DYNAMIC ANALYSIS OF CONDENSER ASSEMBLY OF AUTOMOBILE AIR CONDITIONING SYSTEM USING CAE TOOLS

A thesis report submitted in partial fulfilment of the requirement for the award of

MASTER OF ENGINEERING

IN

CAD/CAM & ROBOTICS

Submitted By

MANJEET SINGH

Roll No. 800981016

Under the Guidance of

Mr. DALJEET SINGH	**Mr. JASWINDER SINGH SAINI**
Assistant Professor,	**Assistant Professor,**
Deptt. of Mech. Engg.,	**Deptt. of Mech. Engg.,**
TU, Patiala.	**TU, Patiala.**

DEPARTMENT OF MECHANICAL ENGINEERING

THAPAR UNIVERSITY

PATIALA-147004, INDIA

July-2011

DECLARATION

I hereby declare that the work in this thesis report entitled "**DYNAMIC ANALYSIS OF CONDENSER ASSEMBLY OF AUTOMOBILE AIR CONDITIONING SYSTEM USING CAE TOOLS**" 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 and **Mr. JASWINDER SINGH SAINI,** Assistant Professor, Mechanical Engineering Department, Thapar University, Patiala, during **July 2010 to June 2011.** 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.

Manjeet Singh
(Manjeet Singh)
Reg. No. 800981016

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

14/7/11

Mr. DALJEET SINGH
Assistant Professor,
Mech. Engg. Deptt,
Thapar University,
Patiala-147004.

14-7-2011

Mr. JASWINDER SINGH SAINI
Assistant Professor,
Mech. Engg. Deptt,
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*** *and* ***Mr. Jaswinder Singh Saini*** *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 & 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.

Manjeet Singh

Reg. No.: 800981016

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 air conditioning condenser assembly for the vibration loads. The condenser assembly has been analyzed using the standard testing conditions. The results revealed that the components of condenser assembly 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 condenser assembly is analyzed and optimized.

INDEX

CONTENTS	Page No.

LIST OF FIGURES

LIST OF TABLES

ABBREVIATIONS

HVAC	Heating, Ventilating and Cooling
MBS	MultiBody System
AC	Air Conditioning
CAE	Computer Aided Engineering
CAD	Computer Aided Design
FEM	Finite Element Method
FVM	Finite Volume Method
BEM	Boundary Element Method
FDM	Finite Difference Method
CFD	Computational Fluid Dynamics
MV	Motionview
ISO	Indian Standard Organization
FRF	Frequency Response Function

CHAPTER 1

INTRODUCTION

This chapter discusses the basic details and concepts of related topics associated with the present work.

1.1 METHODS TO SOLVE ANY ENGINEERING PROBLEM

Any engineering problem can be solved using the following different methods:

a) Analytical method
b) Numerical method
c) Experimental method

The following table 1.1 shows the brief comparison of these methods:

Table 1.1: Comparison of solution methods

Analytical Method	Numerical Method	Experimental Method
Classical approach	Mathematical representation	Actual measurement
100% accurate results	Approximate, assumptions made	Time consuming & needs expensive set up.
Closed form solution	Applicable even if physical prototype not available (initial design phase)	Applicable only if physical prototype is available
Complete in itself	Real life complicated problems	Results cannot be believed blindly & min. 3 to 5 prototypes must be tested

The procedure for solving a problem using Analytical or Numerical approach is as follows:

Step1- Writing governing equation: It represents problem definition or in other words formulating the problem in the form of a mathematical equation.

Step2- Mathematical solution of governing equation.

Final result is summation of above two steps. In analytical approach, the results will be 100% accurate as there is no assumption at either of the two steps. But on the other hand numerical method makes approximation at both the steps and hence it is an approximate method.

Analytical methods are used for only limited number of problems because exact governing equations are available for simple problems. For complex problems governing equations with some assumptions are to be made which are then solved using numerical method.

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 and then interpolate the results for the entire domain. Even before getting the solution, the variation of unknown parameters over a domain is known. For example, when meshing is carried out using linear quadrilateral elements, assumptions made is linear variation of displacement over the domain and for 8 nodded quadrilateral element, assumption is parabolic variation. This may or may not be the case in real life and 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 results, correlation, accuracy and minimization of errors.

1.2 TYPES OF NUMERICAL METHODS

Different types of numerical methods are as under:

a) Finite Element Method (FEM)

FEM is the most popular numerical method.
Applications – Linear, Nonlinear, buckling, Thermal, Dynamic & Fatigue analysis.

b) Boundary Element Method (BEM)

It is a very powerful and efficient technique to solve acoustics or NVH problems. Just like the finite element method it also requires nodes and elements but it considers only outer boundary of the domain. So in this case if the problem is of 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 & thus solving it faster.

c) Finite Volume Method (FVM)

All computational fluid dynamics (CFD) softwares are based on FVM. Unit volume is considered in Finite Volume Method. Variable properties at nodes are pressure, velocity, area, mass etc. It is based on Navier – strokes equation.

d) Finite Difference Method (FDM) :

Finite element and Finite difference share many common things. In general Finite difference method is described as a way to solve differential equation. It uses Taylor's series to convert differential equations to algebraic equation. It is used in combination of BEM or otherwise FVM to solve thermal and CFD coupled problems. FEM, being the most versatile method is to be used in the present work.

1.2.1 Finite Element Method (FEM)

Finite element method is Numerical analysis technique for obtaining approximate solution to number of engineering problem. It has been applied to broad area of continuum mechanics. Because of its flexibility as an analysis tool, it has emerged as much bone to the researchers.

In number of engineering situation, it is necessary to obtain approximate solution rather than exact solution to the problem. For example to find the concentration of pollution in atmosphere, to predict the formation of tornadoes and thunderstorms are the few examples of many partial problems. The derivation of governing equation for these problems can for done without much effort, but there solution by exact method of analysis is difficult task. The difficulty in these problems lies in the fact that either the geometry or some other feature of the problem is irregular or arbitrary. In such cases approximate methods of analysis provide alternate solutions. Examples of approximation methods are finite difference method and variational method such as Rayleigh Ritz method and Galerkin method.

In Finite difference method, the model of a problem gives a point wise approximation to the governing equation. Although finite difference method is simpler but it has several disadvantage. For example we have irregular or non straight boundaries. It is difficult to apply boundary condition. The difficulty to represent geometrically complex domain is to employ non uniform and nonrectangular meshes to more a few.

FEM overcomes the difficulties of all the other methods as it provides the systematic procedure for the deviation of the approximation functions. First a geometrically complex domain is represented as a number of geometrically simple sub domains, called 'Finite Element'. Second, the approximation functions over each finite element are derived using the idea that any continuous function can be represented by a linear combination of polynomials. In other words, FEM has the ability to formulate the solution for each Finite Elements which are then put together to report the entire problem.

1.2.2 Basic Steps in Finite Element Method

The following steps are used for analyzing a problem-

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

Pre-processing

Steps in pre-processing-

a) CAD data generation / import
b) Meshing
c) Boundary conditions

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

Boundary conditions consume least time but it is the most important step. 3 months hard work of meshing & CAD data preparation of engine block would be undone in just 1 day if boundary conditions are not applied properly.

Processing or Solution

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.

Software uses the equation, $[\boldsymbol{F}] = [\boldsymbol{K}]\,[\boldsymbol{\delta}]$

Where,

$[F]$ $=$ Force matrix

$[\delta]$ $=$ Displacement matrix

$[K]$ $=$ Stiffness matrix, the characteristic property of element depends on geometry as well as material.

Steps in Processing

a) Compute the element stiffness matrix

After the continuum is discretized with the discrete element shapes and the number, 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: 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 softwares.

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 a square matrix with size as *[No. of columns] = [No. of rows] = Total No. of nodes in the body* × *Degree of freedom.* It is always a symmetric matrix.

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

Post processing is viewing results, verifications, conclusion and thinking about what steps could be taken to improve the design

1.2.3 Applications of Finite Element Method

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

1) Linear static analysis
2) Dynamic analysis
3) Buckling analysis
4) Thermal analysis
5) Fatigue analysis
6) Optimization
7) CFD analysis
8) Crash analysis

Linear Static Analysis

It is the simplest and most commonly used type of analysis. Linear means straight line. $\boldsymbol{\sigma} = \boldsymbol{\varepsilon} E$ is an equation of straight line ($y = m\,x$) 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, Catia etc.

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.

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 softwares: Nastran, Anys, Abaqus etc.

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 softwares: Ansys, Nastran, Abaqus, I-deas NX etc.

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: MSC Fatigue, FEMFAT, FE SAFE, LMS etc.

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: OptiSstruct, Tosca,Nastran, Ansys etc.

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.

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: LS-Dyna, Pamcarsh, Radioss, Abaqus-Explicit, Madymo etc.

1.3 DYNAMIC ANALYSIS

Two basic aspects of dynamic analysis differ from static analysis. First, dynamic loads are applied as a function of time. Second, this time-varying load application induces time-varying response (displacements, velocities, accelerations, forces, and stresses). These time-varying characteristics make dynamic analysis more complicated and more realistic than static analysis. The equations of motion the simplest representation of a dynamic system are a single degree-of-freedom (SDOF) system shown in Fig.1.1. In an SDOF system, the time-varying displacement of the structure is defined by one component of motion. Velocity and acceleration are derived from the displacement.

In the given Fig.1.1, m is mass, b is damping (energy dissipation), k is stiffness (restoring force), p is applied force, u is displacement of mass, $\dot{u}$ is velocity of mass and $\ddot{u}$ is acceleration of mass.

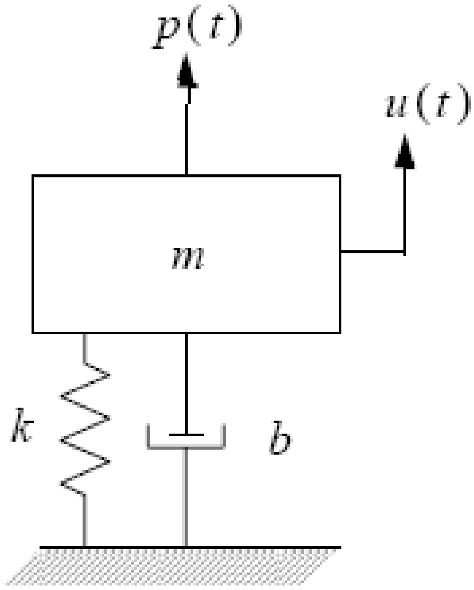


Figure 1.1: Single degree of freedom

1.3.1 Dynamic and Static Degrees-of-Freedom

Mass and damping are associated with the motion of a dynamic system. Degrees-of-freedom with mass or damping are often called dynamic degrees-of-freedom; degrees-of-freedom with stiffness are called static degrees-of-freedom. It is possible (and often desirable) in models of complex systems to have fewer dynamic degrees-of-freedom than static degrees-of-freedom. The four basic components of a dynamic system are mass, energy dissipation (damper), resistance (spring), and applied load. As the structure moves in response to an applied load, forces are induced that are a function of both the applied load and the motion in the individual components. The equilibrium equation representing the dynamic motion of the system is known as the equation of motion.

Equation of Motion This equation, which defines the equilibrium condition of the system at each point in time, is represented as

$$m\ddot{u}(t) + b\dot{u}(t) + ku(t) = p(t) \tag{1.1}$$

The equation of motion accounts for the forces acting on the structure at each instant in time. Typically, these forces are separated into internal forces and external forces. Internal forces are found on the left-hand side of the equation, and external forces are specified on the right-hand side. The resulting equation is a second-order linear differential equation representing the motion of the system as a function of displacement and higher-order derivatives of the displacement.

Inertia Force: An accelerated mass induces a force that is proportional to the mass and the acceleration. This force is called the inertia force.

Viscous Damping: The energy dissipation mechanism induces a force that is a function of dissipation constant and the velocity. This force is known as the viscous damping force. The damping force transforms the kinetic energy into another form of energy, typically heat, which tends to reduce the vibration.

Elastic Force: The final induced force in the dynamic system is due to the elastic resistance in the system and is a function of the displacement and stiffness of the system. This force is called the elastic force or occasionally the spring force.

Applied Load: The applied load on the right-hand side of Eq. (1.1) is defined as a function of time. This load is independent of the structure to which it is applied (e.g., an earthquake is the same earthquake whether it is applied to a house, office building, or bridge), yet its effect on different structures can be very different.

Solution of the Equation of Motion: The solution of the equation of motion for quantities such as displacements, velocities, accelerations, and/or stresses all as a function of time is the objective of a dynamic analysis. The primary task for the dynamic analyst is to determine the type of analysis to be performed. The nature of the dynamic analysis in many cases governs the choice of the appropriate mathematical approach. The extent of the information required from a dynamic analysis also dictates the necessary solution approach and steps. Dynamic analysis can be divided into two basic classifications: free vibrations and forced vibrations. Free vibration analysis is used to determine the basic dynamic characteristics of the system with the right-hand side of Eq. (1.1) set to zero (*i.e.*, no applied load). If damping is neglected, the solution is known as undammed free vibration analysis.

1.3.2 Dynamic Analysis Process

Before conducting a dynamic analysis, it is important to define the goal of the analysis prior to the formulation of the finite element model. Consider the dynamic analysis process to be represented by the steps in Fig. 1.2. The analyst must 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.

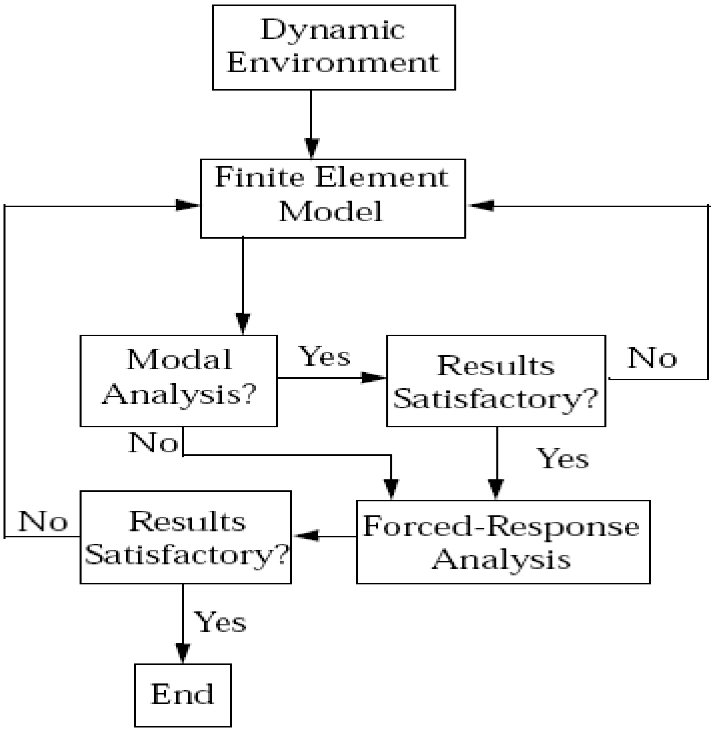


Figure 1.2: Overview of the dynamic analysis.

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:

1. Define the dynamic environment (loading).
2. Formulate the proper finite element model.
3. Select and apply the appropriate analysis approach to determine the behavior of the structure.
4. Evaluate the results.

CHAPTER 2
LITERATURE REVIEW

Following paragraphs presents the review of work done on dynamic and structure analysis of different components related to automobile.

Cameron *et al.* [1] studied the NVH behavior of trimmed body components in the frequency range 100–500 Hz. The work focused on the application and validation of numerical methods for predicting the acoustic and structural NVH behavior of trimmed body components in an automotive context. In particular, the level of modeling refinement and accuracy necessary to establish a reliable finite element analysis model for comparative purposes in the development of alternative designs was investigated. Specifically, the roof structure of a passenger car was investigated from various performance aspects, using both structural and acoustic excitation. The roof was initially tested in situ, with and without interior lining, to provide a reference for subsequent component tests. It was then detached from the car, mounted in a stiff frame and tested in a transmission window using both acoustic and structural excitation. A finite element model of the detached component was developed using shell and solid elements for the structure and solid elements for the interior lining. Predictions were carried out to evaluate the STL as well as the vibrational frequency response due to a force applied to the structure. Special attention was given to the modeling of the headliner as well as the air gap separating the headliner from the outer sheet metal. A sensitivity study of various headliner properties was performed in addition to a comparison between solutions calculated using standard Nastran elements and augmented poro-elastic elements via the software package CDH/EXEL.

The main objective of the work was to establish a datum reference for alternative designs. From this aspect, the validation of the numerical modeling methodology, in particular the level of detail and accuracy used was a crucial step. It was found that the predictions agreed very well with the measured data. As an additional, very interesting result, it was also found that the in situ testing correlated well with the transmission suite testing.

