OPTIMIZATION AND EXPERIMENTAL TESTING OF CAR DOOR HANDLE USING 3 POINT BENDING TEST AND 3D PRINTING TECHNOLOGY

Brahmanand[1], Prof.Pawankumar R.Sonawane[2],Dr.Kiran C.More[3],
[1] *P.G.Student,Mechanical Engg.Department, D.Y.Patil Institute of Engg.&Technology,Pune,MH,India*
[2] *Asst.Professor,Mechanical Engg.Department, D.Y.Patil Institute of Engg.&Technology,Pune,MH,India*
[3] *Asst.Professor,Mechanical Engg.Department, D.Y.Patil Institute of Engg.&Technology,Pune,MH,India*

ABSTRACT

Doors are the most essential hardware used by human beings daily. Doors are used for sense of security to ourselves. To operate the door, we need door handle and it's the most important part in door. Door handles are used for opening and closing of a door with minimum effort. Various types of door handle such as lever handle, doorknob and pull handles are the different kind of handles we came across in our day-to-day life. There are various designs of door handles are available so that we are unaware of door handles selection criteria. The aim of this work is to study different material and internal design of car door interior handle. For that purpose, we used FEA software for understanding which material and which shape have more strength and optimize design. In this project we used two different structure one is honeycomb structure and other is auxetic structure for inside section of car door interior handle. Optimized car door interior handle will be manufacture using 3D printing technology. The behaviors of these structures under three-point bending were investigated by using UTM.

Keyword : - *Door handles , minimum effort, strength,optimize design, honeycomb structure and auxetic structure, 3D printing technology etc….*

---

1. INTRODUCTION

Door handles are the only hardware used for opening and closing of doors. Doors are used by every individual and for its functionality door handles are used. Doors are used for security purposes of our belongings. There are many kinds of doors like passage, closet, dummy doors etc. similarly there are various kinds of door handles used for variety of doors and their functionality. Door handles are installed on doors to simply open and close the door with minimum effort.As its name suggests, a door handle is used to open and close car doors. It is found on both the exterior and interior sides of automobile doors, although they are used differently on each panel. The one on the outside is pulled to open the car door, while the inside door handle is used to release the door latch before you can push the door to let yourself out. In this project interior door handle is taken into consideration.
Car interior door handles are designed ergonomically for the comfort of the driver and occupants
So, the interior car door handles are designed aesthetically and ergonomically.
Ergonomic Considerations in Design Ergonomics is defined as the relationship between man and machine and the application of anatomical, physiological and psychological principles to solve the problems arising from man-machine relationship.
Door handles can be made from durable plastic and metals like **aluminum** alloy, zinc alloy, and magnesium alloy.

2. PROBLEM STATEMENT:

The internal door handles of an automobile is typically made of different materials. Unlike the materials used on the exterior side of the vehicle door, the material on the interior side serves a greater purpose other than just aesthetic appeal.

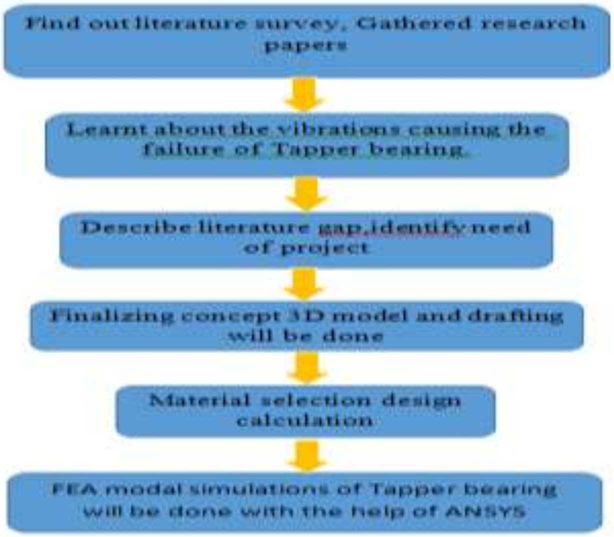
3. OBJECTIVES:

- Modeling car door interior handle with two different internal structure in CATIA V5 software.

- Analyzing for stresses and deformation in car door interior handle of vehicle using ANSYS 19 software.
- To manufacturing of car door interior handle of vehicle using3D printing technology.
- The behaviors of car door interior handle under three-point bending were investigated by using UTM.
- Experimental testing and correlating results.

.

4. METHODOLOGY:



5. DESIGN:

5.1.CAD:-

Computer-aided design (**CAD**) is the use of computer systems (or workstations) to aid in the creation, modification, analysis, or optimization of a design. CAD software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing. CAD output is often in the form of electronic files for print, machining, or other manufacturing operations. The term **CADD** (for *Computer Aided Design and Drafting*) is also used.

Its use in designing electronic systems is known as electronic design automation (**EDA**). In mechanical design it is known as mechanical design automation (**MDA**) or **computer-aided drafting** (**CAD**), which includes the process of creating a technical drawing with the use of computer software.

CAD software for mechanical design uses either vector-based graphics to depict the objects of traditional drafting, or may also produce raster graphicsshowing the overall appearance of designed objects. However, it involves more than just shapes. As in the manual drafting of technical and engineering drawings, the output of CAD must convey information, such as materials, processes, dimensions, and tolerances, according to application-specific conventions.

CAD may be used to design curves and figures in two-dimensional (2D) space; or curves, surfaces, and solids in three-dimensional (3D) space.

CAD is an important industrial art extensively used in many applications, including automotive, shipbuilding, and aerospace industries, industrial and architectural design, prosthetics, and many more. CAD is also widely used to produce computer animation for special effects in movies, advertising and technical manuals, often called

DCC digital content creation. The modern ubiquity and power of computers means that even perfume bottles and shampoo dispensers are designed using techniques unheard of by engineers of the 1960s. Because of its enormous economic importance, CAD has been a major driving force for research in computational geometry, computer graphics (both hardware and software), and discrete differential geometry.

The design of geometric models for object shapes, in particular, is occasionally called *computer-aided geometric design* (*CAGD*)

USES:

Computer-aided design is one of the many tools used by engineers and designers and is used in many ways depending on the profession of the user and the type of software in question.

CAD is one part of the whole Digital Product Development (DPD) activity within the Product Lifecycle Management (PLM) processes, and as such is used together with other tools, which are either integrated modules or stand-alone products, such as:

- Computer-aided engineering (CAE) and Finite element analysis (FEA)
- Computer-aided manufacturing (CAM) including instructions to Computer Numerical Control (CNC) machines
- Photorealistic rendering and Motion Simulation.
- Document management and revision control using Product Data Management (PDM).

CAD is also used for the accurate creation of photo simulations that are often required in the preparation of Environmental Impact Reports, in which computer-aided designs of intended buildings are superimposed into photographs of existing environments to represent what that locale will be like, where the proposed facilities are allowed to be built. Potential blockage of view corridors and shadow studies are also frequently analyzed through the use of CAD.

