CAE ANALYSIS OF HVAC MODULE OF CAR AIR CONDITIONING SYSTEM

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

MASTER OF ENGINEERING
IN
CAD/CAM ENGINEERING

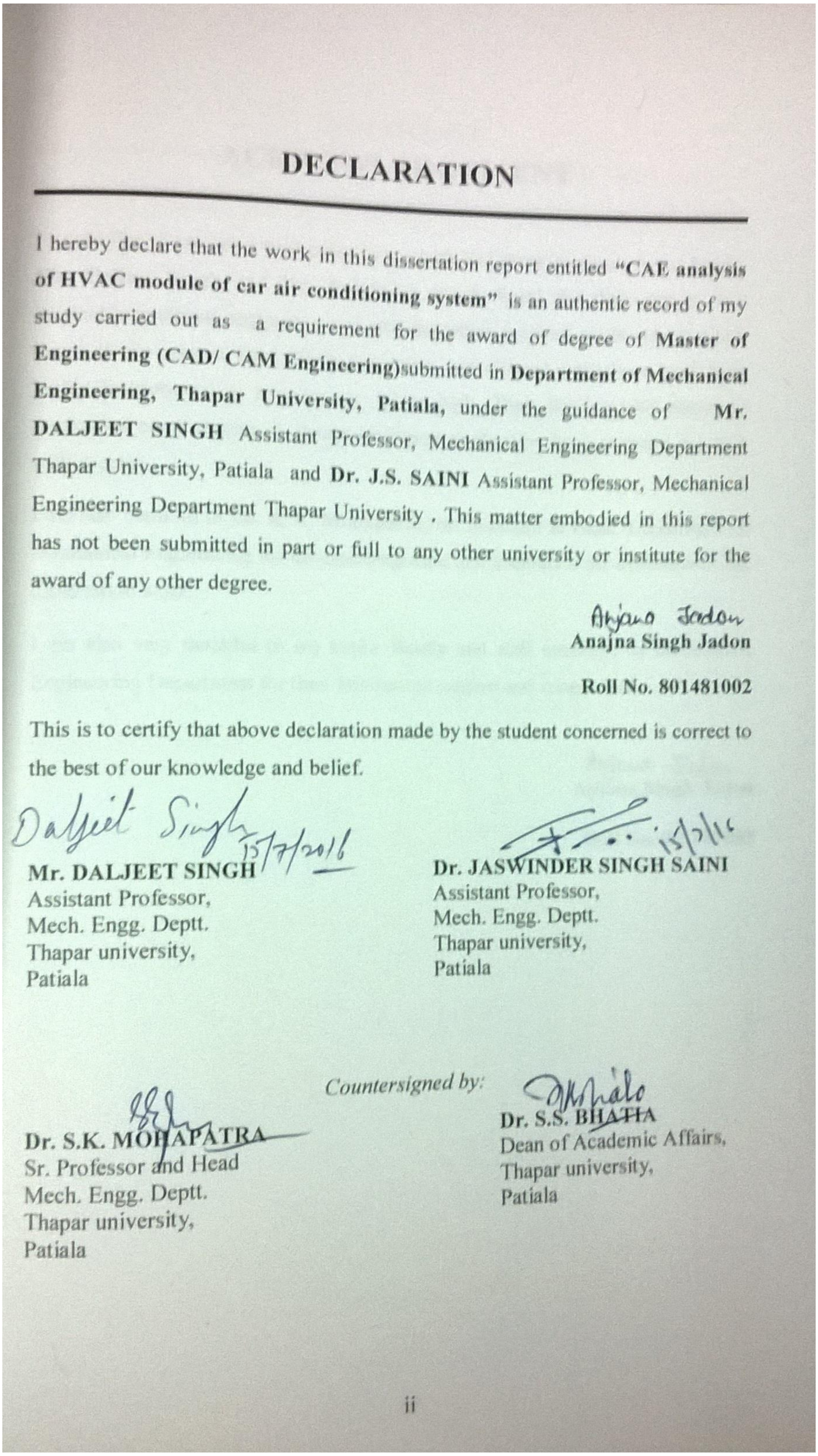
Submitted by
Anjana Singh Jadon
Roll no. 801481002

Under the guidance of

Mr. Daljeet Singh
Assistant Professor,
Mech. Engg. Deptt.,
TU Patiala.

Dr. Jaswinder Singh Saini
Assistant Professor,
Mech. Engg. Deptt.,
TU Patiala.

MECHANICAL ENGINEERING DEPARTMENT
THAPAR UNIVERSITY
PATIALA- 147004, INDIA
JULY 2016



DECLARATION

I hereby declare that the work in this dissertation report entitled **"CAE analysis of HVAC module of car 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 Engineering)** submitted in **Department of Mechanical Engineering, Thapar University, Patiala,** under the guidance of **Mr. DALJEET SINGH** Assistant Professor, Mechanical Engineering Department Thapar University, Patiala and **Dr. J.S. SAINI** Assistant Professor, Mechanical Engineering Department Thapar University . This 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.

Anjana Jadon
Anajna Singh Jadon

Roll No. 801481002

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

Daljeet Singh 15/7/2016
Mr. DALJEET SINGH
Assistant Professor,
Mech. Engg. Deptt.
Thapar university,
Patiala

15/7/16
Dr. JASWINDER SINGH SAINI
Assistant Professor,
Mech. Engg. Deptt.
Thapar university,
Patiala

Countersigned by:

Dr. S.K. MOHAPATRA
Sr. Professor and Head
Mech. Engg. Deptt.
Thapar university,
Patiala

Dr. S.S. BHATIA
Dean of Academic Affairs,
Thapar university,
Patiala

ACKNOWLEDGEMENT

I would like to express a deep sense of gratitude and thank profusely to my guides **Mr. Daljeet Singh** and **Dr. Jaswinder Singh Saini** for their sincere and invaluable guidance, suggestion 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 suggestion blended with an innate intelligent application have crowned my task with success.

I am also thankful to **Dr. S. K. Mohapatra**, Sr. Professor & Head, Department of Mechanical Engineering for his encouragement and inspiration for execution of the dissertation work.

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

Anjana Singh Jadon

Roll No. 801481002

ABSTRACT

The aim of the automobile industry, because of the stiff competition these days is to provide better quality and least cost of any component or part used in automobile vehicles. The change of design of any part or component can be done by using the different Computer Aided Engineering (CAE) tools in less time, thereby reducing the time to market.

In the present work, the vibration characteristics of heating, ventilating and cooling system by using the CAE tools such as Altair hyper works are determined. Both static and dynamic analysis are done on the HVAC module of the car. Dynamic analysis consisted of two analyses i.e. Normal Modes Analysis and Frequency Response Analysis. The Normal Modes Analysis is performed to determine the different mode shapes of the HVAC at the natural frequencies and the frequency response analysis is performed to determine the stresses in all the directions. The above methodology is used to analyse the HVAC module and a new optimized model is proposed, that is having lesser mass and reduced stress as compared to the original HVAC module.

TABLE OF CONTENTS

LIST OF FIGURE

LIST OF TABLE

CHAPTER 1 INTRODUCTION

Due to increasing competition in the domestic and international car manufacturing, the automobile manufacturers are always in the need of a new technique that can be easily implemented at an acceptable cost. The quality enhancement along with the addition of new features with short duration computer aided engineering (CAE) tools help a lot to achieve the goal and prove to be very useful in automobile industries.

1.1 Computer Aided Engineering

CAE is the use of computers to design, analysis, and simulate the process engineering problem to promote or advance design for improving their functioning. This consists simulation, validation and optimization of the product, process and manufacturing. The analysis result of CAE is based upon the assumptions, and boundary condition, *i.e.* constraints, loads, moment etc.

Main objectives of CAE are as follows:

i. It reduces the cost and time of the product development and improves the quality and endurance limit.
ii. Decisions related to design can be easily made that are based on their impact on performance.
iii. Design of product can be evaluated and much refined using computer simulations rather than physical prototype testing of that product which as a result saves money and time.
iv. It provides performance insights earlier in the development process stage, the time when design changes are less expensive to make.
v. It helps engineering teams manage any risk and understand the performance implications of their design of a product.
vi. When properly integrated into the product and manufacturing development, CAE enables earlier problem resolution, which reduces the costs associated with product life cycle.
vii. Graphical representation of results can be done.
viii. Analysis is simplified through a Graphical User Interface (GUI).

1.1.1 Process of CAE

Following are the step involved in the CAE process:

i. The product of system is designed by using the CAD software and the parameters such as physical properties, geometry, and the boundary condition are defined.

ii. The entity is determined and analysed by using CAE processes like FEA, Noise Vibration and Harshness (NVH), and CFD etc.

iii. The results are obtained in the CAE software and according to optimum results: parameter and properties of the system or object are calibrated.

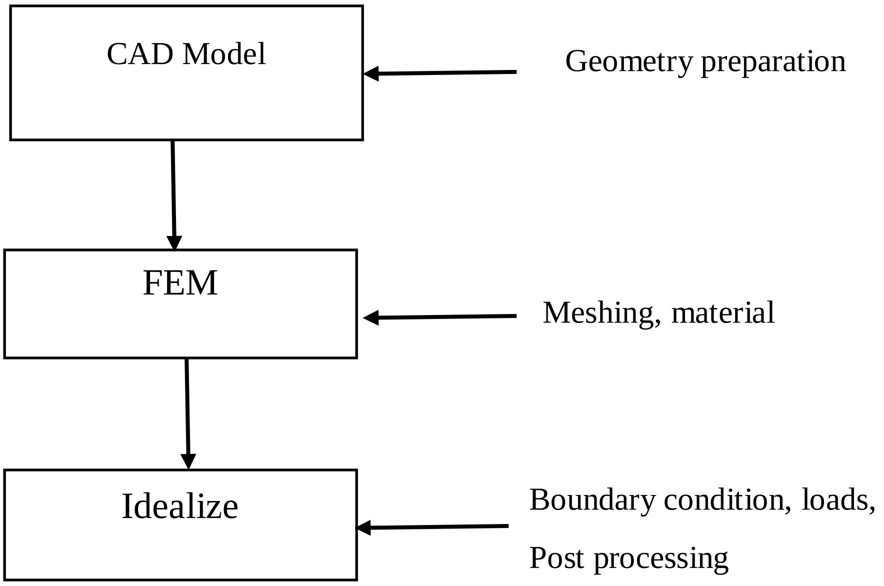


Figure 1.1: Process of CAE

1.2 Different type of Analysis

There are different type of analysis can be done on a component. These are discuss in the following results.

1.2.1 Static structural analysis

It determines the effect on the part/CAD model under the static condition. It doesn’t consider any type of load which is changing with respect to time or time varying loads. But it can include steady inertia load which is comparable to the static such as gravity and rotational velocity. It is used to determine the structural deformation, stresses due to the internal forces etc. under static loading condition.

1.2.1.1 Types of load analysis

The different type of the load that can be applied during static structural analysis are:

a) Pressure and forces
b) Rotational velocity or gravity that is steady static inertial forces.
c) Non zero displacement
d) Effect of heating

1.2.1.2 Types of static structural analysis

Static structural analysis are of two types of which depends upon the material behaviour, loading and environmental conditions.

- **Linear static structural analysis**

It is a proportional analysis. In linear static structural analysis, the stress is directly proportional to the deformation. It obeys Hooke's law. Linear static analysis follows equation (1.1).

$$[K]\{u\} = \{f\} \tag{1.1}$$

Where $[K]$ is the stiffness of the system, $\{u\}$ is the displacement

$\{f\}$ is the applied forces.

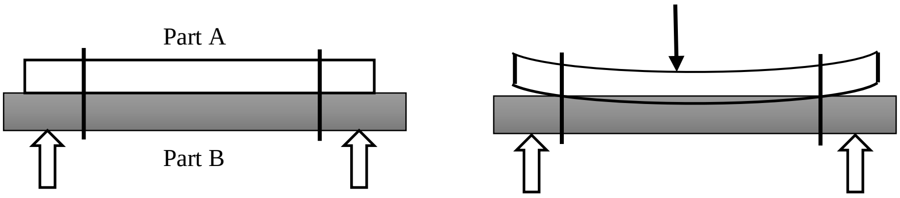


Figure 1.2: Linear static analysis

Figure 1.2 shows the linear static structural analysis on the simply supported beam by using the external load. The maximum stresses was occurred in the mid of the beam.

- **Non Linear static structural analysis**

In Nonlinear static structural analysis, the stresses and displacement do not vary linearly with respect to the applied forces. It can be due to number of reason such as irregularities in the geometry, material and boundary condition.

a) Non-linearity due to geometry

When the stiffness of the structure changes due to geometric deformations. The cause of non-linearity due to geometry are given below:

i. Large strain
ii. Large deflection or large rotation
iii. Stress stiffening
iv. Spin softening

b) Non linearity due to material

When the materials go beyond the Yield’s strength, then it doesn't behave linearly with stress and displacement due to given condition.

i. Permanent deformations in the material
ii. Cracking : due to reducing the stiffness of the member
iii. Energy dissipation

c) **Non- linearity due to boundary conditions**

The boundary condition could be added or removed from the model due to boundary non-linearity as the analysis progresses. This type of non-linearity involves setting of contacts in the model which get engaged or disengaged as a response to applied loads.

1.3 Dynamic analysis

Dynamic analysis is a real time analysis of a finite element model which is varied with respect to time during the load condition and applied boundary condition. It is related to inertia force which is developed in the part/component. Different type of dynamic analysis depends upon the type of inertia loading and response of the finite element model.

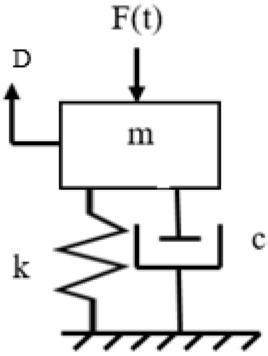


Figure 1.3: Single degree of freedom

Equation (1.2) gives the dynamic response of a structure

$$[M]\{\ddot{D}\} + [C]\{\dot{D}\} + [K]\{D\} = \{F\} \tag{1.2}$$

Where [M] is the structural mass

$\{\ddot{D}\}$ is the nodal acceleration

$[C]$ is the structural damping

$\{\dot{D}\}$ is the node velocity vector

$[K]$ is the structural stiffness

$\{D\}$ is the nodal displacement vector

$\{F\}$ is the applied time varying nodal load vector

Equation 1.2 gives the response of the structure which is varying with respect to time by considering inertia and damping forces as shown in Figure 1.3.

1.3.1 Types of Dynamic Analysis

The different type of analysis on the different material and applied different type of boundary condition of the system are as follows

- **Modal analysis**

In this analysis, the frequencies and mode shape of the finite element model/part/component are determined. Deformation in the model/component is due to natural frequencies, the natural frequencies and mode shapes are given by the Eigen values and Eigen vectors. It is also known as eigenvalue analysis. Because of this analysis, the period of vibration of the any structure are determined. In the case of experimentation, the model/part is fixed on the shaker table and excited by the electromagnetic shaker. This is a time consuming process which is not valid for all the structure. The solution is obtained by removing the inertia and damping forces from the equation (1.2).