Douville *et al*. [2] studied On-resonance transmissibility methodology for quantifying the structure-borne road noise of an automotive suspension assembly. Methodology was presented for the analysis of the structure-borne noise transmission paths for an automotive suspension assembly. First, a fullyinstrumented test bench consisting of a wheel/suspension/lower suspension A-arm assembly was designed in order to identify the vibro-acoustic transmission paths (up to 250 Hz) for white noise excitation of the wheel. Second, frequency response function measurements between the excitation signal and each suspension/chassis linkages were used to characterize the different transmission paths that transmit energy through the chassis of the car. Finally, a synthesis of the major resonances of the suspension was drawn, with the objective of indicating which suspension transfer paths contribute the most to the structural forces transmitted to the chassis. On-resonance force transmissibility factors (ORTF) were calculated to provide an overall classification of the system resonances to the vibration transmission through the individual suspension linkages and in all axes.

Jong *et al.* [3] explained park panel contribution analysis by using acoustic reciprocity vehicle vibration & noise was evaluated by sound pressure level in cavity. In order to reducing sound pressure level, vehicle vibration was to reduce. Vehicle Vibration is affected by engine force, road profile and Structure of Vehicle etc. Engine force and road profile is impossible to change, so to reduce SPL (Sound Pressure Profile) is to lower level of panel vibration. After running normal mode analysis, Mode shapes showed that which modes were global or local. Modal participation to SPL (Sound Pressure Level) got from Modal Participation factor analysis. But the contribution of panel vibration to SPL was not found. This work focused on how to get contribution of panel vibration. Sensitive position of panel was found by applying acoustic reciprocity. To verify results, mass was added to sensitive panel position. SPL was reduced after mass was added. Also similar result got from result of modal participation factor analysis. The method was applicable to predict position of sensitive panel vibration. All calculations were performed using MSC.Nastran.

But the method could not find panel position. Therefore, the method was not proper. Also NTF (Noise Transfer Function) was not proper to find affective panels. To find affective panel to sound pressure Peak, Acoustic reciprocity method was used. After comparing Modal participation factor, the result of Panel participation factor was agreed with acoustic reciprocity method, it was proven to agree with two results. Therefore the acoustic reciprocity method was applicable to find sensitive panel position to sound pressure peak. This method benefit was to decide position of deadening material and development of Body.

Kang *et al.* [4] studied improving the durability of automobile suspension systems by optimizing the elastomeric bushing compliance. The durability of the suspension system is a critical component in the development of a vehicle because it is consistently and directly exposed to dynamic loads while the vehicle is in motion. In most cases, attempts are made to improve the durability of a vehicle suspension system by changing the shapes of its parts and components. However, in this work, proposal was given to improve the overall durability performance of a vehicle suspension system by modifying the compliance of the elastomeric bushings, which pivotally connected the lower control arm to the vehicle frame. It was relatively easy and cost effective to change the compliance of the bushing components because they were made of rubber or elastomeric materials. The following procedure was used for the present analysis. Firstly, dynamic loads were obtained based on a multi-body dynamic analysis while the vehicle was driven on a virtual proving ground test. Secondly, the stress distribution of the vehicle suspension system was calculated using finite element analysis to obtain an optimal combination of the elastomeric bushing compliance and provided the maximum improvement to the stress distribution on the basis of a robust design approach. Finally, the durability performance of each component in the vehicle suspension system was evaluated using quasi-static durability analyses. The fatigue life of the optimal model was improved by 36–39 per cent when the knuckle and lower control arm passed over bumps, and by 30–41 per cent when they passed over potholes. The proposed optimal design process to improve the elastomeric bushing compliance made it easy to evaluate and improve the durability performance of the vehicle suspension system in the initial stages of the vehicle development process. However, the approach had a drawback that the durability design was not reflected in the initial stages of development because the load history was obtained experimentally from a prototype.

Kindt *et al.* [5] studied on the development and validation of a three-dimensional ring-based structural tyre model. It presented a structural model for an unloaded tier, based on a three-dimensional flexible ring on an elastic foundation was developed. The ring represented the belt and the elastic foundation represented the tier sidewall. The model was valid up to 300 Hz and included a sub model of the wheel and the air cavity. This made the model potentially suitable for the prediction of structure-borne interior noise. Unlike most ring models, which only considered in-plane modes, the presented model also predicted the modes that involve torsion of the belt in circumferential direction. The parameterization of the model, which did not require detailed knowledge of the tyre construction, was based on the main geometrical properties of the tyre and a limited modal test. Comparison between measured and calculated responses showed that the tier–wheel model described the dynamic behaviour with acceptable accuracy. Since the model was physical, it could be applied to describe other operational conditions such as loading and rotation.

The work described a physically based structural tyre model for the analysis of the tier dynamic behavior up to 300 Hz. The model described all modes that appear in this frequency range, unlike most ring models that were limited to in-plane modes. The main assumption was that the dynamic behavior of a tier in this frequency range could be approximated by a flexible three-dimensional ring on an elastic foundation. The model was implemented as a finite element model. The fully assembled tyre model included a wheel and air cavity model, which was necessary to describe all physical phenomena below 300 Hz. It was shown that the wheel flexibility had a significant influence on the lowest tyre modes. Therefore, a detailed wheel model was needed to obtain the required accuracy of the fully assembled model. The parameterization of the model was based on simple geometrical properties of the tier and the experimental modal parameters of three tyre modes. Despite the drastic, well considered simplifications, comparison between measured and calculated responses showed that the tyre–wheel model described the dynamic behavior with acceptable accuracy. Because the model was physical, it could also be applied to predict the dynamic behavior under different operating conditions, such as loading and rotation.

Liu *et al.* [6] studied an edge-based smoothed finite element method (ES-FEM) for static, free and forced vibration analyses of solids. The work presented an edge-based smoothed finite element method (ES-FEM) to significantly improve the accuracy of the finite element method without much changing to the standard FEM settings. The ES-FEM could use different shape of elements but preferred triangular elements that could be much easily generated automatically for complicated domains. In the ES-FEM, the system stiffness matrix was computed using strains smoothed over the smoothing domains associated with the edges of the triangles. Intensive numerical results demonstrated that the ES-FEM possessed the following excellent properties: (1) the ES-FEM model possessed a close-to-exact stiffness: it was much softer than the ‘‘overly-stiff’’ FEM and much stiffer than the ‘‘overly-soft’’ NS-FEM model; (2) the results were often found super convergence and ultra-accurate: much more accurate than the linear triangular elements of FEM and even more accurate than those of the FEM using quadrilateral elements with the same sets of nodes; (3) there were no spurious non-zeros energy modes found and hence the method was also temporally stable and worked well for vibration analysis and (4) the implementation of the method was straightforward and no penalty parameter was used, and the computational efficiency was better than the FEM using the same sets of nodes. In addition, a novel domain-based selective scheme was proposed leading to a combined ES/NS-FEM model that was immune from volumetric locking and hence worked very well for nearly incompressible materials. These properties of the ES-FEM were confirmed using examples of static, free and forced vibration analyses of solids.

An edge-based smoothed finite element method (ES-FEM) was proposed for stable and accurate solutions. The method was applied to static, free and forced vibration analyses of 2D solid mechanics problems. Through the formulation and numerical examples, following conclusions were drawn:

(a) ES-FEM could use general n-sided polygonal elements including triangular element the extension of the method for 3D problems using tetrahedral elements was also straightforward.

(b) The ES-FEM using triangular elements was stable and accurate without using any parameter for stabilization. The formulation was straightforward and the implementation was as easy as the FEM, without the increase of degree of freedoms. The ES-FEM often showed super convergence behaviour with ultra- accurate results: The numerical results of the ES-FEM using triangular elements were even more accurate than the FEM using quadrilateral elements with the same sets of nodes.

Migeot *et al*. [7] discussed about predicting car body dynamics at higher frequencies.

A new solver for the prediction of narrow-band frequency response functions was developed.

An approximation of the entire FRF was obtained by factorizing the matrix at a single frequency.The procedure was adaptive and provided results within a user specified tolerance. It could easily be integrated within MSC.Nastran. The cost of calculating the FRF with new procedure was a tiny fraction of the cost of the standard frequency sweep and provided a much better frequency resolution.

Reis *et al.* [8] studied on structural analysis of brake system's air reservoir of heavy duty vehicles. The work described the use of computer simulation supported by experimental data on the structural analysis process of a heavy duty commercial vehicle. In this work the occurrence of cracks in the brake system's air reservoir prototype of a heavy duty commercial vehicle during track tests were investigated.

The FEA software MSC.Nastran was employed and specific simulations were carried out for the reservoir and its brackets which were inserted in the full vehicle model. The results obtained for the initial configuration were in agreement with the failure that occurred in the test track. Geometric and welding alterations in accordance to design criteria were proposed. New failures have not been verified in experimental tests since these modifications were implemented.

The numerical results for the original version of the fixing supports really identify the failure occurred in the track test. The analyses had shown that the final configuration complies with the design criteria and, therefore, could be used in the production vehicle. New failures, in the analyzed component were not verified in the track tests since the new configuration were adopted. The presented work had described the successful use of numerical simulation in the design processes of a heavy duty commercial vehicle. The numerical simulation allowed different concepts of the air reservoir to be tested before a new prototype were constructed. As a consequence, development cost and time of such component could be reduced. From this, it was concluded that numerical simulation is a powerful tool to be used in the development phase of new vehicles, in conjunction with experimental data and validation procedures.

Scigliano *et al*. studied [9] the verification, validation and variability for the vibration study of a car windscreen modeled by finite elements. The main purpose was to investigate verification, validation and variability issues applied to an industrial component: a car windscreen. The windscreen is a sandwich structure whose stacking sequence contains five layers. The two external thick layers are made of glass, while the three thin intermediate layers are made of appropriate polymers. In this framework, two main objectives were identified. This work focused the attention on the study of the dynamics of an acoustic windscreen under free–free and real boundary conditions. An application of the verification and validation methodology was presented to assess the capability of finite element models to predict the natural frequencies of the acoustic windscreen, in presence of intra variability due to temperature variation. Indeed, intra variability of glue and polymers' elastic properties lead to intra variability of the dynamic behavior of the windscreen. Experimental campaigns, in free–free and real boundary conditions, were performed in a climatic chamber. The effect of the temperature changes on the windscreen vibration behavior was evaluated and the component experimental intra variability estimated. A numerical study was performed as well. Three different numerical models were considered: a simplified model in free or clamped conditions, a trimmed body model. The verification stage, including convergence studies, concluded that the multilayer shell model approach was valid at low temperature, when polymers were relatively stiff. On the contrary, at higher temperature, polymers were very flexible and shell models lead to significant errors due to considerable transverse shear effects. Finally, the main result was that a solid model must be used for the windscreen to correctly reproduce the physics at different temperatures. A validation stage, involving numerical and experimental results, was performed to evaluate the predictive capability of the developed numerical models. Validation metrics, which assess the mean value and the variability level of the frequencies, were proposed. The finite element models led to very satisfactory results for the mean values of the frequencies. The general trends of the experimentally observed intra variability were also well reproduced. Nevertheless, further investigations were necessary to improve the predictive capability of the numerical model that currently underestimates the experimental intra variability. This discrepancy was essentially due to the complex non-linear behavior of polymers.

The work focused the attention on the experimental and numerical study of the dynamics of an acoustic windscreen in free–free and real boundary conditions. An application of the verification and validation methodology was presented to assess the capability of finite element models to predict the natural frequencies of the acoustic windscreen, in presence of intra variability. In free–free conditions the experimental campaign proved that the system was extremely sensitive to temperature changes. The natural frequencies and the damping ratios showed intra variability up to, respectively, 40% and 10%. Variability of the polymers elastic properties due to temperature variations leads to variability of the frequencies. For the verification stage, the main result was that the multilayered shell model approach was valid at lower temperature, when polymers were relatively stiff. On the contrary, at higher temperature, polymers were very flexible and shell models led to significant errors because transverse shear effects were considerable. Therefore, a solid finite element model was necessary. For the validation stage, two validation metrics were proposed. They assesed the mean value and standard deviation of the natural frequencies, respectively. The solid finite element model led to very satisfactory results for the mean value of the frequencies. The general trends of the intra variability experimentally observed were also well reproduced. However, the numerical approach underestimated the variability level.

Wendong *et al.* [10] studied numerical simulation on XIALI car dynamic response. Based on the white body's finite element model, the dynamic response finite element model of TJ7100 car created with MSC.Patran and MSC.Nastran. The body response Power Spectral Density (PSD) was calculated under real road PSD and compared with test result. The finite element model was reliable and could be applied to car response evaluation.

It can be seen from the literature review, very less amount of work is done on the air conditioning assembly of the automobile. Dynamic analysis of component is to be done for better results. In the present work, a condenser assembly is taken which is dynamically analyzed and optimized.

CHAPTER 3

ANALYSIS OF CONDENSER ASSEMBLY-1

3.1 PROBLEM DEFINITION

The various automobile parts, mounted on the engine or body of the vehicle are subjected to dynamic loading from the road vibrations. Condenser assembly is one such component, which is the integral part of automobile air conditioning system, shown in Fig. 3.1

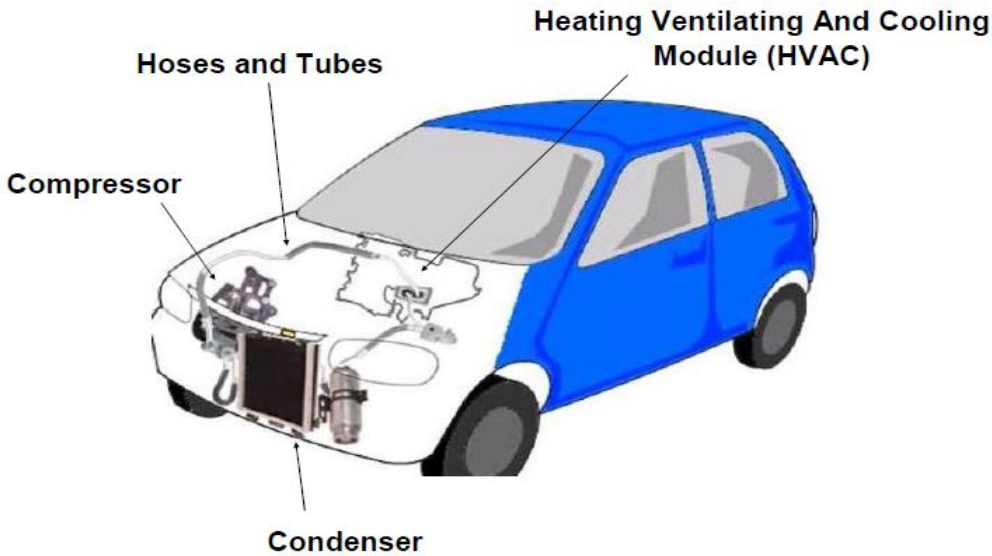


Figure 3.1: Components of car AC

3.2 DESCRIPTION OF PARTS AND BASIC LAWS OF THE AIR-CONDITIONING

Before discussing the methodology for analysis a brief description of an air-conditioning system is presented here because it is useful for the understanding of the problem background and the working conditions.

3.2.1 Air Conditioning

The process of treating air so as to control simultaneously its

1) Temperature
2) Humidity
3) Cleanliness
4) Distribution (air movement)

Meeting the requirements of the conditioned space is called air conditioning.

3.2.2 Elements of the Air-Conditioning Systems

Air conditioning system is defined as an assembly of the different parts used to produce a specified condition of air within a required space. The basic elements of an air conditioning system are:

a) Fans: for moving the air.
b) Filters: for cleaning air, either fresh, re-circulated or both.
c) Refrigerating plant: Connected to heat exchange surface, such as finned coils or chilled water sprays.
d) Means for warming: the air, such as hot water or steam heated coils or electrical elements.
e) Means for humidification and/or dehumidification.
f) Control system: to regulate automatically the amount of cooling or warming.
g) Refrigerant: In the air conditioning system heat is extracted from the given volume which is done by the flow the heat transfer media in the 'refrigerant cycle' called 'refrigerant'. So a 'refrigerant' is defined as any substance that absorbs heat through the expansion or vaporization and loses it through the condensation in a refrigeration system. For example air, water, ammonia, Freon etc. Freon gas is the most commonly used heat transfer media:
    1) R22
    2) R134a
    3) R407C
    4) R410a

3.2.3 Basic Components of the Refrigeration Cycle

A refrigeration cycle has the following components, as shown in the Fig. 3.2

1) Compressor: To compress the refrigerant.
2) Discharge line :To flow the refrigerant fro compressor to condenser
3) Condenser: To condense the refrigerant.
4) Liquid line: To flow the liquid refer giant from condenser to expansion valve.
5) Expansion valve: To expand the refergiant.

