

# Development and Analysis of Automotive Car Door Crop by using FEA Method

Yashas S

Undergraduate Student,

Department of Mechanical Engineering, Visvesvaraya Technical University,  
East West Institute of Technology, Anjana Nagar, Bengaluru, Karnataka 560-091

**Abstract:-** Door Assembly module is pre-assembly of various components which can be directly mounted onto the door inner panel. These door modules, when directly shipped to the OEM's (original equipment manufacturer), to save assembly time and cost in mass manufacturing. So in this project work carried out with help of FEM by using Hyper mesh as per the quality criteria by using Nastran as a solver for test loads.

Using Finite element method to determine the natural frequency of door trim with respect to considering the mass of other structure, now for this project work we are considering the LH side door. In the door assemblies having some major parts are there in that Door inner trim part is the major one, it will be supporting the door from inner and outer side also including electrical assembly parts like sensors and motor for windshield.

Generally the initial mass of Door assembly in automotive car will be 10.2 kg, which need to be improved to reach the natural frequency of the door trim, by adding stiffeners to reach the target mass of 12.8 kg and reached the desired natural frequency of door trim. once determining the natural frequency door trim purpose is to check which material will be used for manufacturing of the door trim, for the consideration purpose material will be used for door trim is steel, plastic, Aluminum based on their young's modulus.

**Key words:** Car Door, FEM, Hyper mesh.

## I. INTRODUCTION

Door modules are preassemblies of various components on a carrier plate which can be directly mounted onto the door inner panel. Those door modules, when directly shipped to the OEM's, save assembly time and cost in mass manufacturing. Long fiber which is reinforced thermoplastic polypropylene (PP-LFT) with glass filled fiber material is used for various automotive applications as it has improved structural and material properties over conventional materials like steel. Door modules are made up of PP-LFT material is manufactured using injection and moulding method.

As the engineering profession has developed further, analytical techniques were introduced to improve the design procedures. As the components became more and more complex, the more difficult it was to apply analytical techniques to predict fatigue life. Instead, the numerical approaches have been developed, which make it possible to effectively analyse complex structures and obtain solutions faster. These numerical approaches use both the data obtained from multiple experimental testing together with the analytical correlations developed.

For the effective usage of these numerical approaches, knowledge of Finite Element Analysis (FEA) is required. In a same progressive manner as fatigue, FEA is now an

integral part of structural design. FEA allows the design analysts and engineers to create virtual models and perform simulations in the preliminary design stage. As such, a considerable amount of cost and time are been saved. FEA is being done by using Hyper mesh for pre-processing, Nastran as solver and Hyper view for post- processing.

### 1.1 Modal analysis

Modal analysis is a field of measuring and analysing the dynamic response of structures or fluids without excitation. It is a study of the dynamic properties of the structures under a vibrational excitation.

In the Structural engineering, modal analysis uses overall mass and stiffness of a structure to find the various periods at which it will naturally resonate.

#### 1.1.1 Types of analysis:

1. Linear static analysis.
2. Non-linear static analysis.
3. Dynamic analysis.
4. Buckling analysis.
5. Thermal analysis.
6. Fatigue analysis.
7. Optimization.
8. CFD analysis.
9. Crash analysis.
10. NVH analysis.

#### Linear static analysis:

Linear means a straight line. In real life, a material after crossing the yield point materials follows a non-linear curve but software always follows the same straight line. The Components brake into two separate pieces after crossing the ultimate stress but software based analysis never show failure in this fashion. It shows the single unbroken part with the red colour zone at the location of failure. The linear solver assumes that all material behaves in linear elastic manner.

#### 1.2 Terms of vibration:

**Natural frequency:** It is the frequency at which a system or body tends to oscillate in the absence of external driving force or damping force.

Free vibrations of an elastic body are called natural vibrations and occur at frequency called as natural frequency.

It is given by mathematical equation,  $F_n = 1/T$  Hertz.

Where,

T is Time period in seconds.

**Frequency:** It is the rate at which something occurs over a particular period of time i.e. the number of cycles per unit time.

**Amplitude:** While the body is said to be in or under the influence of vibration, the maximum displacement of a vibrating body from the mean position is known as vibration. It is denoted by 'A'.

**Resonance:** It is a phenomenon in which vibrating system or external force drives another system to oscillate with greater amplitude at a specific preferential frequency.

Frequencies at which the response amplitude is a relative maximum are known as the system's resonance frequency. At a resonant frequency, small periodic driving forces have the ability to produce large amount of amplitude oscillations, due to the storage of vibrational energy.

**Degrees of freedom:** A minimum number of independent coordinates required to determine completely the positions of all parts of a system at any instant of time defines the number of degrees of freedom of the system.

Systems with finite numbers of degree of freedom are known as discrete systems, and those systems with infinite number of degrees of freedom are known as continuous systems. Any structure in space will possess six degrees of freedom as shown in the figure below.

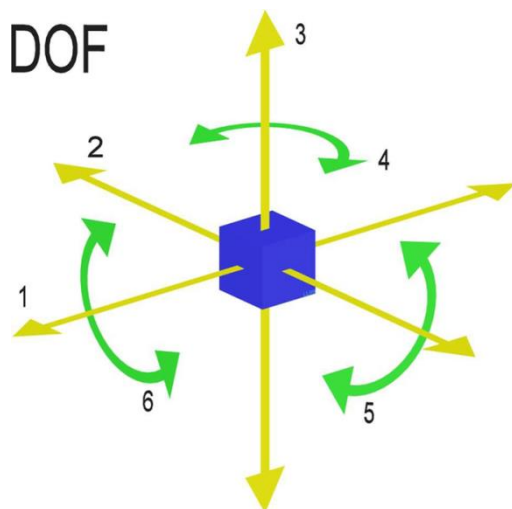


Fig. 1.1: Degrees of freedom.

### 1.3 Concepts of vibration:

Any motion that repeats itself after a fixed interval of time is known as vibration. A swinging pendulum and a motion of plucked string are typical examples of vibration.

A vibratory system, generally includes a means of storing for potential energy (spring or elastic), a means of storing for kinetic energy (mass or inertia), and a means by which energy is gradually lost (damper).

Classification of vibration:

1. Free and forced vibration.
2. Damped and un-damped vibration.
3. Linear and non-linear vibration.
4. Deterministic and random vibration.

### Free vibration:

If a system, after an initial displacement or disturbance, is left to vibrate on its own, that vibration possessed by the body is known as free vibration. No external force acts on the system.

### Forced vibration:

If a system is subjected to external force i.e., external load then the vibration possessed by the body is known as forced vibration. If the frequency of external force coincides with any of the natural frequencies of the system, there occurs a condition known as resonance. And the system undergoes dangerously large oscillations. While the frequency of vibrations caused meet the natural frequency of the system or instrument, then it causes vigorous damage to the continuum of the system.

