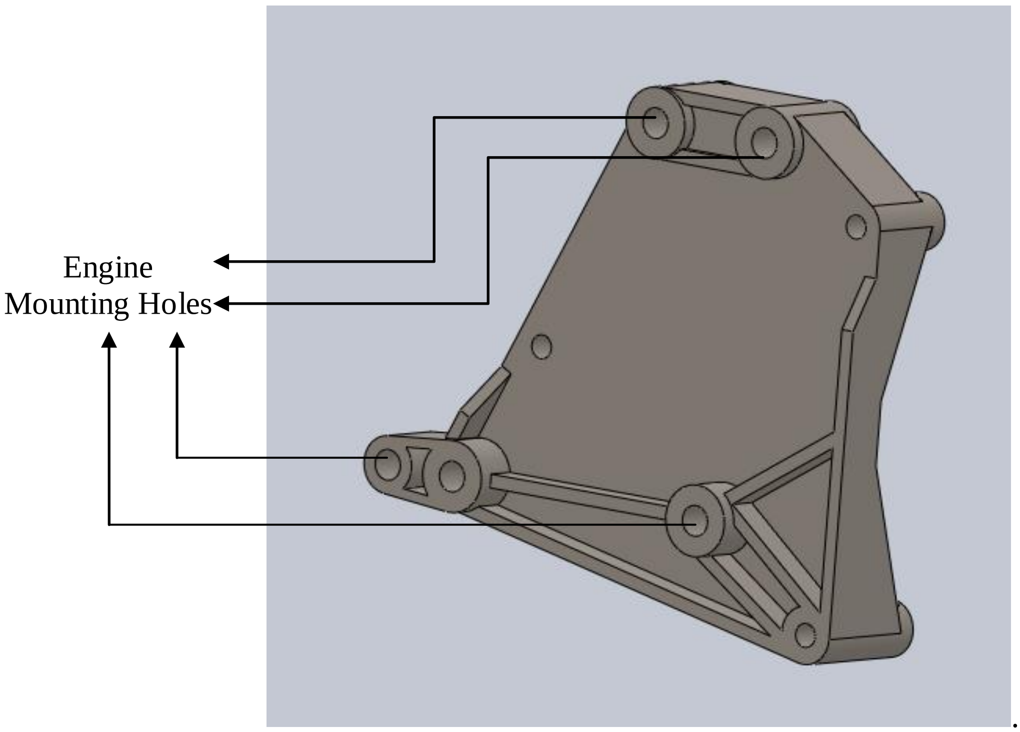
d) Rear View

Fig. 4.2 (a), (b), (c) & (d): Detailed Drawings of the Compressor Mounting Bracket

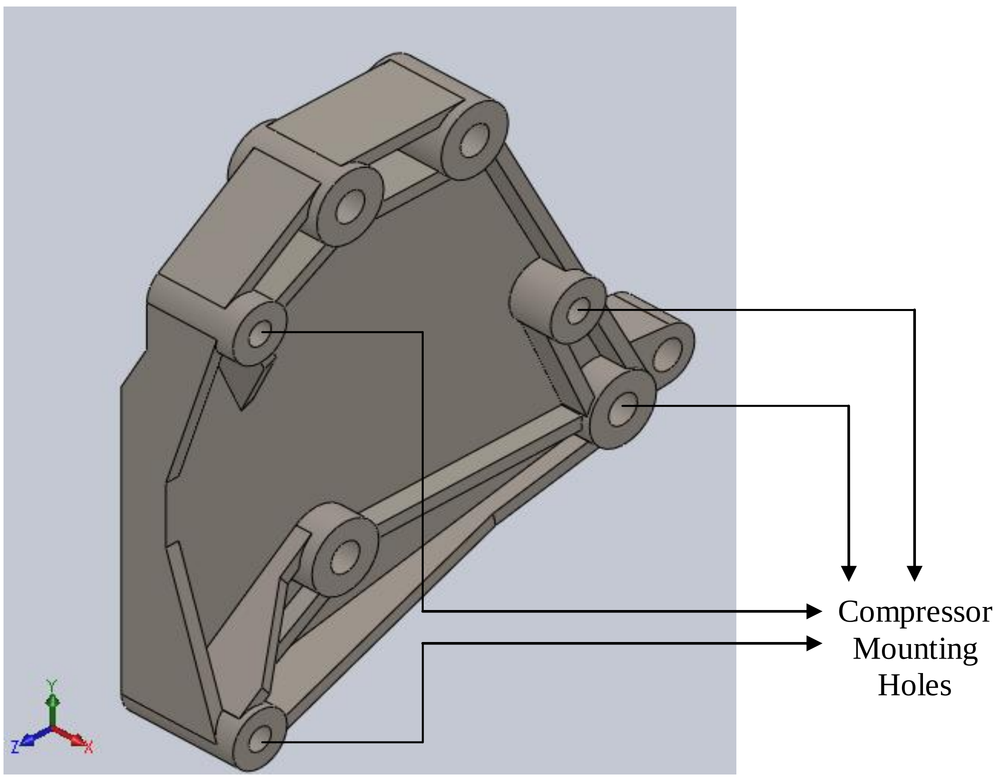
CAD Model

From the reverse engineered data, the component is modeled in Solid Works software. The generated CAD model is shown in Fig. 4.3

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.)



a.) Engine mounting location



b.) Compressor mounting location

Fig 4.3: CAD model of mounting 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 compressor mounting bracket was done according to the geometry.

Quality Checks for Tetra Meshing

The ideal shape for a tetrahedron element is an equilateral tetrahedron (all equilateral triangle faces). Following are the various quality parameters:

- **Tetra Collapse:**

  Defined as the distance of a node from the opposite face divided by the area of the face multiplied by 1.24 as shown in Fig. 4.4

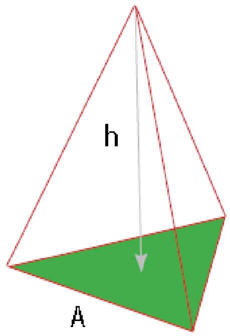


Fig. 4.4: Tetra collapse

Ideal Value = 1.0 (Acceptable > 0.1)

*Tetra collapse = h * 1.24 / A*

- **Skewness :**

  Skew in tetra is calculated by finding the minimum angle between the vector from each node to the opposing mid-side and the vector between the two adjacent mid-sides at each node of the element. Ninety degrees minus the minimum angle found is reported as shown in Fig. 4.5

  Triangle skew = 90 – α

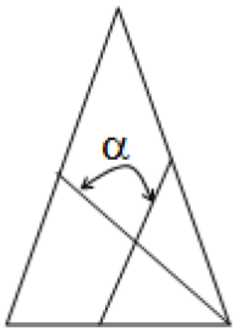


Fig. 4.5 Skewness

- **Volumetric Skewness**

  Create a sphere passing through the corner nodes of the tetra; fit an ideal (equilateral) tetra in it. Find the volume of the ideal and actual tetra elements.

  Ideal value = 0 (Acceptable < 0.7)

$$Volumetric\ Skew = \left[\frac{(V_{ideal} - V_{actual})}{V_{ideal}}\right]$$

- **Aspect ratio:**

  It is a ratio of the longest edge to the shortest normal dropped from a vertex to the opposite face, normalized with respect to the shortest normal dropped from a vertex to the opposite face of a perfect tetrahedral element. The aspect ratio of an ideal tetrahedral element is 1.0.

- **Jacobian:**

  In simple terms, the jacobian is a scale factor arising because of the transformation of the coordinate system. Elements are transformed from global coordinates to local coordinates to reduce the solution time.

  Ideal value = 1.0 (Acceptable > 0.5)

Tetrahedral elements of the component are organized in the ‘component collector’. From this component collector, size of tetra element can be controlled. The property and loads are assigned to the elements. Tetrahedral element meshing of the bracket is shown in Fig 4.6.

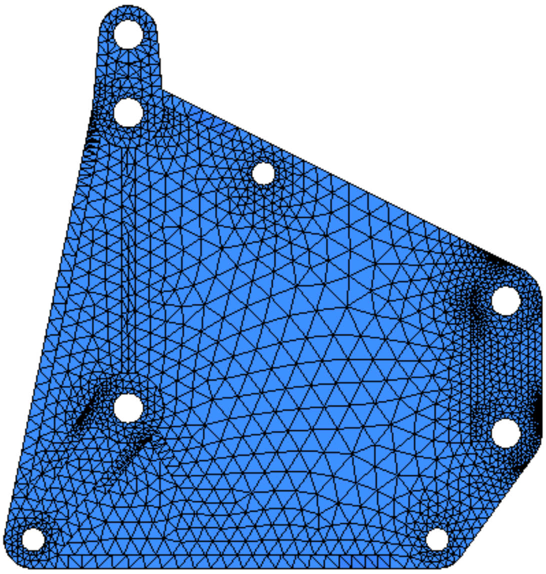


Fig. 4.6: Tetrahedral meshing of Bracket

4. 4 Dynamic Analysis

Dynamic analysis for the compressor 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**
  1) It is the basic design property. For specific components like engine mounting bracket or chassis mounting bracket, it is one of the important design approval criteria.
  2) 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.
  3) To control the noise and the vibration in components.

- **Conditions for the normal mode Analysis**

  Following conditions are applied for the normal modes analysis:

  1) ***No external force:*** As per the requirement of natural frequency external forces is not applied.
  2) ***Constraints:*** Natural frequency analysis is carried out as per actual constraint.
  3) ***Damping:*** It is neglected for the natural frequency calculations.
  4) ***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 this section.

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

1) **Defining the Material**: To define a specific material, the material Collectors are used in the software. The material used is “aluminium” with the following properties: linear elastic, isotropic and temperature independent. The value of different properties are:-
   - Modulus of elasticity, E = 6.6 e +04, (N/m^2)
   - Modulus of rigidity, G = 2.7 e +04, (N/mm^2)
   - Poisson ratio, ν (Nu) = 0.3,
   - Density, ρ = 2.7 e -09, (kg/m^3)

2) **Property (collector):** These collectors contained the properties that are assigned to the components. It assigns card image and material.
   - ***Card Image*** - PSOLID to assign solid elements for 3D meshing
   - ***Material*** – assign aluminium (Al) from material collector

3) **Rigid body element (RBE):** Rigid body element can be used on bolt locations to make a rigid region. It does not require a solid bolt, it use a line element and RBE elements to represent the stud and bolt respectively, as shown in Fig. 4.7. Tensile, bending, and

thermal loads can be transferred through the RBE nodes.