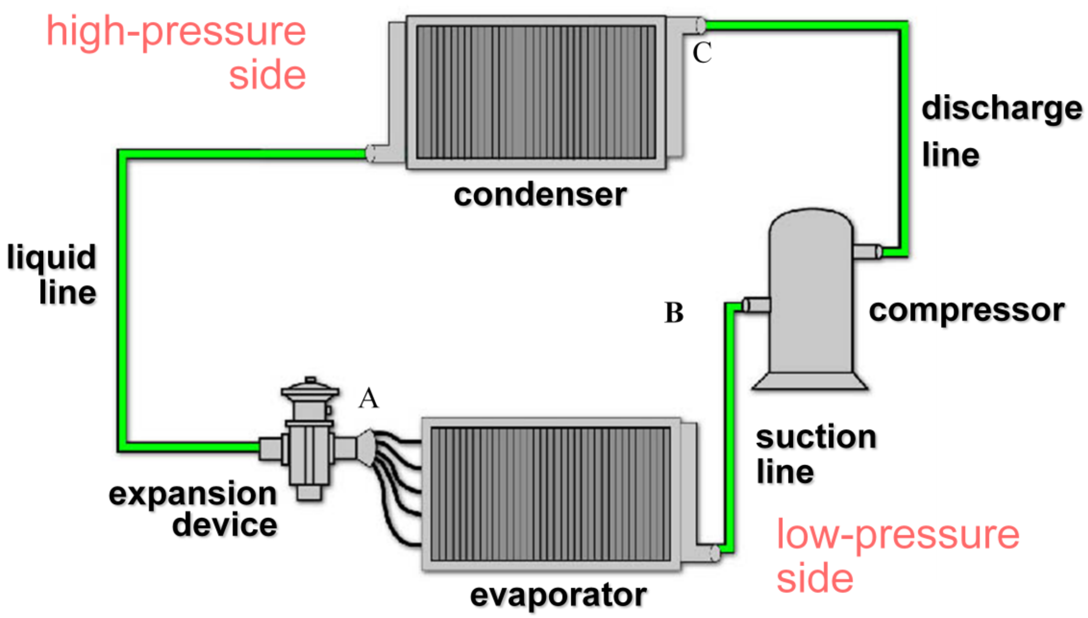


Figure 3.2: Basic components of refrigeration cycle

6) Evaporator: To extracted the heat from the required space.
7) Suction line : To flow the refer giant from evaporator to the compressor

3.2.4 Condenser

A condenser is a heat exchanger is which de superheating of high temperature vapour changes the phase from vapour to liquid and sub-cooling of condensate occurs. However in some cases condensation may not be accompanied with de superheating of vapour or sub cooling of condensate.

The heat exchanger should have the following features:

1) It should be compact, strong and light.
2) It should be easily cleaned and adaptable for replacement of worn out tubes.
3) The manufacture should be quick.
4) The pressure drop should be minimum.

I. *Types of condenser*-Condenser are of three types

1) Air cooled condenser: use the air for the cooling medium.
   a) Natural convection condenser.
   b) Forced-air circulation condenser.
2) Water cooled condenser: use the water for the cooling medium.
   a) Shell and tube condenser.
   b) Shell and coil condenser.

c) Double tube condenser.

3) Evaporative condenser: is the combination of both the above ie. Use the water and air for the cooling.

II. *Selection of a condenser* -The selection of the condenser depends upon the following factors:

1) The capacity of the refrigerating system.
2) The type of refer giant used.
3) The type of cooling medium available.

III. *Factor affecting the condenser capacity*- The condenser capacity is the ability of the condenser to transfer the heat from hot water refer giant to the condensing medium. Following are the factors on which the heat transfer capacity of the condenser depends:

1) Material – higher the ability of a material to transfer the heat, the smaller will be the size of condenser.
2) Amount of contact- the condenser capacity is directly proportional to the amount of contact between the condenser surface and the condensing medium.
3) Temperature difference: with the increase in the temperature difference condenser capacity increase. Normally the temperature difference cannot be controlled.

3.3 SIMULATION OF CONDENSER ASSEMBLY-1

This section discusses the methodology of analysis of the condenser assembly using HyperWorks9.0**.** First a condenser assembly is selected for the required purpose. Following steps are performed for the simulation of the condenser

- Reverse engineering of the condenser assembly and CAD modelling.
- Meshing required for all analysis.
- Dynamic analysis (normal modes and modal frequency response).
- Static analysis and comparison with dynamic analysis.

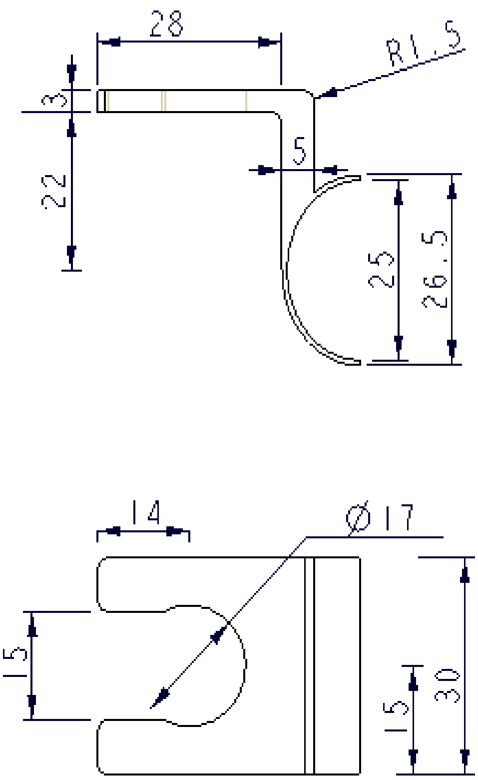
3.4 REVERSE ENGINEERING

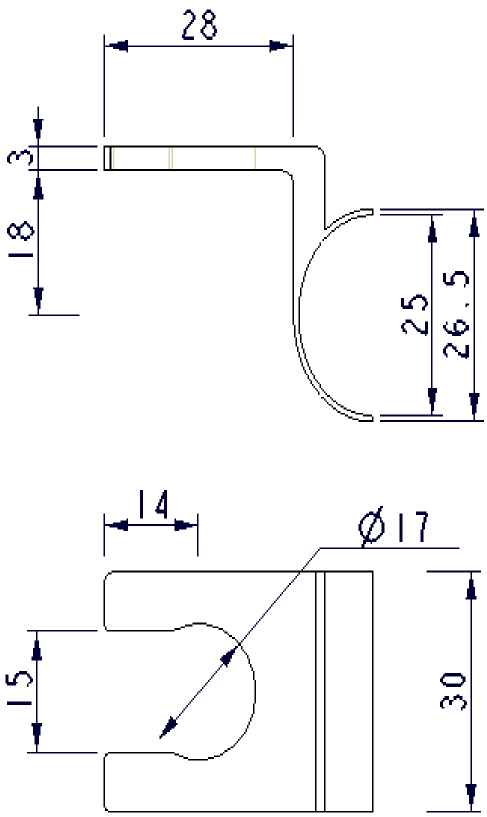
Reverse engineering of the condenser assembly, shown in Fig. 3.3, is done using different measuring device. The detailed drawings are shown in the Fig. 3.4. Condenser assembly has the following components:

1) Small brackets (2).
2) Big brackets (2).

3) Cylinders with Side plates (1).

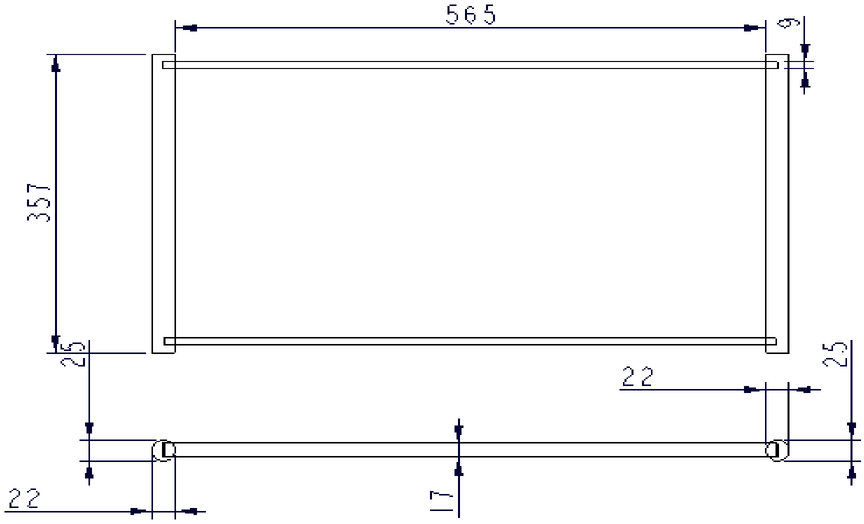
Figure 3.3: Actual image of condenser assembly-1





a) Big bracket drawing

b) Small bracket drawing



c) Cylinders with side plates

Figure 3.4: Drawings of the components of the condenser assembly

CAD Model

From the reverse engineered data all the components are modeled in Pro-E software, further edited in the 'HyperMesh'. The generated CAD model is shown in Fig. 3.5.

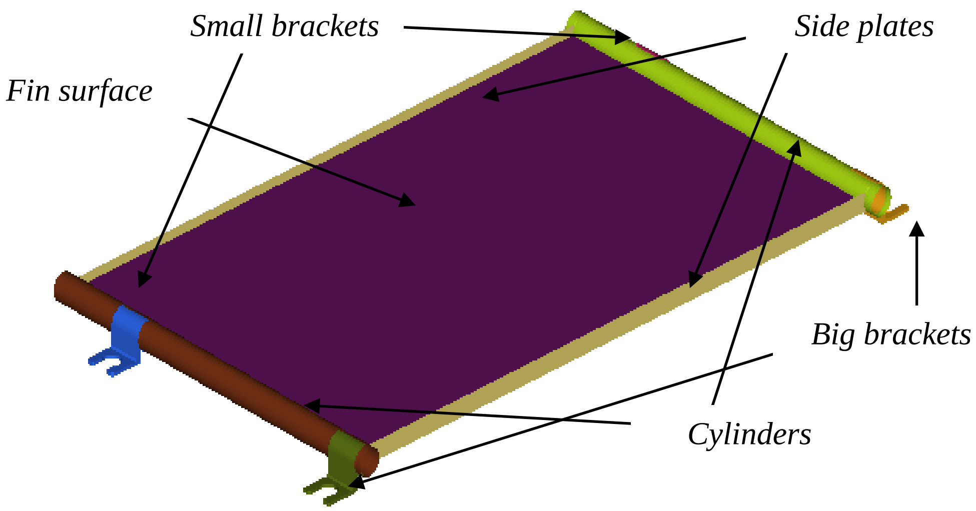


Figure 3.5: CAD model of condenser assembly

3.5 MESHING

Meshing of the condenser assembly is done according to the geometry of the different components and their connectivity with the other components. Elements of the each component of the condenser are assigned in the 'component collector'. The property and loads are assigned to these collectors.

The condenser has the following different components:

1) **Left and Right Cylinders**: They are used for the circulation of the refrigerant from the condenser. The quadrilateral elements are used for mesh generation as shown in the Fig. 3.6.

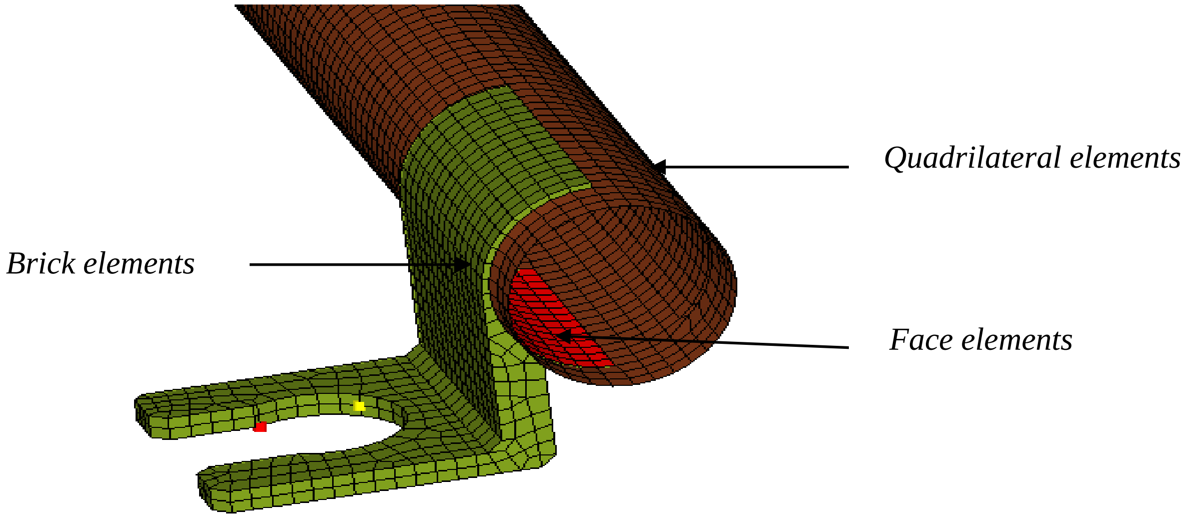


Figure 3.6: Meshing of the left and right cylinders

2) **Side Plates**: Condenser has two side plates which support the fins surface. Quadrilateral elements are used for meshing. As shown in the Fig. 3.7, side plates are merged with the cylinder elements for connectivity.

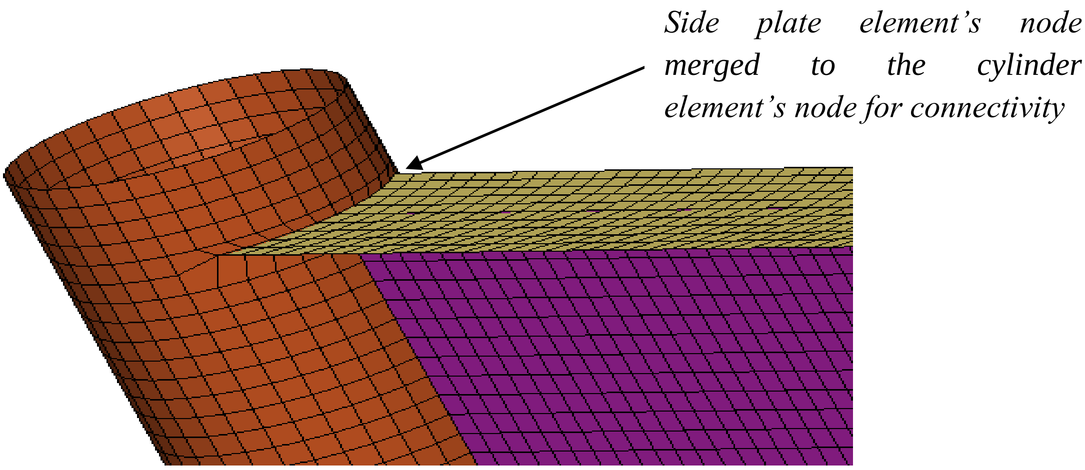


Figure 3.7: Meshing and connectivity of side plate

3) **Fin Surface:** The condenser has the fins in the centre of its body. Quadrilateral elements are used for meshing. Elements are connected to the elements of side plates and two cylinders as shown in the Fig. 3.8.

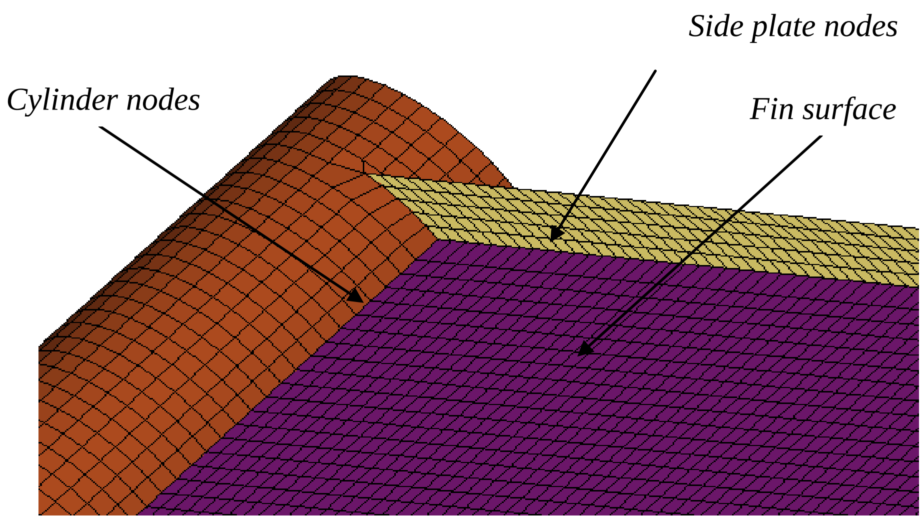


Figure 3.8: Fin surface and its merged nodes with other components

4) **Solid Brackets:** These brackets are used to mount the condenser. Condenser has four brackets which are meshed using brick and pentagonal elements as shown in Fig. 3.6.