### 1.4 FEM (Finite element method):

The Finite element method is the numerical method of solving engineering problems. It is also referred to as FEA (Finite element analysis).

**General procedure of FEM:** Discretization: Since the model or structure is an infinite structure for which the problem cannot be solved. Therefore the infinite structure is converted into finite structure by discretizing a part of structure assuming that structure is divided into equal parts.

**Selection of displacement models or nodes:** Nodal displacements are determined. Displacement within the structure is computed. Some values are approximated in order to achieve least error. And model should satisfy few basic requirements to minimize errors.

**Derivation of element matrix:** Nodes are subjected to equivalent loads. Material and geometrical properties of the structure material is specified. Forces at the nodes refer to nodal displacement which relates to stiffness of the structure. By this stiffness matrix is derived, which is denoted by "K". Stiffness matrix refers to the matrix that represents the system of linear equations that must be solved in order to ascertain an approximate solution to the differential equation.

**Derivation of overall matrix:** While there has to be displacement at the node it has to be same for all the adjacent elements. So by this combined element matrix can be determined.

**Solution for unknown displacements:** While the specified boundary conditions are applied as of equation given below the unknown displacement values can be determined.

$$[K]\{Q\} = [F]$$

Where, K is stiffness matrix.

Q is nodal displacement. F is force matrix.

### Areas where FEM can be applied:

1. Structure analysis: Cantilever, Simply supported beam.
2. Solid mechanics: a gear, an automotive train.
3. Dynamics: shear tower, earthquake.
4. Thermal analysis: heat radiation of finned surface, thermal stress brake.
5. Electrical analysis: piezo actuator, electrical signal propagation.

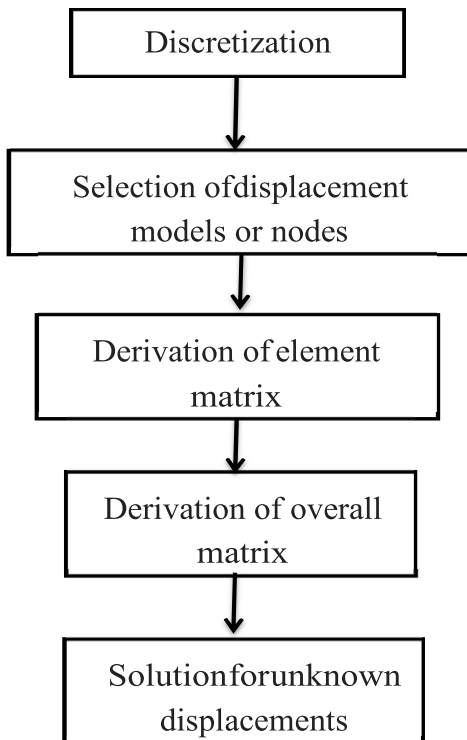


Fig. 1.2 Procedure of FEM.

### 1.5 Software based FEA:

For any software there are 3 steps

#### 1.5.1 Pre-Processing

- CAD data
- Meshing
- Boundary condition

Create the model first using the modeling software that we want to do test or to build prototype. Then import the CAD data & discretize the whole model into smaller, using finite elements (i.e. meshing the CAD model using smaller elements). Then applying boundary conditions (i.e. constraints & loads to be applied to the model) to know the required solution. After completion of pre- processing (i.e., a,b,c), software internally forms mathematical equation of the form  $[F]=[K]*[\delta]$ .

#### 1.5.2 Processing

Processing means solving the given problem (i.e. internally software carries out matrix formation, inversions, multiplications) & gives solution for unknowns (i.e. Displacements, Strains & Stress for static analysis).

#### 1.5.3 Post Processing

Post processing is visualization of results, verification, conclusion & thinking about what step could be taken to improve the design. The results can be viewed in various formats: graph, value, animation etc.

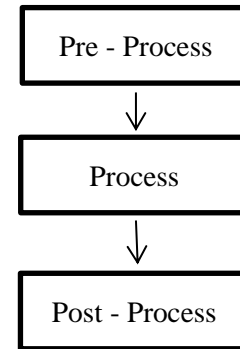


Fig. 1.3 Steps followed for FEA.

### 1.6 Introduction to software used:

- Pre Processor as HYPERMESH
- Processor as NASTRAN
- Post processor as HYPERVIEW

#### 1.6.1 Hyper mesh

Hyper Mesh is a high-performance finite element pre-processor to prepare even the largest models, starting from import of cad geometry to exporting analysis run for various disciplines. If element size data of meshing is not specified by the customer convergence study is conducted to determine element size. Care should to be taken to decide the type of element to be used like 1D, 2D or 3D elements. Special considerations has to be taken while deciding the elements at welded regions, solder joints etc., with a focus on engineering productivity, Hyper Mesh is the user- preferred environment for:

- Solid Geometry Modelling.
- Shell Meshing Model.
- Morphing Detailed Model.
- Setup Surface Geometry.
- Modelling Automatic Mid-Surface.
- Generation Batch meshing.

#### 1.6.2 Nastran

Nastran is acronym formed from NASA Structure Analysis; Nastran is a finite element analysis (FEA) program that was developed by NASA in the late 1960s by Stephen Burns of the University of Rochester. Generally used Nastran versions are MSC (MacNeal-schwendler) Nastran or CSC (Computer Sciences Corporation) Nastran NX Nastran. Nowadays, Nastran is widely used in the fields of aerospace, automotive and maritime industries. Nastran is written primarily in FORTRAN and contains over million lines of code. These modules are controlled by an internal language called the Direct Matrix Abstraction Program (DMAP).

Few most common solution sequence codes are

- 101-Linear static
- 103-Modal or Dynamic
- 105-Buckling 106-Non-Linear Static
- 107-Direct Complex Eigenvalue
- 108-Direct Frequency Response
- 109- Direct Transient Response

- 110-Modal Complex Eigen value
- 111-Modal Frequency Response
- 112-Modal Transient Response
- 129-Nonlinear Transient
- 144-Static Aero elastic analysis
- 145-Flutter/Aero servo elastic Analysis
- 146-Dynamic Aero elastic Analysis
- 153-Non-linear Static coupled with Heat transfer
- 159- Non-linear transient coupled with Heat transfer
- 187-DDAM
- 200-Design Optimization and sensitivity analysis
- 600-Non-linear static and dynamic(implicit)
- 601-Implicit Non-linear
- 700-Explicit Non-linear

### 1.6.3 Hyper view

It is actually a post processor which is used to view the results visually and graphically. Various types of results are viewable in hyper view like stress plots, displacement plots, various modes of vibrations, etc. From these we can draw conclusions on the components analyzed whether it is safe and can be used for the intended applications.

## 1.7 Material Properties:

### Density:

A material density is defined as its respected mass per unit volume. It is, essentially, the measurement of how tightly matter is crammed together. Take the mass (m) and divide by a volume (v):  $\rho = m / v$

The SI unit of the density is kilogram per cubic meter (kg/m<sup>3</sup>). It is also frequently represented in CGS unit of grams per cubic centimeter (g/cm<sup>3</sup>).