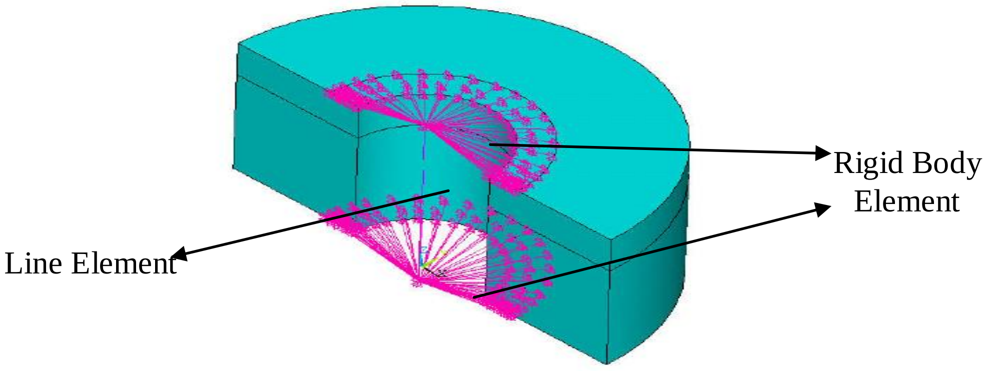


Fig. 4.7 Rigid Body Element

- ***RBE2*** – is creates a rigid region by generating constraint equations to relate nodes in the region. Nodes in the rigid region must be assigned a geometric location. It is used to translate only degree of freedom at the dependent nodes. It requires two type of nodes:

  Master:

  A node which represents the fixture is a "master node". This node is an independent node and was created at the fixture mounting position as shown in Fig. 4.7 (a).

  Slave:

  All the other selected nodes are on mounting bracket (at bolt location) represent the "slave nodes" as shown in Fig 4.7 (a). Slave nodes are dependent nodes.

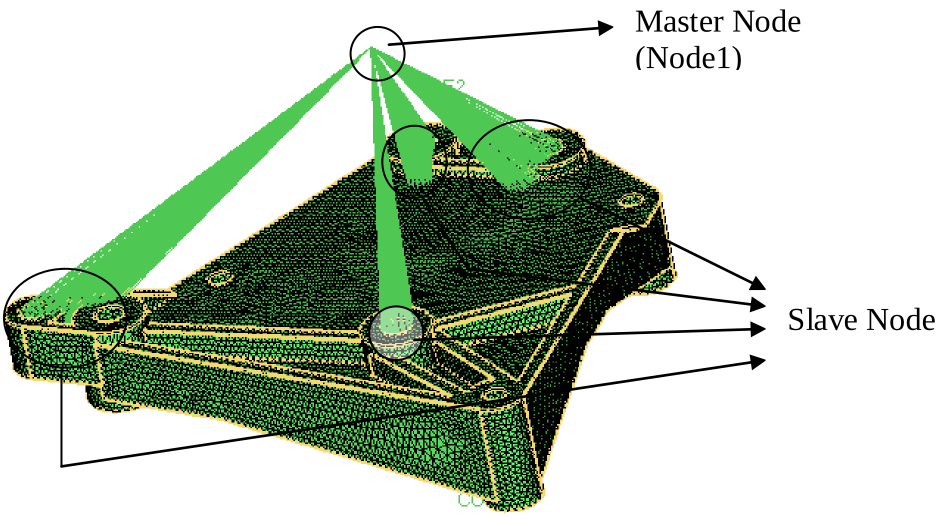


Fig 4.7 (a) – Rigid Body Element (RBE2)

- ***RBE3*** – element also creates a rigid region. In RBE3, master node is dependent node and slave nodes are independent nodes. It represents the mass of compressor at master node, which is created at the centre of gravity of compressor.

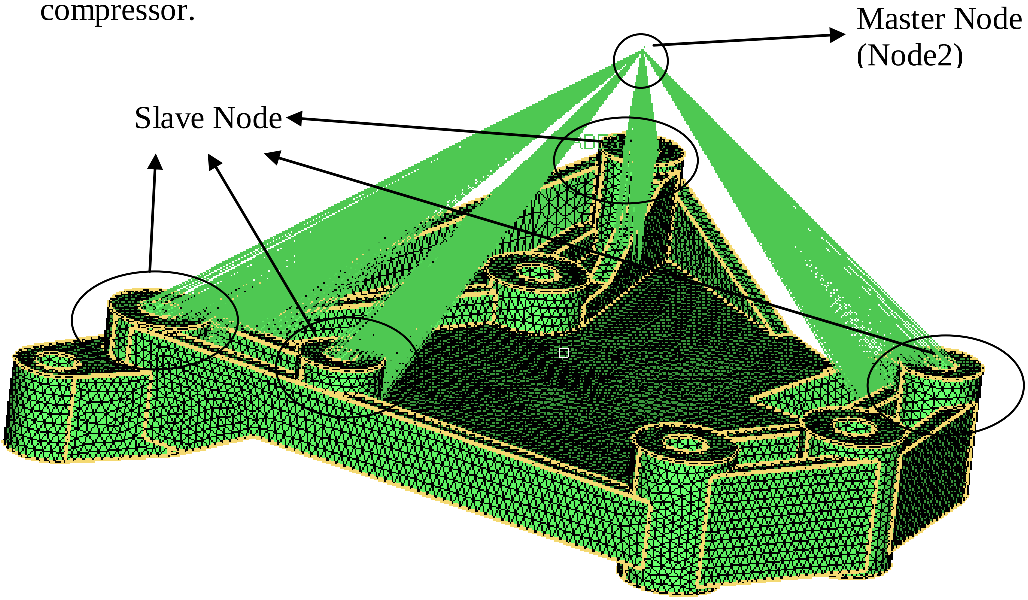


Fig 4.7 (b) – Rigid Body Element (RBE3)

The RBE3 is used to distribute the effect of a mass (acting at the dependent node) over a number of other nodes (the independent nodes), without adding stiffness to the structure.

4) **Load Collectors**: Two load collectors are used in the normal mode analysis. In the load collector a load and constraints can be applied on the bracket of the compressor assembly.

a) Eigrl (load collector): EIGRL card image defines the data that is needed to perform real eigen value analysis (vibration or buckling) with the Lanczos Method. EIGRL is used for the calculation of the number of modes and natural frequencies.

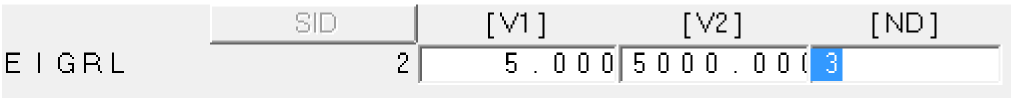


Fig. 4.8 EIGRL card

Where,

V1 & V2 = frequency range (lowest and highest frequency)

ND = number of desired modes

b) **Constraints (load collector):** In this collector node1 is fixed in all directions as shown in (Fig. 4.9).

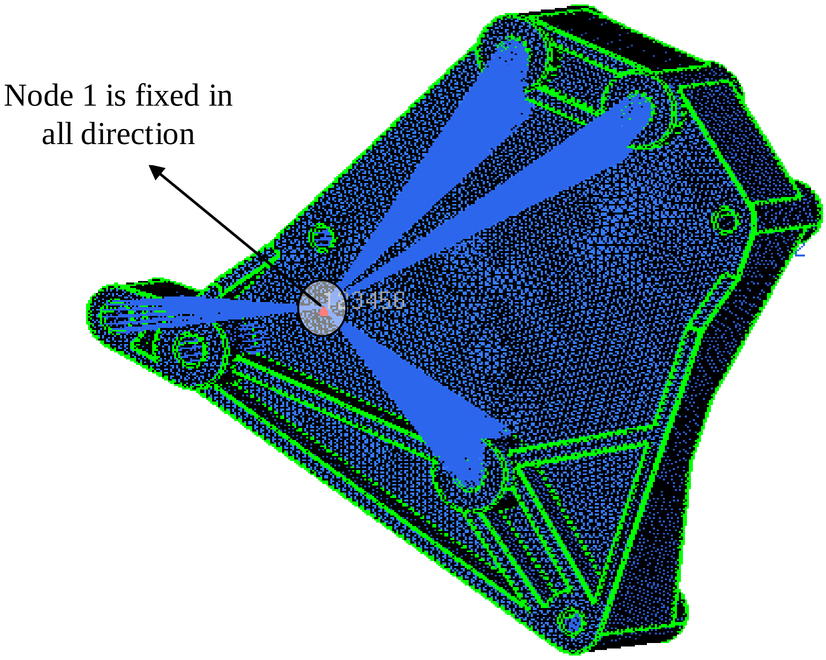
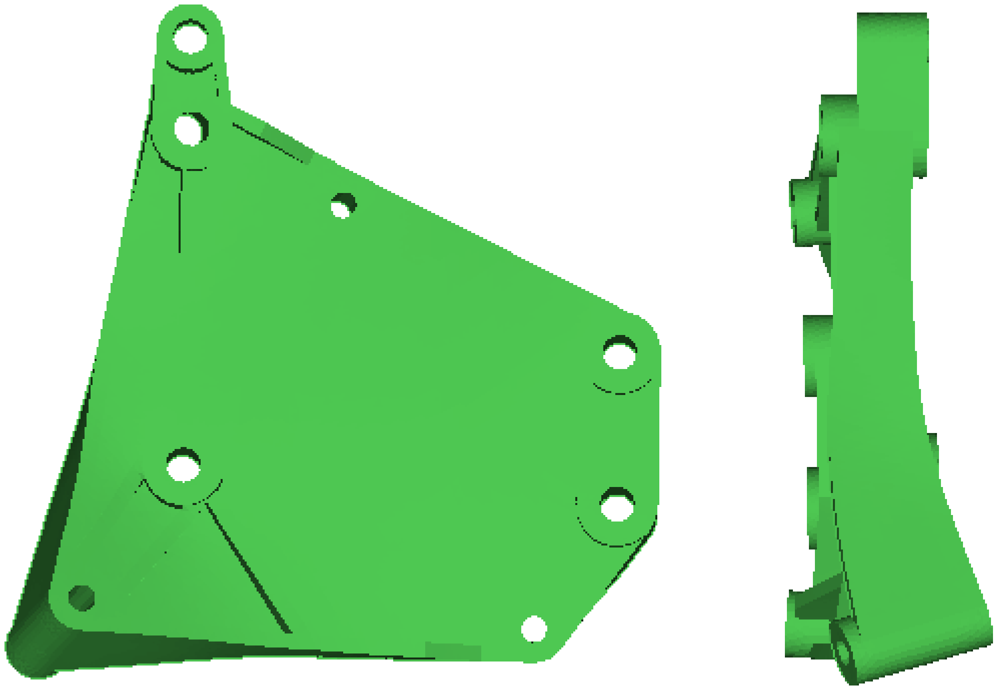
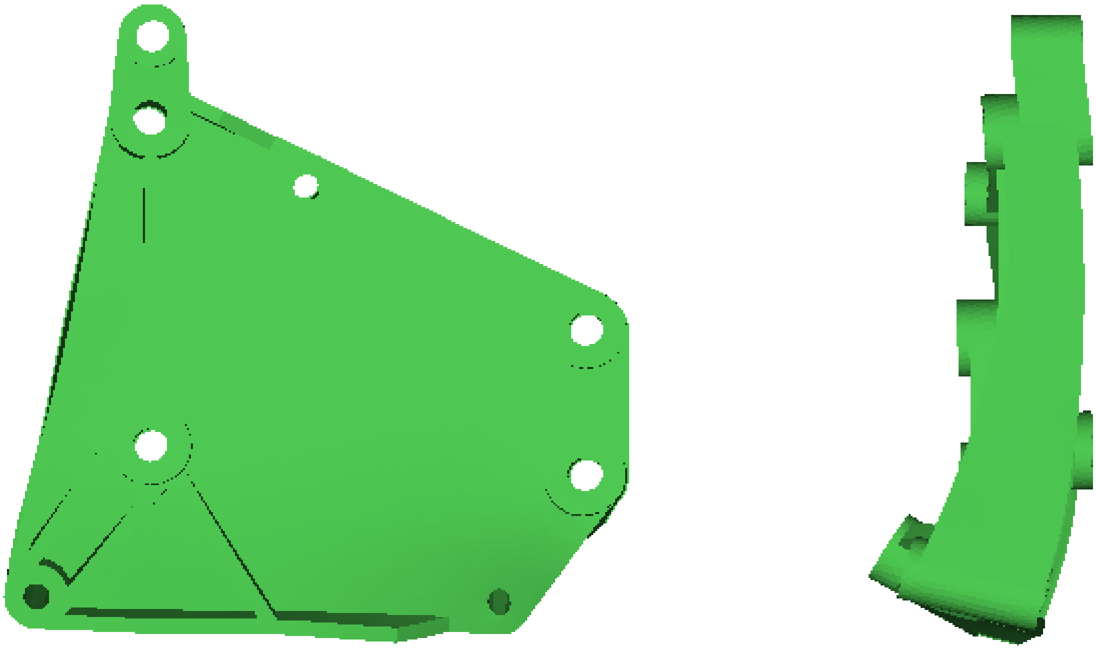