3.5.1 Meshing Quality

The quality of mesh is verified using software in-built function. The software symbolises the poor quality with red ink. The quality generated by software is shown in Fig. 3.9.

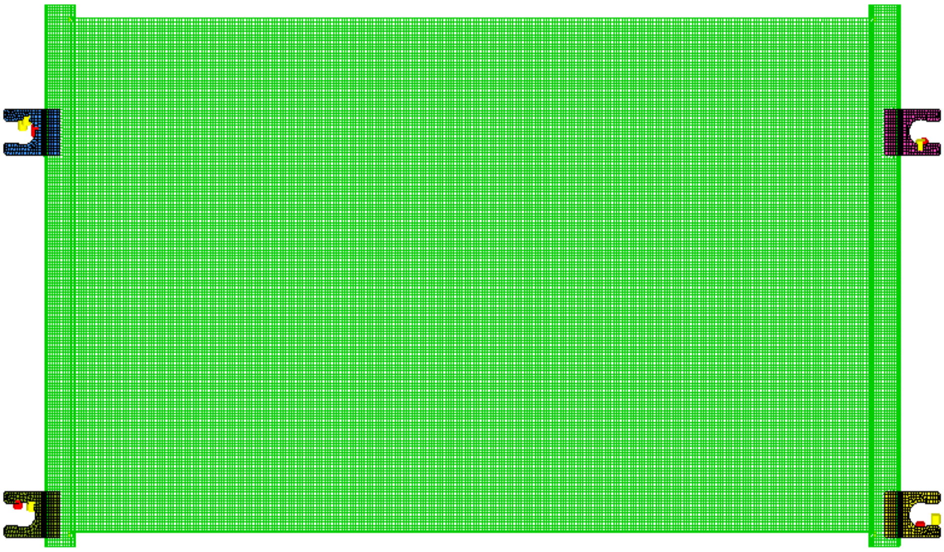


Figure 3.9: Elements quality

The controlled parameters for elements quality are shown in Fig. 3.10.

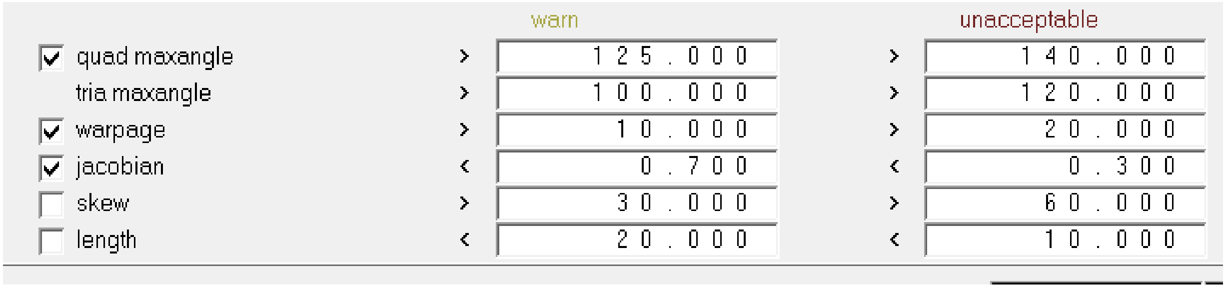


Figure 3.10: Parameters for the elements quality

3.6 DYNAMIC ANALYSIS

Dynamic analysis for the condenser assembly 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 parameter of the second step.

3.7 Normal Modes Analysis

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

I. Requirement to evaluate natural frequency analysis

1) It is the basic design property. For specific components like condenser, it is one of the important design approval criteria.
2) Resonance: It is produced when external frequency become 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

I. 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 no external force is 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.

3.7.1 Normal Mode Analysis of the Condenser Assembly-1

The modeling and meshing of the components is explained in the previous sections. Boundary conditions are applied and results are generated.

Boundary conditions: Collectors are used for each component of the condenser assembly. Meshed components of the condenser are grouped in these collectors. Boundary conditions are assigned to the collectors. The following sub-steps are performed in this step:

1) ***Defining the Material***: To define a specific material, the material Collectors are used in the software. Following two material collectors are used to define the materials.
    a) ALU: Collector in which aluminium properties (Modulus of elasticity, $\mathrm{E} = 6.6\mathrm{e}4$, Modulus of rigidity, $G = 2.7e4$, Density, $\rho = 2.7e - 9$) are assigned.
    b) STEEL: Collector in which properties of the steel are assigned ($\mathrm{E} = 2.1\mathrm{e}5, G = 8.7e4$, $\rho = 7.9e - 9$).
2) ***Property (collector)****:* These collectors contained the properties that are assigned to the components. Five property collectors are made to assign the property to the different component of the condenser assembly.
    a) Hexa element (property collector): The following property are assigned to this collector for solid brackets
    Type of element = 3D
    Card image =PSOLID
    Material = ALUMINIUM
    b) Cylinder (property collector): This property is assigned to the two cylinders.
    Thickness = 4
    Type of element = 2D
    Card image =PSHELL
    Material = ALUMINIUM
    c) Side plate (property collector): This property is assigned to the component side plate.
    Thickness = 5
    Type of element = 2D
    Card image =PSHELL
    Material = ALUMINIUM

d) Mid surface (property collector): This property is assigned to the mid surface collector with thickness 3. Other parameters are same as that of the side plate.

3) Load Collectors: Two load collectors are used in the normal mode analysis. In the load collector a load, constraints etc. and can be applied on the different components of the condenser.

a) Eigrl (load collector): In this collector EIGRL card image (real eigenvalue extraction data lanczos method. Represented in Hyper Mesh as a specific load collector) is used for the calculation of the number of modes and natural frequencies

b) Constraints (load collector): In this collector all bottom nodes of the four brackets are fixed in all directions, shown in Fig. 3.11.

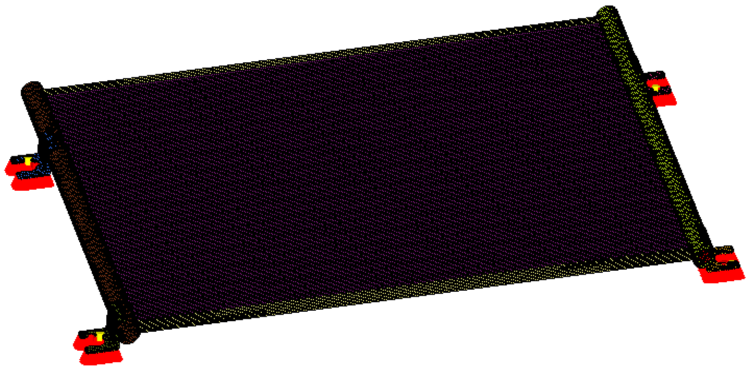
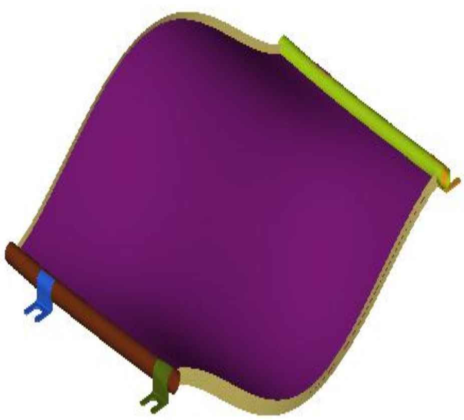


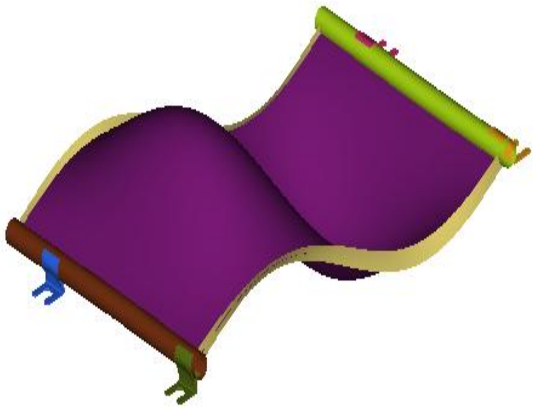
Figure 3.11: Constraint nodes (shown in red colour)

Normal modes analysis is done for five natural frequencies and mode shapes. The results obtained from analysis are shown in Fig. 3.12.

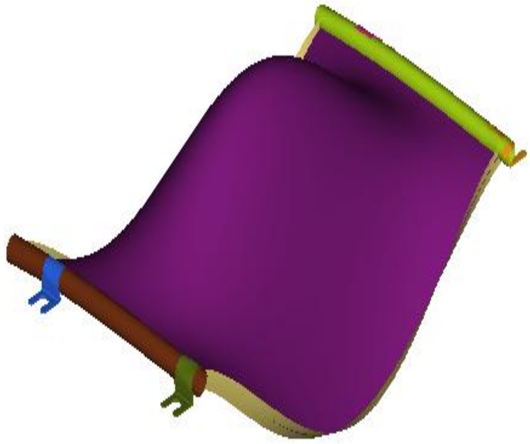
a) Mode1
$Frequency = 70.97Hz$



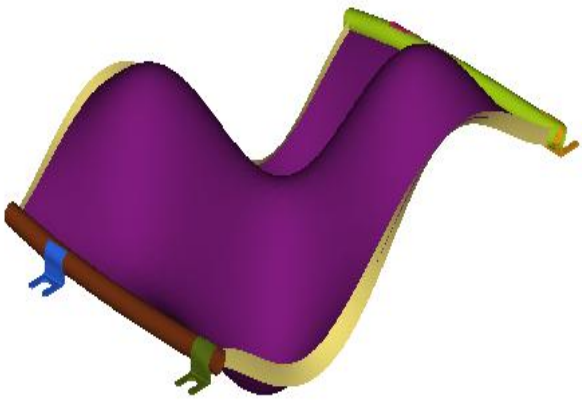
b) *Mode 2*
$Frequency = 129.21Hz$



c) *Mode 3*
$Frequency = 150.94Hz$



d) *Mode 4*
$Frequency = 195.14Hz$



e) *Mode 5*
$Frequency = 295.70Hz$

Figure 3.12: Natural frequencies and mode shapes

3.8 MODAL FREQUENCY RESPONSE ANALYSIS OF CONDENSER ASSEMBLY-1

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 condenser along all three axes are done. CAD data and meshing has been explained in the previous sections and is same for all analysis of condenser assembly-1.

1) Modal frequency response analysis along x-axis

In this analysis, force is applied in the x-axis direction and condenser is free to move in this direction. Forces are applied according to industry standards [11] for the testing of the automobile mounting equipments.

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

a) Cons (load collector***):*** In this collector nodes of brackets are fixed in all direction, except x direction.

b) Unit-Load: In this collector load type DAREA is used Scale (area) factors for dynamic loads represented in HyperMesh as a constraint load type. This is applied along the x direction and on the entire four brackets bottom nodes. The value of the force is taken as the 27 N on each bracket.

c) Eigrl Load (Collector): In this collector EIGRL card image is used, for modes are to be calculated in the frequency range from 20 to 1000 Hz..

d) Tabled 1(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. 3.13.

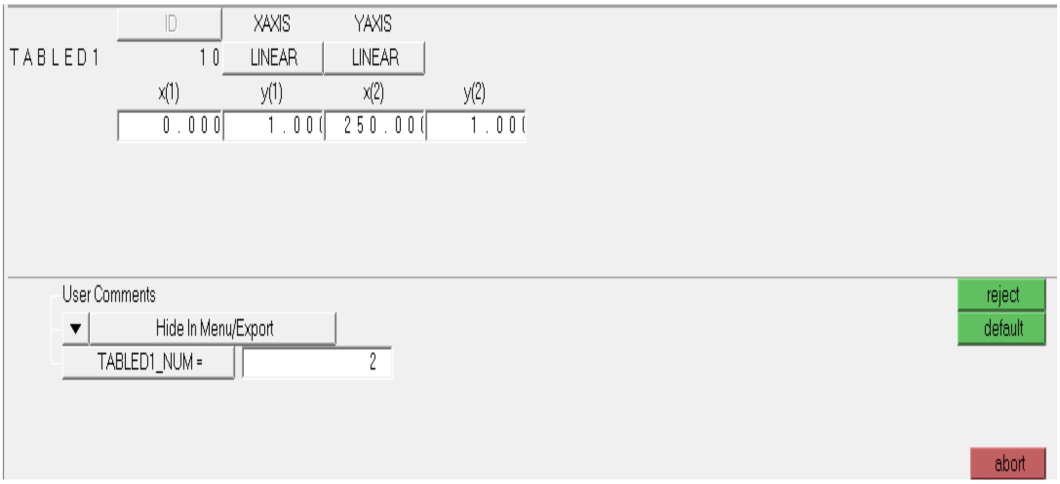


Figure 3.13: Tabular functions

e) Rload 2(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 parameter is given by editing this collector are-

Initial excitation frequency = 20

Increment for the next frequency =20

Number of frequencies = 49

The results obtained, from the analysis, are shown in the Fig. 3.14. Stresses (in MPa) produced in x-axis due to dynamic loading with excitation frequency of 60 Hz, and are very low. Mounting brackets has the highest values of the stress as compared to the other components.

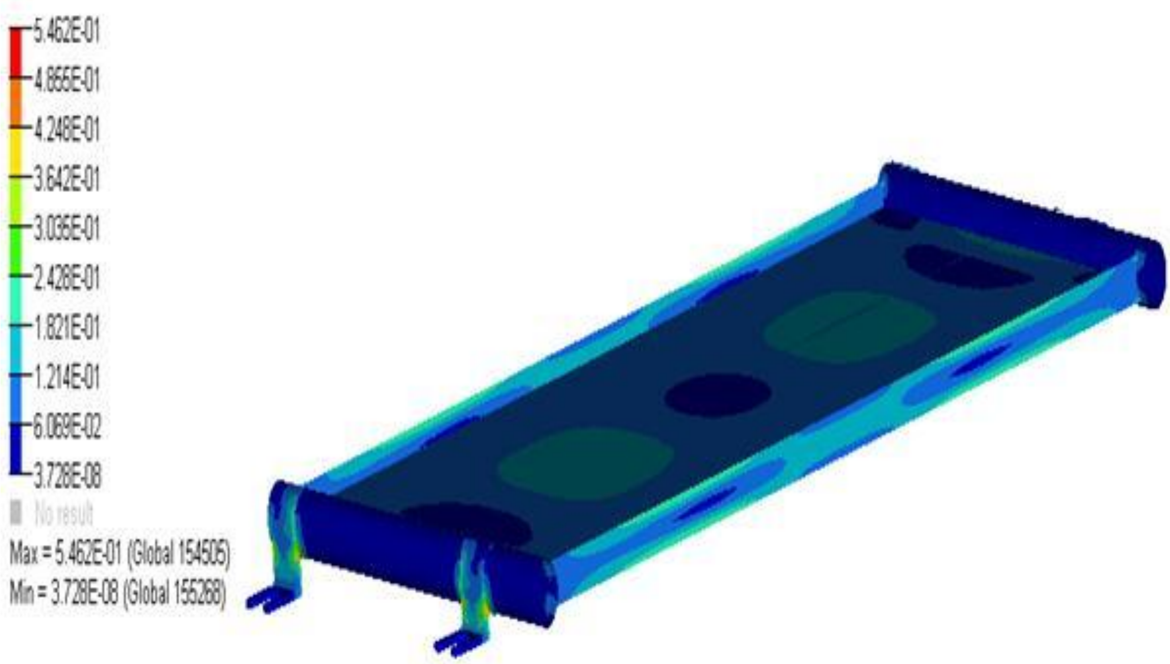


Figure 3.14: Stresses in condenser along x-axis

2) Modal frequency response analysis along y-axis

In this analysis force is applied in the y-axis direction and condenser 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 these collector bottom nodes of 'brackets component collector are constrained in all direction, except y direction.

b) Unit-Load: Dynamic loads, is applied along the y direction and on ‘brackets’ bottom nodes of magnitude 27 N.

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. 3.15. The stresses produced in condenser due to the dynamic load in y-axis with excitation frequency of 60 Hz are very low.