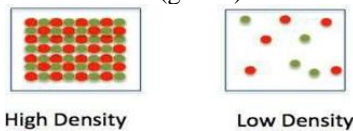


Fig 1.4: Density

### Poisson's ratio:

Poisson's ratio is defined as a negative of the ratio of a lateral strain to the axial strain for a uniaxial stress state. If the tensile load is applied to a material, then the material will elongate on the axis of the load. Conversely, if a load is compressive, the axial dimension will decrease. If volume is constant, the corresponding lateral contraction or expansion must occur. This lateral change will bear a fixed relationship to an axial strain. The relationship, or a ratio, of lateral to the axial strain is called Poisson's ratio.

## 1.8 Theories of failure:

- a) Maximum principal stress theory.
- b) Maximum shear stress or stress difference theory.
- c) Strain energy theory.
- d) Shear strain energy theory.
- e) Maximum principal strain theory.

### 1.8.1 By Default all the post processors gives von-Mises stress plot. What is von-Mises stress and the maximum principle stress?

The stress – strain diagram is plotted from an uniaxial tensile test. These curve is helpful in designing dimensions of component via if tensile force is known, based on yield stress 1 can is easily determined safe area of C/S ( $A=F/\text{stress yield}$ ), but component is subjected to a multiracial loading (normal and shear together) nature of stress- strain curve will not be the same (graph will be different), it indicates for different combinations of loading ( $F_x, F_y, F_z, M_x, M_y, M_z$ ) different graphs should be referred to. Therefore it is not just particle to conduct so many tests. Then how components can be designed when it is subjected to multi-axial loading using available test data for uniaxial loading? It is achieved via theories of failure, which gives us equivalent max. Normal or max. Shear stress energy of the component subjected to multi-axial loading. It is then equated with respective value at yield point (uniaxial tensile test).

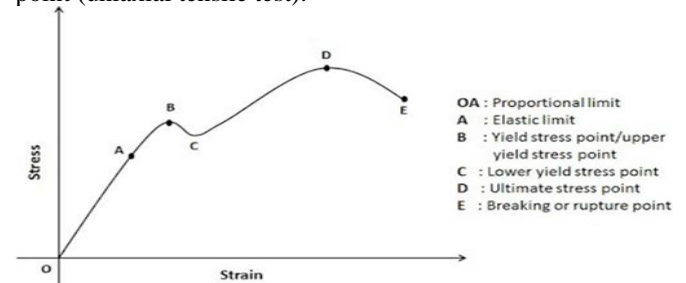


Fig 1.8: Stress-Strain Diagram

**Max principle stress:** Max. Value of the normal stress, the plane on which max. Principal stress acts, shear stress value is zero.

### 1.8.2 Why von-Mises stress is recommended for a ductile and principal stress for brittle material?

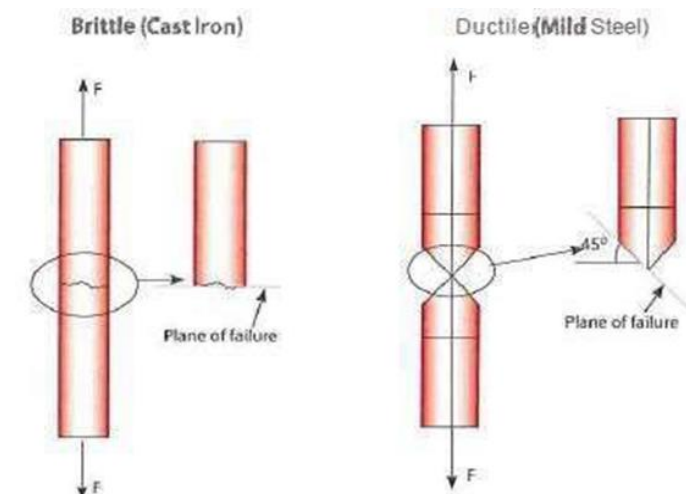


Fig 1.9: von-Mises for brittle and ductile material.

### Brittle:

Failure of a cast iron rod subjected to uniaxial load is along a plane perpendicular to the axis of loading. Clearly failure is due to normal stress. Out of different theories of



failure max. Principal stress theory is one which is based on normal stresses. Hence for brittle material components max. Principal stress is used.

**Ductile:**

Mild steel fails at the plane inclined 45 degree to axis of loading normal stresses cannot act on this plane and the only other possible is shear stress. Out of different theories of failure its max. shear stress and a von-Mises stress which are based on shear stress. Von-Mises stress gives a better correlation with experimental results and hence preferred for ductile materials.

Max. Shear stress is sometimes preferred over von-Mises stress for designing rotating components such as turbine blade and compressor components.

**1.8.3 What is absolute principal stress and signed von-Mises stress?**

**Absolute principal stress:** This term is so commonly used in fatigue analysis. Fatigue calculations are based on an amplitude and mean stress. It has been observed that if the calculations are just based on only the maximum principal stress or only minimum principal stress then stress range is less and leads to higher fatigue life. Remedy is to find max. value out of a two at the point over given period of time and then find the stress range or amplitude and mean stress based on this data (say max. principal stresses at a node at time 1 sec. is +250 N/mm<sup>2</sup> and min. principal is around -400 N/mm<sup>2</sup> then the absolute principal stress is equal to -400 N/mm<sup>2</sup>, collect the data over a period of 10 sec and then find stress range based on max. and min. absolute stress values out of the 10).

**Signed von-Mises stress:** von-Mises and max. Shear stress values are always positive. If those values are used for fatigue calculations then stress range would be reduced to half resulting in higher fatigue life. Remedy is to find out the sign of absolute principal stress at the point at a given time instance and assign it to corresponding value of von-Mises or max. Shear stress. Say absolute max.

Principal stress is -300 N/mm<sup>2</sup> at 2 seconds and corresponding von-Mises stress at time 2 sec is 315 N/mm<sup>2</sup> then signed von-Mises stress would be -315 N/mm<sup>2</sup>.

**II. LITERATURE SURVEY**

**2.1. H. M. A. Hussein and Ahmad Gemeal :**

Conducted analysis on automotive car doors since they not only determine the general guidelines of car style, but also are vital for passenger's safety by protecting humans from side crashes. Predetermined values from the previous works conducted on a similar existing SUV vehicle were used as targets to be achieved by Finite Element Analysis (FEA) of car doors. The modal analysis of the rear and the front doors reveal that the lateral stiffness and vertical displacement analysis the reinforcement must be brought to 1.2 mm at the inner panel. The stress analysis of the rear door suggests addition of a terminal flange in its area. And the stiffness of a door can be increased by percentage of St.42 or by other steels of high strength without increasing weight.