Thereafter, it is assumed that the structure is subjected to a sinusoidal function of the peak amplitude. Let the displacement, {D} is of the form as shown in equation (1.3).

$$\{\boldsymbol{D}\} = \{\boldsymbol{A}\}\boldsymbol{sin\omega t} \tag{1.3}$$

Where A is the amplitude of the displacement

ω is frequency

By differentiating and solving equation (1.3): the equation (1.4) is obtained.

$$([K] - \lambda[M])\{A\} = \{0\} \tag{1.4}$$

Where λ corresponds to ω^2 and {A} associated with every value of λ. The natural frequency is obtained from the eigenvalues which depend upon the degrees of freedom. Normally, first few eigenvalues are considered because the finite element model gives the approximation of the structure. Therefore, the higher eigenvalues and eigenvectors that is mode shapes are mostly neglected. The natural frequency of a structure is given by equation (1.5).

$$f_i = \frac{\omega_i}{2\pi} \tag{1.5}$$

Where, f_i is the i^{th} natural frequency.

ω_i is $\sqrt{\lambda_i}$ (1.6)

The linear combination of all of the normal modes and displacements gives the corresponding deformation or vector displacements of a structure under forced and free vibrations.

• Modal frequency response analysis

When the system was excited by using the steady state oscillation then it is known as the modal frequency response analysis. It includes rotary parts such as turbine blade, helicopter blades, and those components which have vibrations. The input of the system is defined in the frequency domain and applied forces is known as forcing frequencies. The forces can be enforced motions applied or applied forces. The frequency of the response to the harmonic input is also harmonic and it occurs at the same frequency. The frequency of the forces is given by equation (1.7).

$$[\boldsymbol{F}] = \{\boldsymbol{F_o}\}\boldsymbol{e^{i\omega t}} \tag{1.7}$$

Where Fo is the peak force amplitude,

ω is the harmonic frequency.

The general equation of dynamic motion is given by equation (1.8).

$$(-\boldsymbol{\omega}^2[\boldsymbol{M}] + \boldsymbol{i\omega}[\boldsymbol{C}] + [\boldsymbol{K}])\{\boldsymbol{D_O}\} = [\boldsymbol{F_0}] \tag{1.8}$$

• Transient response analysis

When the input of the system is changed with respect to time varying excitement than it is known as transient response analysis. And the excitation is also the function of the time. In general, this equation is solved by using the inertia forces and damping forces on the basis of time scale. Stresses, deformation and strain are varying with respect to time. This analysis is of two types, which depend on the structure and the nature of the loading.

a) Direct transient response analysis

In this analysis, a numerical integration of the complete coupled equation of motion, *i.e.* solving the equation by using the direct integration method. It takes more computational time for complex geometry problem.

b) Model transient response analysis

Modal transient response analysis is the natural extension of the normal mode analysis. This method is used to the different mode shapes of the structure to reduce the size, uncouple the equation of motion, *i.e.* when modal or no damping used in the structure. It gives the higher accuracy, high frequency excitation and have many iterations of time for small model.

1.4 Optimization

Optimization is a technique which is used to get the best optimum result under the given condition. For the optimum result, change the geometric parameter, shapes, size of the component are done.

The objective function for the problem of optimization consists of the parameters that are to be maximized or minimized. The optimal solution obtained depends upon some design variables that can be expressed with numerical value.

Types of the optimization method

a) Shape optimization

It is provided a common and systematic frame work for optimizing the structure described by various possible physical and mechanical model. In this optimization, the outer boundary of the model/structure is improved as shown in Figure 1.4.

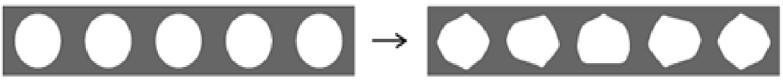


Figure 1.4: Shape optimisation

b) Size optimization

It is used to define the optimum thickness and cross section dimension of the model/structure. It determined the ideal thickness of the structure on the basis of performance and boundary condition *i.e.* applied on the structure. Size optimization is shown in Figure 1.5.

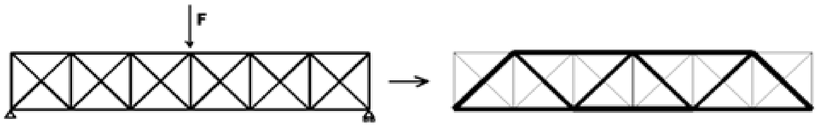


Figure 1.5: Size optimisation

c) Topology optimization

It is used to optimizes the material layout for the given load and boundary condition within a given design space. It minimized the time, costly design iteration, design development time and overall cost of the model. The topology optimization is shown in Figure 1.6.

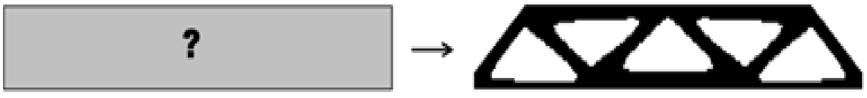


Figure 1.6: Topology optimisation

d) Topography optimization

It is almost similar to the topology optimization, but it is applied only on 2D element. It is mostly used for the sheet metal parts. In this optimization, thickness of the part is not changed, but varying the topology of the surface for finding the optimization results.

CHAPTER 2 LITERATURE REVIEW

A review of work done on dynamic structural analysis of different components related to automobiles is presented below.

2.1 Literature Review

Park *et al.* [2] studied the reduction of the weight of the armrest frame of the car by using design optimization. Automobile manufacturing requires a reduction of weight not only to increase performance and decrease fuel consumption, but also to reduce manufacturing cost and strengthen competitiveness. They designed the armrest frame of the car by using size, shape and topology optimization. The design optimization process of a short fiber-reinforced plastic armrest frame to minimize its weight by replacing the steel frame with a plastic frame. The analysis was carried out with the equivalent mechanical model and design of experiment (DOE) method. Instead of considering the whole structure, it was divided into three simpler regions to reduce the complexity of the problem through examining its structural characteristics and load conditions. It used polypropylene plastic which is light in weight, low cost but with lesser yield strength as compared to the steel frame.

After design optimization, the weight of the optimum plastic-based armrest frame is reduced by about 18% compared to the initial design of a plastic frame and is decreased by 50% in comparison with the steel frame and also with reduced manufacturing cost. Some prototypical armrest frames were also made by injection molding and tested.

Wijaya *et al.* [3] investigated the dynamics analysis, *i.e.* modal and transient analysis on the air conditioning system hose assembly using the finite element method. Its results were also validated experimentally. The A/C hose consisted of 2 braid layer and 3 rubber layers. The hose connected the A/C to the engine side and supplied the high refrigerant fluid. Boundary condition of the hose in such a way that at one end, the movement of the hose restricted in all the direction and at the other end, the hose is free to move only in y-direction.

It was found that the swing and bending modes of the hose occurred in first six natural frequencies. In transient analysis, maximum stress occurred in the (middle layer) reinforced braid layers of hose component.

Singh *et al.* [4] studied the component of the condenser assembly of the air conditioning system of the automobile using CAE tools. Computer aided engineering was used provide the optimized design of condenser assembly. It provided the linear static structural analysis, different mode shape on first five natural frequencies and modal frequency response analysis in the x-direction, y-direction and z-direction of the condenser assembly.

The weight was reduced by 2% after making a small change in the design and it stresses were reduced in all directions.

Singh *et al.* [5] proposed a new design of the cam of the heating, ventilation and cooling system. The mechanism consisted of many doors opening and closing for controlling the air flow in the heating, ventilating and cooling system. The developed design was faster and cheaper from the previous with the lesser torque required as compared to existing design.

Jeyraj *et al.* [6] studied the durability of the muffler mounting bracket. The bracket failure of the three wheeler vehicle was analyzed and find that the bracket fails after covering the average distance 10,000km due to durability test. Its boundary condition was, engine cradle fixed at both the ends and mass of the engine and muffler center of mass applied on the engine cradle by using the rigid link. Maximum stresses occurred near weld portion between the cradle and muffler. Force was applied in the downward vertical direction (gravitational direction).It used different type of design and optimization for the bracket and collar bush.

The optimized bracket fails after covering average distance, i.e. 1, 00,000km and also reduced bracket mass and induced stress.

Chavan *et al.* [7] studied the natural frequencies of the engine mounting bracket of a car using finite element method. The mostly stresses and deformation induced in the engine mounting bracket of car were due to the combustion force, reaction of bearing, gear transmission, etc. ANSYS software to be used to design and develop a modified bracket design. The bracket was tested for three types of material, i.e.

Aluminum alloy, Magnesium alloy, and Grey cast iron and the model was meshed using hex quadrilateral and triangular elements.

It was observed, that the bracket of grey cast iron had maximum stress as compared to the other two materials used in the bracket. AL alloy and Mg alloy had approximately equal natural frequency (259.96Hz and 257.95Hz), but stresses were more occurred in the Al alloy *i.e.* 535.76MPa as compared to Mg alloy had 366.58MPa. So magnesium alloy was the best material for the mounting bracket of the engine.

Tsai *et al*. [8] developed a technique for determining the material distribution of structure to get the required Eigen mode shape for a problem involving maximum value of fundamental Eigen frequencies. The solid isotropic method with penalization (SIMP) was used for topology optimization. The weighted constrained also added to maximize the fundamental natural frequency, in the bound formulation and also the additional constraints such as modal assurance criteria (MAC) were added in the optimization to achieve the desired Eigen mode.

Loh *et al.* [9] performed various finite element analysis on motor bracket, designed to support motor and fan in air conditioning products, to determine the structural and fatigue response because of harmonic excitations over a range of frequencies. Only the elastic deformation were covered in the study.

Different modifications with comparison of each analysis, were presented in effort to increase the strength of the P-tac motor bracket.

Deshmukh *et al.* [10] proposed and optimised the engine mounting bracket to reduce its rib weight. The various analysis such as static analysis, modal analysis of engine and harmonic analysis of the bracket were performed using FEA software package ANSYS 15.0. The results of deformation and stresses for both actual and proposed model of engine mounting bracket were also compared. The optimized bracket is 12.7% reduced the weight as compared to the initial bracket.

Chen *et al.* [11] proposed study in which a finite element model of a light truck was built and verified against cab model and frontal impact test. The survivor space was evaluated by examining the contact between deform cab and manikin model by simulating to crash tests. Some structural improvement was designed, such as filling structural foams in the side panel and A-pillars, reinforcing the rear wall.

Basavaraj *et al.* [12] studied the dynamic behaviour of the mounting bracket of engine by the nonlinear effects using LS-Dyna. It created the element model of the mounting bracket and determined the response of the multidimensional effect on the bracket.

The CAD model was generated in Hypermesh and simulation was done using LS-Dyna. The six different type of arm design were performed on the dynamic behavior of the bracket and compared with the hysteretic damping material model.

The analysis result of four arm symmetry engine mount curve followed the exactly the experiment test at the highest natural frequency. Frequency of the engine mounting bracket was increased from 1.2Hz to 1.8Hz, for the rubber material in the engine mount.

Eads *et al.* [13] optimised the weight and the cost of exhaust design using finite element method (FEM). The static and dynamic loading condition were used for optimization considering the gravitational load of exhaust.

After optimization, the reduction of weight under dynamic analysis and static analysis were 22% and 25% respectively.

Rao *et al.* [14] studied the dynamic characterization of the exhaust isolator used in automobile, such as variation of the damping and the stiffness by varying the frequency generated by electrohydraulic and shaker excitation. The number of samples of the isolator with different material using random excitation and swept sine were considered. The study shown that result of loss factor and stiffness were similar between both shaker and electrohydraulic excitation at high frequency.

Pan *et al.* [15] performed shape and topology optimization for the design of the bracket used to mount the engine to achieve more strength and structural rigidity with reduced mass. The topology optimization was done considering manufacturing constraints and also the die direction constraints. After optimization the strength and structural rigidity was improved by 50% with a 12% reduction of mass of the optimized design.

Cevik ***et al.*** **[16]** optimized the crank shaft using shape optimization method. The shape optimization was done to decrease the mass and its second moment of inertia with least effect on the stiffness and strength of crank shaft under the dynamic conditions. In the process of optimization the multi body simulation were also used and hence the reduction of 15% of mass obtained of the crank shaft. Without effecting the stresses value the mass moment of inertia was also reduced.

Chang ***et al.*** **[17]** optimized the compressor bracket using topology optimization. The bracket was modeled considering all the interference with the adjacent part of the vehicle and the material chosen was aluminum die casting. For optimization static and model analysis performed using CAE tools and validated experimentally. After optimization, there was reduction by 60% in the average stresses with marginal increase in the mass and first natural frequency was also increased by 29Hz.

Raghavan ***et al.*** **[18]** optimized the hydro mount bracket using finite element approach (FEA) to reduce its weight. This bracket was used to protect the fluid filled in the inner mount of the shock load. The optimization was done by performing static analysis using ETA-VPG preprocessor. Weight saving calculation was also done on the proposed and optimized design. The results were also verified experimentally and there was reduction by 8% in weight in the proposed design.

2.2 Gaps in literature

From the literature review it was found that a lot of work has been done in the area of analysis and optimization for the different parts of the automobile. But, still there is scope for work in analysis of HVAC assembly of the automobile. The following are the gaps found during literature review:

i. Limited work on frequency response analysis for automotive plastic components.
ii. Work on vibration simulation and stress analysis for plastic components not reported, especially for HVAC of an automobile.
iii. Different structural optimization techniques like shape, size and topology optimization have not been applied on these components.

2.3 Objectives of the present work

By considering the above mentioned gaps in the literature review, following objectives were identified:

i. Linear static structural analysis to be performed on the HVAC case.
ii. Modal analysis and Modal Frequency Response analysis to be performed to analyze the stresses and displacement at corresponding natural frequencies.
iii. Comparison of stresses induced due to vibration produced at different frequencies using different material for HVAC module.
iv. Optimization of the HVAC to be done in order to reduce the overall mass, keeping the stresses within the prescribed limit.

CHAPTER 3 ANALYSIS OF HVAC

The Heating Ventilating and Cooling unit (HVAC) is a very important part in automobile. It controls the humidity of the air and maintain the temperature, inside the vehicle cabin. The HVAC unit is shown in Figure 3.1.