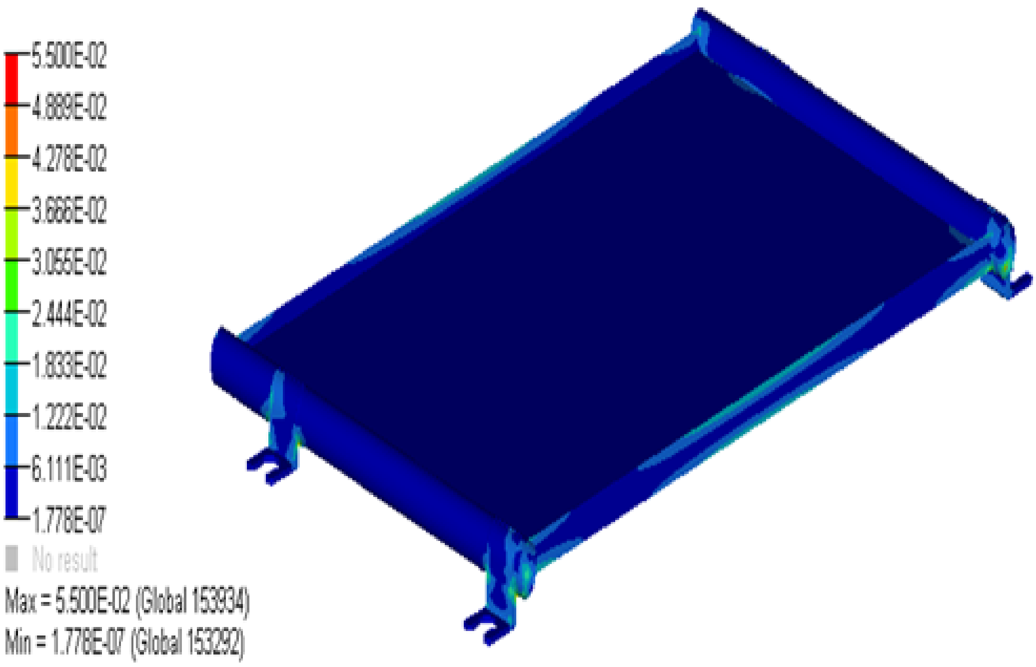


Figure 3.15: Stresses in condenser along y-axis

3) Modal frequency response analysis along z -axis

In this analysis force is applied in the z-axis direction and condenser 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 these collector bottom nodes of brackets are constrained in all direction, except z direction.
b) Unit-Load: A dynamic load is applied along the z direction and on ‘brackets’ bottom nodes of magnitude 27 N.

The remaining four loads are applied in the same way as discussed in the previous analysis.

Result obtained, the stresses produced in condenser due to the dynamic load in z-axis with excitation frequency of 60 Hz is shown in Fig. 3.16. It can be seen from Fig. 3.17 that the maximum values of the stresses is produced on the brackets.

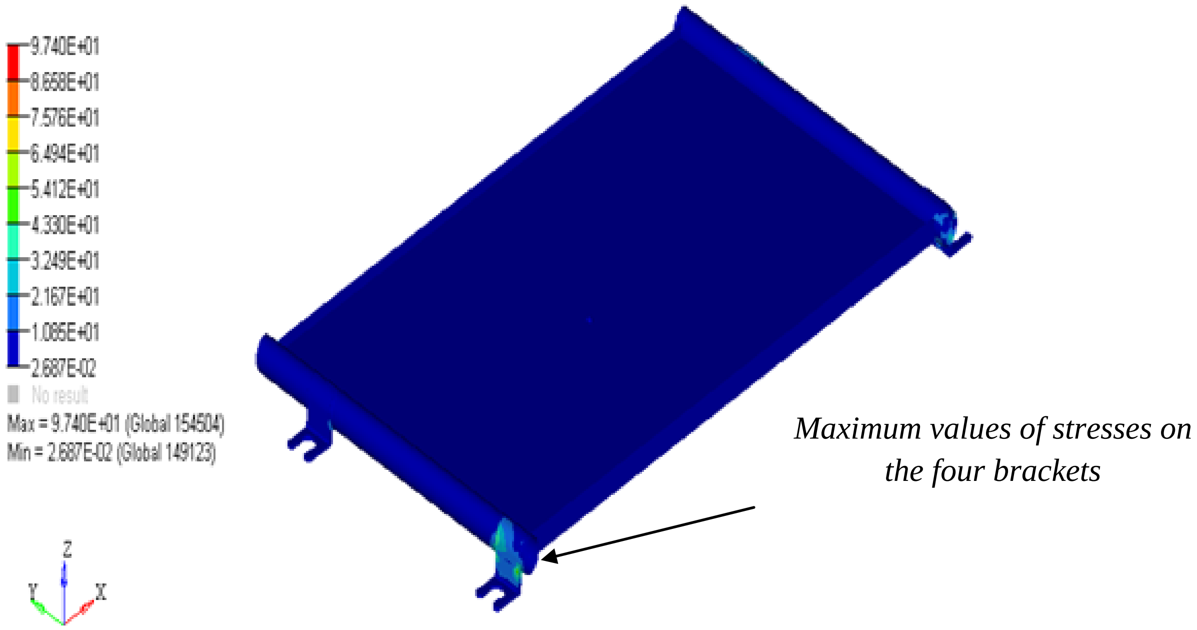


Figure 3.16: Stresses in condenser

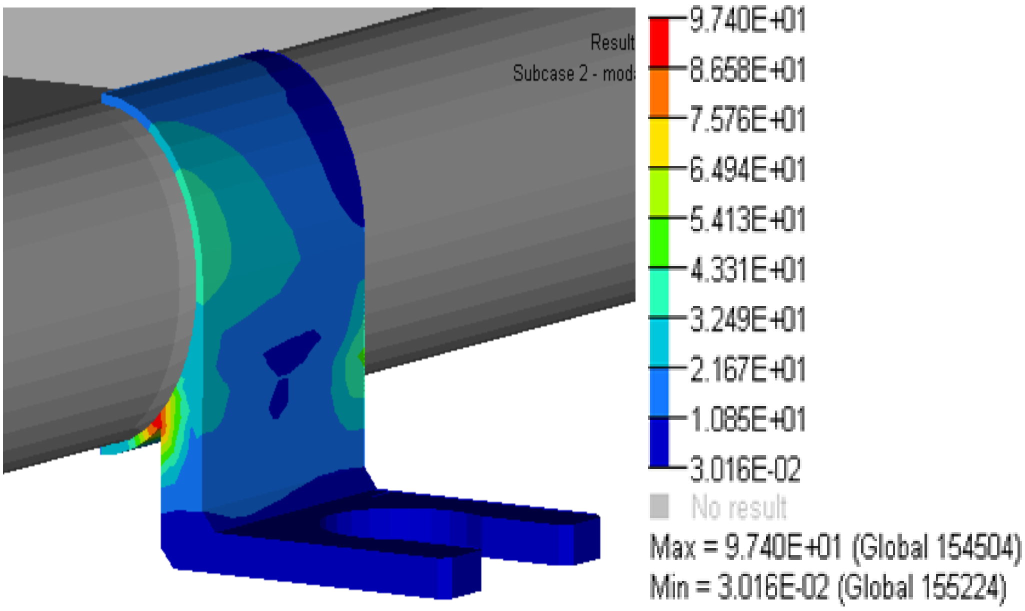


Figure 3.17: Stresses in bracket

Frequency stress plot is shown in Fig. 3.18, which represent the modal frequency response of the condenser assembly. Peak stresses seen in the plot represents the resonance in the component.

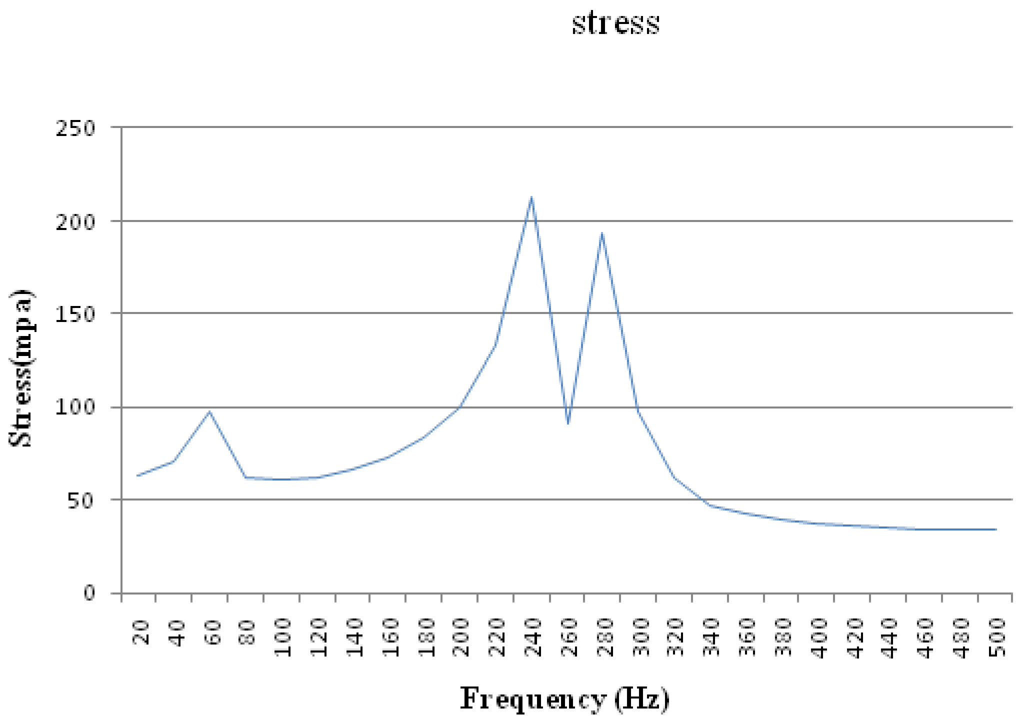


Figure 3.18: Stress vs. Frequency plot

3.9 LINEAR STATIC ANALYSIS OF THE CONDENSER ASSEMBLY-1

Linear statics analysis is done on the condenser to analyze the stress as generated under the same magnitude of load and the boundary conditions used in dynamic analysis. The linear statics analysis is done only in the z-axis direction, because the highest values of stress in previous analysis are produced in z-axis direction. In boundary conditions, only load collectors are different, but other boundary conditions (material and property collector) are applied in the same way as in modal frequency response analysis.

Load Collectors: Two load collectors are used in the liner static analysis.

a) Load (load collector): In this load collector, 27 N forces are applied on the each bracket (equally divided on each node of the bracket) along z-axis direction.
b) Constraints (load collector): In this collector all bottom nodes of the four brackets are fixed in all directions, except z-axis direction as shown in Fig. 3.19.

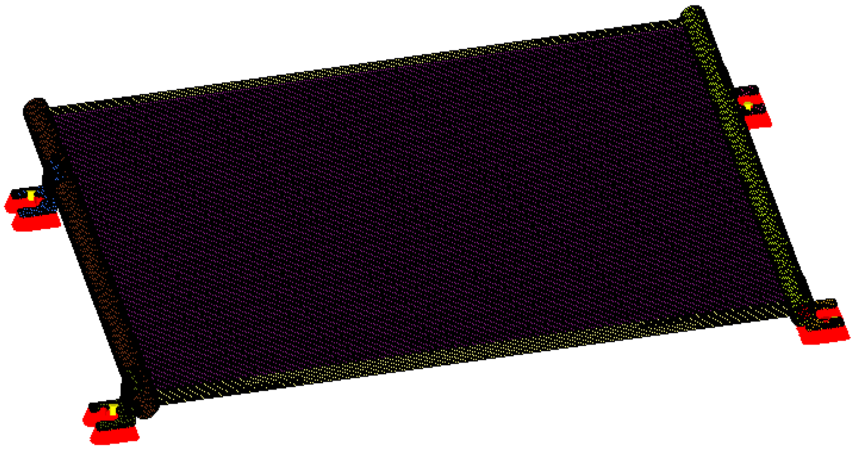


Figure 3.19: Constrained nodes in static analysis (shown in red colour)

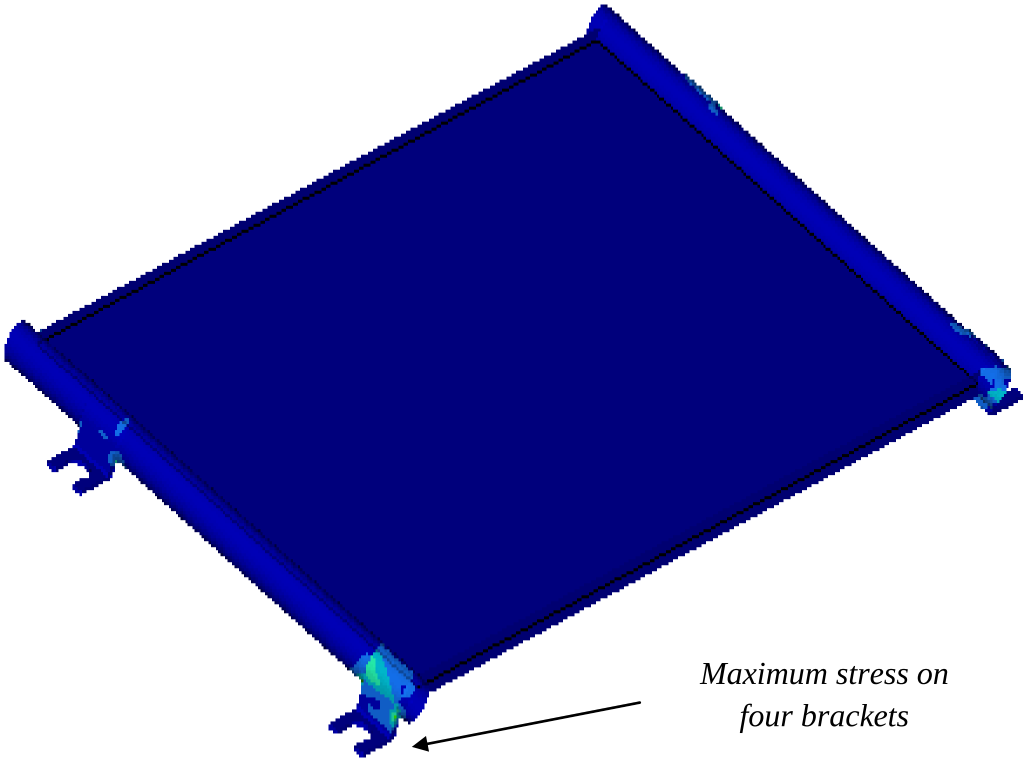


Figure 3.20: Stresses in condenser due to static load

The stresses produced, due to the static load in the condenser assembly-1, shown in Fig. 3.20 are lower than the stresses produced due to dynamic load. It can be seen from the Fig. 3.21., that the maximum stresses are produced in the brackets.

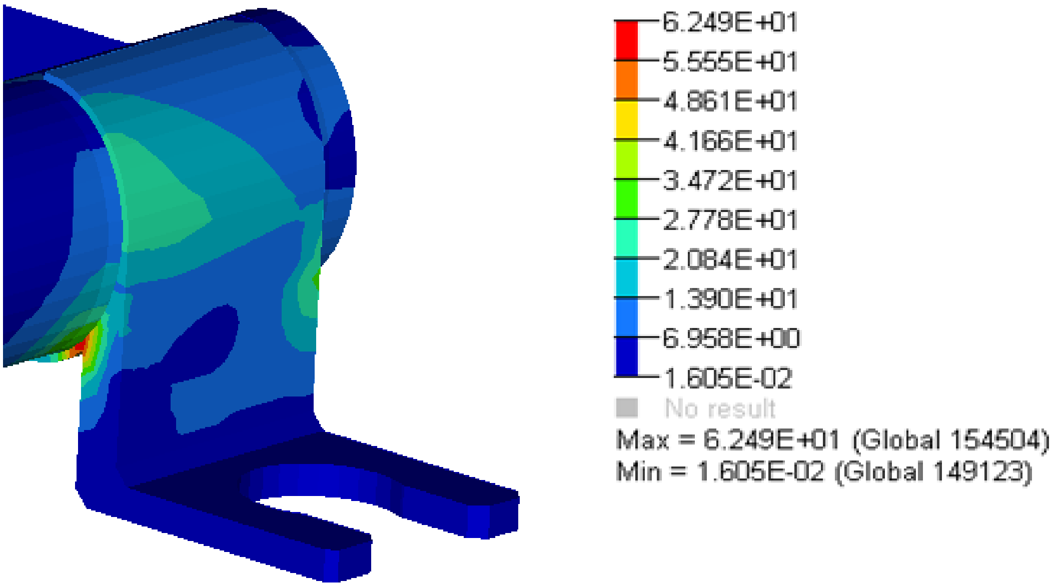


Figure 3.21: Stresses in bracket

CHAPTER 4
ANALYSIS OF CONDENSER ASSEMBLY-2

As explained in the previous chapter same methodology is used for the condenser assembly-2. The analysis is further extended by optimizing this assembly. Following steps are performed:

1) CAD data of the model.
2) Meshing required for all analysis.
3) Dynamic analysis (normal modes and modal frequency response).
4) Static analysis.
5) Optimization.