**2.2. O.M. Terciu, I. Curtu :**

He has done research on FEA of an automotive door trim panel made of lignocelluloses composites in order to determine the stresses and displacements in case of a door slam simulation. On his research a new lignocelluloses composites made of polymers reinforced with woven fabrics of natural fibers and wood particles was created. The mechanical properties of these material were determined experimentally and used as input data in FEM simulations. The finite element method result emphasized that new material improves component stiffness compare with classical materials used. The modeling part of the complex geometry requires specialized software to achieve the required model. Since the composite material has high of fixing areas to the metal structure there is equivalent stress distribution and the displacement of the lignocellulose material is 43% smaller than the other polypropylene panel when compared by Finite Element Method. Since smaller displacement results it is due to high rigidity and low weight of the material this decreases the overall weight by 1.81kg to 1.49kg.

**2.3. S. Vinay Seeba, S. Srikari :**

He had studied The design and analysis methodology design adopt plastic door module plate to replace existing steel door module plate, this door modules where directly shipped to the OEM's, which saves assembly time and cost in mass manufacturing. The application of long fibers reinforced thermoplastic has improved structural and material properties over conventional materials like steel. The final design of plastic door module is 33% lesser in weight with improved structure, when compared to steel. The same can be adopted to replace semi- structural automotive parts currently made from steel.

**2.4. M. Raghuveer, Ganti Satya Parkash :**

He had conducted finite element analysis based on implementation or replacing the presently used material steel with composite materials like Aluminum, Carbon Epoxy, S-glass epoxy, E-Glass epoxy, etc. such that the weight of the door can be reduced and impact analysis was conducted on door for different loads by varying the materials. Therefore results show the improvement in the strength of the door as the stresses of the solid work is far less than the material yield stress which indicates us the design is safe. Since the strength is increased by decreasing the weight the manufacturing cost of the car door is increased by using composite material by replacing steel. By above results it can be decided that FEA is a good tool to reduce the time consuming theoretical work.

**2.5. Tan Wei, Wang Yan, Li Lingyang :**

He had studied the Design of The Vehicle Door Structure Based on Finite Element Method. The performance of the door assembly is very significant for the vehicle design. In the present paper, the FEM is used in the development processes of the door assembly. The modal characteristic, stiffness, strength, and anti-extrusion of a newly developed passenger vehicle door assembly were calculated and evaluated by several finite element

analysis software's. The structural problems were discovered by finite element analysis and had been modified and finally achieved the expected door structure performance target of this new vehicle, and its issue in focus is to predict performance of the door assembly by powerful finite element analysis software, and optimize the structure to meet the design targets and it is observed that this method have been used to forecast the performance of vehicle door efficiently when it was designed.

### III. SUMMARIZATION

1. Rear Modal analysis of the rear and the front doors reveals that they is in target. Front doors lower opening must be split into two; Rear door stress analysis suggest the need to extend the latch reinforce or adding the terminal flange in this area. Anti-intrusion bar and mirror reinforcement increases stiffness in a hinged area for both front and rear doors rather than for rear door only, from the strength point of view: there is still wide opportunity to increase the stiffness of doors by increasing the percentage of St. 42 are already used and/or other steels of high strength without increasing weight.

2. Modeling parts with a complex geometry involves the usage of specialized software to achieve a virtual model; From the equivalent stress distribution of an analyzed component can be seen that these areas have high levels on fixing the areas to a metal structure; The displacements of lignocellulose materials door trim the panel obtained by FEM are smaller by 43% than that of the polypropylene panel; Small values of displacements resulting for lignocelluloses composite component are due to high rigidity and also low weight material layer given its lower thickness of the panel, these are gradually decreasing from 1.81kg to 1.49kg.

3. This study helped in designing the plastic door module plate for the car body application using 30% glass fiber PP-LFT (STAMAX®) material, topology and the shape optimization techniques. A final design of the plastic door module is 33% lesser in weight with improved structural properties compared to the existing steel door module plate. The design and the analysis methodology adopted here to design plastic door module plate to replace existing steel door module plate can be adopted to design and the analysis plastic parts which replaces the semi structural automotive parts currently made from steel.

4. Result shows an improvement in the strength of the door as the maximum limit of the stresses. The value of the von-misses stresses that comes out from the solid works is far less than that of material yield stress so our design is safe. The strength of a car door is also increased (weight Reduction) from change of material from steel to Eglass epoxy and sglass epoxy. The cost of the car doors are increased sufficiently by using the composite materials for the car door manufacturing but this decreases the risk from the collisions.

5. There is no physical test in the development of the process of this new vehicle door products, just using the FEM to the discovery of the design problems and will be able to make the target recommendations for an improvement are very effective, quick guide engineer to improve the product quality. This method can reduce the parts of prototype producing and the number of physical tests to shorten the development cycle and reduce the development investment.

6. Modal analysis of the front doors reveals that they are in target. Modal analysis result is obtained for 10 different modes, in that first 5 modes are local modes and the remaining is global modes. The modal frequencies are obtained that is >20 Hz which well away from the engine excitation of 16Hz. This result is for an ideal condition of engine. (Engine at 1000 rpm and load at ideal), Harmonic analysis is performed with and without the stiffener for automotive car door. In a harmonic analysis displacement were obtained for both conditions. With stiffener displacement were less and a safe design.

### IV. OBJECTIVES

- Finite element analysis is very good tool to reduce the time consuming theoretical work.
- It is recommended that the structural analysis optimization has to made for doors.
- Hyper mesh software is used for modal and linear static analysis.
- The natural frequencies of the door trim should be found, and it is suggested that the free natural frequencies are not as same as forced frequencies so it doesn't reach the resonance state.
- Stiffeners are added to the door trim so that the required natural frequencies are obtained.
- Then three materials are selected and have been analyzed with respect to their displacement and stress with applied loads.
- And the suitable materials are selected for manufacturing process of the car door trim.

### V. METHODOLOGY

#### 5.1 Meshing parameters considered for Door trim Analysis:



Fig. 5.1: FE modeling of door trim.

Elements are used for the meshing of door trim is having size of 5mm and remaining quality parameters is explained in the table number 2, Each critical capture

properly to represent the CAD, elements which are used for meshing in both quad and tria element. These 2d shell elements have only translation DOF there is no rotational DOF, for only solid elements, elements information will be provided below.

## 5.2 Quality parameters considered while meshing of doortrim:

- **Warpage:** In shell & hexahedron elements only applicable to determine the warpage. In shell element also if third node is out of plane by a certain angle, error due to warpage can occur, in hexahedron element it will be deviated from the plane from 3 node plane to 4 node plane.
- **Maximum Angle:** Between the two edges of the obtuse angle in quad element should be more than the limit, which will be not accepted for the software.
- **Minimum Angle:** Between two edges in quadrilateral obtuse angle is less than limit.
- **Aspect Ratio:** The ratio of maximum length to minimum length should be less than limit.
- **Jacobian:** The differential matrix which obtained from the formulation of any stiffness called the Jacobian.