Fig 4.9 Boundary condition on the bracket

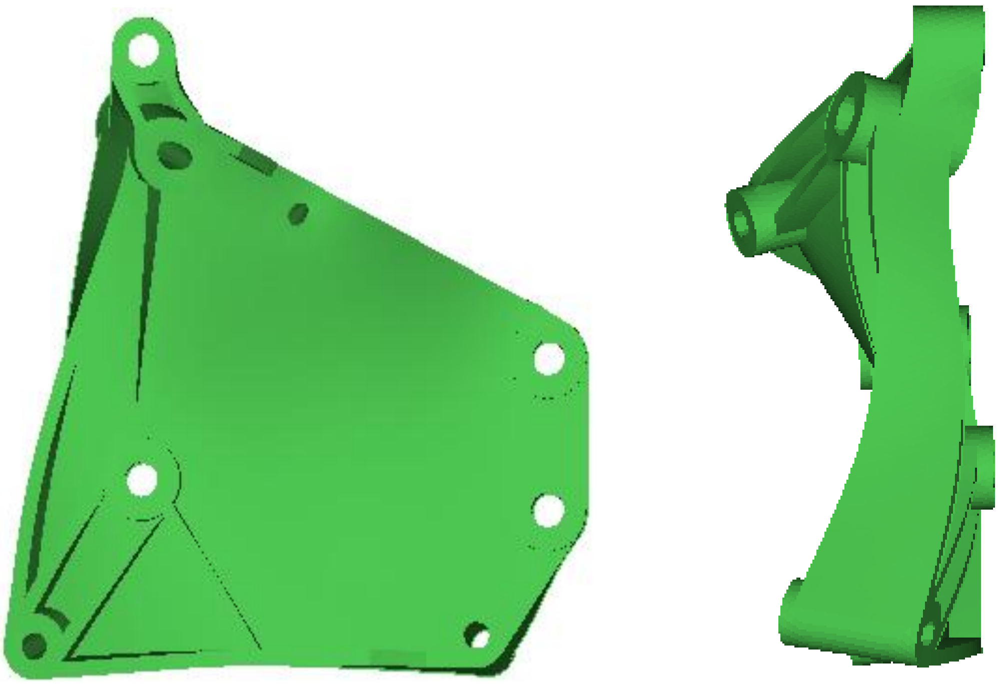
Normal modes analysis is done for natural frequencies and mode shapes. The results obtained from analysis are shown in Fig. 4.10.



Mode 1 – 362.48 Hz



Mode 2 – 632.82 Hz



Mode 3 – 933.99 Hz

Fig 4.10: Natural frequencies and mode shapes

4.6 Modal Frequency Response analysis of Compressor 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 ω becomes equal to natural frequency of the system, high value vibrations are produced due to resonance. Modal frequency response analyses of compressor mounting bracket along all three axes are done. CAD data and meshing has been explained in the previous sections and is same for all the analyses of the compressor mounting bracket.

a) Modal frequency response analysis along x-axis

In this analysis, force is applied in the x-axis direction and compressor is free to move in this direction. Forces are applied according to industry standards **(JIS1601D)** for the vibration testing of the automobile air conditioning equipment.

Boundary Conditions- For the components of the compressor mounting bracket, material and property are assigned in the same way as discussed in the previous section (normal mode analysis). The following load collectors are used in modal frequency response analysis along x axis direction.

a) **Cons (load collector):** In this collector, node1 of RBE2 element is fixed in all directions, except the x direction.

b) **Unit-Load:** In this collector, load type DAREA is used for dynamic loads represented as a constraint load type. This is applied along the x direction and on the node1 of RBE2 element. The value of acceleration load applied is 30g.

c) **Eigrl Load (Collector):** In this collector, EIGRL card image is used, for modes to be calculated in the frequency range from 50 to 1500 Hz.

d) **Tabled1 (load collector):** In this load collector frequency range table is created. The TABLED1 (Dynamic load tabular function) card image is used. The tabular function is given as shown in Fig. 4.11.

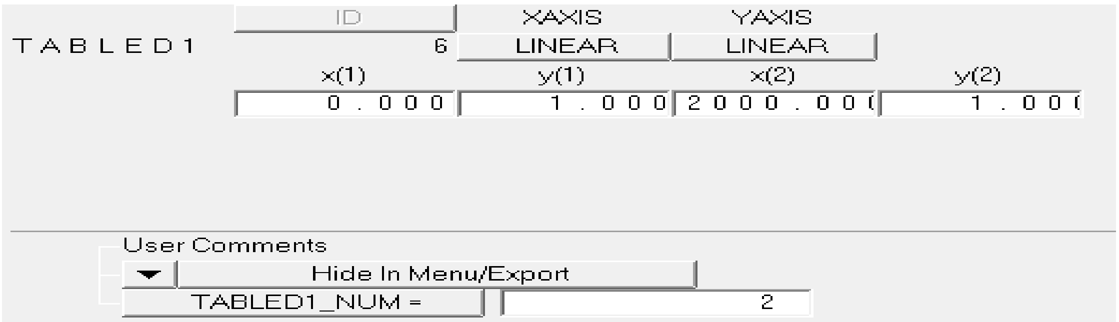


Fig. 4.11 Tabular function

e) **Rload2 (load collector):** This collector is used to create a frequency response dynamic load. In this collector ‘RLOAD2’ card image is used. This card image is used to give the excitation load ‘unit-load’ and tabular function ‘tabled1’.

f) **FREQ1 (load collector):** In this load collector ‘FREQ1’ card image is used. This card image is used to give the alternative form of frequency list. The following parameters are given by editing this collector:
Initial excitation frequency = 357
Increment for the next frequency =1
Number of frequencies = 10

g) **Damping Ratio:** the ratio of actual damping coefficient to the critical damping coefficient is known as damping ratio. It is a dimensionless measure describing how oscillations in a system decay after a disturbance. Damping ratio is taken as 6% for this analysis.

The results obtained, from the analysis, are shown in the Fig. 4.12. Stresses (in MPa) produced in x-axis due to dynamic loading with excitation frequency of 362.47 Hz are low as compared with the yield strength of the material (200 MPa).