4.1 CAD DATA

CAD data of this condenser assembly-2 is in the IGES format. This IGES format is imported into the HyperWork for the simulation.

4.2 MESHING

Condenser assembly-2 has many different components which are assembled, meshing is done according to their geometry and their connectivity with other components. Elements of the each component of the condenser are assigned in the 'component collector'. The property and loads are assigned to these collectors.

1) **Solid Brackets –** These two brackets are meshed with brick elements as shown in the Fig. 4.1. Symmetrical meshing is done around the hole for better elements quality.

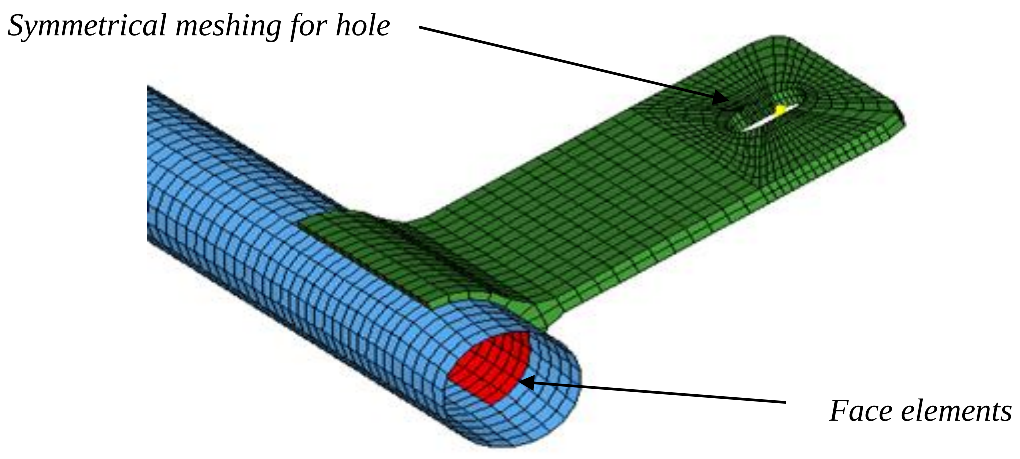


Figure 4.1: Solid bracket meshing

2) **Right and Left Cylinder (Dia 20)** –These cylinders are meshed with quadrilateral elements. At the both ends, element nodes are merged with side plate element nodes for the connectivity with the side plates, shown in the Fig. 4.2. Connectivity of right cylinder (dia20) with the lower thin bracket is done with elements which are common to both cylinder and bracket at the surfaces as shown in the Fig. 4.3

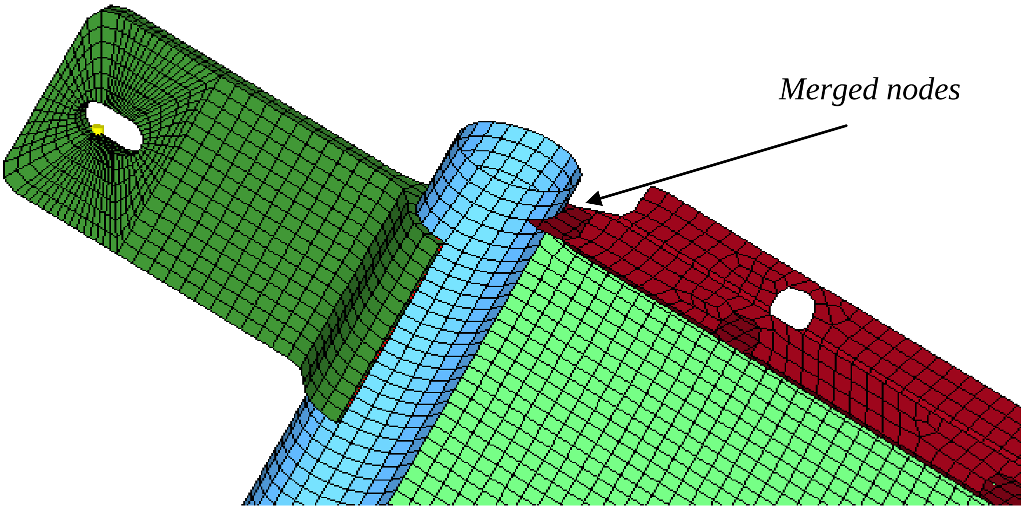


Figure 4.2: Left cylinder meshing

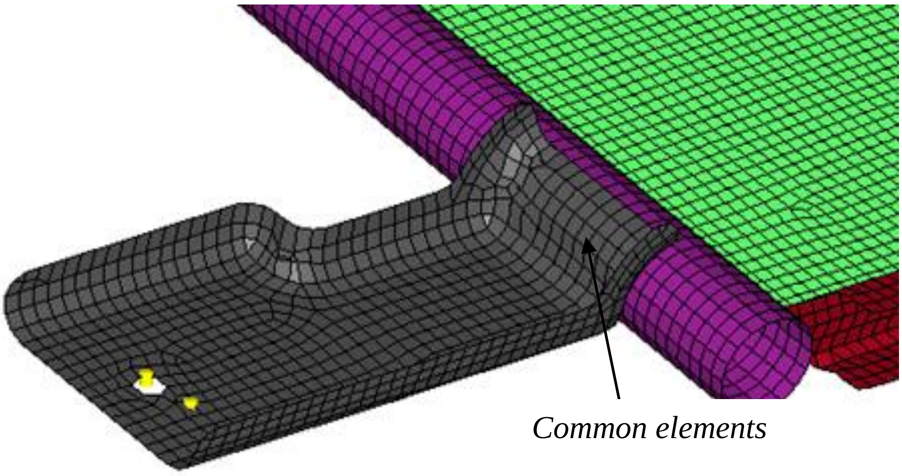


Figure 4.3: Right cylinder meshing

3) **Side Plates –** Both plates are meshed with the quadrilateral elements.

4) **Cylinder (Dia 32)** – This cylinder is meshed with quadrilateral elements. The cylinder (dia 32) is jointed with left cylinder (dia 20). These joints are made using quadrilateral elements as shown in Fig. 4.4.

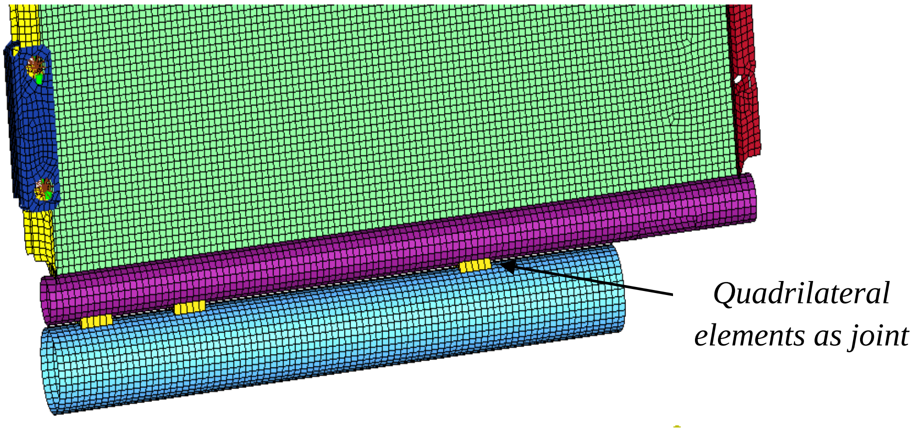


Figure 4.4: Meshing and connectivity of cylinder (dia 32)

5) **Lower Thin Bracket –**Thin bracket is meshed with quadrilateral elements and connected to left cylinder (dia 20) by some common elements as shown in the Fig. 4.3.
6) **Upper Thin Bracket –** This bracket is meshed with the quadrilateral elements and triangular elements. The one end of this bracket is joined with the side plate and the mounting bracket as shown in the Fig. 4.5.

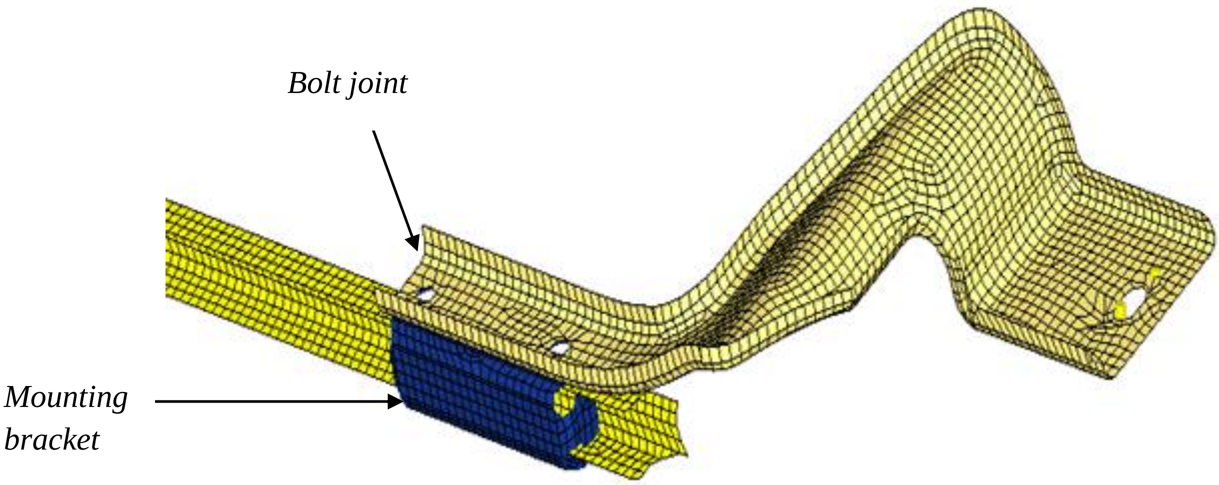


Figure 4.5: Meshing and connectivity of upper thin bracket

7) **Mounting Bracket** – Mounting bracket is meshed with quadrilateral element. This bracket is used to make the joint between side plate and the lower thin bracket as shown in the Fig. 4.5.
8) **Fin Surface –** The condenser has the fins in the centre of its body. Quadrilateral elements are used for its meshing. It is connected to the side plates at its top and bottom edges and cylinders (dia 20) at its left and right edges as shown in the Fig. 4.6.

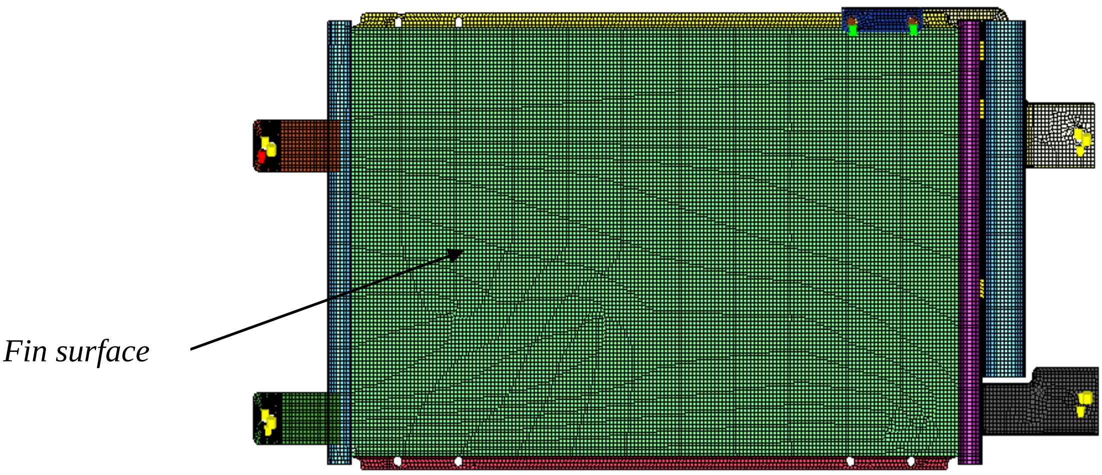


Figure 4.6: Fin surface and meshed model

4.2.1 Quality Index

The quality of mesh is verified using software in-built function. The software symbolises the poor quality with red ink. The quality generated by software is shown in Fig. 4.7.

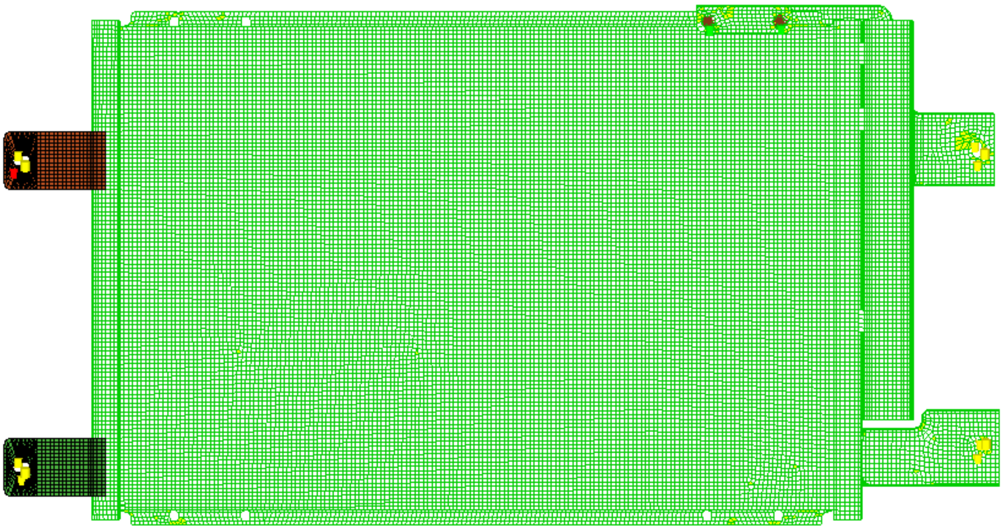


Figure 4.7: Elements quality for condenser assembly-2

4.3 DYNAMIC ANALYSIS

Dynamic analysis for the condenser assembly-2 is done which is divided in two steps:

1) Normal modes analysis
2) Modal frequency response analysis

4.4 NORMAL MODES ANALYSIS OF THE CONDENSER ASSEMBLY-2

CAD data and meshing of all the components of condenser assembly-2 are explained in the previous sections. The boundary conditions are applied and results are generated.