CAD has been proven to be useful to engineers as well. Using four properties which are history, features, parameterization, and high-level constraints. The construction history can be used to look back into the model's personal features and work on the single area rather than the whole model. Parameters and constraints can be used to determine the size, shape, and other properties of the different modeling elements. The features in the CAD system can be used for the variety of tools for measurement such as tensile strength, yield strength, electrical or electromagnetic properties. Also its stress, strain, timing or how the element gets affected in certain temperatures, etc.

TYPES:

There are several different types of CAD, each requiring the operator to think differently about how to use them and design their virtual components in a different manner for each.

There are many producers of the lower-end 2D systems, including a number of free and open-source programs. These provide an approach to the drawing process without all the fuss over scale and placement on the drawing sheet that accompanied hand drafting since these can be adjusted as required during the creation of the final draft.

3D wireframe is basically an extension of 2D drafting (not often used today). Each line has to be manually inserted into the drawing. The final product has no mass properties associated with it and cannot have features directly added to it, such as holes. The operator approaches these in a similar fashion to the 2D systems, although many 3D systems allow using the wireframe model to make the final engineering drawing views.

3D "dumb" solids are created in a way analogous to manipulations of real-world objects (not often used today). Basic three-dimensional geometric forms (prisms, cylinders, spheres, and so on) have solid volumes added or subtracted from them as if assembling or cutting real-world objects. Two-dimensional projected views can easily be generated from the models. Basic 3D solids don't usually include tools to easily allow motion of components, set limits to their motion, or identify interference between components.

There are two types of *3D Solid Modeling*

1. *Parametric modeling* allows the operator to use what is referred to as "design intent". The objects and features created are modifiable. Any future modifications can be made by changing how the original part was created. If a feature was intended to be located from the center of the part, the operator should locate it from the center of the model. The feature could be located using any geometric object already available in the part, but this random placement would defeat the design intent. If the operator designs the part as it functions the parametric modeler is able to make changes to the part while maintaining geometric and functional relationships.
2. Direct or Explicit modeling provide the ability to edit geometry without a history tree. With direct modeling, once a sketch is used to create geometry the sketch is incorporated into the new geometry and the designer just modifies the geometry without needing the original sketch. As with parametric modeling, direct modeling has the ability to include relationships between selected geometry (e.g., tangency, concentricity).

Top end systems offer the capabilities to incorporate more organic, aesthetics and ergonomic features into designs. Freeform surface modeling is often combined with solids to allow the designer to create products that fit the human form and visual requirements as well as they interface with the machine.

5.2.FINITE ELEMENT ANALYSIS:

5.2.1.Introduction of Finite Element Analysis

The finite element analysis (FEA) is a problem-solving approach for the practical (engineering) problems. The problems are first converted to matrix and partial differential equation forms. Eventually the partial differential and integral equations are being solved to reach the solution of the problem. The volume of the equations to be solved is usually so large that arriving solution without using computer is practically impossible. And, that's why the need of different FEA packages is felt. There are many FEA packages available for different applications. Some popular FEA packages are Pro Mechanica, ANSYS, NASTRAN, and Gambit etc.

In mathematics, the finite element analysis (FEA) is a numerical technique for finding approximate solutions to boundary value problems for partial differential equations. It uses subdivision of a whole problem domain into simpler parts, called finite elements, and variational methods from the calculus of variations to solve the problem by minimizing an associated error function. Analogous to the idea that connecting many tiny straight lines can approximate a larger circle, FEA encompasses methods for connecting many simple element equations over many small sub domains, named finite elements, to approximate a more complex equation over a larger domain.

Finite element analysis (FEA) is a useful and powerful technique for determining stresses and strains in structures or components too complex to analyze by strictly analytical methods. With this technique, the structure or component is broken down into many small pieces (finite number of elements) of various types, sizes and shapes. The elements are assumed to have a simplified pattern of deformation (linear or quadratic etc.) and are connected at "nodes" normally located at corners or edges of the elements. The elements are then assembled mathematically using basic rules of structural mechanics, i.e. equilibrium of forces and continuity of displacements, resulting in a large system of simultaneous equations. By solving these large simultaneous equations system with the help of a computer, the deformed shape of the structure or component under load may be obtained. Based on that, stresses and strains may be calculated. The finite element analysis (FEA) is probably the most versatile way of calculating stress intensity factors. This method primarily involves the evaluation of displacements at nodal points of the body which has been idealized into a system of elements connected at the nodal points. The FEA has become a powerful tool for the numerical solution of a wide range of engineering problems. The FEA has been extensively used to solve problems involving irregular regions and complicated modals.

Steps of Finite Element Analysis:

FEA solution of engineering problems, such as finding deflections and stresses in a structure, requires three steps:

1. Pre-processing

2. Solution

3. Post processing

A brief description of each of these steps follows

Step1: Pre-processing

Using a CAD program that either comes with the FEA software or 3D CAD modeling tools like Pro-E, Catia, and solid Edge etc. provided by another software vendor, the structure is modeled. The final FEA model consists of several elements that collectively represent the entire structure. The elements not only represent segments of the structure, they also simulate its mechanical behaviour and properties.

Regions where geometry is complex (curves, notches, holes, etc.) require increased number of elements to accurately represent the shape; whereas, the regions with simple geometry can be represented by coarser mesh (or fewer elements). The selection of proper elements requires prior experience with FEA, knowledge of structure's behaviour, available elements in the software and their characteristics, etc. The elements are joined at the nodes, or common points. In the pre-processor phase, along with the geometry of the structure, the constraints, loads and mechanical properties of the structure are defined. Thus, in pre-processing, the entire structure is completely defined by the geometric model. The structure represented by nodes and elements is called "mesh".

Step 2: Solution

In this step, the geometry, constraints, mechanical properties and loads are applied to generate matrix equations for each element, which are then assembled to generate a global matrix equation of the structure. The form 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.

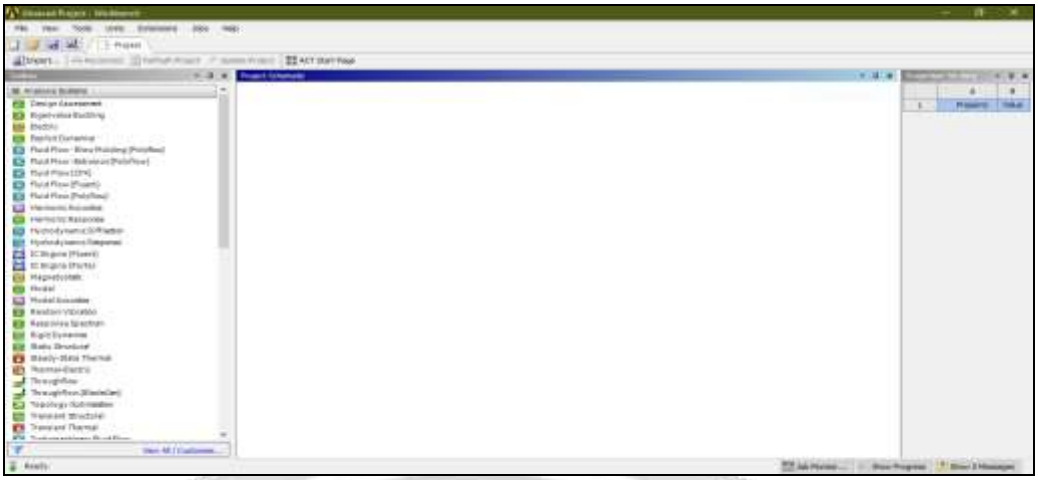
The equation is then solved for deflections. Using the deflection values, strain, stress, and reactions are calculated. All the results are stored and can be used to create graphic plots and charts in the post analysis.