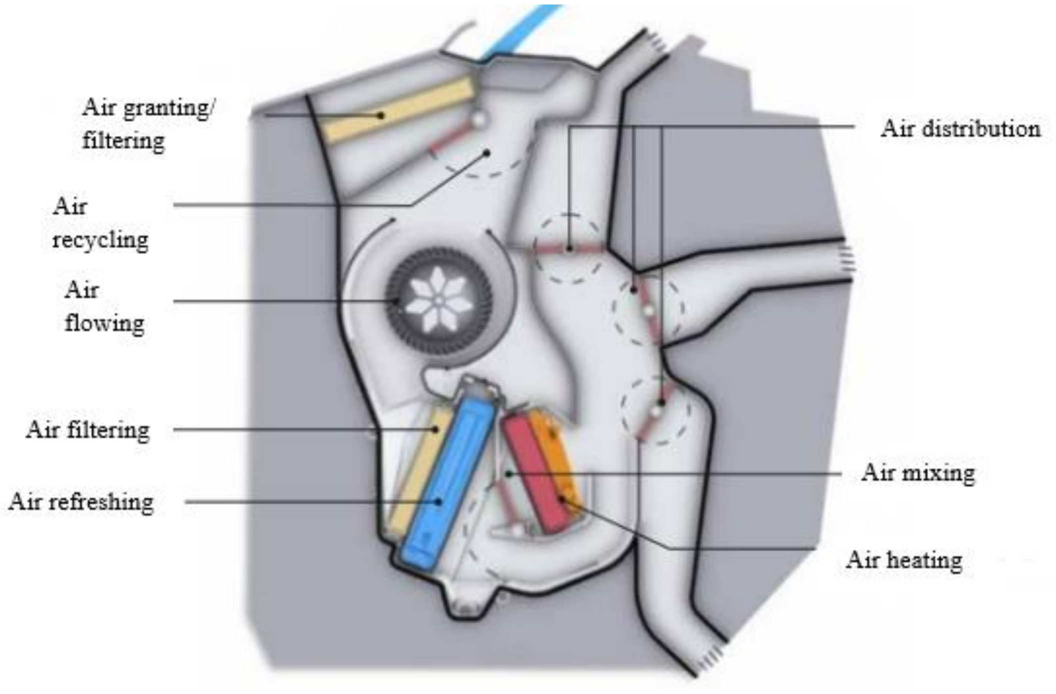


Figure 3.1: HVAC unit of a passenger car [19]

It consists of the following different parts:

i. Blower – it provides the air movement in the HVAC. It pulls the air in through the center by using the centrifugal force and throw it towards the outside.

ii. Heating coil – it delivers the warm air, based on the requirement of the temperature. It has running coolant in the coils and transfers the heat from the engine coolant to the air in the HVAC system.

iii. Cooling coil - it delivers cooled and dehumidified air and acts like a heat exchanger. When the outside air passes through the cooling coil, and it transfers its heat by using a refrigerant.

iv. Air distribution duct- Flaps are used to control the head, lap and foot air distribution in the HVAC system. The flaps are actuated by Cam-Follower arrangement.

3.1 Working of the HVAC

3.1.1 Air circulation

The air goes into the cabin through the air inlet between the wind screen and bonnet. The vent grill is provided to the air inlet, to protect from the external bodies such as insects, leaves, dust, etc. The Figure 3.2 shows the circulation of the air in the car

Figure 3.2: Air circulation in the car [19]

The air extractor are made of the rubber membranes that act as a valve. Air extractor location can be in different position such as

i. In the wheel arch
ii. At the rear of the vehicle
iii. In the body work at the rear
iv. In the center pillar

3.1.2 Propelling the air

The air in the cabin is circulated by a motor driving fan known as blower. It consists of a motor drive, a fan, and a power control device. The blower speed controls the ventilation power and it is of two type *i.e.* passive resistive device and power transistors. Figure 3.3 shows the blower position in the HVAC in the car.

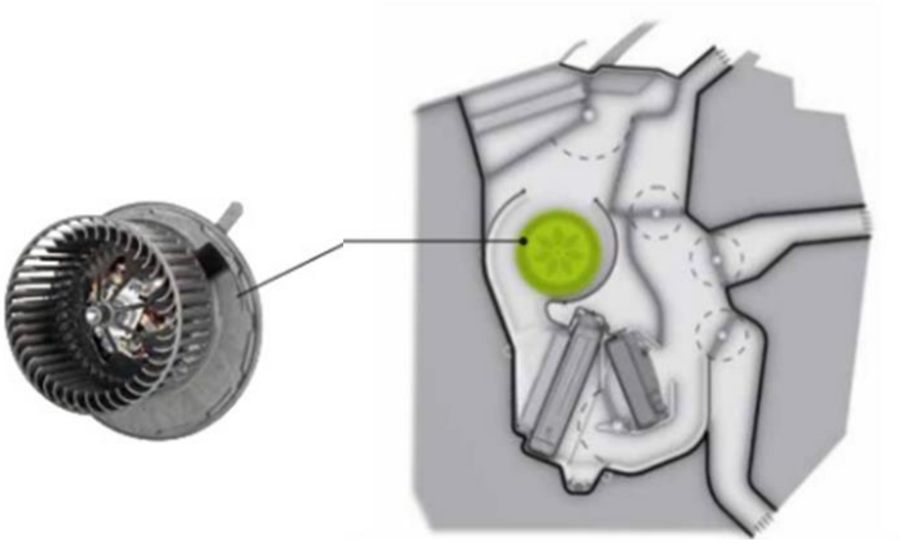


Figure 3.3: Blower in the HVAC [19]

3.1.3 Filtering particle air suspended in the air

Pollutant particles will also be trapped before air entering in the cabin. The largest particle such as leaves, insects etc. are prevented by the air intake grill. The following functions must be performed by the cabin air filter-

i. Air free from the pollution, gases.
ii. Prevent dust, dirt etc.

3.1.3.1 Location of the filter

The filter shape and size depend on the design of the heating, ventilation cooling system. Mostly it is located in the two position *i.e.*

i. Before entering the blower
ii. And between the evaporator and blower

Figure 3.4 shows the position of the filtration coil in the HVAC.

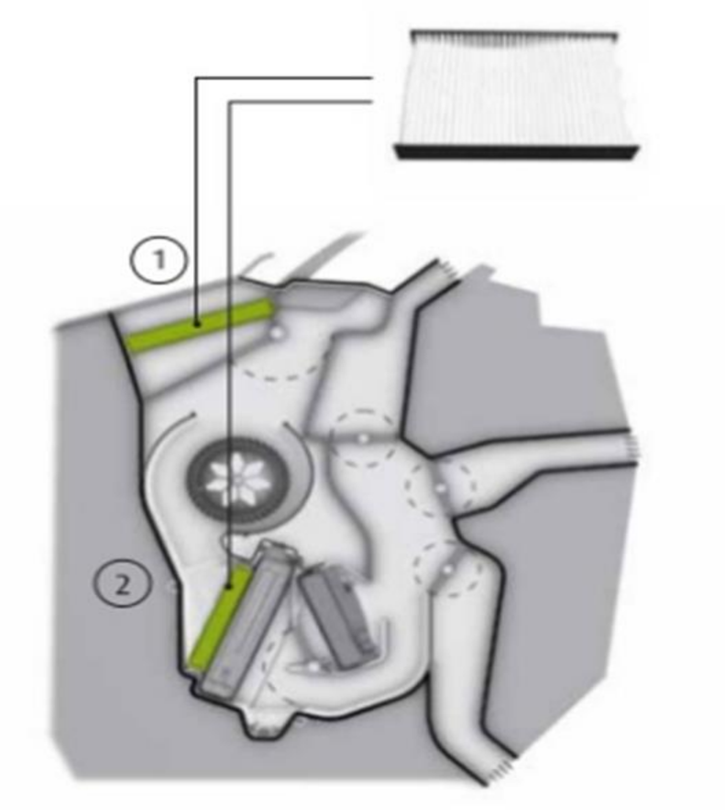


Figure 3.4: Filtration coils [19]

3.1.4 Heating coils and cooling coils

- **Heating coils**

The heating coil is fitted to the HVAC assembly for increasing the temperature of the cabin. The heating coil shown in the Figure 3.5.The function of the heating core are as follows-

i. An air/water exchanger
ii. Increase the cabin temperature
iii. Demists and defrost the glasses

- **Cooling coils**

The outer intake air is cooled by the cooling coil or evaporator and it transfer its heat to the refrigerant. Cooling coil works as a heat exchanger and also intricate in the dehumidification of the air. It is usually made of copper and are enclosed by the aluminum fins for increasing the heat transfer rate. The cooling coil of HVAC is shown in the Figure 3.6.

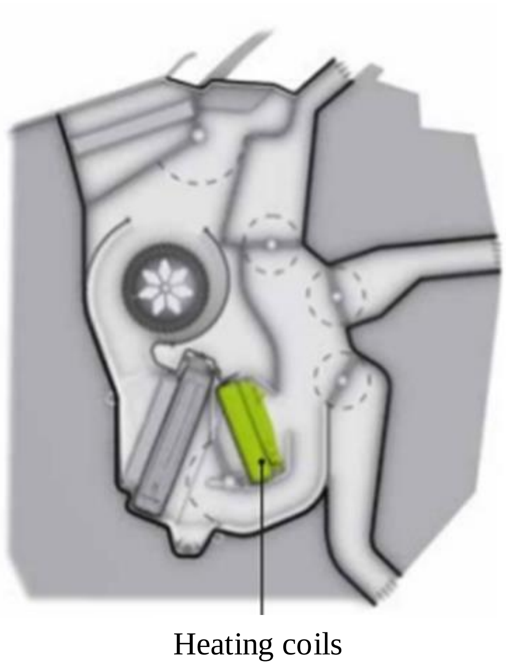


Figure 3.5: Heating coils [19]

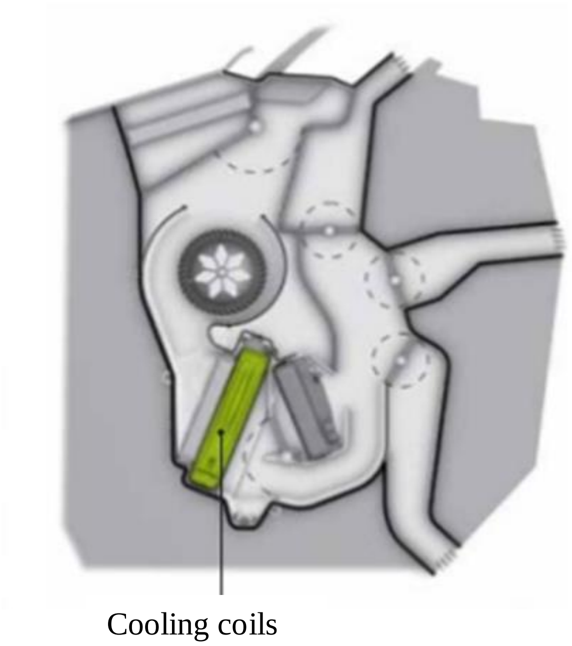


Figure 3.6: Cooling coils [19]

3.1.5 Air distribution

The air distribution depends on the system design. It is of two types

i. Manual air conditioning system- In this system the distribution of air *i.e.* head/lap/foot is controlled by the flap which is driven by a cam profile.

ii. Automatic air conditioning system- In this system the distribution of air is controlled by the driven motor mixing flap.

The air distribution point of HVAC is shown in the Figure 3.7.

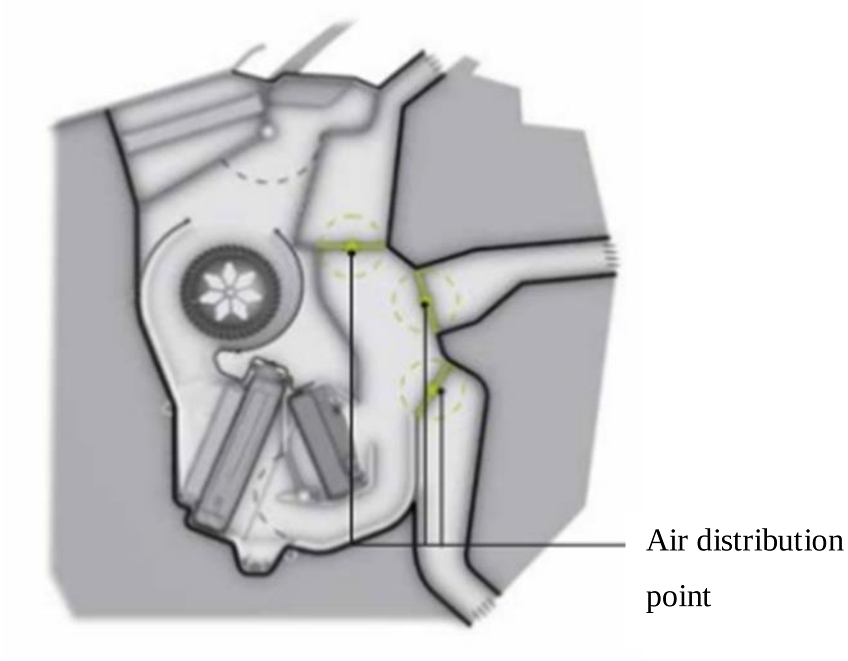


Figure 3.7: Air distribution [19]

3.2 Properties of the material of the HVAC

The polypropylene material was used in the HVAC module. Its properties are given in Table 3.1.

Table 3.1: Properties of material

Properties	**Values**
Elastic Modulus	2e+09 N/m^2
Poisson’s Ratio	0.4
Density	1200 kg/m^3
Ultimate Tensile Strength	40 MPa

3.3 CAD Model

By using the reverse engineering technique of 3D scanning, the CAD model of the HVAC was developed. The CAD model is shown in Figure 3.8. The different orthographic views of CAD model are shown in Figure 3.9 to Figure 3.11.