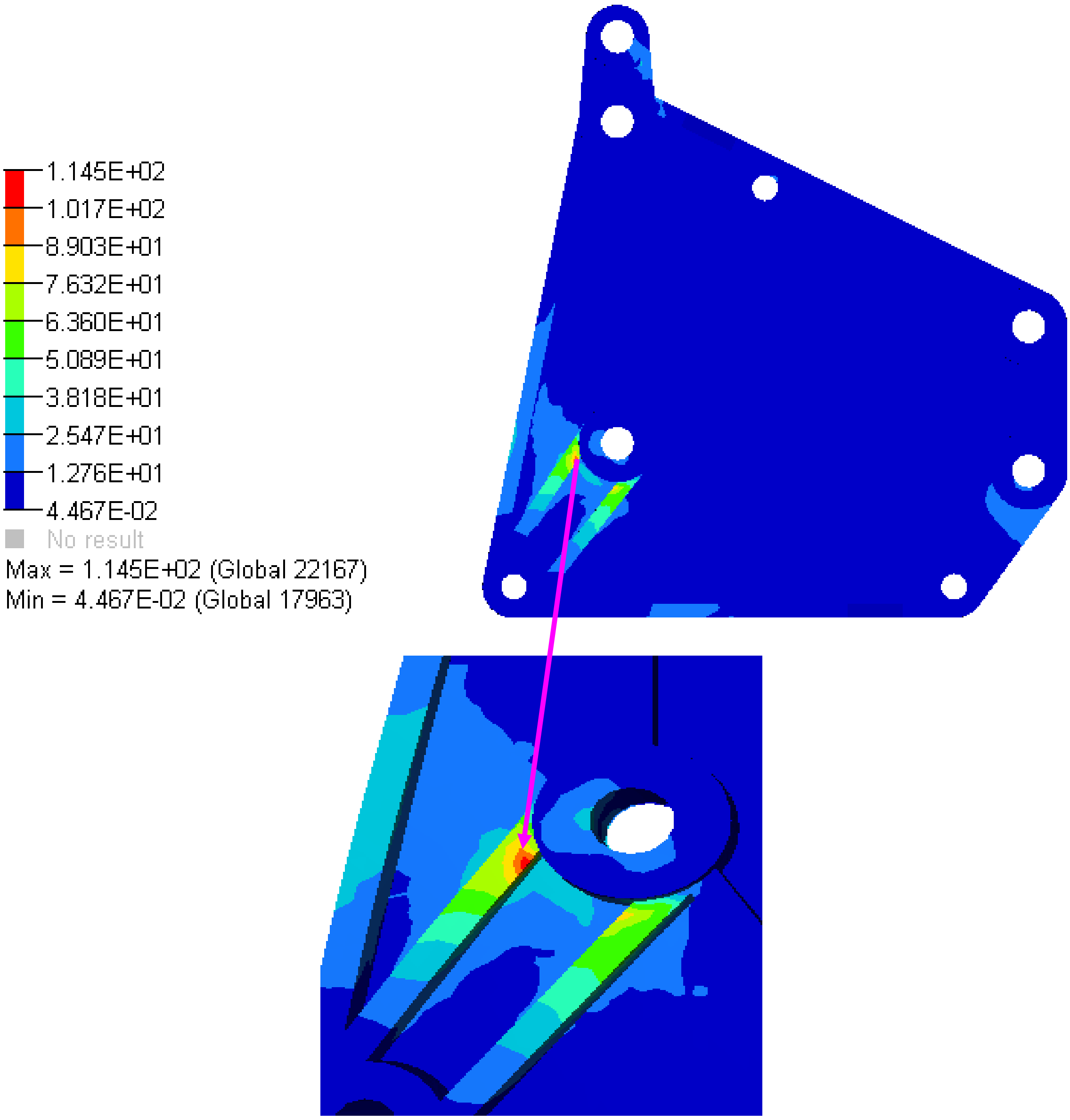


Fig 4.12: Von Mises Stress in mounting bracket along x-axis

b) Modal frequency response analysis along y-axis

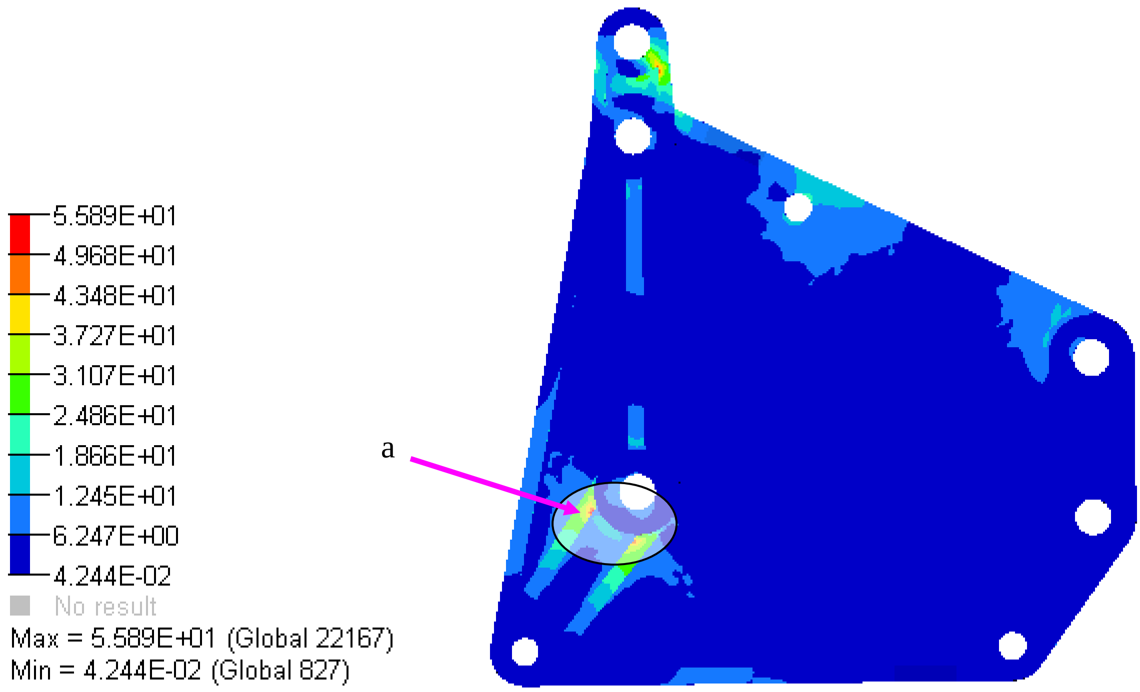
In this analysis force is applied in the y-axis direction and compressor is free to move in this direction. Except first two load collectors; all boundary conditions are applied in same way as for modal frequency response analysis along x-axis,

Load collectors- To apply the load and constraints along y-axis the following load collector are created.

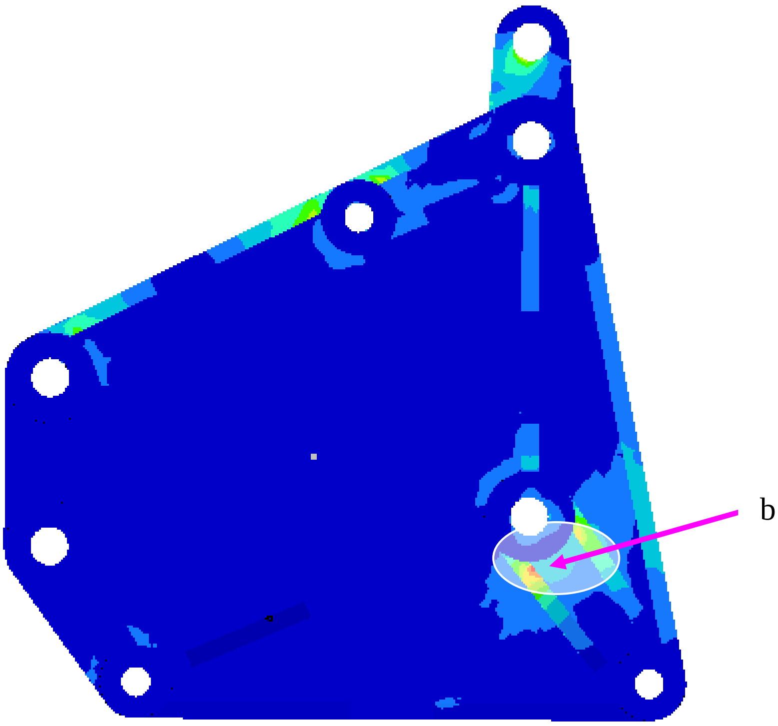
a) **Constraint-** In this collector, nodes1 of 'RBE2 element' are constrained in all direction, except the y direction.

b) **Unit-Load:** In this collector, load type DAREA is used for dynamic loads represented as a constraint load type. This is applied along the y direction and on the 'node1' of RBE2 element. The value of acceleration load applied is 30g.

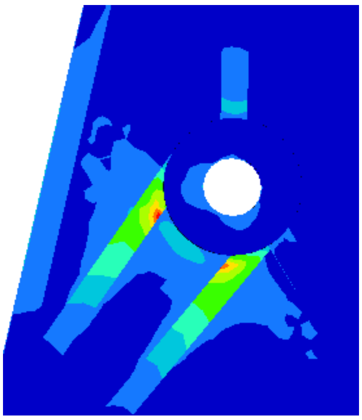
The remaining loads are applied in the same way as discussed in the previous analysis**.** Results obtained from the analysis are shown in the Fig. 4.13. The stresses produced in mounting bracket due to the dynamic load in y-axis with excitation frequency of 362.47 Hz are low.



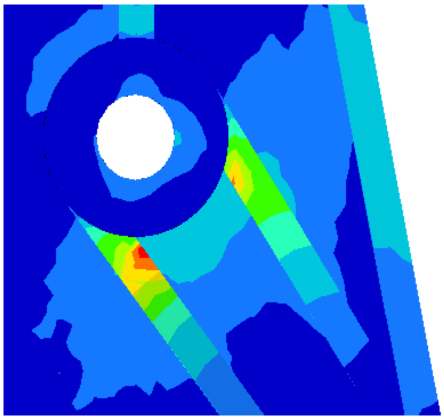
a) Von Mises Stress in Front view



b) Von Mises Stress in Rear view



a) Von Mises Stress on Front Side



b) Von Mises Stress on Rear Side

Fig 4.13: Von Mises Stress in mounting bracket along y-axis

c) Modal frequency response analysis along z -axis

In this analysis, force is applied in the z-axis direction and compressor is free to move in this direction. Except first two load collectors; all boundary conditions are applied in same way as for modal frequency response analysis along x- axis,

Load collectors- To apply the load and constraints along z-axis the following load collector are created.

a) **Constraint-** In this collector, nodes1 of 'RBE2 element' are constrained in all direction, except the z direction.

b) **Unit-Load:** In this collector, load type DAREA is used for dynamic loads represented as a constraint load type. This is applied along the y direction and on the 'node1' of RBE2 element. The value of the acceleration load applied is 30g.

The remaining four loads are applied in the same way as discussed in the previous analysis**.** Results obtained from the analysis are shown in the Fig. 4.14. The stresses produced in mounting bracket due to the dynamic load in z-axis with excitation frequency of 362.47 Hz are high as compared to x-axis and y-axis.

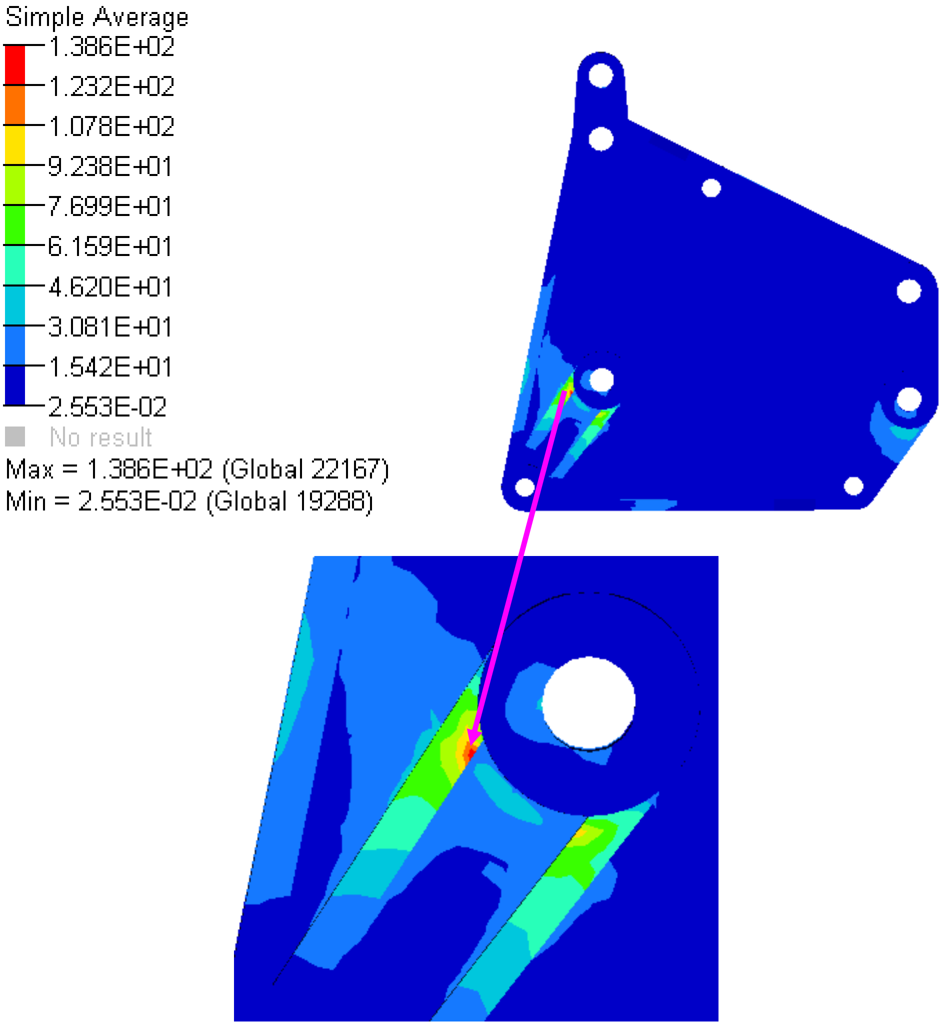
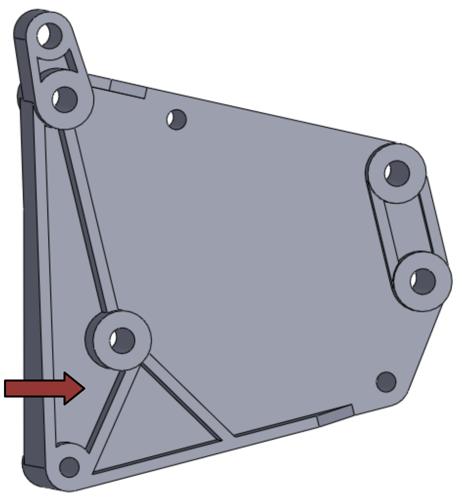