Quality parameters	
Warpage in degree	30
Aspect ratio	5
Skew in degree	60
Min. Length	1
Max. Length	8
Jacobian	0.6
Min. Quad Angle	40
Max. Quad Angle	140

Table 5.1: Quality parameters used for meshing of Door trim.

## 5.3 1D & 2D element information used in Door trim modeling

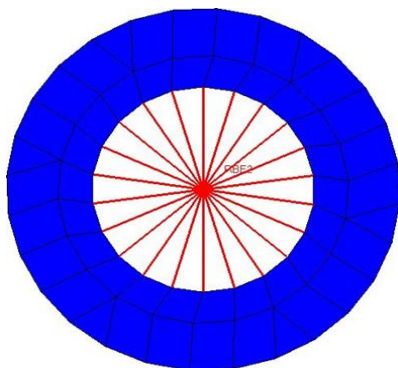


Fig. 5.2: Representation of rigid element.

### 1) Rigid body element (2D element):

#### Properties of rigid elements:

- Rigid element in 1d element, which used to represent the bolt connection and connecting with different components.
- Rigid element will have 6 degrees of freedom, which can release as per the requirement.
- Six degree will be derived into 3 translations and 3 rotation.
- Property of rigid element is to constrain the region of element at that region will not get any displacement or

stress.

e) Rigid element will add the stiffness to original structure, while not in RBE3.

f) RBE2 element distributes the force and moment equally among all the connected nodes Irrespective of position of force or moment application.

g) RBE3 element is a constraint equation to distribute force and moment as per the distance.

### Types of rigid elements:

- RBE2.
- RBE3.
- Bar.
- Beam.
- Rod.
- Spring.

## 5.4 Boundary condition for door trim

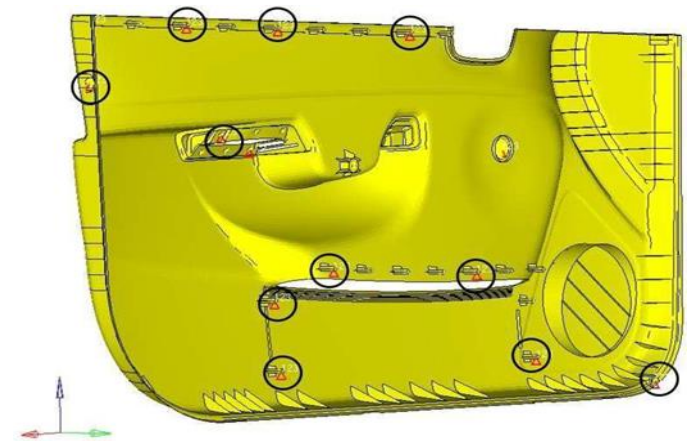


Fig. 5.3: Door trim boundary condition using SPC.

As the door trim is constrained by SPC only in translation direction, SPC located exactly blotted region where which will connect to the remaining door parts. The above Fig shows the typical door trim constrained using SPC.

**Note:** SPC means single point contact (which will be provided with rigid element RBE2).

Single point contact given to node that will be fixed by one or more DOF, the value may be zero or non-zero. In the outside the step we use the non-zero value. Zero SPC can be called in both outside and inside the step. The world SPC can be written in any software \*BOUNDARY\*. Totally 6 degree of freedom (1=translation in x, 2= translation in z, 4=rotation about x, 5= rotation about y, 6= rotation about z), the Rotational degrees of freedom can be applied to beam and shell elements only (11).

## 5.5 Material information for door trim analysis:

Usually door trim part is assigned by plastic material, the properties of plastic material is listed below.



Material	Material properties		
	E MPa	Poisson's Ratio	$\rho$ Kg/mm <sup>3</sup>
Plastic	1500	0.3	7.80E-9

Table 5.2: Plastic material information (standard table from client)

### 5.6 Static Analysis:

The linear static analysis is applied to analyze displacements, strains or stress of structures. This analysis is valid for linear static mechanical problems without any non-linear (geometrically nonlinear, contact, material nonlinear).

Static nonlinear problems can be solved by the nonlinear analysis whereas transient problems are solved by methods.

The static linear analysis formulates structural equilibrium on the initial configuration without consideration of deformations. Hence, it is only valid for problems showing small displacements.

### 5.7 Linear material properties:

Solving of linear problem in all material will behave in a linear elastic manner. Some materials shows non-linear elastic parametric behavior, yield should be above this kind of problem, they still result require non-linear codes & non-linear structural behavior for solution. If a system is to be loaded beyond its yield point, non-linear analysis would also be required. This shows in the figure below for a material comparison behavior. In this step, the constraints, geometry, loads & mechanical properties are applied to develop matrix equations for all elements, which are combined to generate a

global matrix equation of the structure. Forms of the individual equations, as well as the structural equation is always,

$$\{F\} = [K] [U],$$

Where,

$\{F\}$  = External force matrix.

$[K]$  = Global stiffness matrix.

$[U]$  = Displacement matrix.

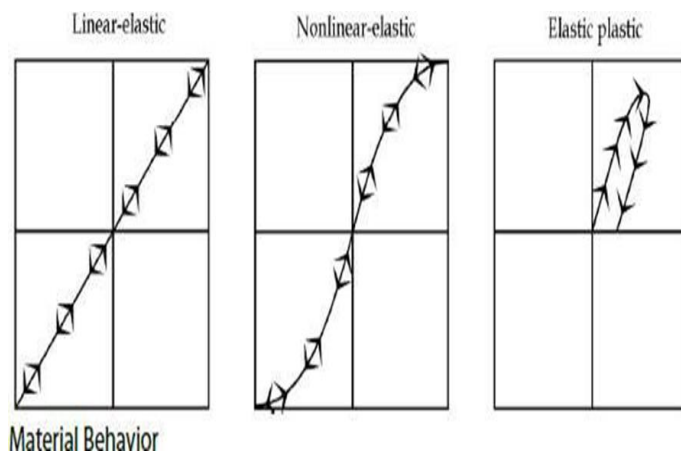


Fig 5.4: Linear, Non-linear, Elastic Material properties.

## VI. RESULTS AND DISCUSSIONS

### 6.1 Model performance of door trim:

Modal analysis is a basic analysis to check the natural frequency of the system, as per the basic analysis door trim having a natural frequency 25Hz, has the model not meeting the modal performance so that we have increased the mass of the door trim, initially door trim model performance was 21.21Hz, so that until reaching natural frequency of door not able to perform the linear static analysis. Increasing the mass of the door trim and by increasing the density and also provide the extra stiffness increase the mass of the door trim, once the mass increase modal performance also increases and thereby attain the desired natural frequency (25Hz).

Modal analysis means, the goal is to find the natural frequencies, mode shapes, Eigen vectors under the free vibration. For the door trim initially undergo the modal analysis in the 1st iteration of modal analysis not reaching the target value of 25Hz, so by increasing the mass of door trim on 2nd iteration then it reaches the natural frequency of 25Hz.