Boundary Conditions

Collectors are used for each component of the condenser assembly. Boundary conditions are assigned to the collectors. The following sub-steps are performed in this step-

1) ***Defining the Material***- Following two material collectors are used to define the materials.
   a) ALU: Collector in which aluminium properties (Modulus of elasticity, $\mathrm{E} = 6.6\mathrm{e}4$, Modulus of rigidity, $G = 2.7e4$, Density, $\rho = 2.7e-9$) are assigned.
   b) STEEL: Collector in which properties of the steel are assigned ($\mathrm{E} = 2.1\mathrm{e}5, G = 8.7e4$, $\rho = 7.9e-9$).
2) ***Property collector***-These collectors contained the properties that are assigned to the components. Following parameters are given to create a property collector
   a) DIA-20 (property collector): The following property is assigned to this collector for both cylinders.
      Thickness = 7
      Type of element = 2D
      Card image = PSHELL
      Material = ALUMINIUM
   b) CONNECTE-AREA (property collector): This property is assigned to the components 'connected-area' with thickness 5. This component collector has the elements which are used to join the component right cylinder (dia20) and cylinder (dia 32). This property has the same parameters as property collector dia-20.
   c) SIDE PLATE (property collector): This property is assigned to the components side plates with the element thickness 5 and other parameters are the same as property collector 'dia-20'.
   d) MOUNT-BKT (property collector): This property is assigned to the 'mounting bracket' collector with element thickness 5 and other parameters are same.
   e) SMALL-THIN-BKT (property collector)-This property is assigned to the component 'right lower thin bracket' with the thickness 5.

f) BIG-THIN-BKT (property collector) - This property collector is assigned to the component ‘upper thin bracket’ with the thickness 5 and following parameters-
   Type of element = 2D
   Card image = PSHELL
   Material = STEEL.
g) HEXA-PROP (property collector)-This property is assigned to the components solid bracket.
   Type of element = 3D
   Card image = PSOLID
   Material = ALUMINIUM
h) MID-PROP (property collector)-this property is assigned to the components fin surface and cylinder (dia-20) with thickness 3.
   Type of element = 2D
   Card image = PSHEL
   Material = ALUMINIUM

After assigning the material and property (collector) a normal modes analysis is done (as discussed in previous chapter) by fixing the bottom nodes of the bracket as shown in the Fig. 4.8

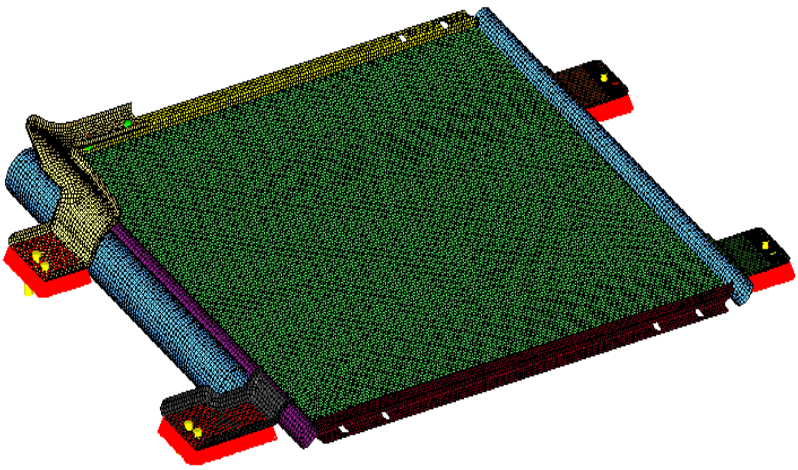
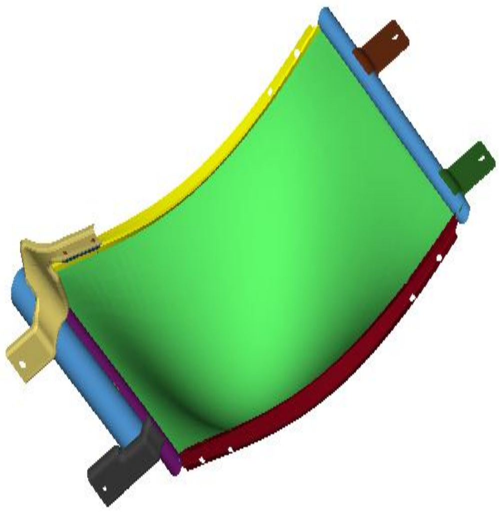


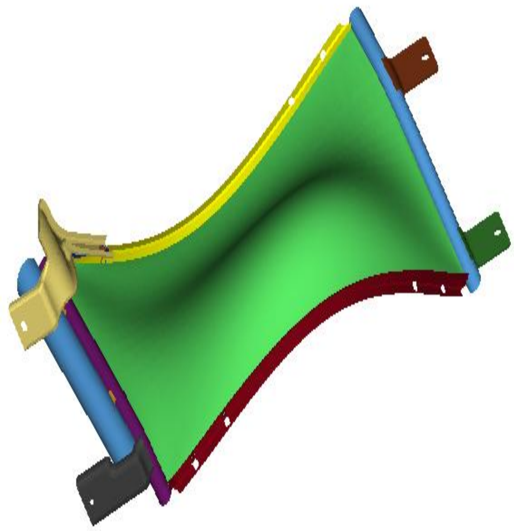
Figure 4.8: Constrained nodes (shown in red colour)

The results obtained after analysis are natural frequencies and mode shapes shown in Fig. 4.9.



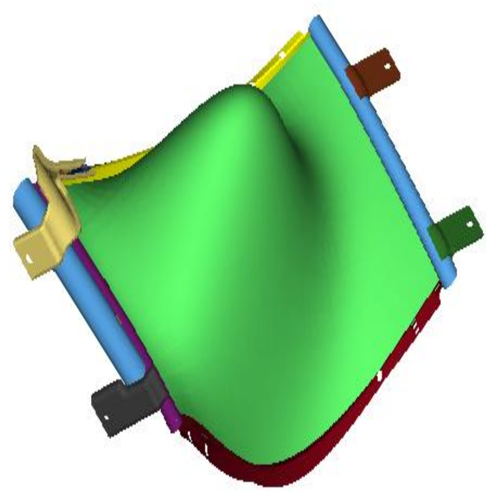
(a) Mode 1

$Frequency = 104.34Hz$



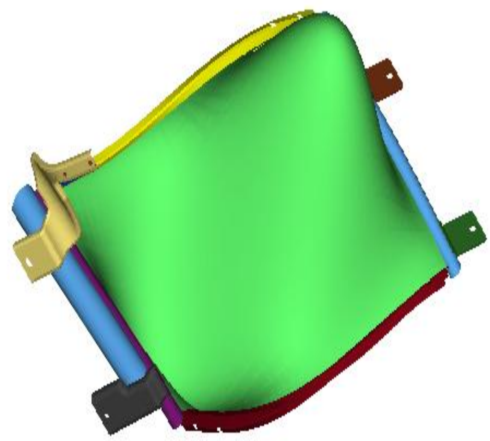
(b) Mode 2

$Frequency = 179.62Hz$



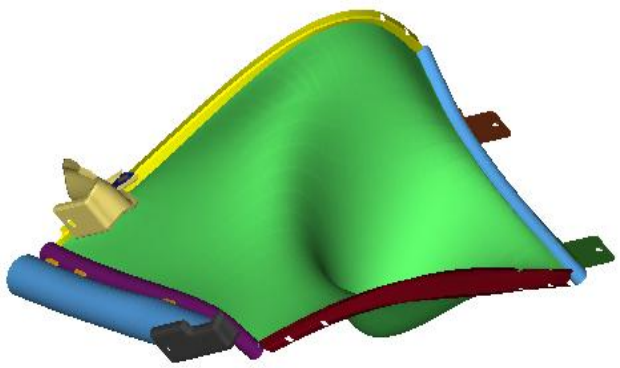
(c) Mode 3

$Frequency = 219.98Hz$



(d) Mode 4

$Frequency = 228.86Hz$



Mode 5

$Frequency = 300.20Hz$

Figure 4.9: Natural frequencies and mode shapes.

4.5 MODAL FREQUENCY RESPONSE ANALYSIS OF CONDENSER ASSEMBLY-2

After assigning the material and the property collector as explained in the previous analysis, the modal frequency response analysis are done in the same way as that modal frequency response analyses for the condenser assembly-1. As explained in the previous chapter, force is applied in all the three direction and three analyses are done according to the load applied. The results obtained from the analyses are discussed below.

4.5.1 Modal Frequency Response Analyses along x-axis

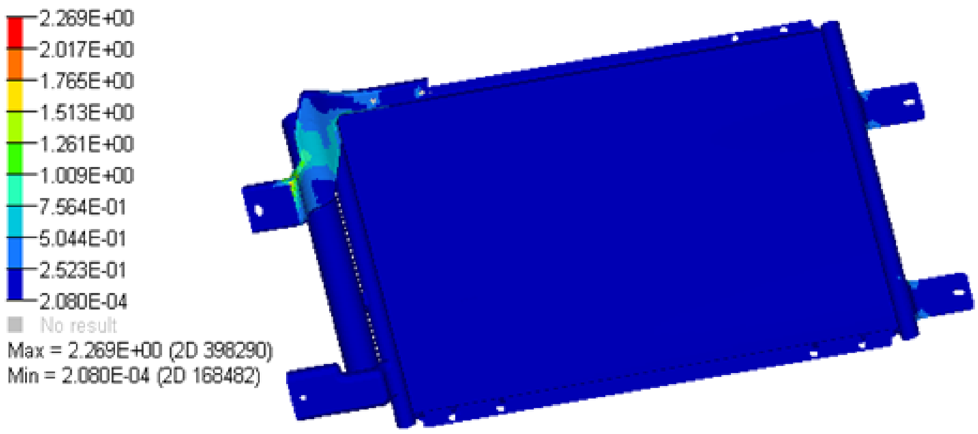


Figure 4.10: Stresses in condenser along x-axis

The results obtained after the analysis are shown in the Fig. 4.10. It can be seen that the stresses (in MPa) produced in x-axis due to dynamic loading with excitation frequency of

60 Hz, are very low. Brackets have the highest values of the stress as compared to the other components.

4.5.2 Modal Frequency Response Analyses along y-axis

Results obtained from the analysis are shown in the Fig. 4.11.

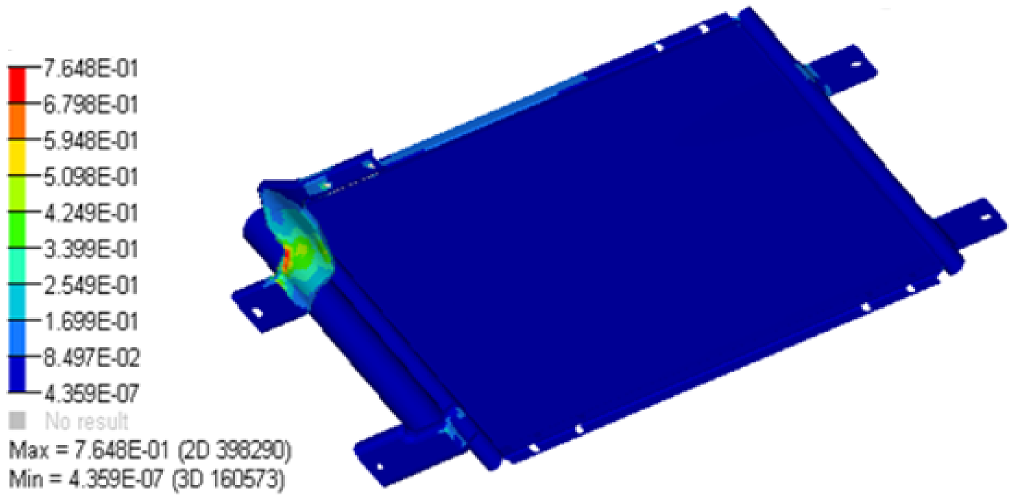


Figure 4.11: Stresses in condenser along y-axis

It can be seen that the stresses produced in condenser due to the dynamic load in y-axis with excitation frequency of 60 Hz are also very low.

4.5.3 Modal Frequency Response Analyses along z-axis

After the analysis, stresses produced in condenser are shown in the Fig. 4.12. It can be seen from Fig. 4.13 that the stresses are produced at the bracket.

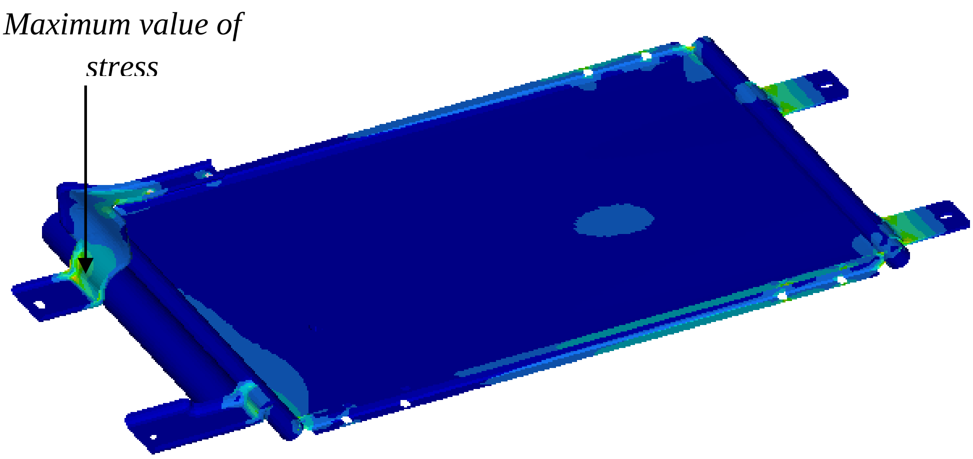


Figure 4.12: Stresses in condenser

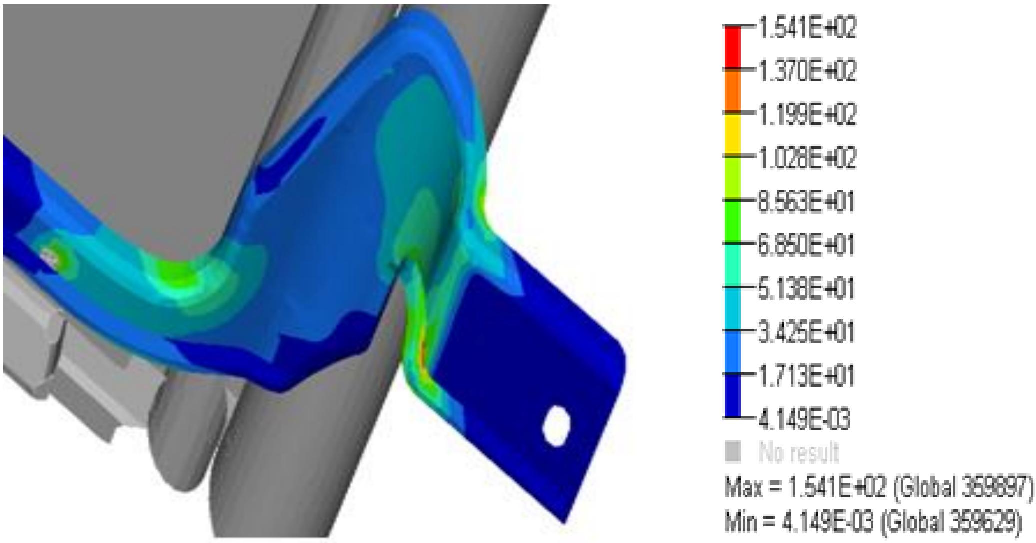


Figure 4.13: Highly stress concentration component in condenser

It can be seen from Fig. 4.14, that the stress values are highly increased when the excitation frequency match with the natural frequency.

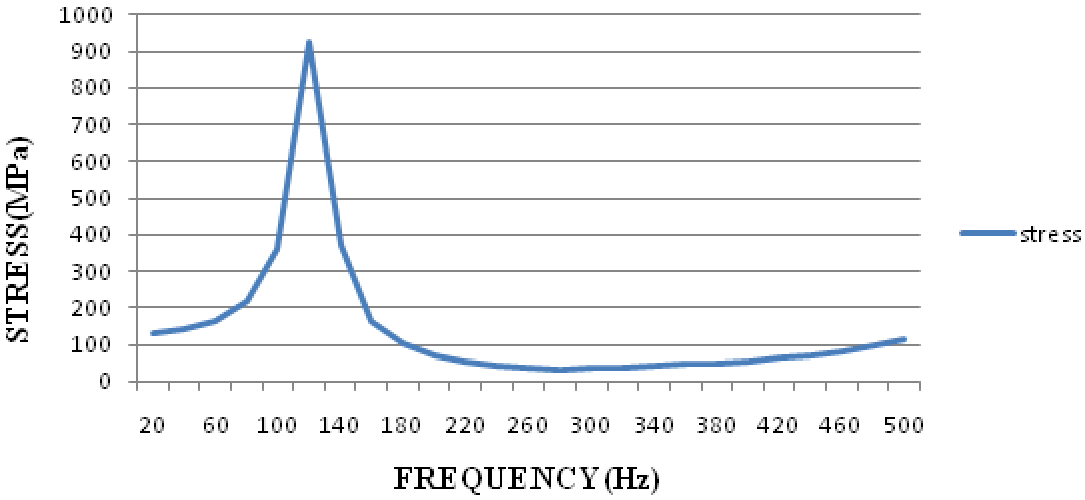


Figure 4.14: Plot of stresses vs frequency

4.6 LINEAR STATIC ANALYSIS OF THE CONDENSER ASSEMBLY-2

Linear statics analysis is done for the condenser assembly-2 in the same way as for condenser assembly-1. Results obtained from the analysis are shown in the Fig 4.15. It can be seen that the stresses are lower than the stresses due dynamic load.