Fig. 4.14: Von Mises Stress in mounting bracket along z - axis

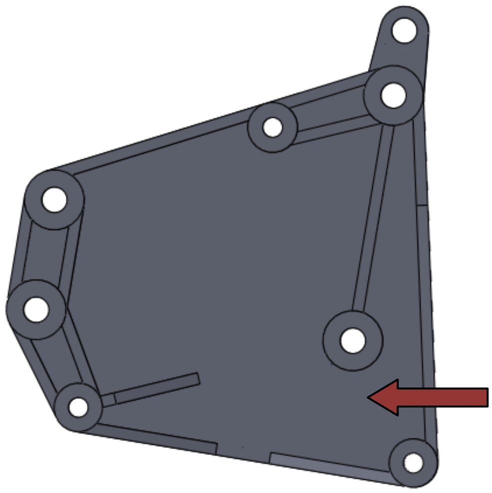
CHAPTER 5

EFFECT OF STIFFENERS IN COMPRESSOR MOUNTING BRACKET

The virtual test rig, that has been created using dynamic analysis methodology proposed in the previous sections, can be used to carry out different studies on the component under consideration. One such study can be carried out to analyze the effect of different 'hole to hole' ribs present in compressor mounting bracket.



a) Front view



b) Rear view

Fig. 5.1: 'Hole to Hole' ribs removed from exits CAD model

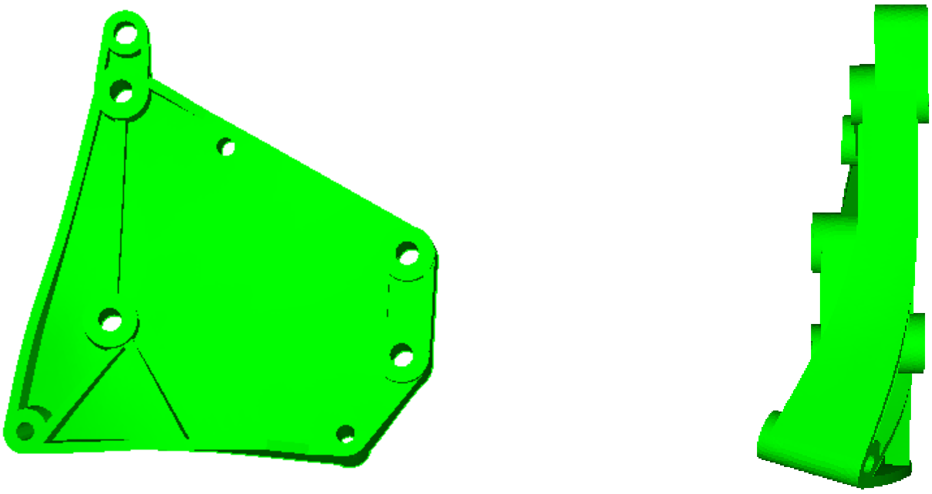
5.1 Dynamic Analysis

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

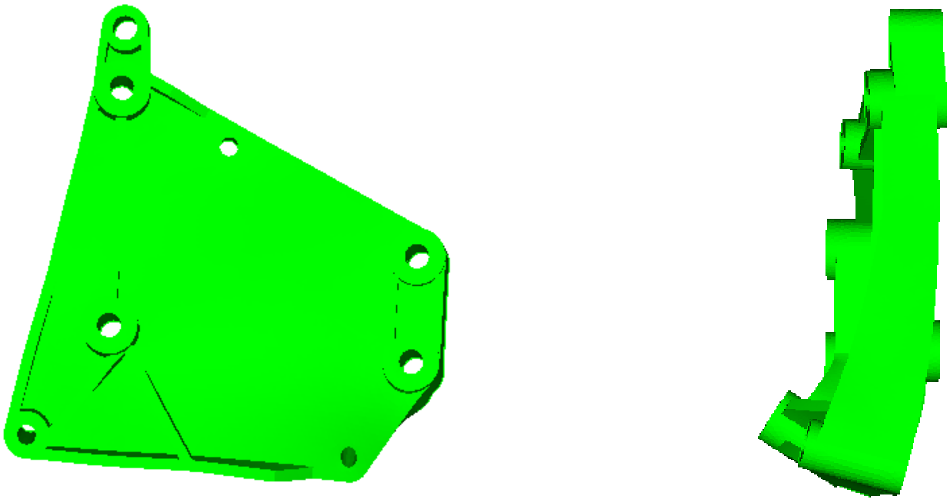
- Normal modes analysis.
- Modal frequency response analysis.

5.1.2 Normal Modes Analysis of the Mounting Bracket

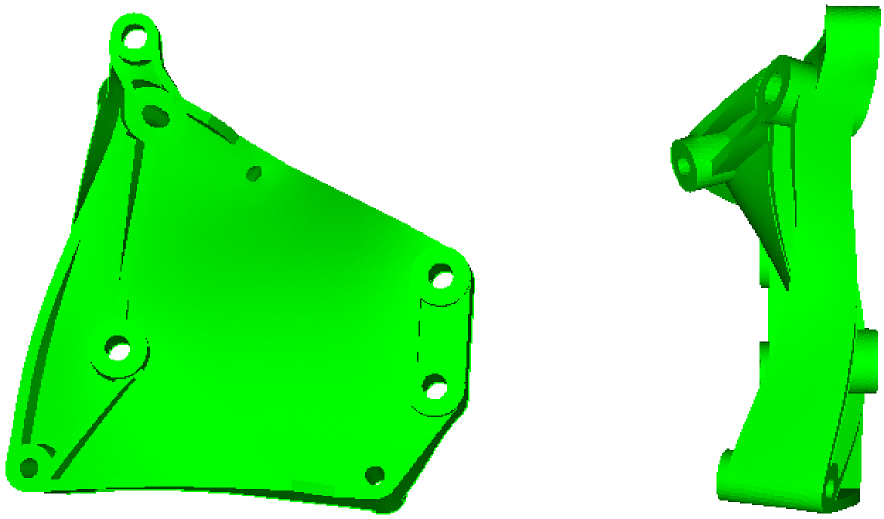
The procedure of normal modes analysis of optimize mounting bracket is same as explained earlier. The results obtained are:



Mode 1 – 283.41 Hz



Mode 2 – 621.77 Hz



Mode 3 – 913.88 Hz

Fig 5.2: Natural frequencies and mode shapes

5.1.2 Modal Frequency Response analysis of Compressor Mounting Bracket

The results obtained are:

Modal Frequency Response Analyses along x-axis

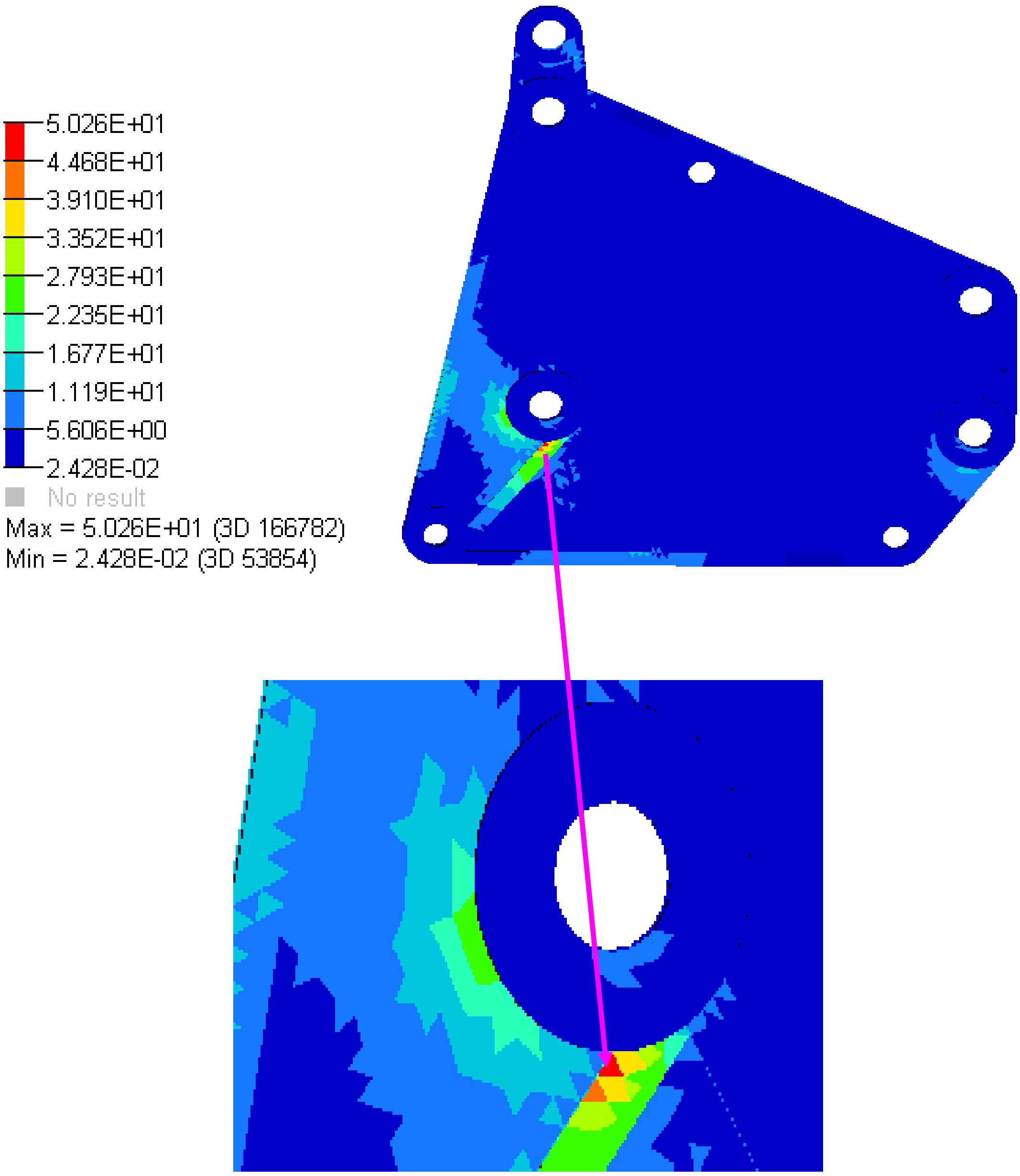


Fig. 5.3: Von Mises Stress in compressor mounting bracket along x-axis

The results obtained, from the analysis, are shown in the Fig 5.3. Stresses (in MPa) produced in x-axis due to dynamic loading with excitation frequency of 283.41 Hz are high and FOS is low as compared to the exiting model.