Sl. no.	Mass in Kg	Natural frequency in Hz
Iteration 1	10.2	21.63
Iteration 2	12.72	26.3

Table 6.1: Natural Frequency of door trim at different iteration.

[NOTE: Target natural frequency of door trim is 25Hz.]

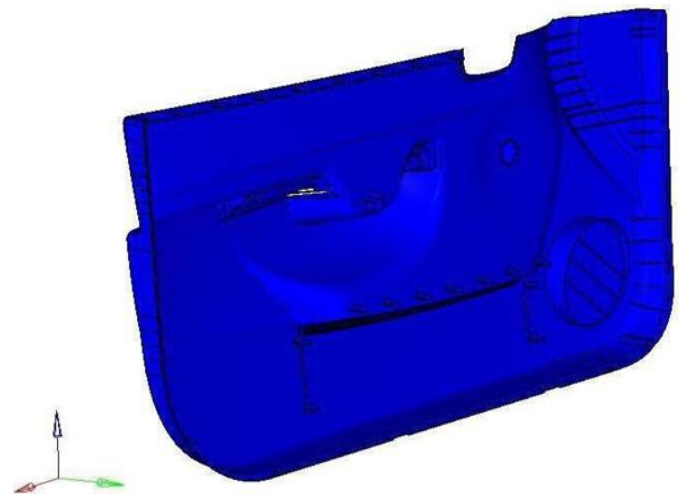


Fig. 6.1: Iteration 1 having mass of 10.2 kg no stiffener is added.

In the figure 6.1, it is seen that door trim has mass of 10.2 kg and modal analysis for the iteration 1 is done.



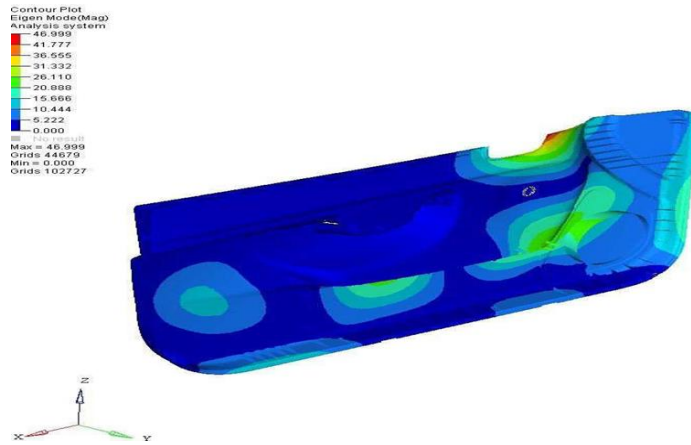


Fig. 6.2: Initial model performance of door trim at 21.3 Hz.

In the figure 6.2, the modal performance result of door trim is obtained as 21.3 Hz.

The desired natural frequency is not obtained from the iteration 1, so in order to achieve the desired natural frequency the mass of the door trim is increased by addition of stiffeners.

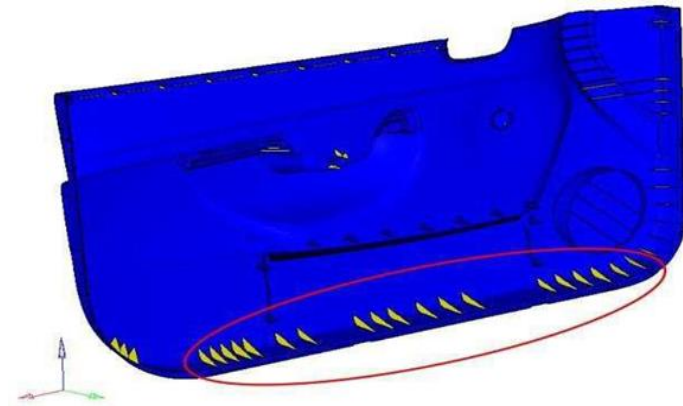


Fig. 6.3: Iteration 2 having mass of 12.72 kg stiffener is added.

Now the increased mass door trim achieves the desired natural frequency as shown in fig. 6.4, and also as the mass is increased by addition of stiffeners, thereby increases the stiffness of the door trim.

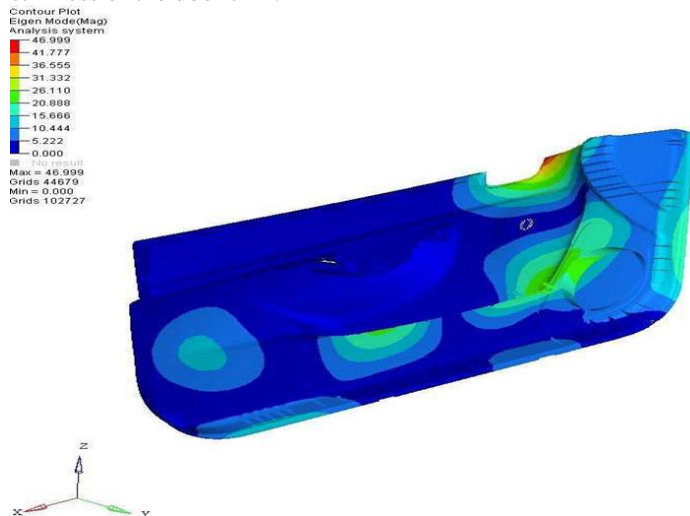


Fig. 6.4: Initial model performance of door trim at 26.21 Hz.

## 6.2 Static analysis of door trim.

Load applied: 100 N

Result: Displacement & Stress plot.

### 1) Displacement Plot:

[NOTE: Target Stiffness is  $15 \pm 0.5$  N/mm].