Step 3: Post processing

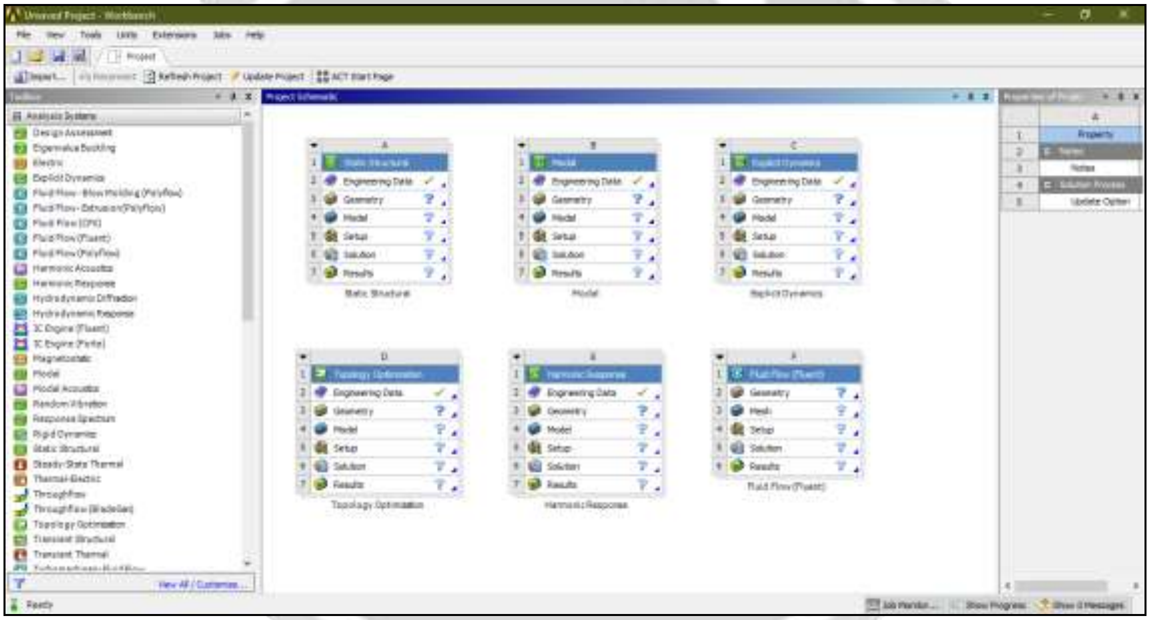
This is the last step in a finite element analysis. Results obtained in step 2 are usually in the form of raw data and difficult to interpret. In post analysis, a CAD program is utilized to manipulate the data for generating deflected shape of the structure, creating stress plots, animation, etc. A graphical representation of the results is very useful in understanding behaviour of the structure.

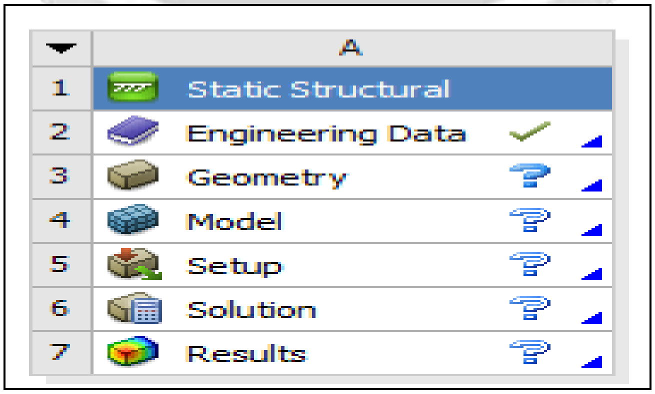
In present research for analysis ANSYS (Analysis System)software is used. Basically ,its present FEM method to solve any problem. Following are steps in detail

1. Geometry
2. Discretization (Meshing)
3. Boundary condition
4. Solve (Solution)
5. Interpretation of results



Workbench contain analysis of different types namely static, modal, harmonic, explicit dynamics, CFD, ACP tool post, CFX, topology optimizationetc. as per problem defined.





	A		
1	Static Structural		
2	Engineering Data	✓	◢
3	Geometry	?	◢
4	Model	?	◢
5	Setup	?	◢
6	Solution	?	◢
7	Results	?	◢

Step 1: Details of material namely copper, steel, grey cast iron, composite material, fluid domain material is defined in engineering data. i.e. ANSYS default material is structural steel.

Step 2: Import of geometry created in any CAD software namely CATIA, PRO E, SOLIDWORK, INVENTOR etc. in geometry section. If any correction is to be made it can be created in geometry section in Design modeller or space claim.

Step 3: In model section after import of component

- Material is assigned to component as per existing material
- Connection is checked in contact region i.e. bonded, frictionless, frictional, no separation etc. for multi body components.
- Meshing or discretization is performed i.e. to break components in small pieces (elements) as per size i.e. preferably tetra mesh and hexahedral mesh for 3D geometry and for 2 D quad or tria are generally preferred.

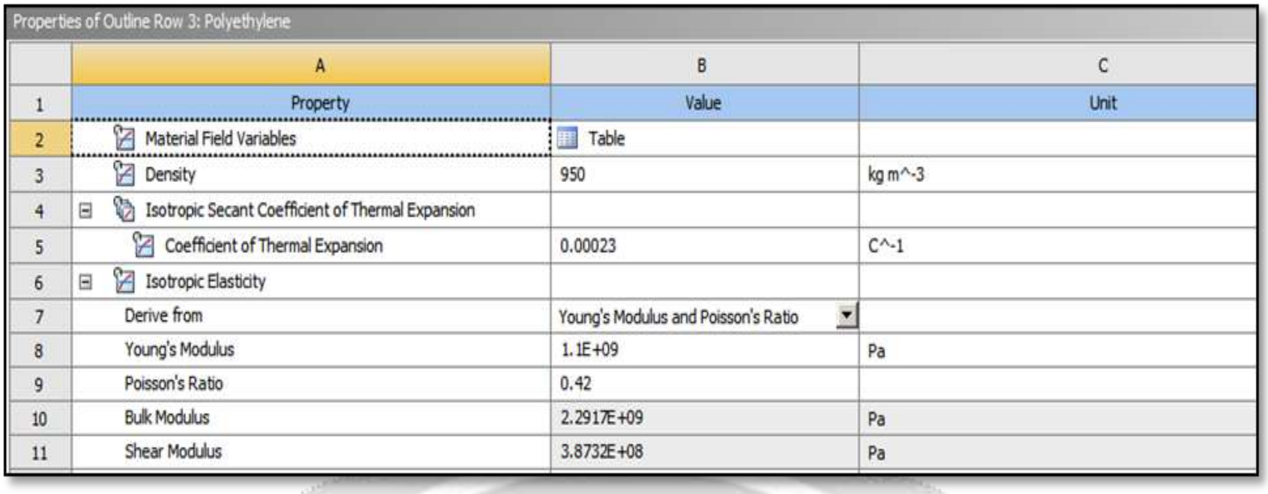
Step 4: Boundary condition are applied as per analysis namely in fixed support, pressure, force, displacement, velocity as per condition.