Modal Frequency Response Analyses along y-axis

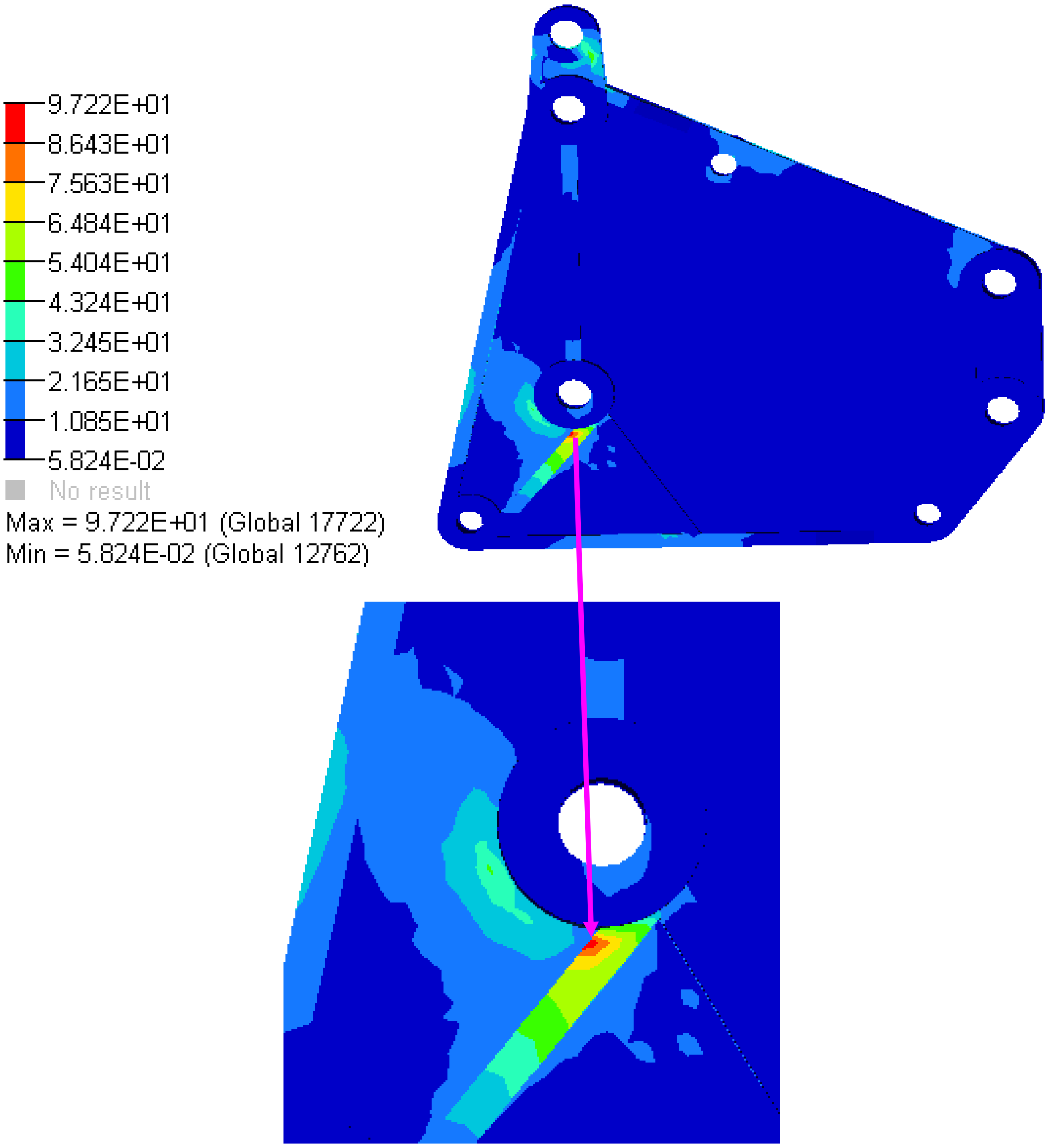


Fig. 5.4: Von Mises Stress in compressor mounting bracket along y-axis

The results obtained, from the analysis, are shown in the Fig 5.4. Stresses (in MPa) produced in y-axis due to dynamic loading with excitation frequency of 283.41 Hz are high and FOS is low as compared to the existing model.

Modal Frequency Response Analyses along z-axis

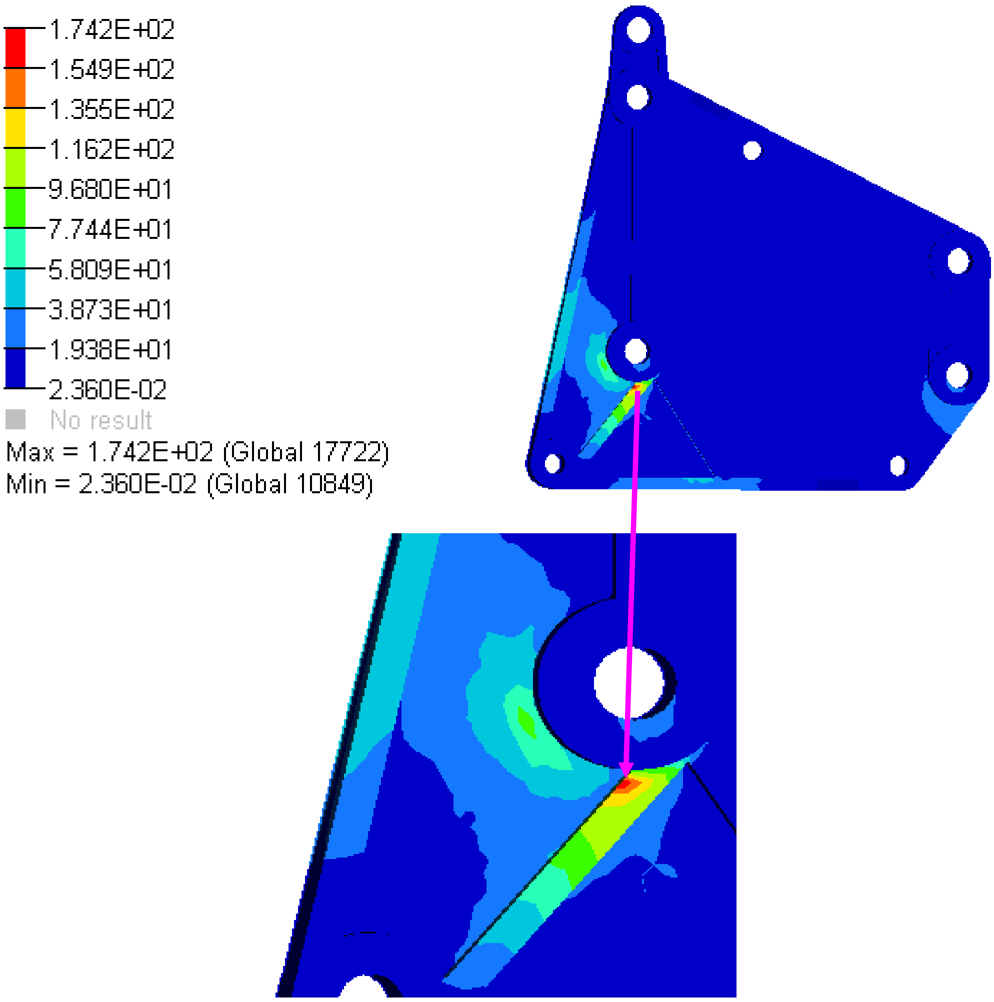


Fig. 5.4: Von Mises Stress in compressor mounting bracket along z-axis

The results obtained, from the analysis, are shown in the Fig 6.4. Stresses (in MPa) produced in x-axis due to dynamic loading with excitation frequency of 283.41 Hz, are high and FOS is low as compared to the existing model.

5.2 Results

A set of ribs is removed from the exiting bracket, and then the component is analyzed and the stress levels analyzed. It is found that after removal of these 'hole to hole' ribs, natural frequency is lowered, and the stresses become high (as shown in Fig. 5.4) and factor of safety (FOS) is lowered.

So, we can say that these ribs are very important in the bracket design and can be used in new design effectively to increase natural frequency and reduce stress in the bracket, without increasing the mass significantly.

CHAPTER 6

OTIMIZATION OF COMPRESSOR MOUNTING BRACKET

The material was removed from the low stress region of the bracket as shown in Fig. 6.1. CAD model was meshed and the methodology proposed for the dynamics analysis was used on the modified bracket.

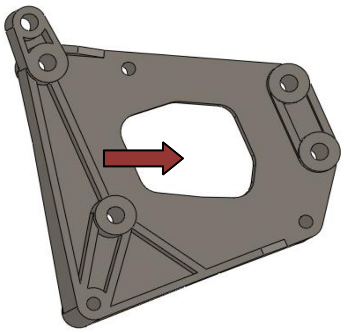


Fig. 6.1 Modified mounting bracket

6.1 MESHING

Procedure of meshing is similar as discuss in previous chapter. Tetrahedral element meshing of the bracket is shown in Fig 6.2.