#### • For Steel:

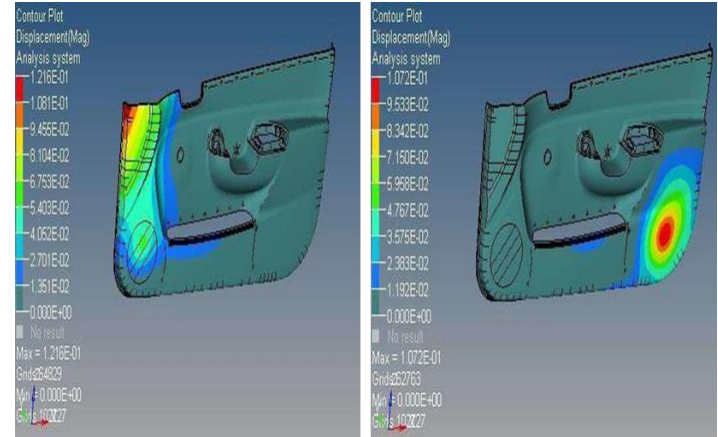


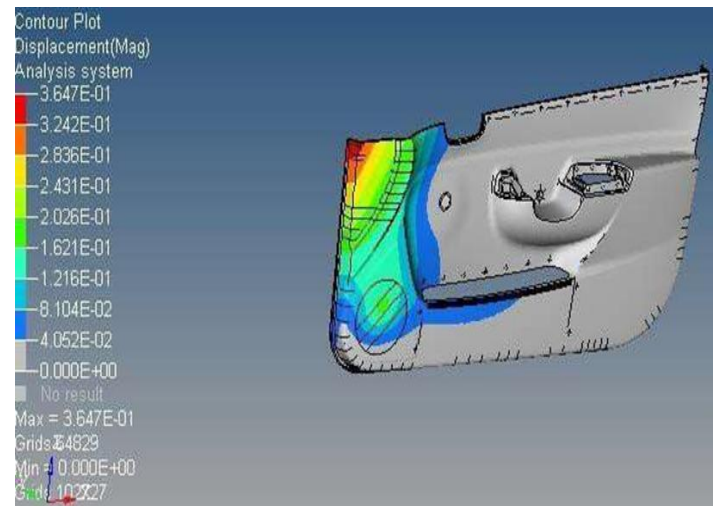
Fig. 6.5: Displacement Plot results for Steel

Loading Points	Load (N)	Displacement (mm)	Stiffness (N/mm)
For Steel:			
1	100	0.12	833.33
2	100	0.17	588.23

Table 6.2: Displacement results for static loading of Steel.

From the static loading the stiffness and displacement values are obtained as shown in fig. 4.5, in which stiffness in both cases is beyond the required  $15 \pm 0.5$  N/mm stiffness. So Steel is not suitable.

#### • For Aluminium:



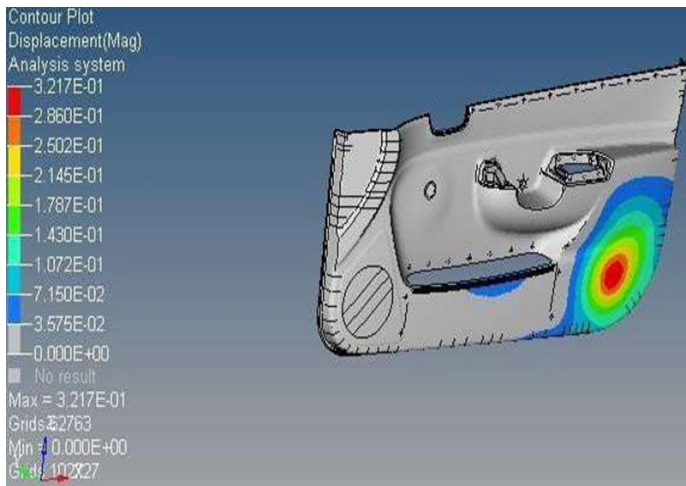


Fig. 6.6: Displacement Plot results for Aluminium.

Loading Points	Load (N)	Displacement (mm)	Stiffness (N/mm)
For Aluminum:			
1	100	0.36	277.78
2	100	0.13	769.23

Table 6.3: Displacement results for static loading of Aluminium.

From the static loading the stiffness and displacement values are obtained as shown in fig. 6.7, in which stiffness in both cases is beyond the required  $15 \pm 0.5$  N/mm stiffness. So Aluminum is not suitable.

• For Plastic:

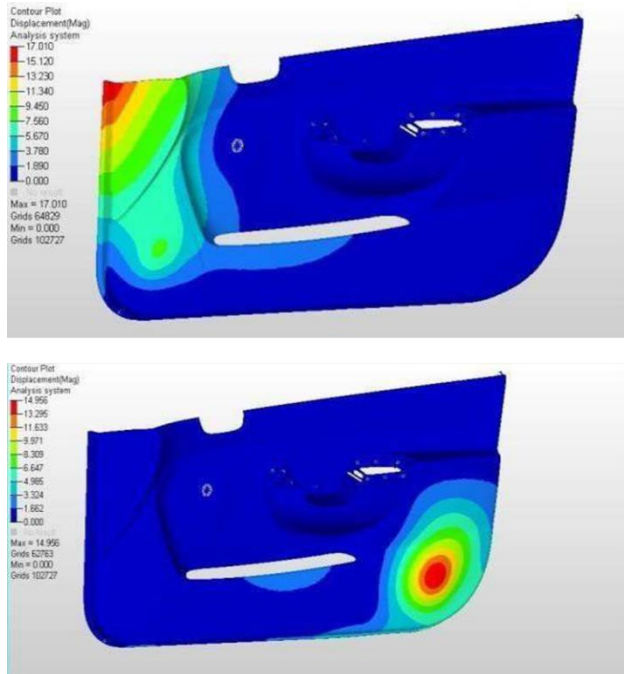


Fig. 6.8: Displacement Plot results for Plastic.

Loading Points	Load (N)	Displacement (mm)	Stiffness (N/mm)
For Plastic:			
1	100	14.95	6.68
2	100	6.46	15.47

Table 6.4: Displacement results for static loading of Plastic.

From the static loading the stiffness and displacement values are obtained as shown in fig. 6.8, the stiffness in both cases is within the required  $15 \pm 0.5$  N/mm stiffness. So Plastic is suitable.

2) Stress Plot:

• For Steel:

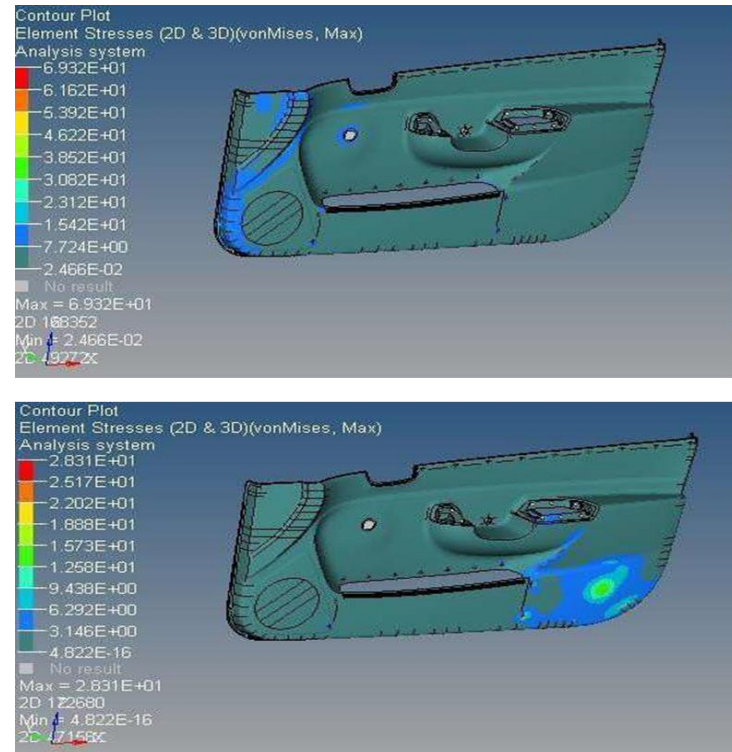


Fig. 6.9: Stress Plot results for Steel.