Step 5: Now problem is well defined and solve option is selected to obtain the solution in the form of equivalent stress, strain, energy, reaction force etc.

6.GEOMETRY:

Figure: Geometry of existing door handle

6.1.Material Properties:



Properties of Outline Row 3: Polyethylene

	A	B	C
1	Property	Value	Unit
2	Material Field Variables	Table	
3	Density	950	kg m^-3
4	⊟ Isotropic Secant Coefficient of Thermal Expansion		
5	Coefficient of Thermal Expansion	0.00023	C^-1
6	⊟ Isotropic Elasticity		
7	Derive from	Young's Modulus and Poisson's Ratio	
8	Young's Modulus	1.1E+09	Pa
9	Poisson's Ratio	0.42	
10	Bulk Modulus	2.2917E+09	Pa
11	Shear Modulus	3.8732E+08	Pa

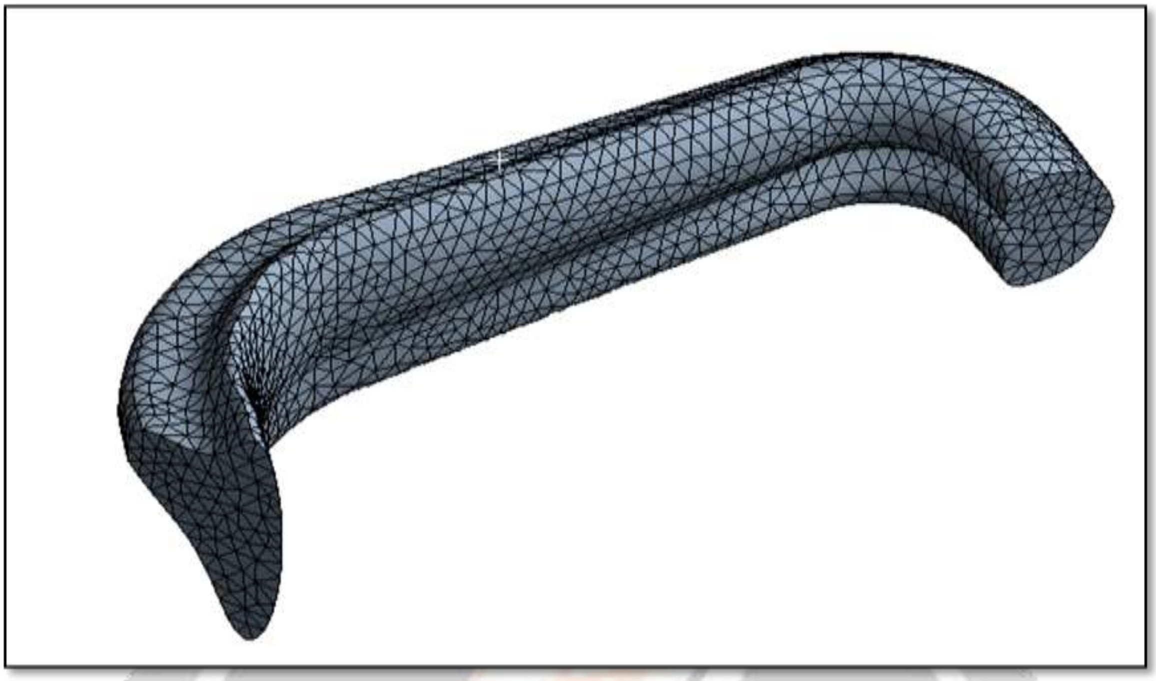
Table:Properties of polyethylene

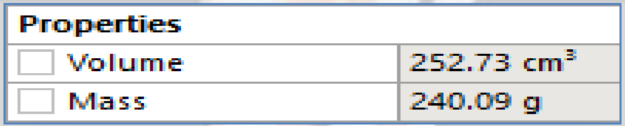
6.2.Meshing:

Meshing is an integral part of the computer-aided engineering simulation process. The mesh influences the accuracy, convergence and speed of the solution. Furthermore, the time it takes to create and mesh a model is often a significant portion of the time it takes to get results from a CAE solution. Therefore, the better and more automated the meshing tools, the better the solution.

For 2-D solid models, meshing technologies from ANSYS provide robust, well-shaped quadratic and triangular meshing along with free and mapped meshing, on even the most complicated geometries. With automatic contact detection and setup, a user requires little training to perform sophisticated analysis. In addition, users can generate pure hex meshes using one of several mesh methods, depending on the type of model and whether the user wants a pure hex or hex-dominant mesh.

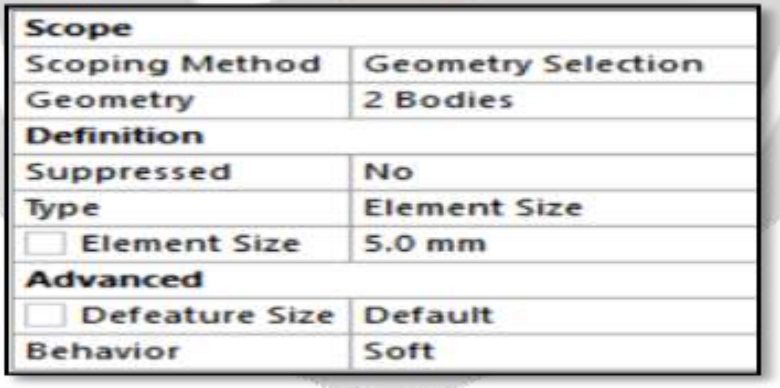
In present FE model, the meshed set using the ‘Global controls’ available with the ‘mesh tool’. Smart mesh set to 3 and triangular mesh is preferred for plate with circular, triangular, square and rectangular cut out. Meshing on test specimen is shown in fig.





Properties	
☐ Volume	252.73 cm^3
☐ Mass	240.09 g

Statistics	
☐ Nodes	30963
☐ Elements	18120



Scope	
Scoping Method	Geometry Selection
Geometry	2 Bodies
Definition	
Suppressed	No
Type	Element Size
☐ Element Size	5.0 mm
Advanced	
☐ Defeature Size	Default
Behavior	Soft

Above Figure & Table showing Meshing and weight details

6.3.Boundary Condition:

A boundary condition for the model is the setting of a known value for a displacement or an associated load. For a particular node you can set either the load or the displacement but not both.

The main types of loading available in FEA include force, pressure and temperature. These can be applied to points, surfaces, edges, nodes and elements or remotely offset from a feature. The way that the model is constrained can significantly affect the results and requires special consideration. Over or under constrained models can give stress that is so inaccurate that it is worthless to the engineer. In an ideal world we could have massive assemblies of components all connected to each other with contact elements but this is beyond the budget and resource of most people. We can however, use the computing hardware we have available to its full potential and this means understanding how to apply realistic boundary conditions.