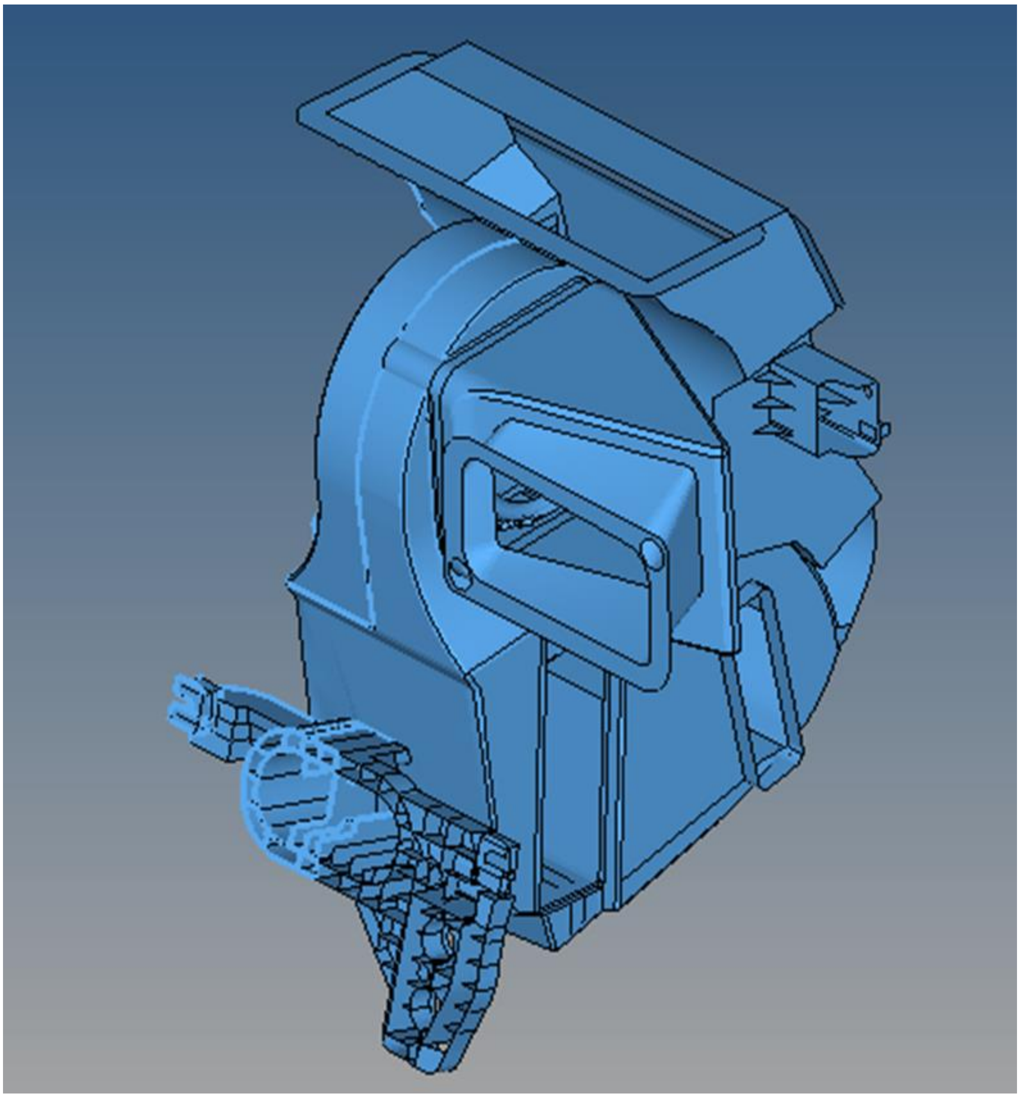


Figure 3.8: Isometric view of the CAD model

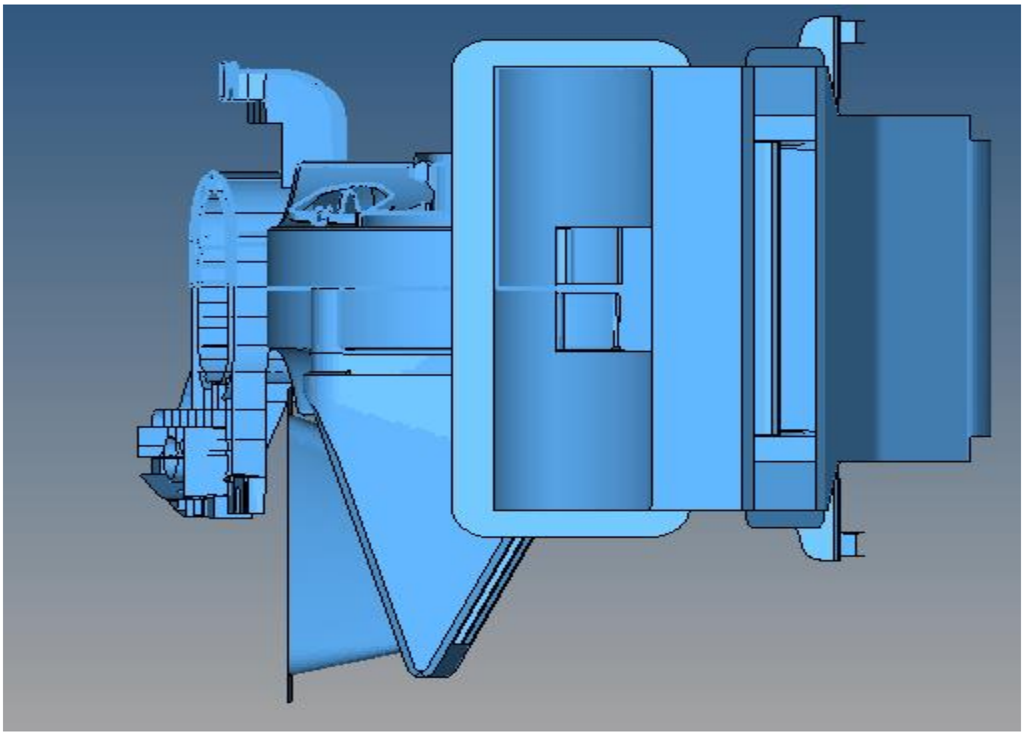


Figure 3.9: Top view of the HVAC

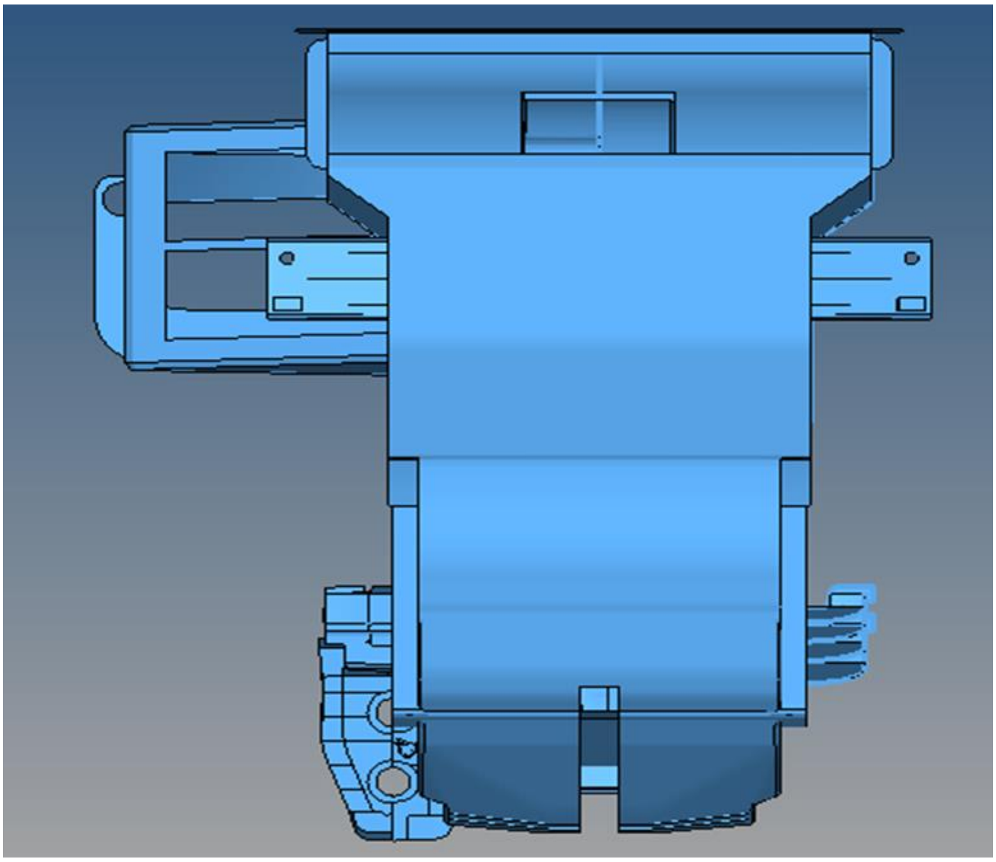


Figure 3.10: Front view of the HVAC

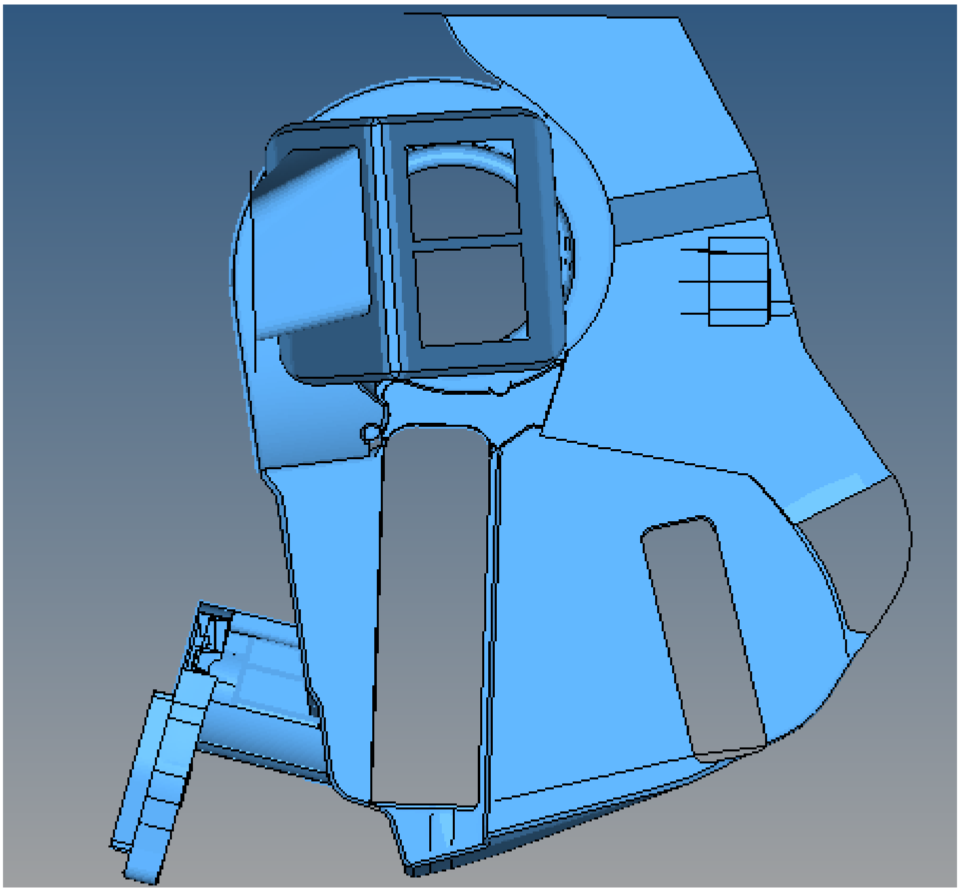


Figure 3.11: Side view of the HVAC

3.4 Mesh generation

Meshing is an important part in simulation of any part using FEA. It converts the whole part into finite elements and then the stresses for each element can be calculated. The results are calculated on each node by solving the governing equations.

As the HVAC assembly is a large and complex part (in part because of the plastic material), meshing of the cleaned up geometry is time consuming and very critical for the successful analysis. The quadrilateral elements were selected for the meshing of the HVAC system in the Altair Hyperworks 12.0 software, as they provide higher accuracy. The meshed model is shown in Figure 3.12.

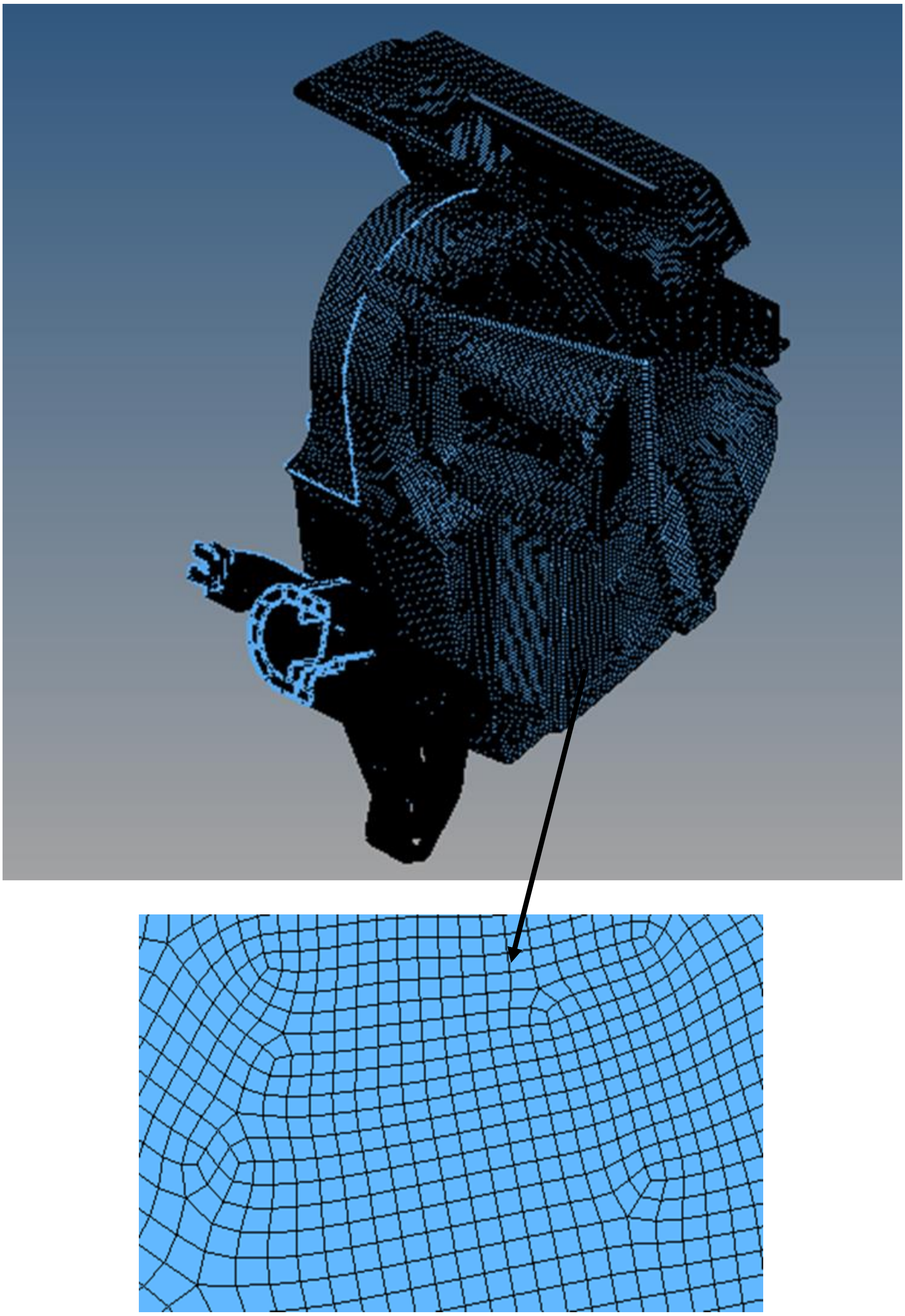


Figure 3.12: Meshed view of HVAC

In the meshing of the HVAC module, washer was added around a mounting location of the HVAC for refining the mesh and improving its quality, as shown in Figure 3.13. The following parameters were used for adding the elements of washer:

a) Number of layers is 3.
b) Width of the element is 2.5.
c) Minimum number of nodes around the hole is 32.