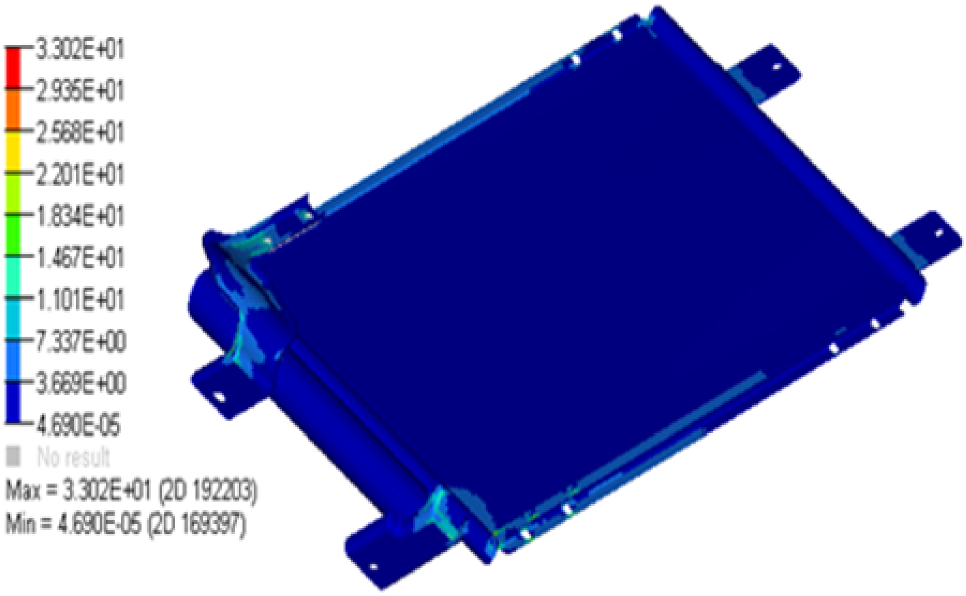


Figure 4.15: Stresses in condenser due to static load

4.7 OPTIMIZATION

The big and small thin brackets are taken for optimizations. Almost 2% mass is reduced from the condenser assembly.

a) **Big Thin Bracket**:

In this bracket stress values is very low at curvature surface as compared to its other surface in Fig 4.13. Big thin bracket before optimization is shown in the Fig 4.16. Material is removed where the stresses are low and the resultant component is shown in Fig 4.17.

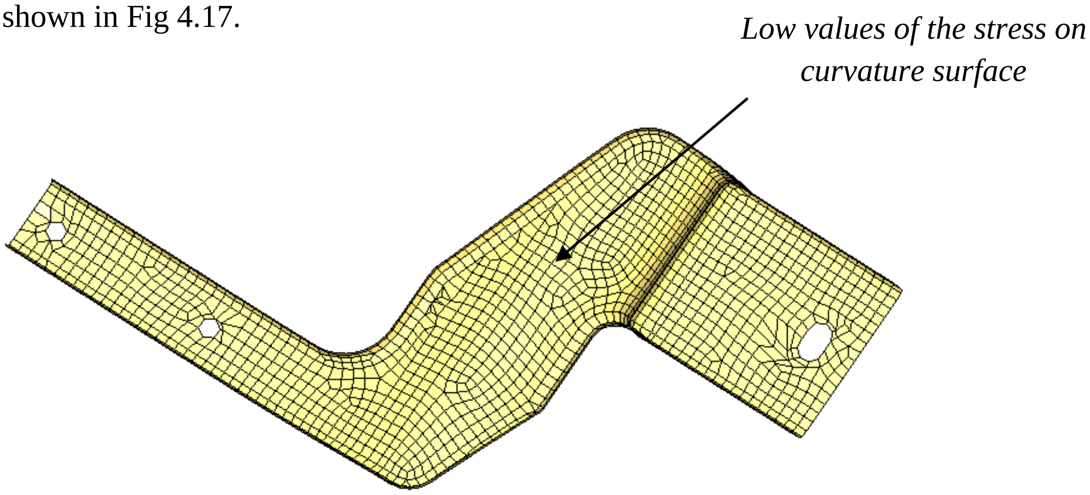


Figure 4.16: Before Optimization

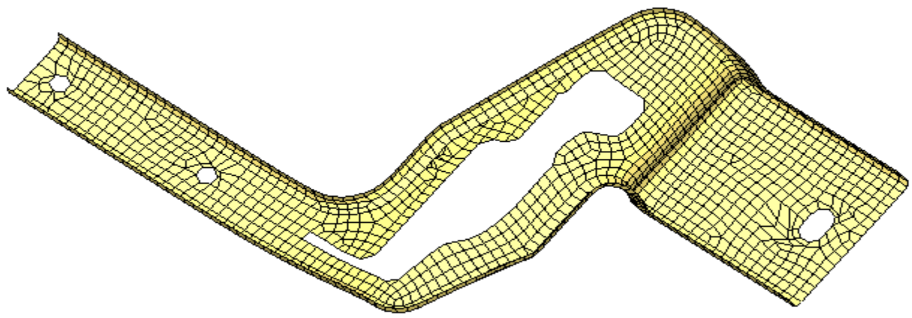


Figure 4.17: After Optimization

b) Small Thin Bracket

In this bracket stress values are very low on surface shown in Fig. 4.18, as compared to the other surface. Material is removed where the stresses are low and and the resultant bracket is shown in Fig. 4.19.

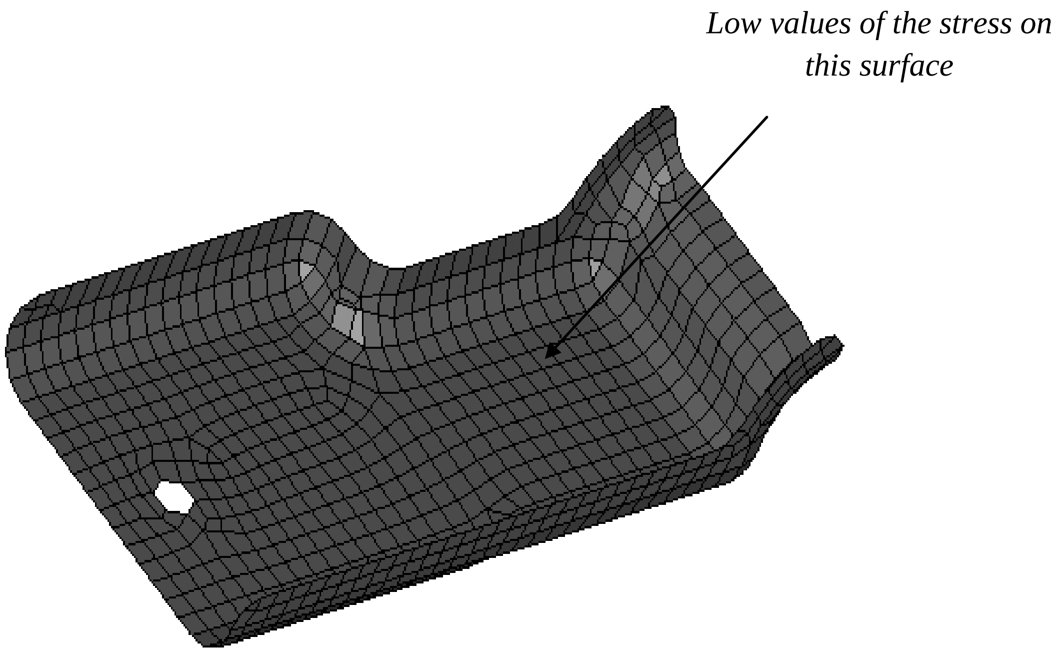


Figure 4.18: Before Optimization

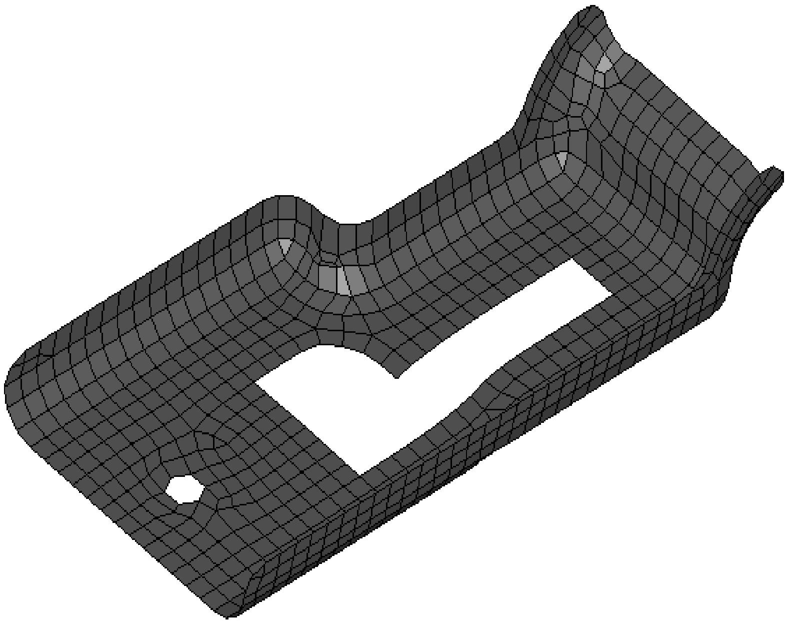


Figure 4.19: After optimization

4.7.1 Modal Frequency Response Analysis of Optimize Condenser Assembly-2

As explained in the analysis, high values stresses are produced when force is applied in z-axis direction, in other (x-axis, y-axis) direction stress value is very low. So to analyze the maximum values of stresses in the optimize condenser assembly-2, modal frequency response analysis is done along in z-axis direction. The analysis of the optimized condenser assembly-2 is done as explained in the previous section. After analysis stresses produced are shown in Fig. 4.20.

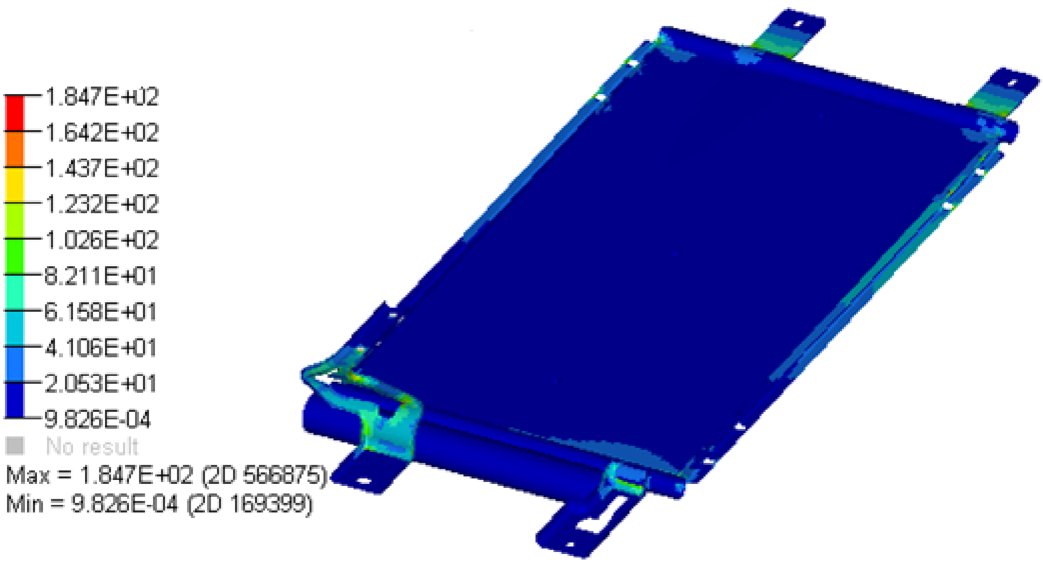


Figure 4.20: Stresses produced in optimize condenser assembly-2

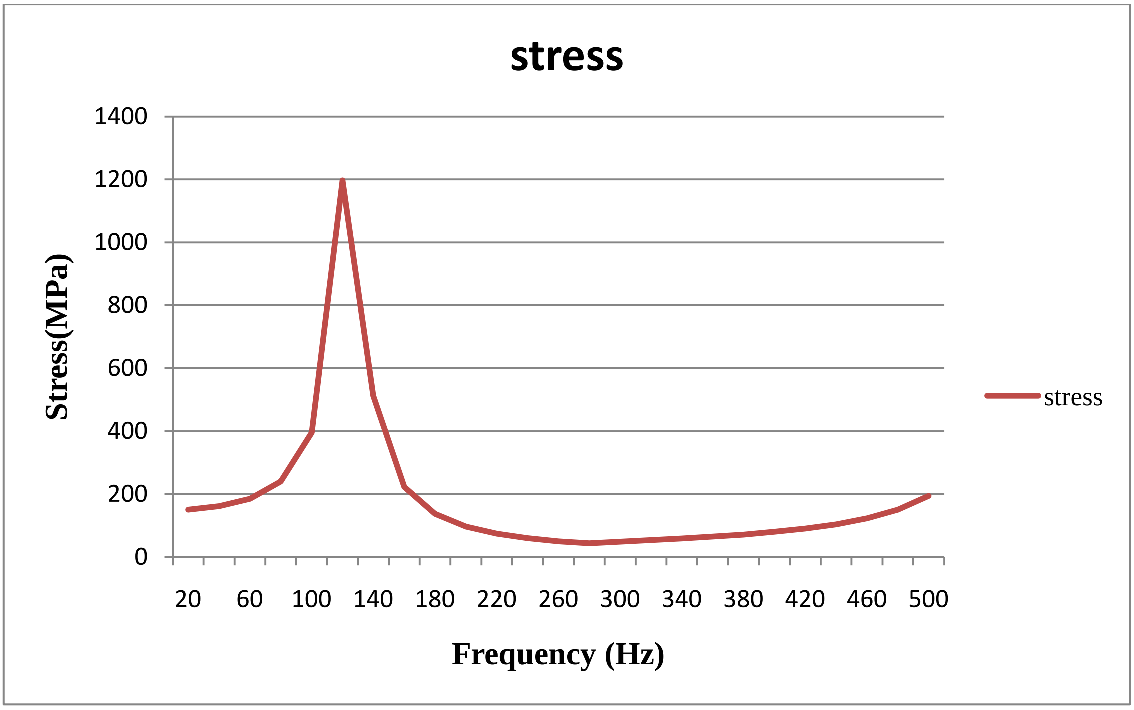


Figure 4.21: Stresses vs Frequency plot

It can be seen from Fig. 4.20 that the stress value is below the yield stress value, so the optimized design is safe. It can also be seen from Fig. 4.21 that the resonance is occurring above the frequency of 60 Hz.

CHAPTER 5

CONCLUSION AND SCOPE FOR FUTURE WORK

5.1 CONCLUSION

Dynamic analysis is done on an automobile condenser assembly using the standard testing conditions. The results obtained revealed that the high values of stresses are produced in the z direction on the mounting brackets of condenser assembly. Thereafter optimization is done for brackets reducing the mass of the same by about 2%.

It is also seen that the stresses increases 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.

5.2 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) This methodology can be practically validated on the Electrodynamic shaker.

c) For the condenser assembly modal transient response analysis can be done to see the effect of vibration (resonance) with time.

REFERENCES

[1] Cameron C. J., Wennhage P. and Goransson P., (2010), "Prediction of NVH behaviour of trimmed body components in the frequency range 100–500 Hz", *Applied Acoustics,* vol. 71, pp. 708–721.

[2] Douville H., Masson P. and Berry A., (2006), "On-resonance transmissibility methodology for quantifying the structure-borne road noise of an automotive suspension assembly", *Applied Acoustics,* vol. 67, pp. 358–382.

[3] Jong C. Y., Sang-Hyun J. and Park J.K., (2002), "Panel Contribution Analysis By using Acoustic Reciprocity", *Proceedings of MSC/NASTRAN conference, Korea.*

[4] Kang B. J., Kim N. W., Kim J. H., Lee Y. H. and Sin H-C, (2007) ,"Improving the durability of automobile suspension systems by optimizing the elastomeric bushing Compliance" , *Journal of Automobile Engineering,* vol. 223, pp. 151–158.

[5] Kindt P., Sas P. and Desmet W., (2009), "Development and validation of a three dimensional ring-based structural tyre model", *Journal of Sound and Vibration,* vol. 326, pp. 852–869.

[6] Liu G.R., Nguyen-Thoi T. and Lam K.Y., (2008), "An edge-based smoothed finite element method (ES-FEM) for static, free and forced vibration analyses of solids" , *Journal of Sound and Vibration,* vol.320, pp. 1100–1130.

[7] Migeot J.L., Coyette J.P. and Meerbergen K., *(*2002), **"**Predicting Car Body Dynamics at Higher Frequencies: Challenges and Possible Solutions", *Proceedings of MSC/NASTRAN conference, Belgium*.

[8] Reis C., Tonioli J., Anjos F. and Argentino M., (2002), "Structural Analysis of Brake System's Air Reservoir of Heavy Duty Vehicles", *Proceedings of MSC/NASTRAN conference, Brazil*.

[9] Scigliano R., Scionti M. and Lardeur P., (2010), "Verification, validation and variability for the vibration study of a car wind screen modeled by finite elements", *Finite Elements in Analysis and Design,* vol. 47, pp.17-29.

[10] Wendong Z., Wang Yan and Jingzhan Li., (2002), "Numerical Simulation on XIALI Car Dynamic Response", *Proceedings of MSC/NASTRAN conference, China.*

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