Fig. Boundary condition

- The maximum weight apply on the door handle is the weight of human. So we consider the weight of human is 100kg with the factor of safety.
- So the maximum force apply on car door is 100*9.81=1000N
- So we apply the 1000N load on car door and fixed both end of the car door.

7. RESULTS:

Total deformation:

The total deformation & directional deformation are general terms in finite element methods irrespective of software being used.

Directional deformation can be put as the displacement of the system in a particular axis or user defined direction. Total deformation is the vector sum of all directional displacements of the systems.

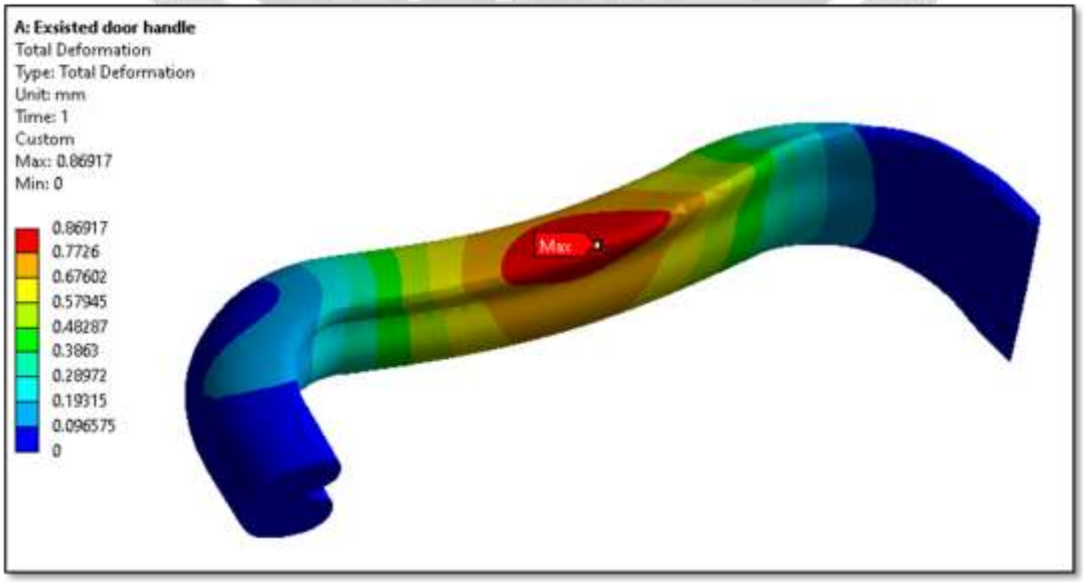


Fig. Total deformation

Equivalent Stress

Equivalent stress is related to the principal stresses by the equation:

$$\sigma_e = \left[\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2} \right]^{1/2}$$

Equivalent stress (also called *von Misses stress*) is often used in design work because it allows any arbitrary three-dimensional stress state to be represented as a single positive stress value. Equivalent stress is part of the maximum equivalent stress failure theory used to predict yielding in a ductile material.

The von Mises or equivalent strain ε_e is computed as:

$$\varepsilon_e = \frac{1}{1+\nu'} \left(\frac{1}{2} \left[(\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_3)^2 + (\varepsilon_3 - \varepsilon_1)^2 \right] \right)^{\frac{1}{2}}$$

Where:

ν' = effective Poisson's ratio

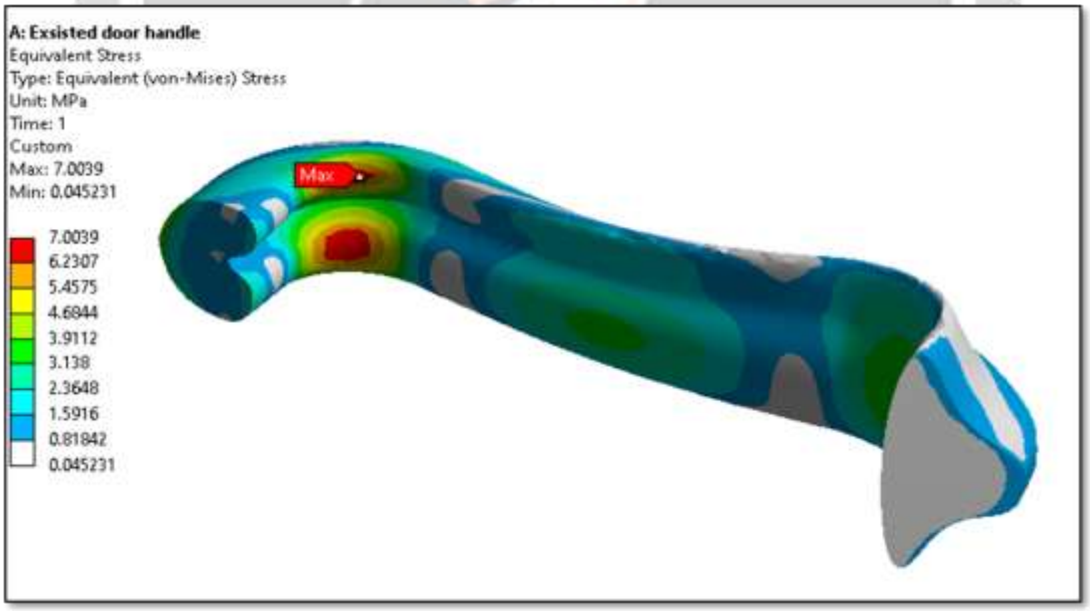


Fig. Equivalent stress

OPTIMIZED CAR DOOR HANDLE PLOT RESULTS:

Geometry:

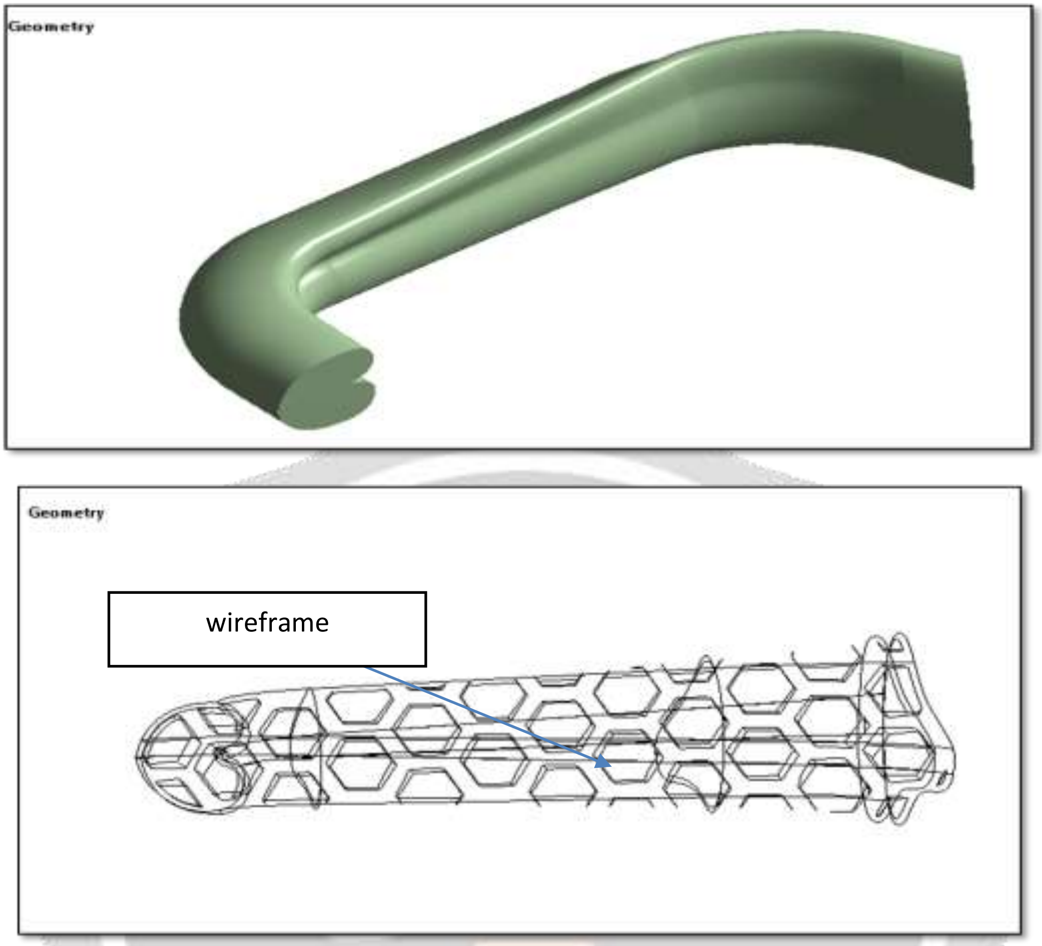
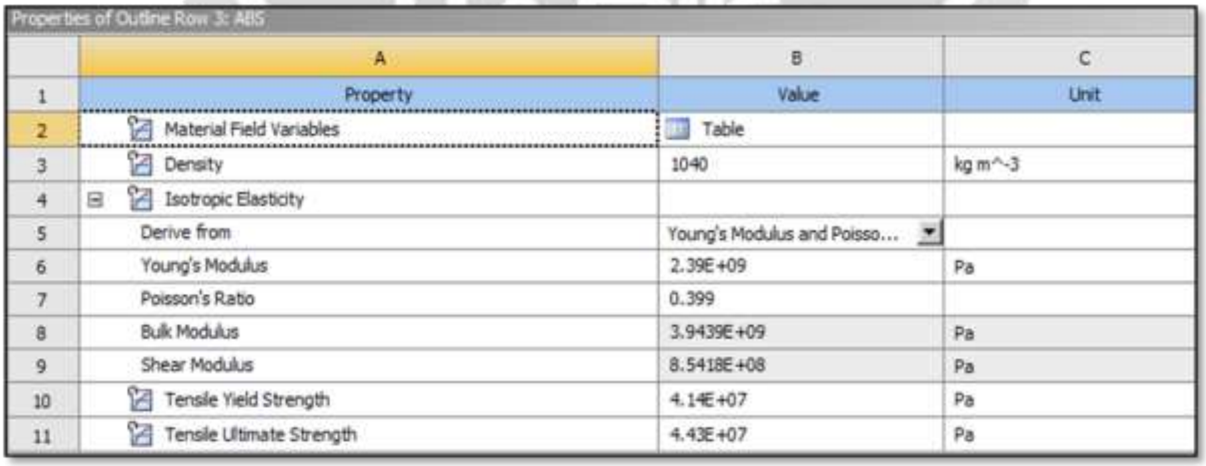


Fig. Cad model of honeycomb structured door handle

Material properties:



Properties of Outline Row 3: ABS

	A	B	C
1	Property	Value	Unit
2	Material Field Variables	Table	
3	Density	1040	kg m^-3
4	⊟ Isotropic Elasticity		
5	Derive from	Young's Modulus and Poisso...	
6	Young's Modulus	2.39E+09	Pa
7	Poisson's Ratio	0.399	
8	Bulk Modulus	3.9439E+09	Pa
9	Shear Modulus	8.5418E+08	Pa
10	Tensile Yield Strength	4.14E+07	Pa
11	Tensile Ultimate Strength	4.43E+07	Pa

Meshing:

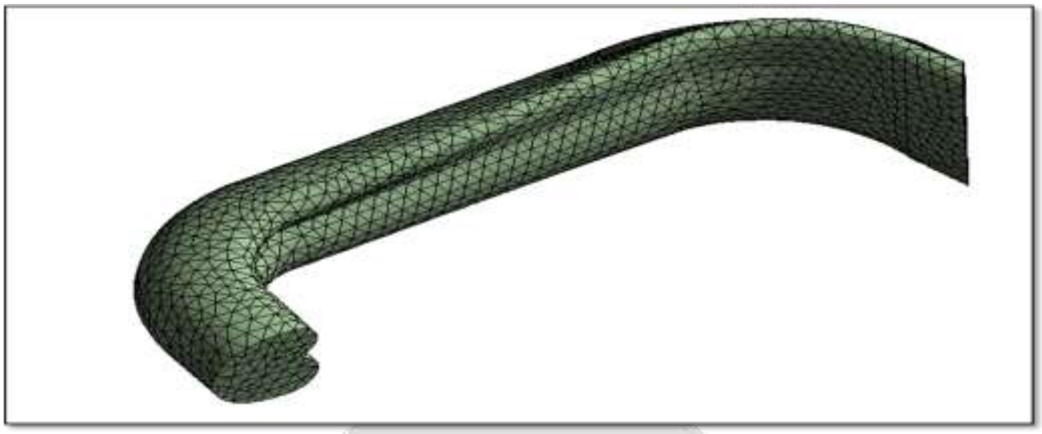
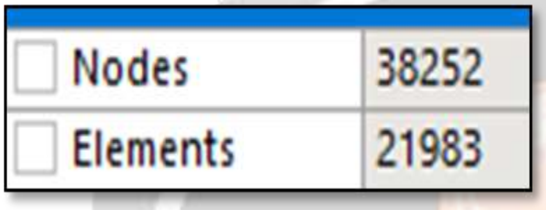
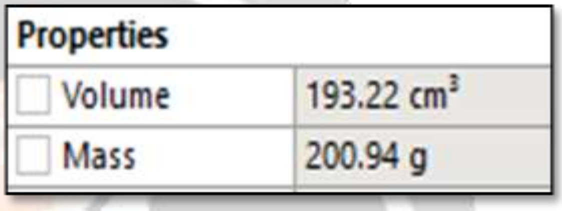