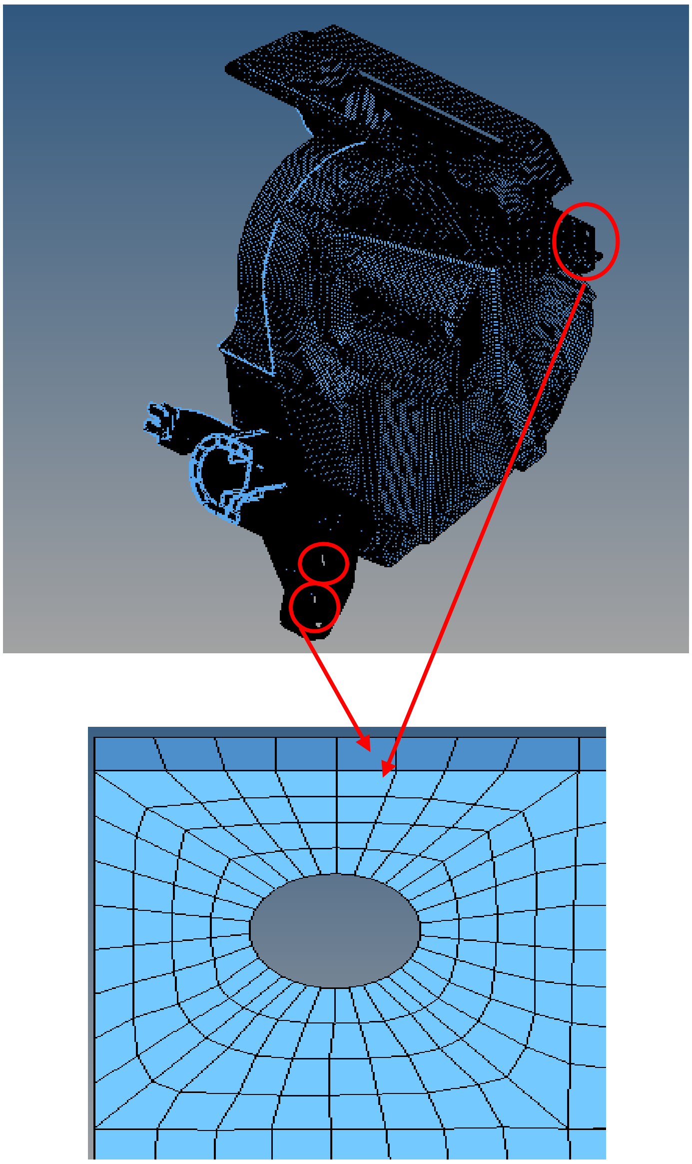
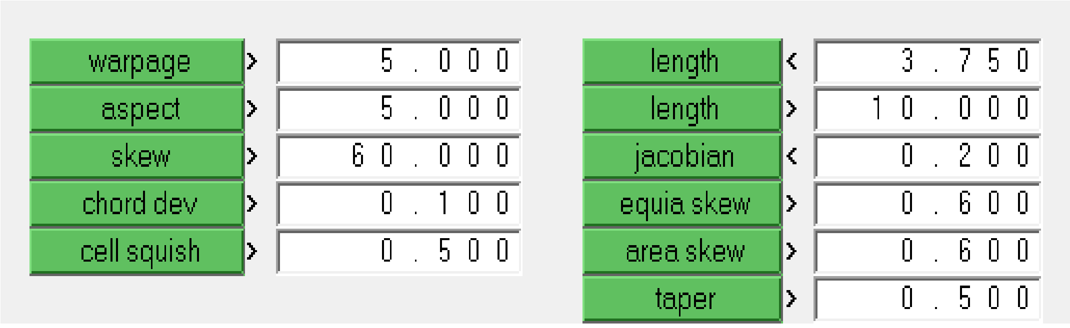


Figure 3.13: Washer in the mounting location

3.4.1 Element quality check

The element quality was checked for the meshed HVAC model by using the software in-built function. These parameters are shown in the Figure 3.14.



warpage	>	5.000	length	<	3.750
aspect	>	5.000	length	>	10.000
skew	>	60.000	jacobian	<	0.200
chord dev	>	0.100	equia skew	>	0.600
cell squish	>	0.500	area skew	>	0.600
			taper	>	0.500

Figure 3.14: Parameters of the element quality

After the meshing of the model, boundary conditions were defined and the analysis run, using the procedure described below:

3.5 Static structural analysis

Linear statics analysis is performed on the HVAC unit of the automobile to determine the stresses.

3.5.1 Static structural analysis in the x- direction

These different steps followed for the linear static structural analysis are discussed below:

1) Material collector- The material properties are given in Table 3.1 as defined.
2) Property collector- It is assigned to all the elements of the HVAC.

Types of element – 2D

Card image – PSHELL

Thickness – 0.25 mm

Material – poly propylene

3) Load collector – In the linear static structural analysis, the following two types of the load collector are used:

- **i.** Constraints – All the five mounting locations of the HVAC are fixed in all the directions as shown in Figure 3.15.
- **ii.** Forces – In this collector the 27N force (equally distributed to all nodes) is applied on bottom node of the mounting location of the HVAC.

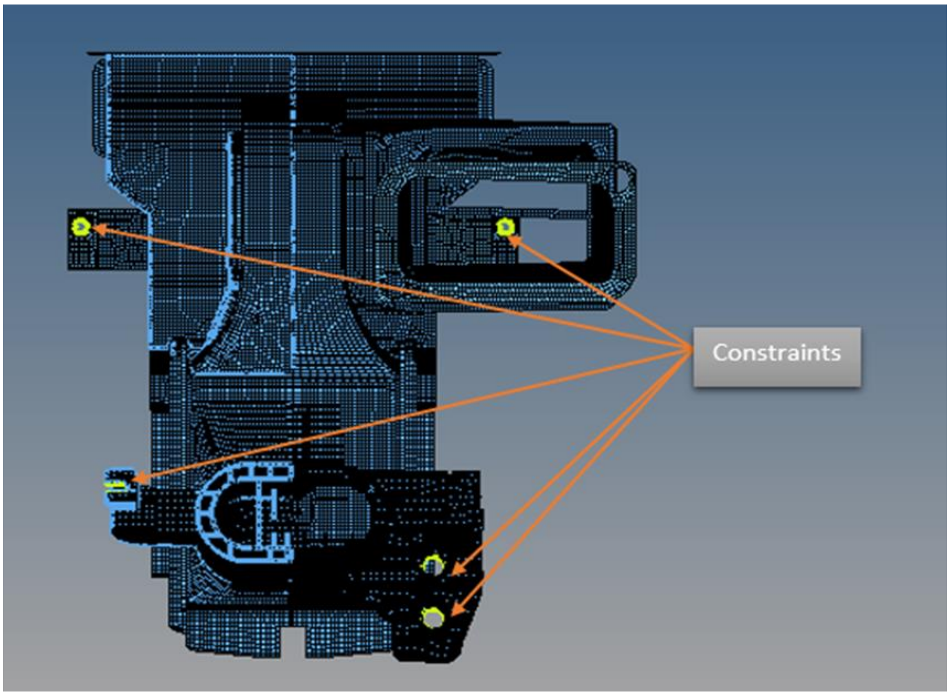


Figure 3.15: Mounting location of the HVAC

After the linear static structure analysis, the maximum stress in x-direction is 3.794 MPA. The stresses are shown in Figure 3.16.

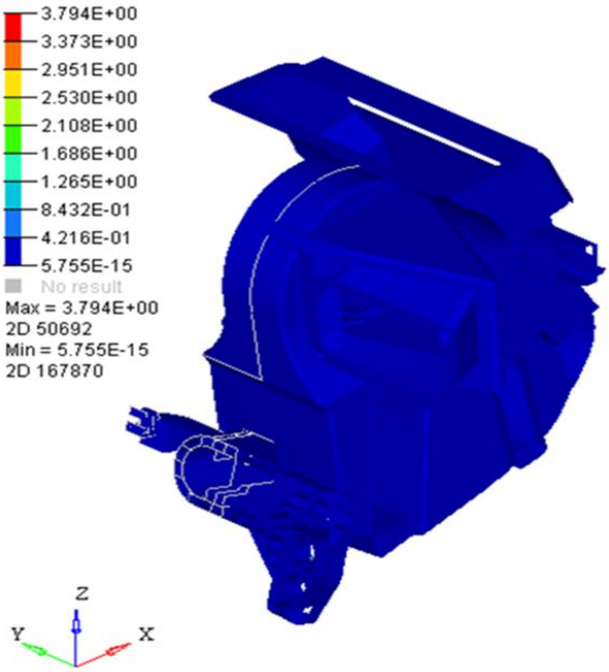


Figure 3.16: Stresses in the x- direction

3.5.2 Linear static structural analysis in the y- direction

In this analysis, the material collector and properties collector are same as the previous analysis, except the force is applied in the y- direction. The results are shown in the Figure 3.17.

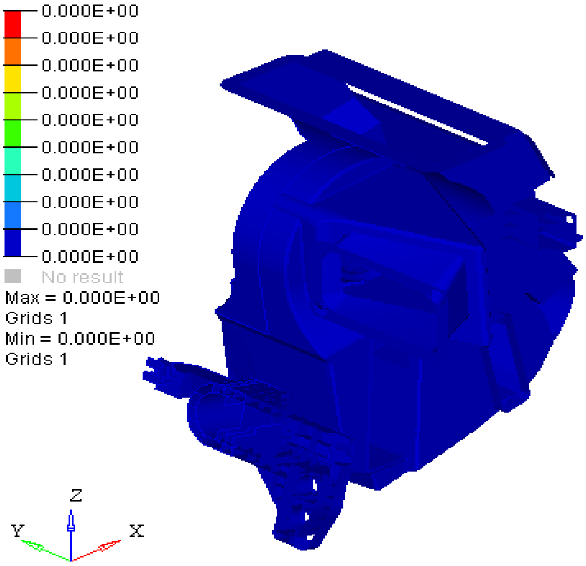


Figure 3.17: Stresses in the y-direction

3.5.3 Linear static structural analysis in the z- direction

The material collector and properties collector are same as the previous analysis, except the force is applied in the z- direction. The results are shown in the Figure 3.18.

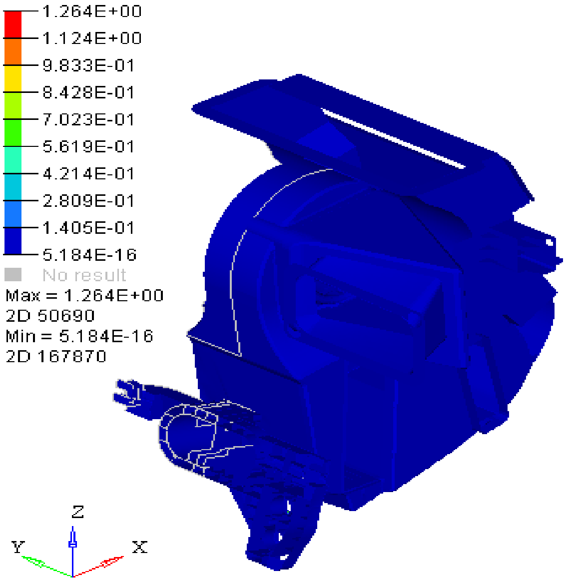


Figure 3.18: stresses in the z- direction

3.6 Dynamic analysis

A dynamic analysis was performed to determine the effect of the vibration of the engine and other automobile parts on the HVAC system. The two types of analysis were done on the HVAC.

i. Normal modes analysis
ii. Modal Frequency response analysis

3.6.1 Normal modes analysis

It was done to determine the mode shapes and the natural frequencies of the system. The following parameters were considered for the Modal analysis:

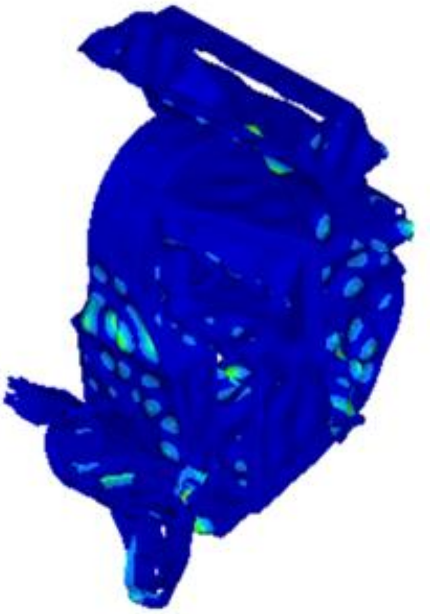
a) External forces: external forces were not considered for finding the natural frequencies.
b) Constraints: These were applied as the actual condition of the system.
c) Damping: It was not considered to determine the natural frequency.
d) Analysis output: After the analysis, natural frequencies and mode shapes were obtained.

The different steps followed for the normal modes analysis are discussed below.

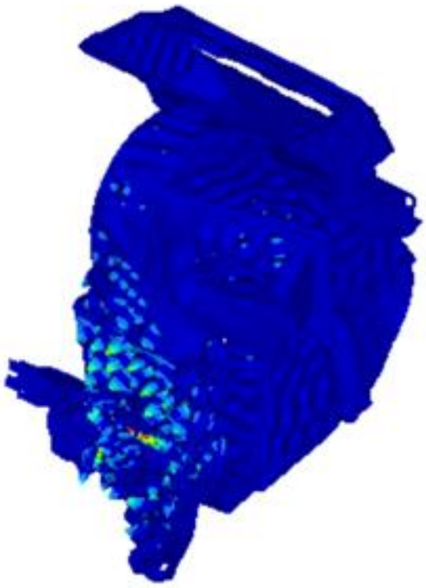
1) **Material collector** – The material collector is same as the previous analysis *i.e.* linear static structural analysis.
2) **Property collector** – The property collector is also same as the previous analysis *i.e.* linear static structural analysis
3) **Load collector**- In the normal modes analysis, the following two types of collector are used :

i. Constraints- All the five mounting location are fixed in all the directions, as shown in Figure 3.15.
ii. Eigrl- In this collector, the data needed to perform the real Eigen value analysis (vibration) using Lanczos method is defined.

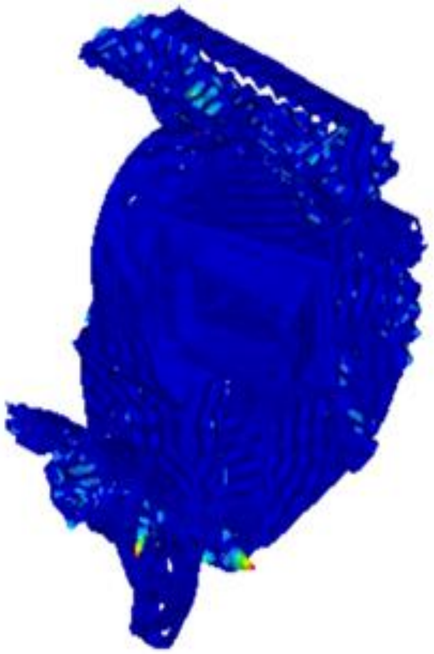
The first five natural frequencies and modes of the HVAC system are obtained after the analysis. These are shown in Figure 3.19.