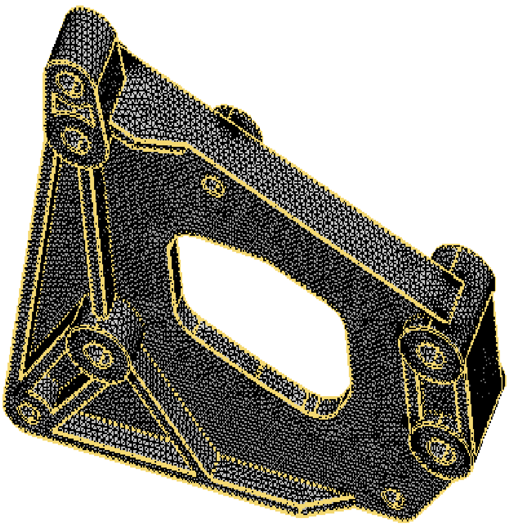


Fig 6.2: Tetrahedral meshing of the modified mounting bracket

6.2 Dynamic Analysis

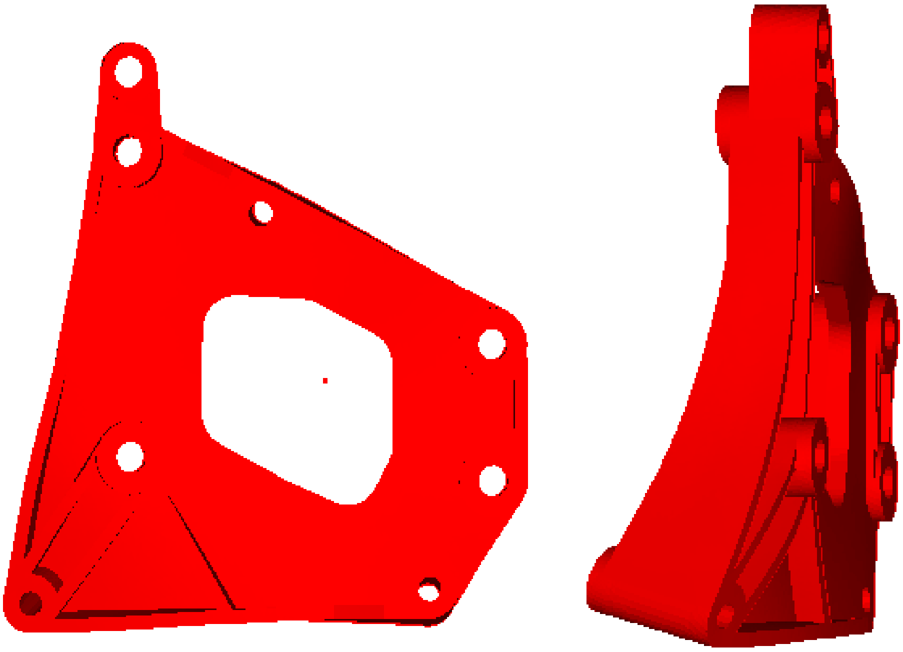
Dynamic analysis for the compressor mounting bracket is done which is divided in two steps:

- Normal modes analysis.
- Modal frequency response analysis.

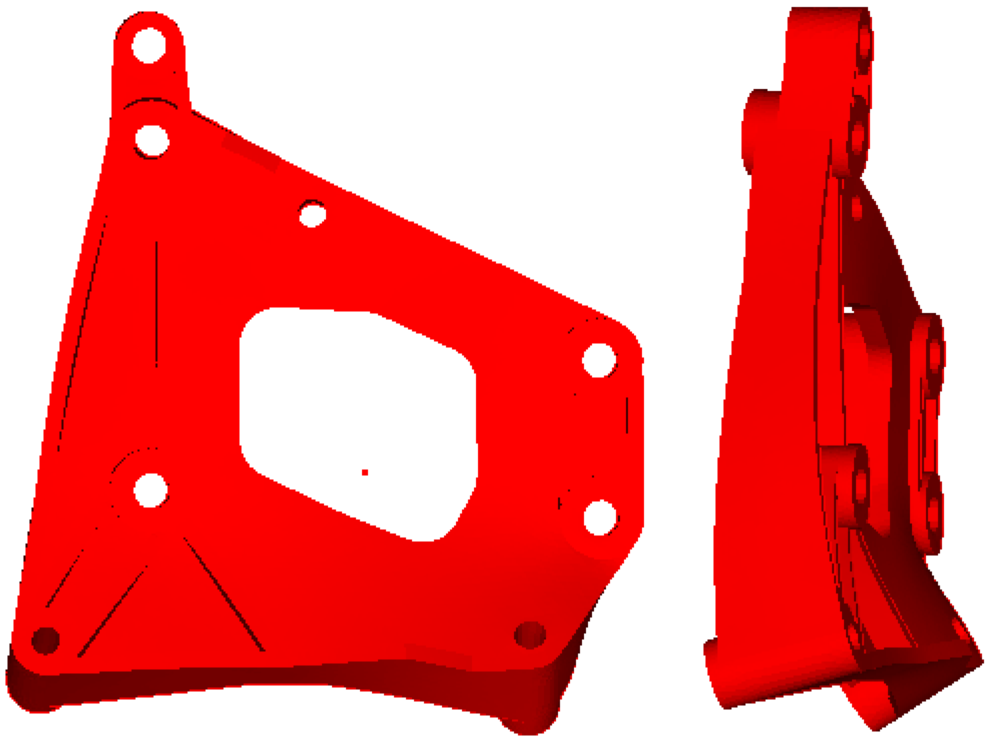
These two steps are related to each other. Output parameters of the first are used as input parameters of the second step.

6.2.1 Normal Modes Analysis of the Mounting Bracket

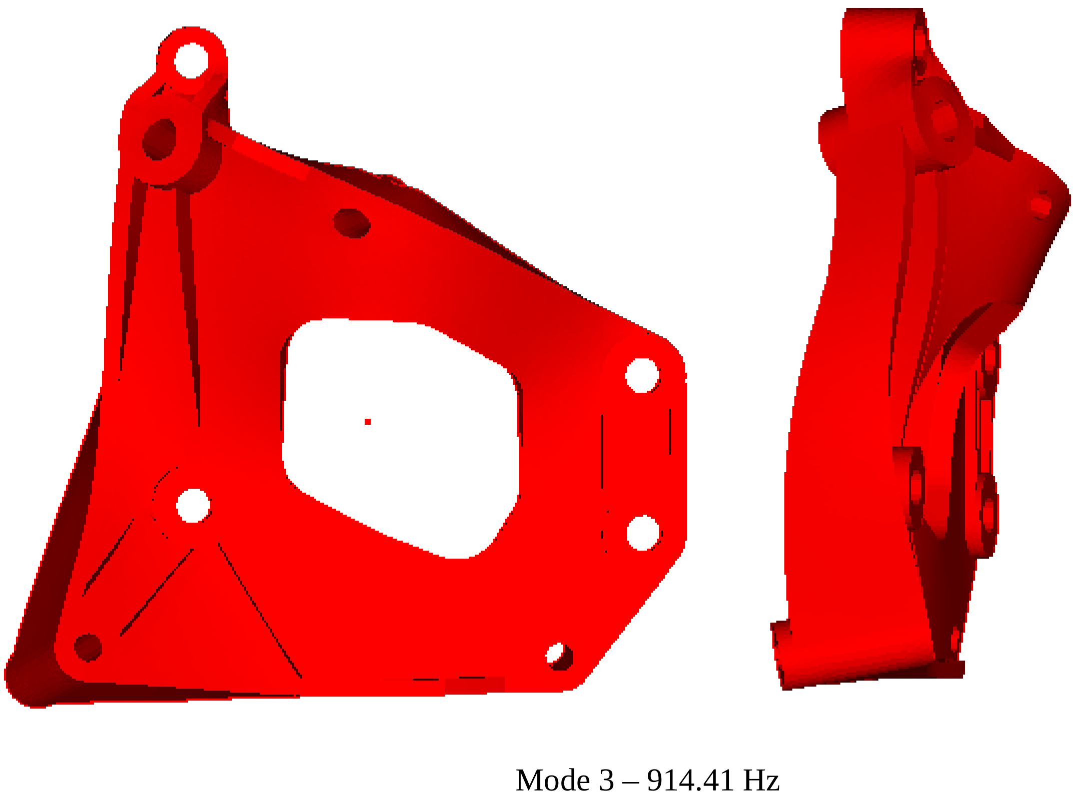
The procedures of normal modes analysis of modified mounting bracket are same as explained earlier. The results obtained are:



Mode 1 – 359.91 Hz



Mode 2 – 625.44 Hz



Mode 3 – 914.41 Hz

Fig 6.3: Natural frequencies and mode shapes

6.2.2 Modal Frequency Response analysis of Compressor Mounting Bracket

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

Modal Frequency Response Analyses along x-axis

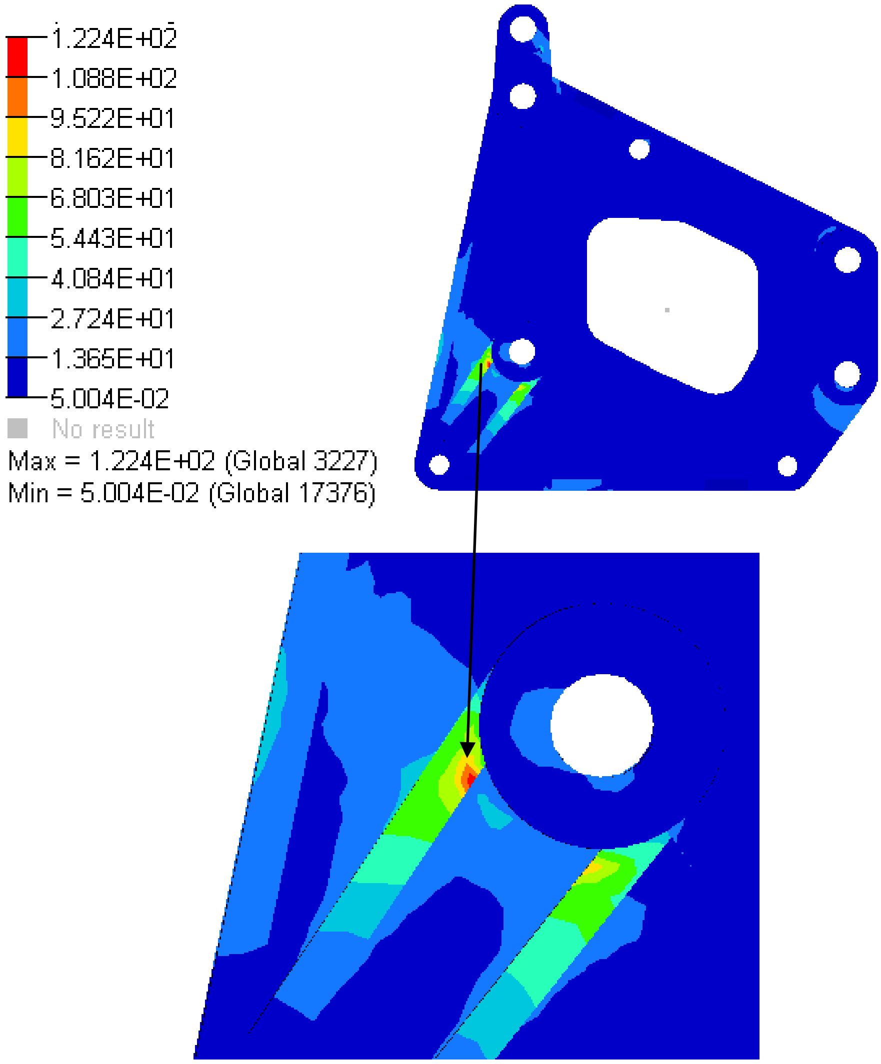


Fig.6.4 Stresses in modified mounting bracket along x-axis

The results obtained, from the analysis, are shown in the Fig 6.4. Stresses (in MPa) produced in x-axis due to dynamic loading with excitation frequency of 359.91 Hz, are still within limits.