Loading Point	Load (N)	Stress Mpa
For Steel:		
1	100	69.32
2	100	28.31

Table 6.5: Stress results for static loading of Steel.

From Static Loading the stress obtained are 69.32 & 28.31 Mpa, at the loading points 1 and 2 respectively for steel, shown in the table 6.5.

• For Aluminum:



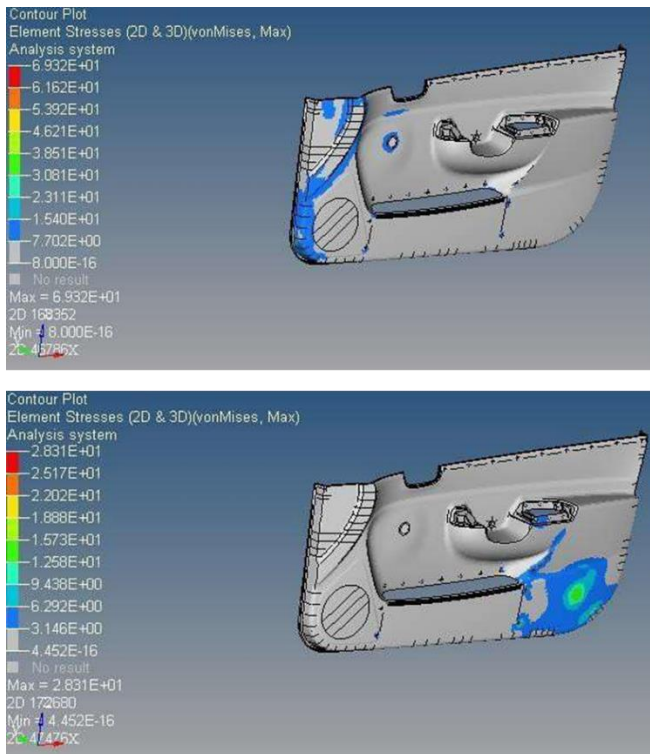


Fig. 6.10: Stress Plot results for Aluminum.

Loading Point	Load (N)	Stress Mpa
For Aluminum:		
1	100	69.32
2	100	28.31

Table 6.6: Stress results for static loading of Aluminum.

From Static Loading the stress obtained are 69.32 & 28.31 Mpa, at the loading points 1 and 2 respectively for Aluminum, shown in the table 6.6.

#### • For Plastic:

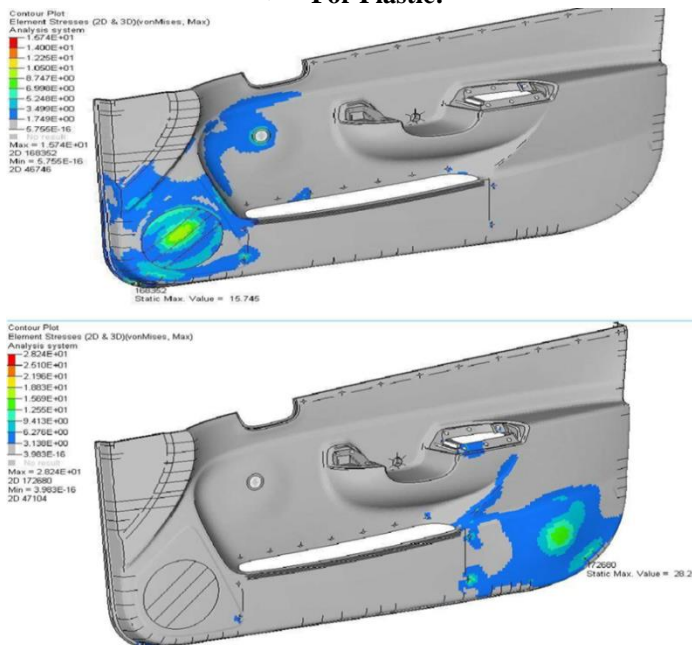


Fig. 6.11: Stress Plot results for Plastic

Loading Point	Load (N)	Stress Mpa
For Plastic:		
1	100	15.745
2	100	28.240

Table 6.7: Stress results for static loading of Plastic.

From Static Loading the stress obtained are 19.03 & 13.38 Mpa, at the loading points 1 and 2 respectively for plastic, shown in the table 6.7 and plastic is suitable.

## VII. CONCLUSION

Modal analysis is based on finite element method to determine the life of door trim, under the loading of 100 N. Simulation results is validated with experimental results. Static analysis was conducted for determining stiffness of door trim, in which results for displacement and stress is obtained.

1. The desired natural frequency of door trim weighing 10.2 Kg is more than 25 Hz, which is not obtained by the door trim. So the weight of door trim is increased to 12.72 kg (by adding stiffeners) by which natural frequency of 26.21 Hz is produced.
2. By having three materials as choice namely Steel, Aluminum, Plastic in order to manufacture door trim it is necessary to determine the best material in terms of requirement and usage.
3. Based on displacement and stress plot it is determined that plastic is the suitable material for the manufacture of door trim.

## VIII. FUTURE SCOPE

1. By the results it is observed that plastic is the better material for the manufacture of automotive door trim, so in order to get better performance of the model the material properties of the plastic are to be enhanced.
2. The further research on door trim samples should be examined and the surface treatment results should be prior in order to determine the life of the component.

## IX. REFERENCES

- [1] H. M. A. Hussein and Ahmad Gemeal, Numerical study of Automotive doors Vol.12, August 2012, No.04
- [2] O.M. Terciu, I. Curtu et al., FEM Modelling of an automotive door trim panel made of lignocelulozic composites in case of door slam simulation COMAT 2012, October 2012.
- [3] S. VinaySeeba, S. Srihari et al., Design and analysis of a plastic door module for car body application : Vol.9, Issue 1, April 2010.
- [4] M. Raghuvier, GantiSathyaPrakash, Design and Impact Analysis of a Car Door Vol. 4, Iss. 9, September 2014, pp. 1-8.
- [5] MSC. Nastran [Computer software]: "Introduction to NASTRAN user guide" Los Angeles (CA, USA), Vol 111, MacNeilSchwendler Corporation, 2011.
- [6] Peterson, R.E "stress Concentration Factors" New york: john Wiley & Sons, Vol 1 2009.
- [7] Kown, Y. W. and Bang H: " The Finite Element Method Using MATLAB" Vol 111, 2010.
- [8] Su, Henkin "Automotive CAE Durability Analysis Using Random Vibration Approach" Proceedings of MSC2nd Worldwide Automotive Conference, Dearborn, MI, Oct.2010.
- [9] "Door Trim Key Life Bench Durability Test", Visteon Internal Test Procedure, TO-WP-0018B, April 28, 1999.
- [10] "Practical Finite Element Analysis byNitin S" Goshala, Vol 1 2008 Edition.