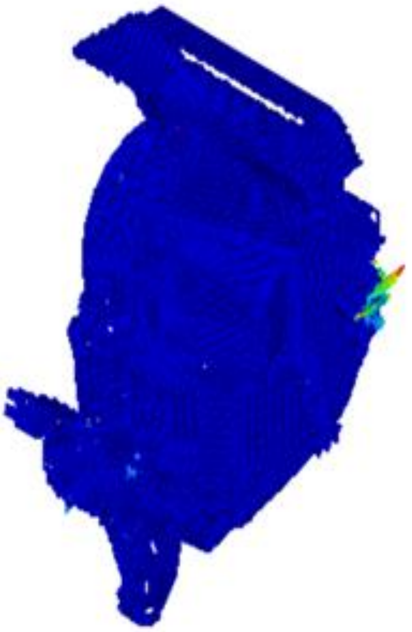
a) Mode1, Frequency 10.074Hz



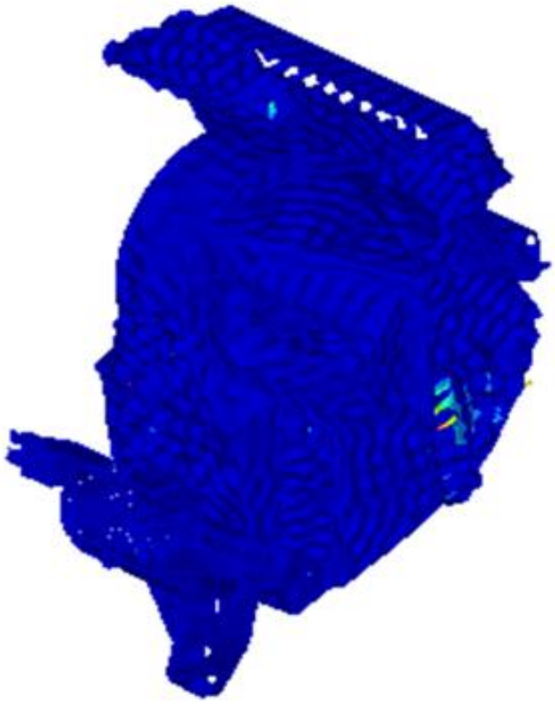
b) Mode2, Frequency 30.116Hz



c) Mode3, Frequency 35.501Hz



d) Mode4, Frequency 50.008Hz



e) Mode5, Frequency 100.040Hz

Figure 3.19: Mode shapes and Frequencies

3.6.2 Modal frequency response analysis

In this analysis, excitation (external force) is given in the form of harmonic response. When the excitation frequency is equal to the natural frequency, the resonance is obtained. The analysis is done in all three translation directions which correspond to the same direction as the excitation force.

a) Modal frequency response analysis in the x- direction

In this analysis, the force was applied in the x- direction by using the RBE3 which is a tool for distributing the applied load and mass in the system. It is having one dependent node and one or more independent nodes. It is shown in Figure 3.20.

Dependent nodes of RBE3 are fixed in all the direction except the x-direction. The 27N force is applied along the x- direction on the dependent node for excitation of the HVAC system. Frequency range was considered 30Hz to 35Hz as per actual testing condition with the number of frequencies range as 5.

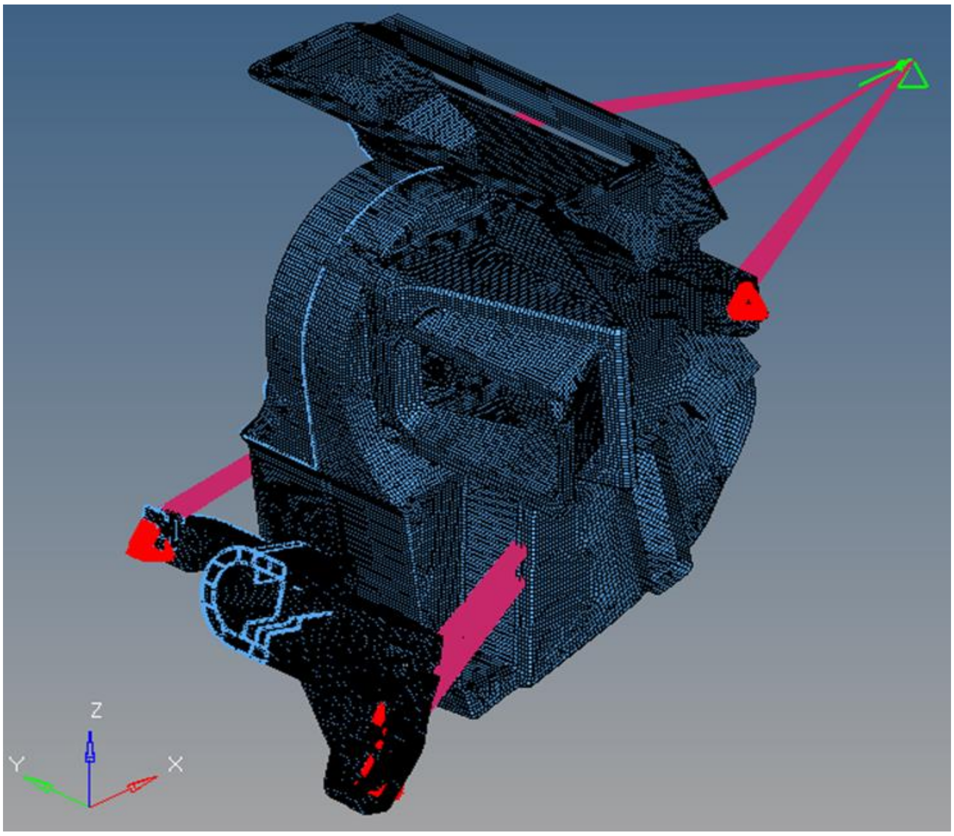


Figure 3.20: Force applied on RBE2

After the analysis, the principle stress that was produced in the x-direction due to dynamic loading with frequency 33.050 Hz, was 27.15 MPa as shown in Figure 3.21. The factor of safety is 1.47.

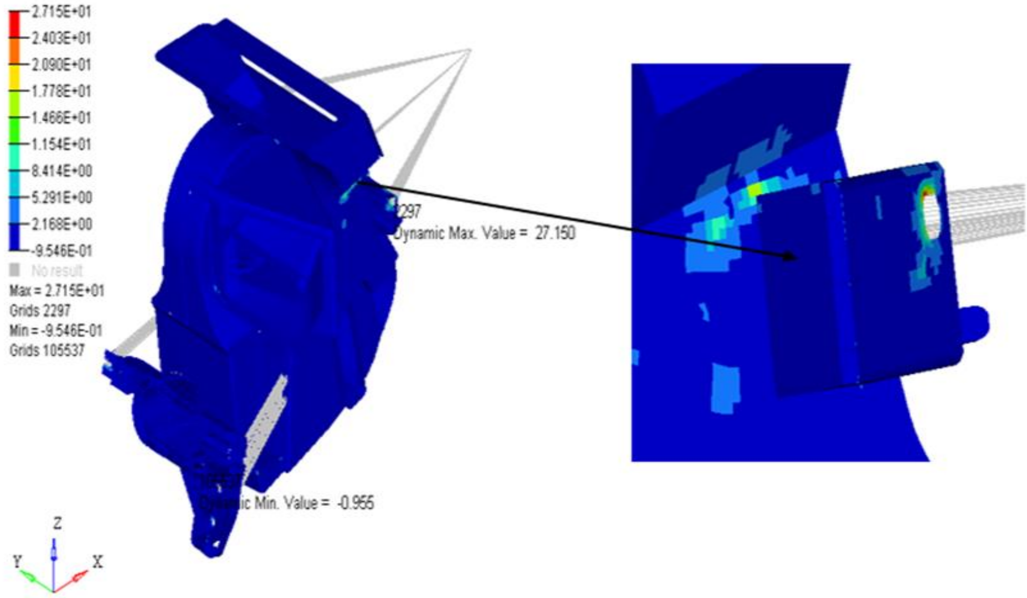


Figure 3.21: Stresses in the x- direction

b) Modal frequency response analysis in the y-direction

The force is applied in the y-direction on the dependent node of RBE3, and zero stress is obtained in the HVAC system as shown in Figure 3.22.

c) Modal frequency response analysis in the z- direction

Dependent node of RBE3 is fixed in all the direction except z-direction. The 27N force is applied along the z- direction of the dependent node for excitation of the HVAC system. Frequency range was considered 30 Hz to 35 HZ and the number of frequency range interval was 5. The stresses in the z-direction are as shown in Figure 3.23.

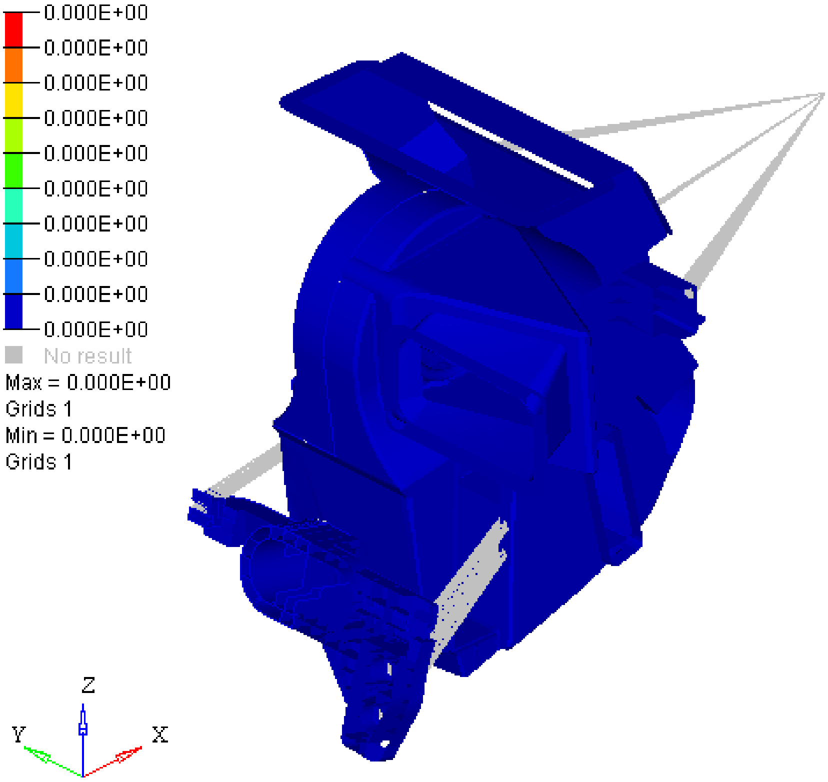


Figure 3.22: Stresses in the y-direction

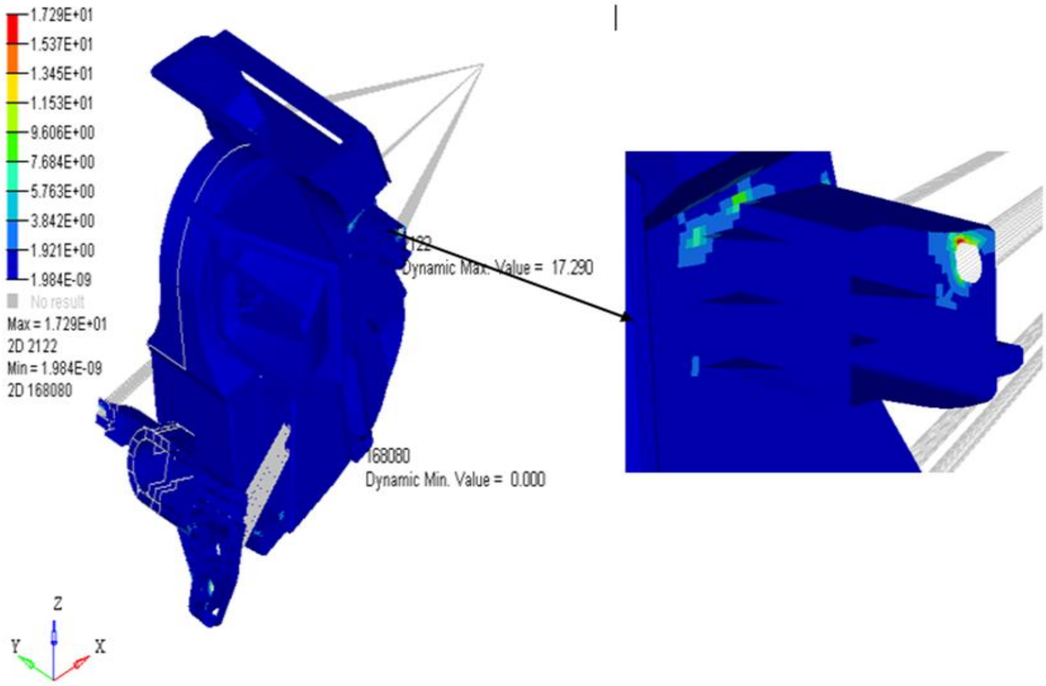


Figure 3.23: Stresses in the z- direction

3.6.3 Modal frequency response analysis by considering the blower mass

Blower is an important part of the HVAC system and it is used to move the air through the system. It is an integral part of the air handling to draw air in the system. It is mounted on the outside wall of the system.

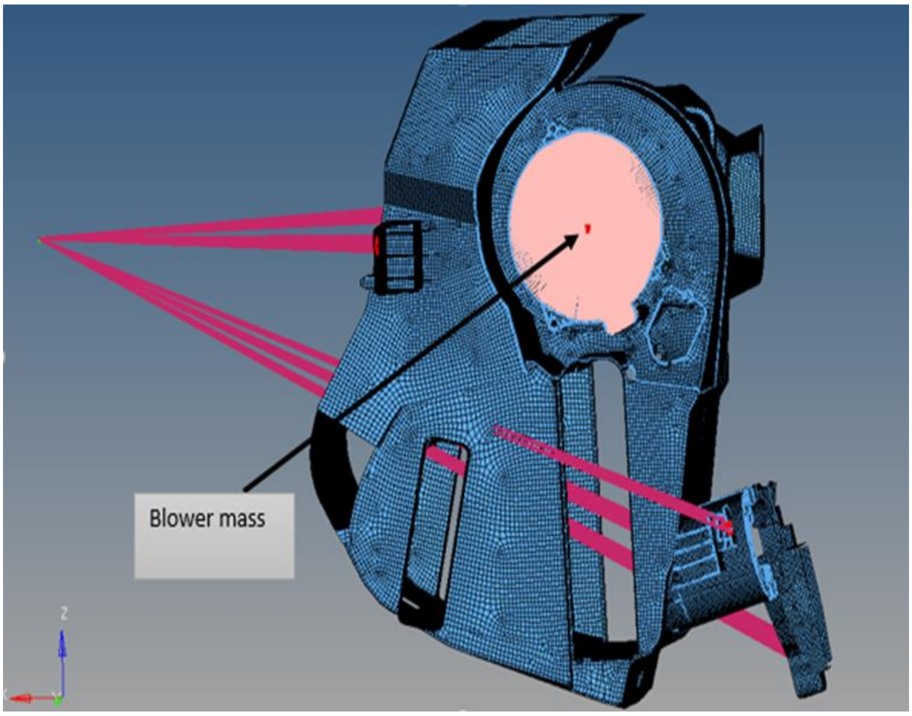


Figure 3.24: Blower mass

The mass is applied where the blower is placed in the HVAC system. RBE3 is used for the distributing the mass and forces. The 0.5 kg mass is applied on the dependent node of RBE3 as shown in the Figure 3.24.

a) Stress in the x-direction

For this analysis, material, properties and the boundary condition are defined as per previous analysis *i.e.* frequency response analysis. The force is applied only in the x- direction. After the analysis, the results are obtained as shown in Figure 3.25.

b) Stress in the y-direction

All the materials, properties and the boundary conditions are same as in the previous analysis except the dynamic force is applied in the y- direction instead of x-direction. The force is applied in the y-direction on the dependent node of RBE3. No stresses are obtained in the HVAC system in the y-direction.

c) Stress in the z-direction

The dynamic force is applied in the z- direction instead of y-direction in this case. The force is applied on the dependent node of RBE3. The stresses are shown in the given Figure 3.26.