Modal Frequency Response Analyses along y-axis

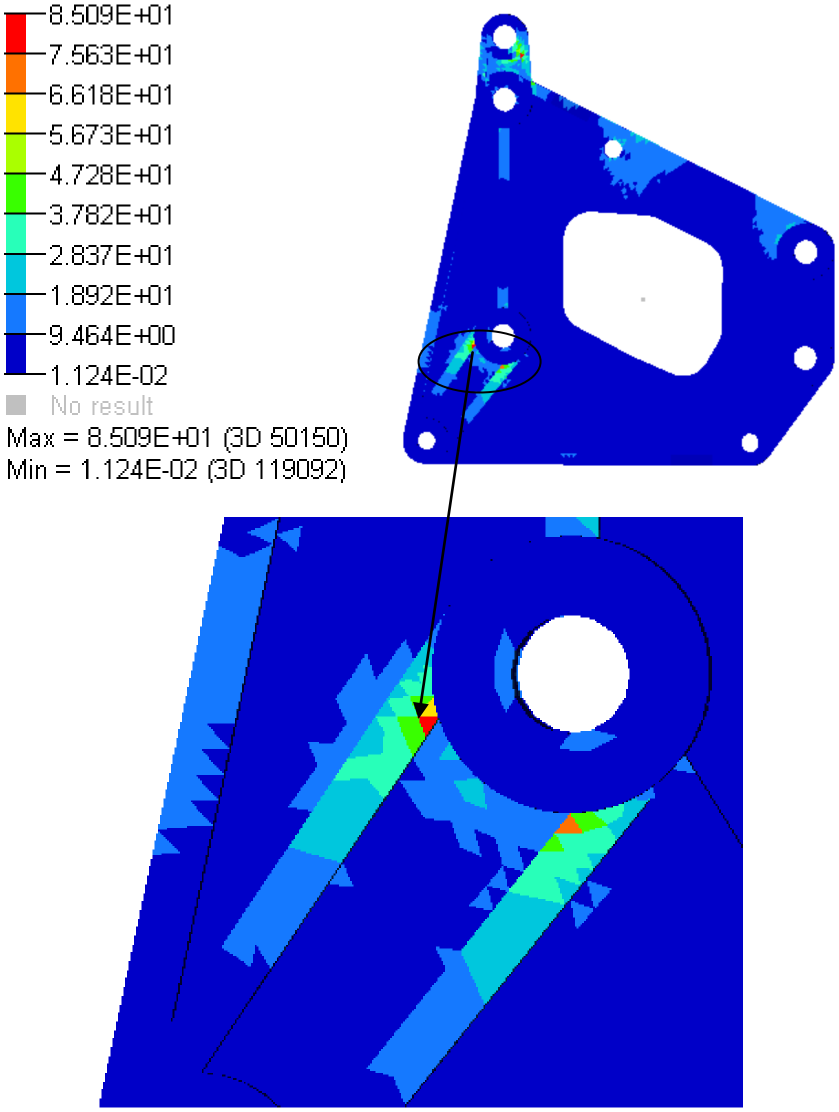


Fig.6.5 Stresses in modified mounting bracket along y-axis

The results obtained, from the analysis, are shown in the Fig 6.5. Stresses (in MPa) produced in x-axis due to dynamic loading with excitation frequency of 359.91 Hz are low and within limits.

Modal Frequency Response Analyses along z-axis

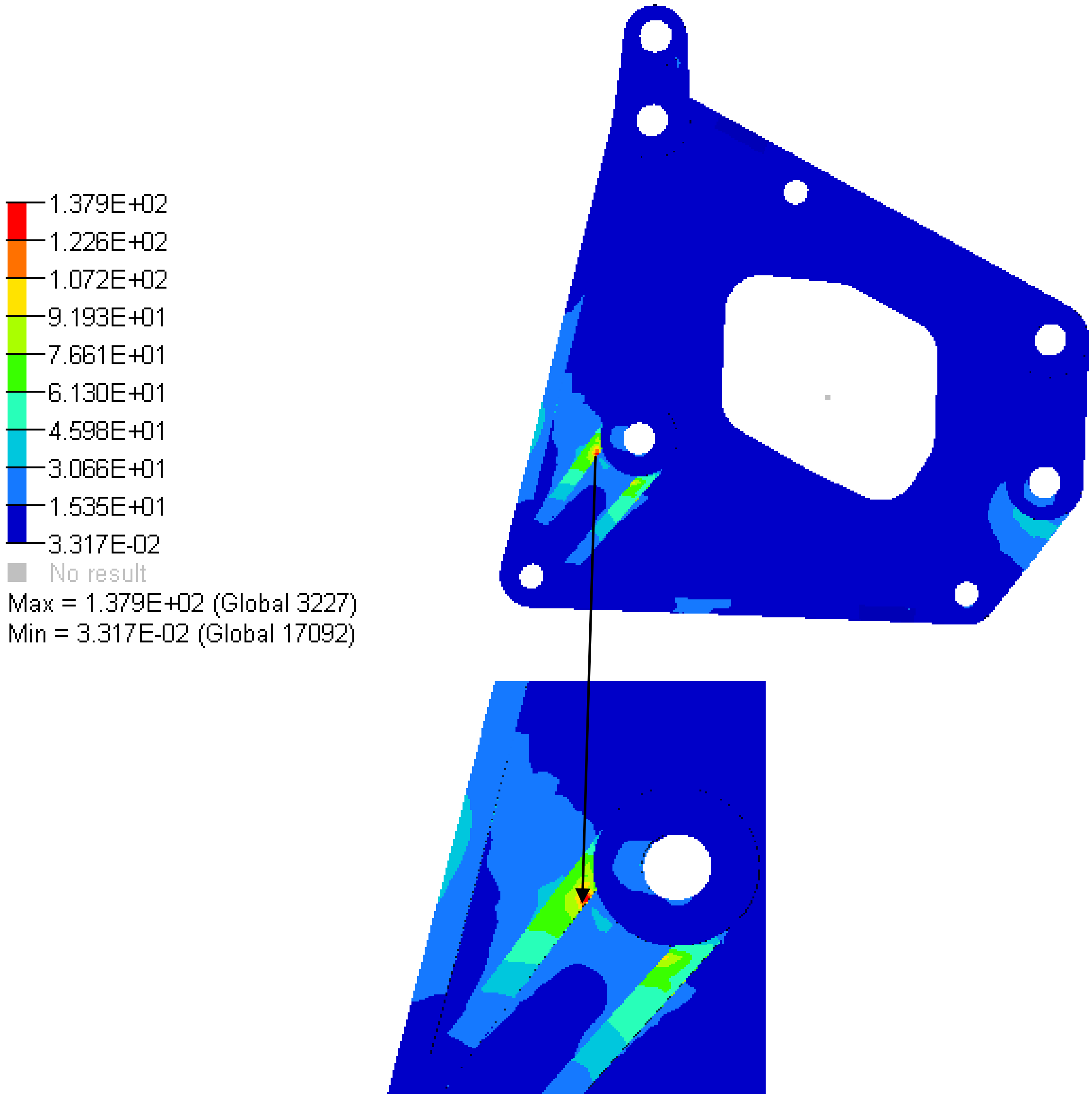


Fig.6.6 Stresses in modified mounting bracket along z-axis

The results obtained, from the analysis, are shown in the Fig 6.5. Stresses (in MPa) produced in z-axis due to dynamic loading with excitation frequency of 359.91 Hz are within acceptable range.

6.3 Results

As seen in Fig. 6.4 – 6.6, the stresses are still low and within the limit so the material can be removed and mass of the bracket reduced. The mass reduction in the modified bracket is 9.4%. Such iterations can be done using the virtual test rig, and improvements suggested.

CHAPTER 7

CONCLUSION AND SCOPE FOR FUTURE WORK

7.1 CONCLUSION

Dynamic analysis is done on an automobile compressor mounting bracket using the standard testing conditions. The results obtained revealed that the high values of stresses are produced in the vertical direction (z direction) on the compressor mounting bracket.

The ribs are one of the most important feature in the bracket design, and can be used in new design effectively to increase the natural frequency and reduce the stress in the bracket, without increasing the mass significantly. Thereafter optimization is done for brackets reducing the mass of the same by about 9.4%. It is also seen that the stresses increase as excitation frequency matches with the natural frequency for the same magnitude of applied load.

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.

7.2 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 practically validated for the value of stresses by using strain gauges, on an electro-dynamic shaker.

d) For the compressor mounting bracket modal transient response analysis can be done to see the effect of vibration (resonance) with time.

REFERENCES

[1] **G. Phani Sowjanya, P. Divakara Rao and Dr. C.Udaya Kiran (2013), "**Finite Element Analysis of Vibration Fixture Made of Aluminum and Magnesium Alloys**",** International Journal of Latest Trends in Engineering and Technology (IJLTET), Vol. 2, 84 – 89.

[2] **Nouby M. Ghazaly, Sufyan Mohammed and Ali M. Abd-El-Tawwab (2012), "**International Journal of Engineering Research and Applications (IJERA)", Vol. 2, 241 - 250.

[3] **Pavan B. Chaudhari and Dr. D. R. Panchagade (2012), "**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.

[4] **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.

[5] **Jeong Woo Chang and Young Shin Lee (2008),** "Topology optimization of compressor bracket", Journal of Mechanical Science and Technology, Vol. 22, 1668-1676.

[6] **Asheber Techane (2007)**, "Dynamics and Vibration analysis of bus body structures", Proceedings of MSC/NASTRAN conference, Brazil.

[7] **Jeong Kim, Joo-Cheol Yoon and Beom-Soo Kang (2007)**, “Finite element analysis and modeling of structure with bolted joints”. Applied Mathematical Modelling, Vol. 31, 895-911.

[8] **Doo-Ho Lee, Jeong-Woo Chang and Chan-Mook Kim (2003),** “Optimal Shape Design of an Air-Conditioner Compressor Mounting Bracket in a Passenger Car”, Vol 01, 1667–1672.

[9] **Sheldon Imaoka (2002)**, “Constant equation - CERIG, RBE3 and RIGID 184”, Proceedings of ANSYS conference, USA.

[10] **Erke Wang, Thomas Nelson and Rainer Rauch (1995)**, “A comparison of all-Hexahedra and all Tetrahedral Finite Element Meshes for elastic analysis”, Proceedings 4th of international conference, 179-181.

[11] **Yasuo O., Nagashima H. and Tanahashi**, "Vehicle technology handbook tests and evaluation", Society of Automotive engineers of Japan.