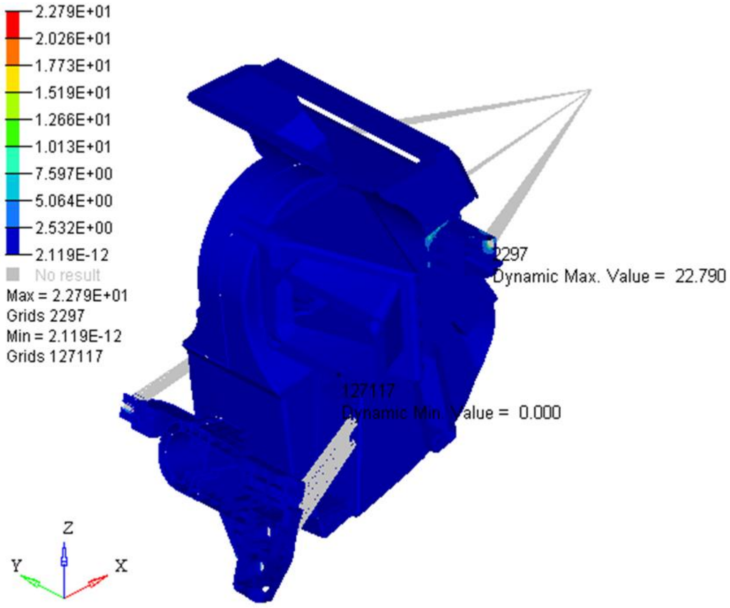


Figure 3.25: Stresses in the x- direction

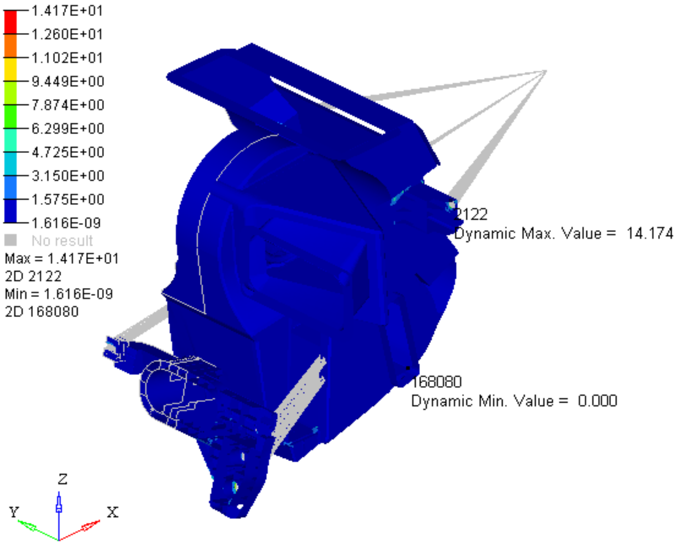


Figure 3.26: Stresses in the z-direction

3.7 Application of the proposed virtual testing for different materials

The proposed CAE model of the HVAC system can be used as a testing rig, for evaluating different materials and design improvements, Different material have been used for the HVAC and the stresses are determined using the modal frequency response analysis. The different materials and properties are shown in Table 3.2.

Table 3.2: Mechanical properties of some materials

S. No.	Types of the material	Elastic modulus (E) in N/mm^2	Poisson ratio (μ)	Density (ρ) kg/mm^3
1.	Nylon6/6 (PA66)	2800	0.40	1.14e-06
2.	PVC (poly vinyl chloride)	2700	0.40	1.4e-06
3.	ABS (acrylonitrile butadiene styrene)	2300	0.35	1.2e-06
4.	PA66-33%GF	7500	0.35	1.12e-06

3.7.1 Frequency response analysis of HVAC with different material

The material properties are assigned, as per Table 3.2.The boundary conditions *i.e.* constraints and forces are applied in the x, y and z-direction respectively. The results of the analysis are shown in the Table 3.3.

Table 3.3: Stresses and displacement in the x, y, z-direction

S. No.	Type of material	Stresses (N/mm^2)			Displacement (mm)		
		direction			direction		
		x	y	z	x	y	z
1.	Nylon6/6 (PA66)	39.89	0	21.28	6.227	0	4.209
2.	PVC	28.64	0	16.50	4.713	0	3.388
3.	ABS	28.33	0	15.65	5.620	0	3.920
4.	PA66-33%GF	52.00	0	61.26	26.80	0	4.700

It was observed from the above Table 3.3, the induced stresses in the x and z-direction of the different material *i.e.* Nylon 6/6, PVC, ABS, and PA 66- 33% GF are higher as compared to the polypropylene material.

CHAPTER 4 OPTIMIZATION OF HVAC

4.1 Optimization design iteration 1

In the optimization, material was removed manually from the model, where the stresses were minimum. The front surface of the HVAC (shown in Figure 4.1) had lower stresses as compared to the other surfaces.

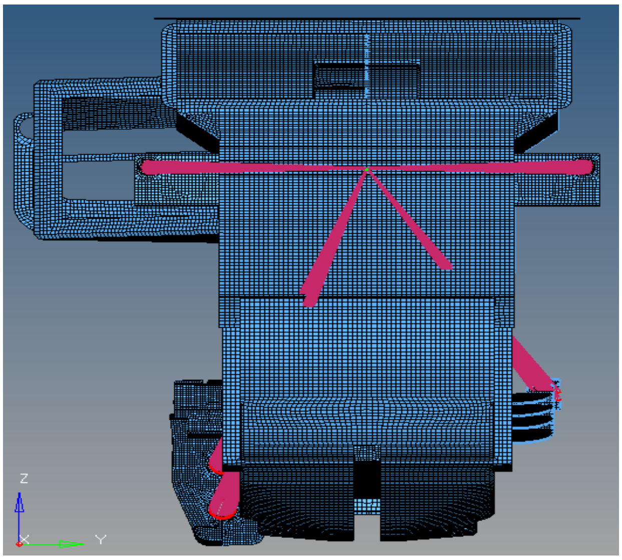


Figure 4.1: Before material removal

After removing the material, the front surface of the HVAC is as shown in Figure 4.2.

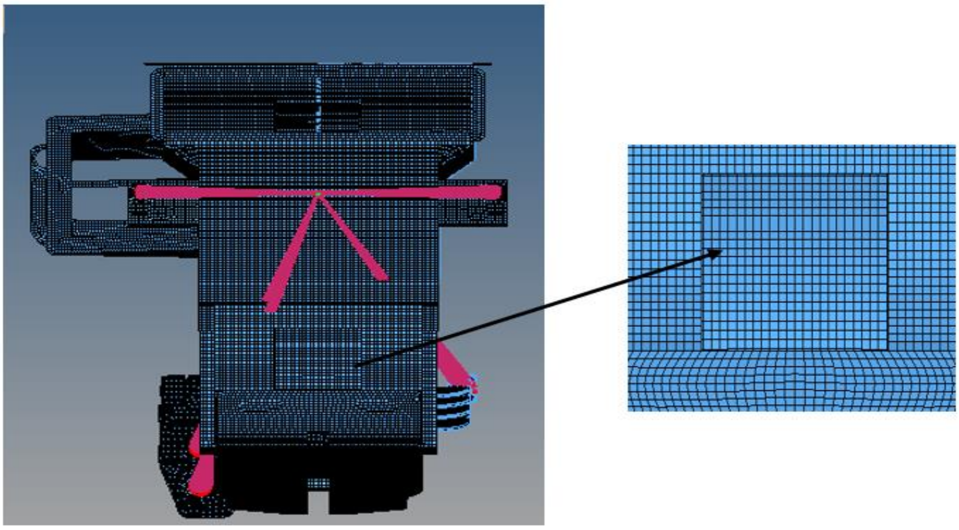


Figure 4.2: After material removal

4.1.1 Modal frequency response analysis of the optimized design 1 of HVAC

a) In the x-direction

The material, properties and load collector are same as the modal frequencies response analysis in the x-direction. The stresses of the optimized HVAC is less than that of as compared the original HVAC in the x-direction. The results are as shown in Figure 4.3.

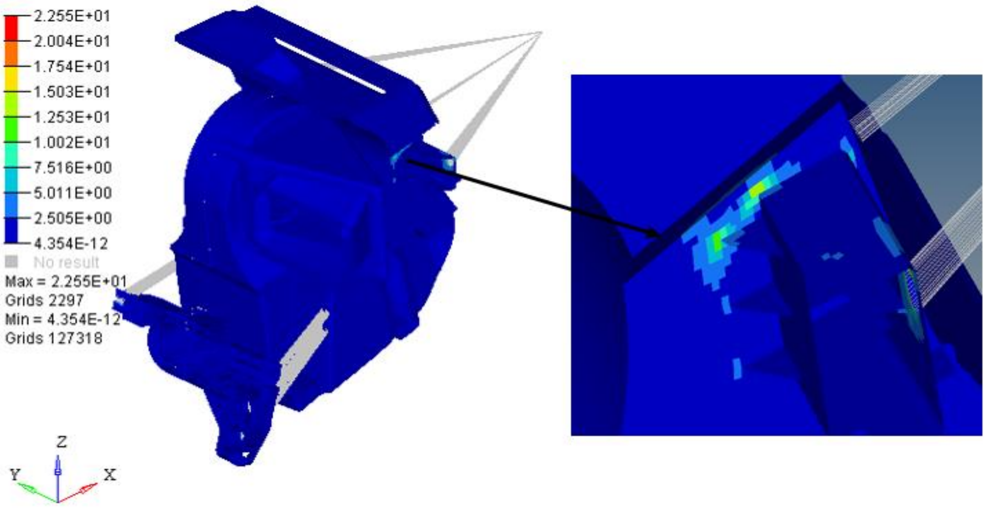


Figure 4.3: Stresses produced in HVAC in x-direction

b) In the y-direction

All the material, properties and the boundary condition are same as the previous analysis expect the force is applied in the y-direction on the RBE3 node. The results in the y-direction are as shown in Figure 4.4.

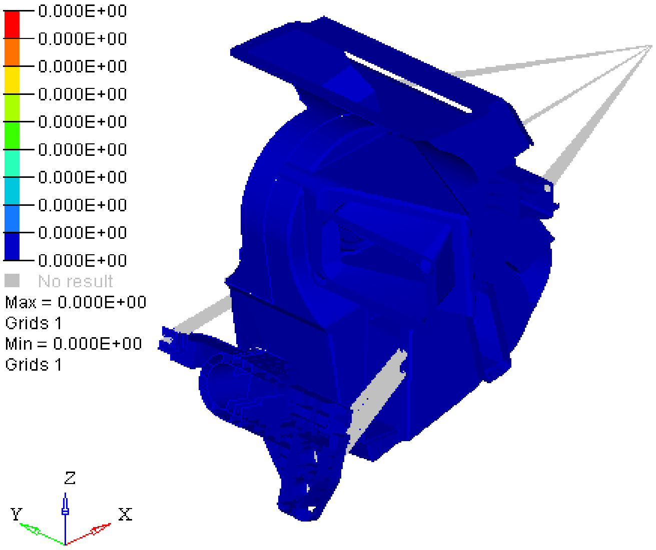


Figure 4.4: Stresses produced in HVAC in y-direction

c) In the z-direction

In this analysis, the force is applied in the z-direction and all the boundary conditions are the same as the previous analysis. The stresses in the z-direction of the optimized HVAC as shown in Figure 4.5.

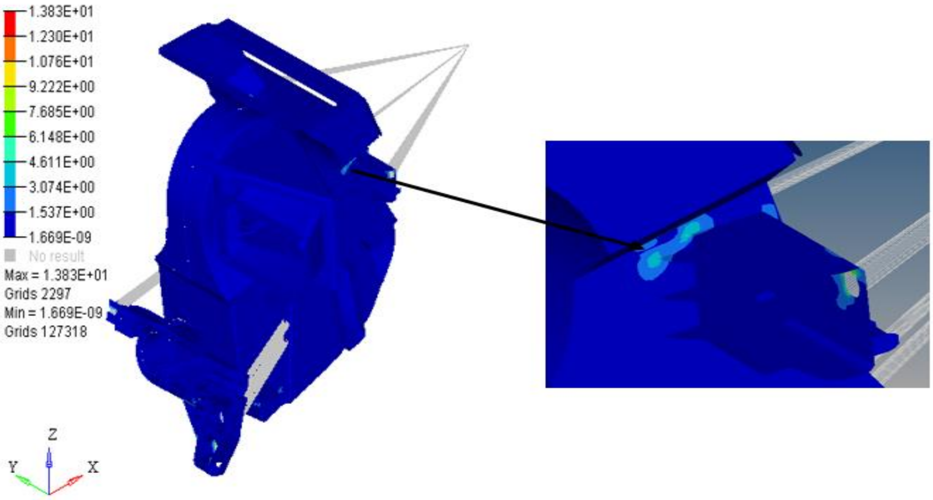


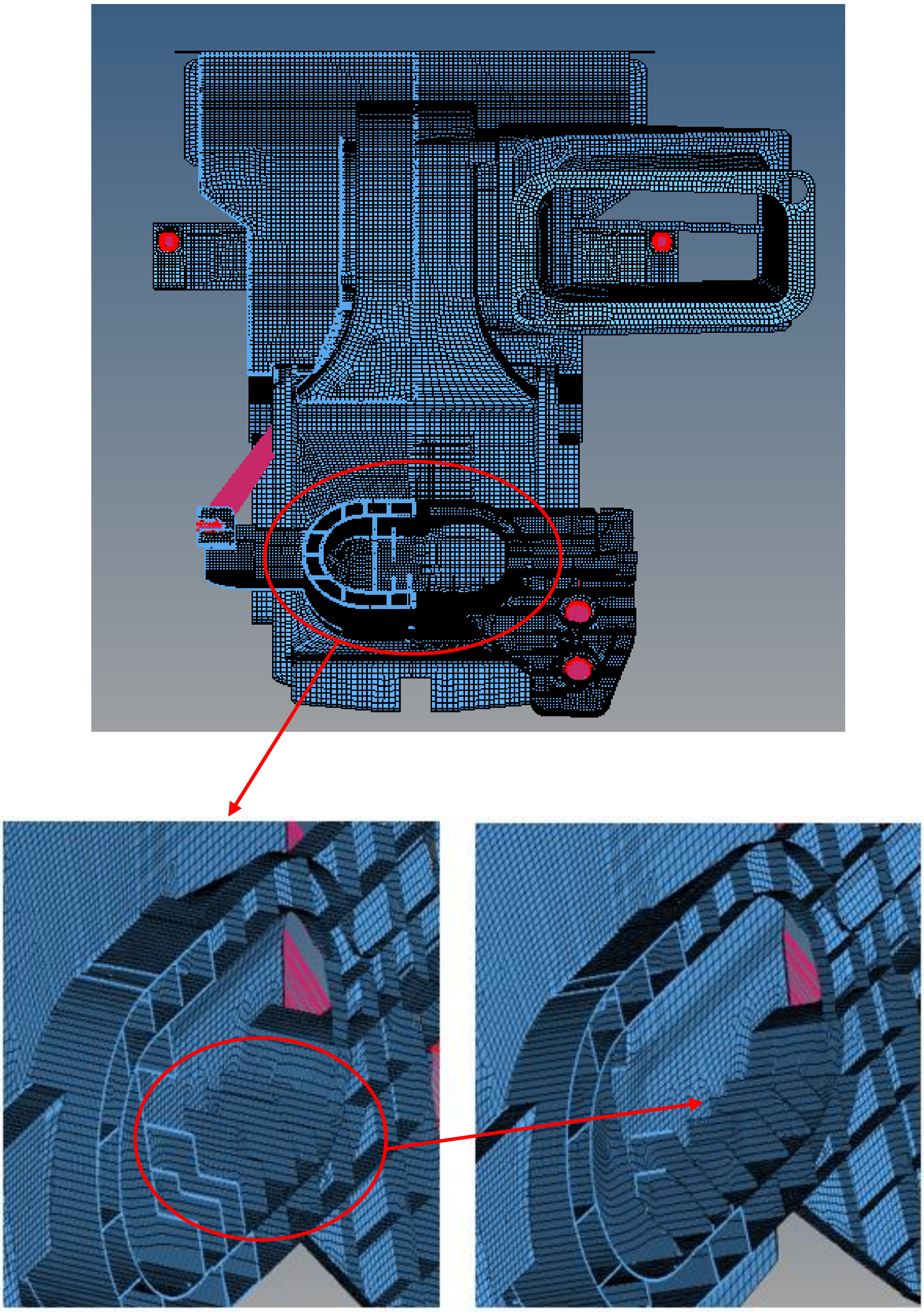
Figure 4.5: Stresses produced in HVAC in z-direction.

4.2 Optimization design iteration 2

The product was further optimized for reduced weight by again removing the unwanted material and performing the frequency response modal analysis in all direction as described below.

The properties of the material *i.e.* polypropylene which is used in the analysis, are given in Table 3.1.

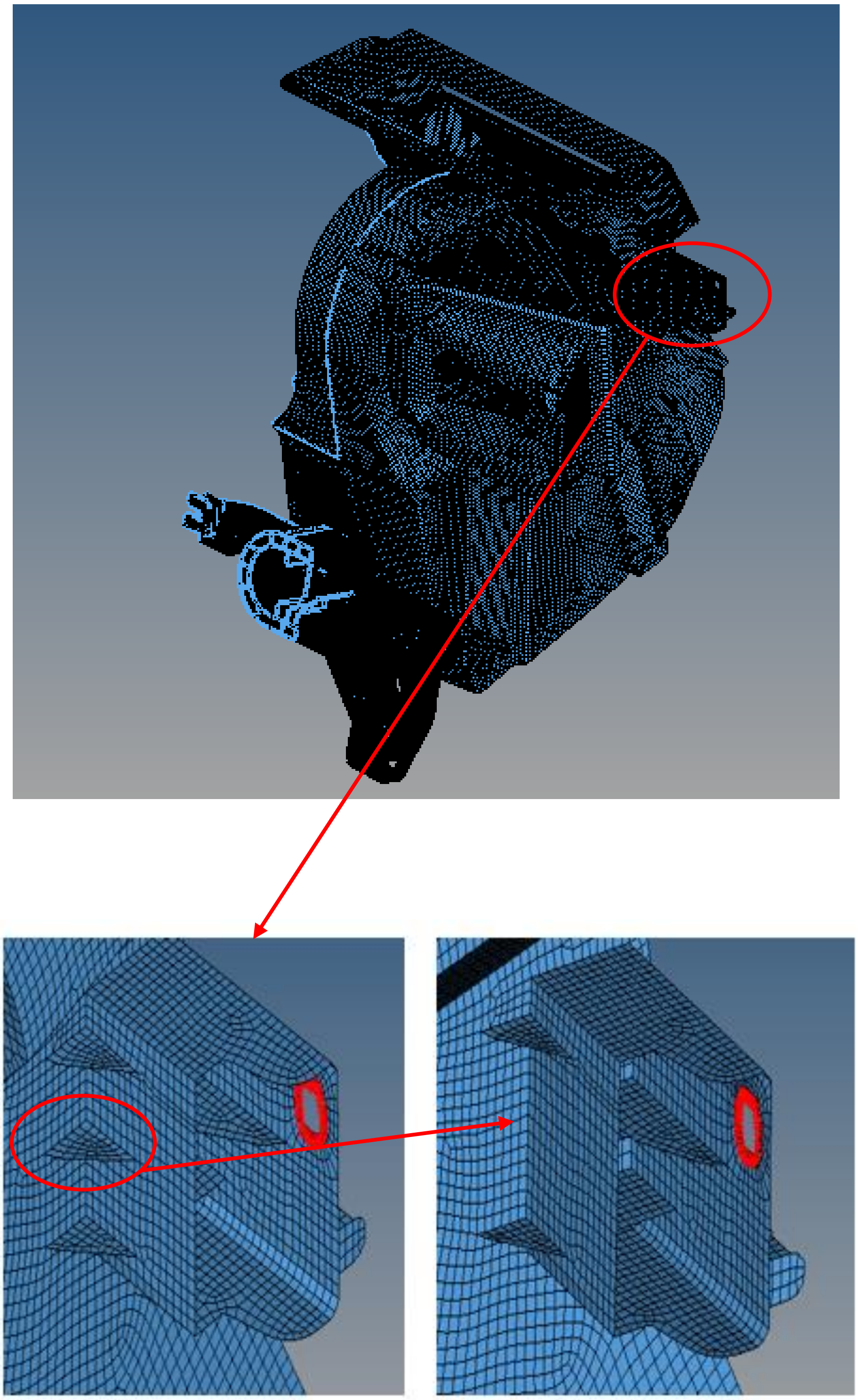
In this optimization design iteration 2, material was removed from two places. First, the shape of ribs was changed as shown in Figure 4.6 and second, removal of ribs was done from the two mounting locations of the HVAC as shown Figure 4.7.



a) Before optimization b) After optimization

Figure 4.6: Modified ribs

After the optimization, stresses were determined by modal frequency response analysis as described below:



a) Before optimization

b) After optimization

Figure 4.7: Modified ribs on mounting location

After the optimization, stresses were determined by modal frequency response analysis as described below:

4.2.1 Modal frequency response analysis of the optimized design 2 of HVAC

a) In the x-direction

The force was applied in the x-direction on the node of the RBE3. After the analysis, it was observed that, the stress obtained after optimization design 2, was slightly higher than the stress obtained after optimization design 1. But it was within the permissible limits as per Table 3.1. The results are shown in Figure 4.8.

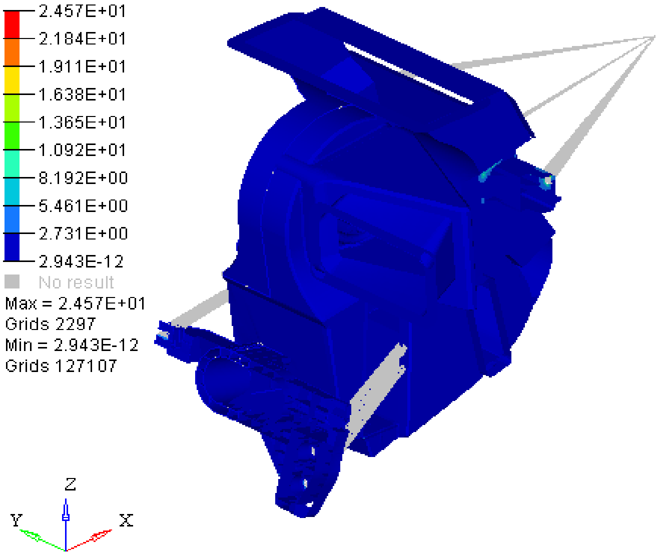


Figure 4.8: Stresses in x-direction

b) In the y-direction

In this analysis, all the boundary conditions were the same as per the previous analysis in the y-direction. It was observed that, the value of stresses in the y-direction did not change in either of the optimized design and the original HVAC.

c) In the z-direction

In this analysis, all the values of the material, properties and the load collector were defined as per the previous analysis in the z- direction. After the analysis, it was observed that, stresses produced in the optimized design 2 are slightly greater than the stresses produced in the optimized design 1, but slightly lesser than the stresses induced in the original HVAC. The stresses are shown in Figure 4.9.

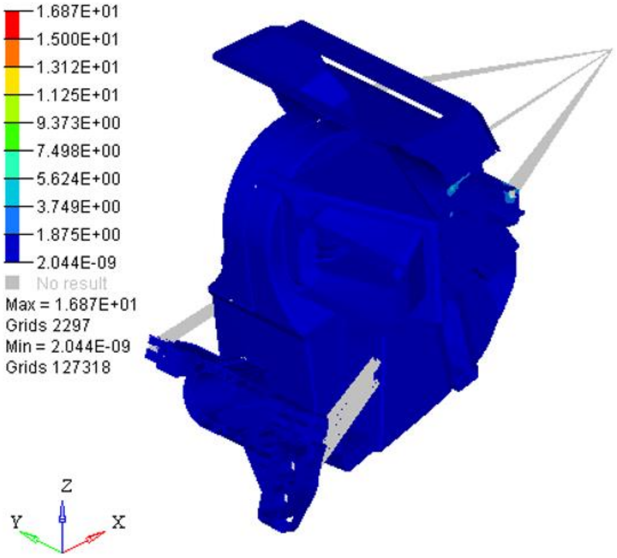


Figure 4.9: Stress in z-direction

It was observed that from the above analysis, the induced stresses in the optimized design 2 are greater than induced stresses in the optimized design, but still it was within the permissible limit in the x and z- direction. And in the y-direction, it was same either of the optimized design. The weight was reduced approximately by 4% from the optimized design 1.

The above design modifications which have been suggested using the proposed methodology should be discussed among the team consisting of tool room engineer, CFD (computational fluid dynamics) analyst and other stakeholders involved in the design of the system, before finalizing it. The CAE tools used also provide a platform for following the practices of concurrent engineering.

CHAPTER 5 CONCLUSION

5.1 Conclusion

Static structural analysis and dynamic analysis (*i.e.* normal modes analysis and frequency response analysis) were performed on the heating ventilating and cooling module of the car. The stresses produced in the dynamic analysis were higher as compared to the linear structural analysis in all the direction except in the y-direction where the stresses were nearly zero for both the cases. The frequency response analysis was also done by using different materials *i.e.* PA66, PVC, ABS and PA66-33%GF. The stresses induced for all the four types of material were seen to be more as compared to polypropylene material

The maximum stress was obtained in the x-direction of the heating ventilation and cooling module.

After optimization of the HVAC module, weight was reduced approximately by 1.5% from the original HVAC for which the induced stresses were also less.

After further optimization of the HVAC module, weight was reduced approximately by 4% from the optimized design 1 of the HVAC.

It can be seen that the CAE tools can be very useful for the analysis / virtual testing of various automobile components. In the presented study, the linear static and dynamic analysis has been illustrated for the plastic components used in HVAC module of the car air-conditioning system. This methodology can be used for the analysis of similar components used in automobiles and other applications.

5.2 Future scope

The presented work can be extended in different directions in the following ways:

a) Transient response analysis for the HVAC module can be done.

b) Fatigue testing can be done on the HVAC module.

c) Experimental verification can be done using electrodynamic shaker.

References

[1] Nitin S Gokhale (2008), Sanjay Deshpande, Sanjeev V Bedekar and Anand N Thite, “Practical Finite Element Analysis”, Finite to Infinite, India, 2008.

[2] H. S. Park and X. P. Dang, “Development of a Fiber-Reinforced Plastic Armrest Frame for Weight-Reduced Automobiles,” International Journal of Automotive Technology, vol. 12, pp. 83–92, 2011.

[3] C. Wijaya, M. C. Yoon, and B. T. Kim, “Numerical Investigation On The Dynamic Characteristics Of An Automotive A / C Hose,” International Journal of Automotive Technology, vol. 13, pp. 433–440, 2012.

[4] M. Singh, D. Singh, and J. S. Saini, “Dynamic Analysis of Condenser Assembly of Automobile Air Conditioning System Using CAE Tools,” Journal of The Institution of Engineers, vol. 94, pp. 135–145, 2013.

[5] B. Singh, D. Singh, and J. S. Saini, “A New Method to Design Cam Used in Automobile Heating, Ventilating and Cooling System,” Journal of The Institution of Engineers, vol. 93, pp. 275–286, 2012.

[6] S. Subbiah, O. P. Singh, S. K. Mohan, and A. P. Jeyaraj, “Effect of muffler mounting bracket designs on durability,” Engineering Failure Analysis, vol. 18, pp. 1094–1107, 2011.

[7] U. S. Ghorpade, D. S. Chavan, V. Patil, and M. Gaikwad, “Finite Element Analysis and Natural Frequency Optimization of Engine Bracket,” International Journal of Engineering Research & Technology, vol.1, pp. 1–6, 2012.

[8] T. D. Tsai and C. C. Cheng, “Structural design for desired Eigen frequencies and mode shapes using topology optimization,” Struct Multisisc Optim, vol.47,pp. 673–686, 2013

[9] S. K. Loh, W. M. Chin, and W. F. Faris, “Fatigue analysis of Package Terminal Air Conditioner motor bracket under dynamic loading,” Material and Design, vol. 30, pp. 3206–3216, 2009.

[10] M. Deshmukh and K. R. Sontakke, “Analysis and Optimisation of Engine Mounting bracket,” International Journal of Scientific Engineering and Research, vol. 35, pp. 2231- 5381, 2016.

[11] D. Y. Chen, L. M. Wang, C. Z. Wang, L. K. Yuan, T. Y. Zhang, and Z. Z. Zhang, “Finite Element Based Improvement Of A Light Truck Design To Optimize Crashworthiness,” International Journal of Automotive Technology, vol. 16, pp. 39–49, 2015.

[12] Y. Basavaraj and T. H. Manjunatha, “Design Optimization of Automotive Engine Mount System,” International Journal of Engineering Science Invention, vol. 2, pp. 48–53, 2013.

[13] K. Eads, K. Haghighi, H. J. Kim and J. M.Grace “ Finite Element Optimization of an Exhaust System”, SAE Technical Paper, Doi: 10.4271/2000-01-0117, 2000.

[14] M. D. Rao, K. J. Wirkner and S. Gruenberg, “Dynamic Charactersistics of Automotive Exhaust Isolators”, Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering, vol. 218, pp. 891–900, 2004.

[15] X. T. Pan, D. Zonni, G. Z. Chai, Y. Q. Zhao and C.-C. Jiang “Structural Optimization for Engine Mount Bracket”, SAE Technical Paper, Doi: 10.4271/2006-01-2665, 2007.

[16] M. Cevik, E. Kanpolat and M. Rebbert “Shape Optimization of Single Cylinder Engine Crankshaft”, SAE Technical Paper, Doi: 10.4271/2011-01-1077, 2011.

[17] J. W. Chang and Y. S. Lee “Topology Optimization of Compressor Bracket”, Journal of Mechanical Science and Technology, vol. 22, pp. 1668–1676, 2008.

[18] K.S. Raghavan, T. Howard, J. Buttles and N. Law, “Weight Reduction of Structural Vibration Isolation HydroMount Bracket through Design Analysis and Use of Advanced High Strength Steels”, SAE Technical Paper, doi: 10.4271/2009-01-1230, 2009.

[19] http://www.slideshare.net/ValeoService/valeo-air-conditioning-valeoscope-technical-brochure-ac-handbook-part-1-998321. (Accessed on 25/06/2016).