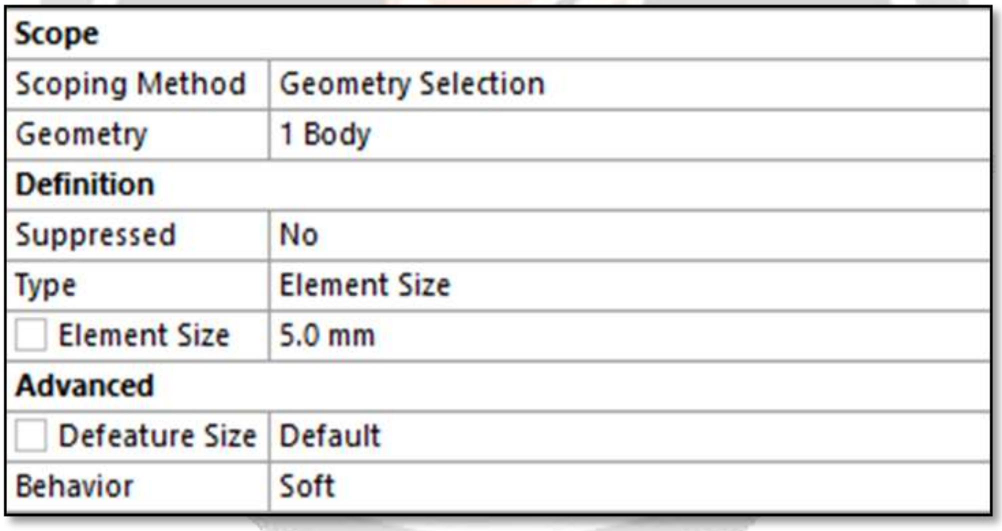
Fig. Meshing Details



☐ Nodes	38252
☐ Elements	21983



Properties	
☐ Volume	193.22 cm^3
☐ Mass	200.94 g



Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
☐ Element Size	5.0 mm
Advanced	
☐ Defeature Size	Default
Behavior	Soft

Boundary condition:

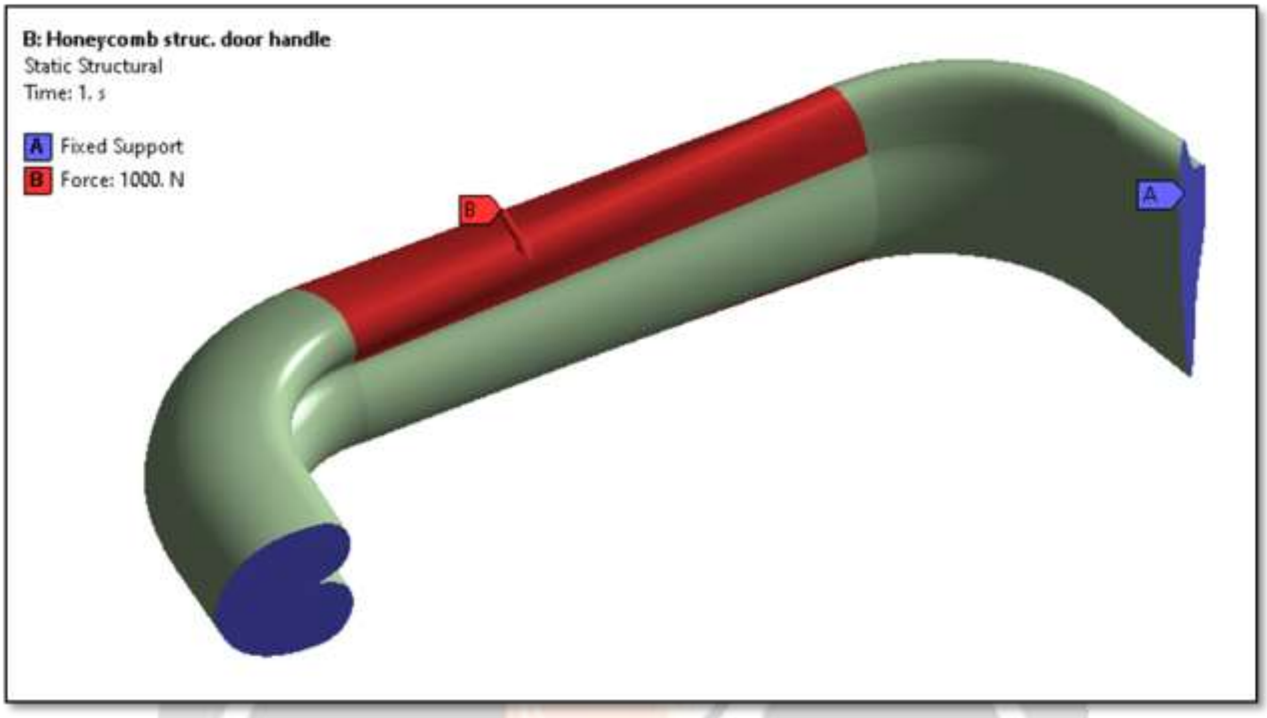


Fig. Boundary condition

Deformation plot:

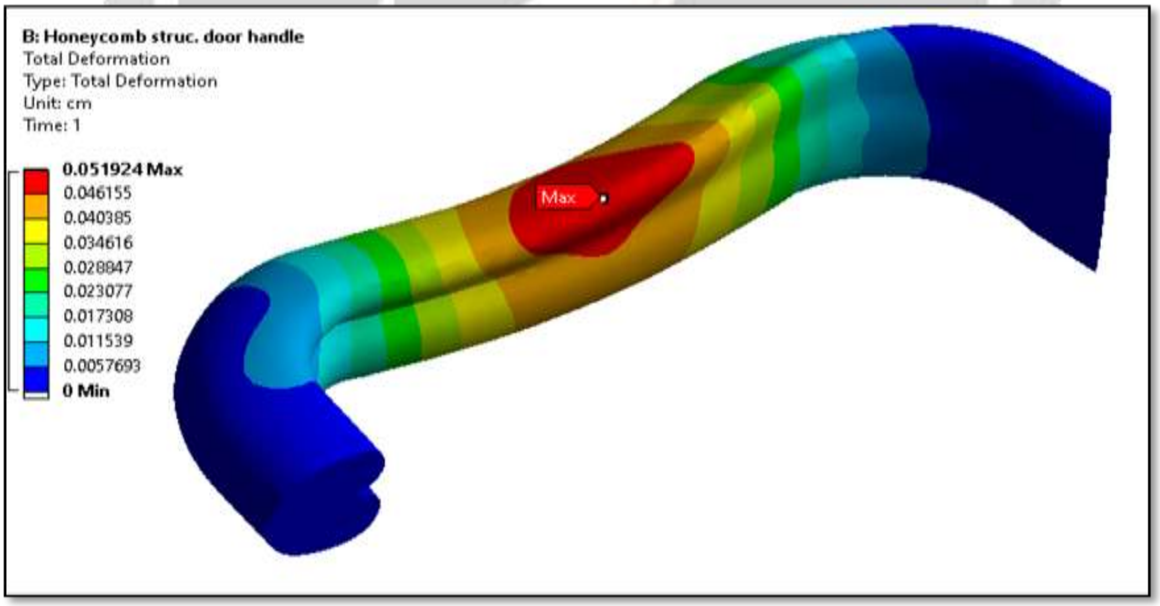


Fig. Total deformation plot

Equivalent stress:

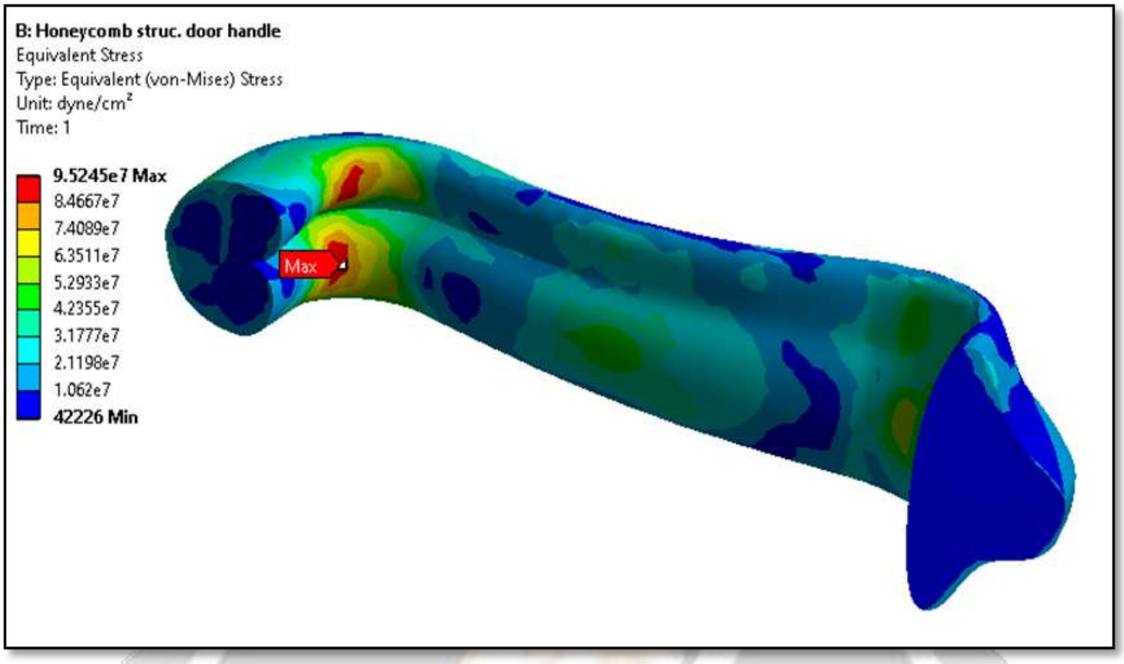
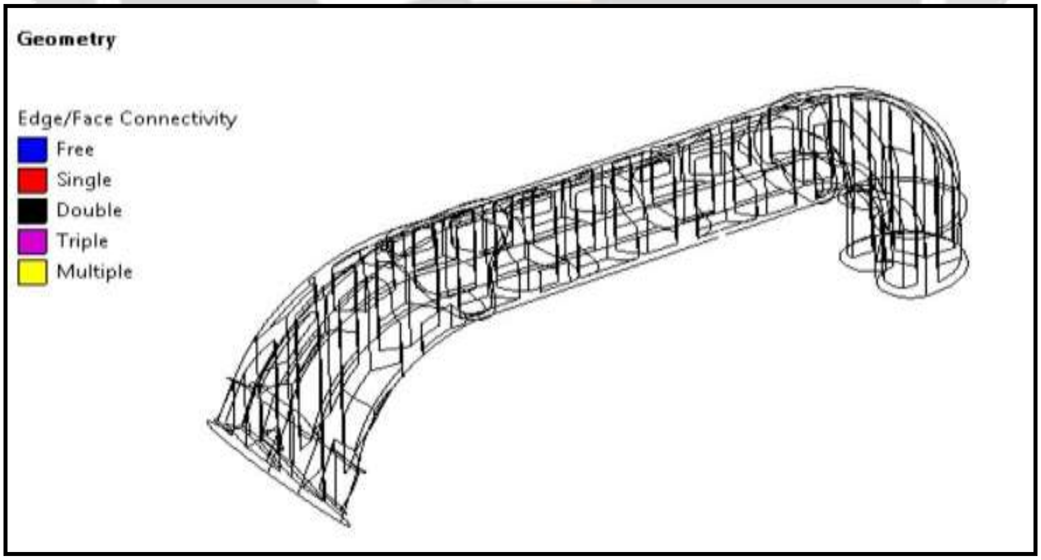


Fig. Equivalent stress plot

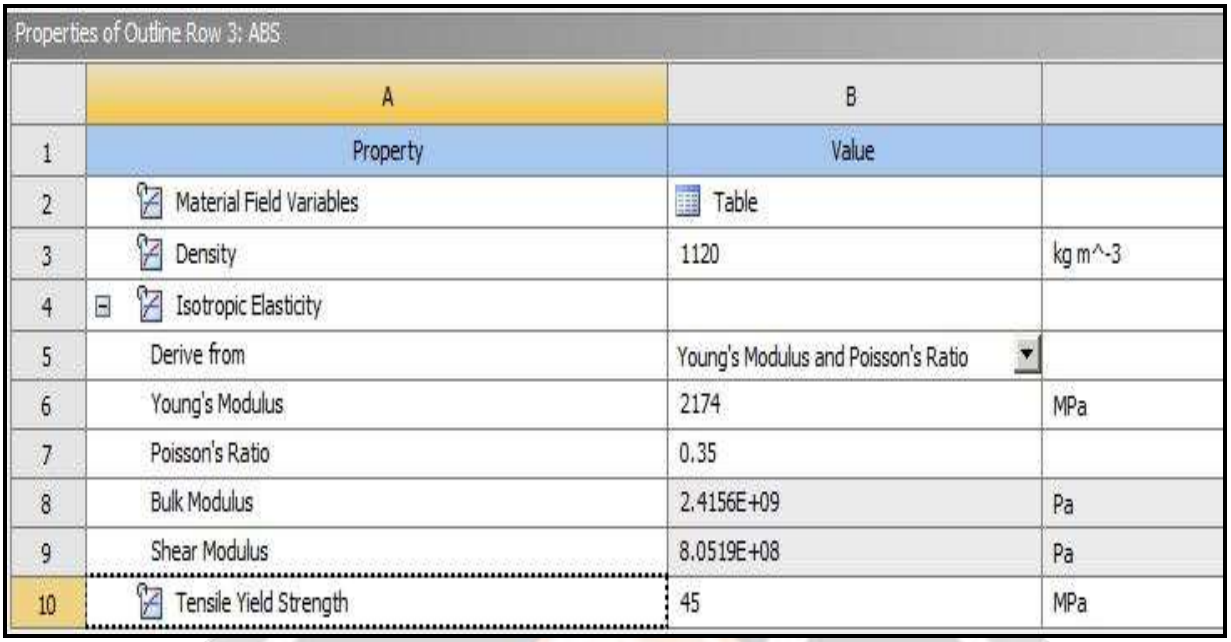
Experimental testing result plots :

Geometry:



Wire-frame geometry of car door handle in honeycomb structure

Material properties of 3D printed interior car door handle:

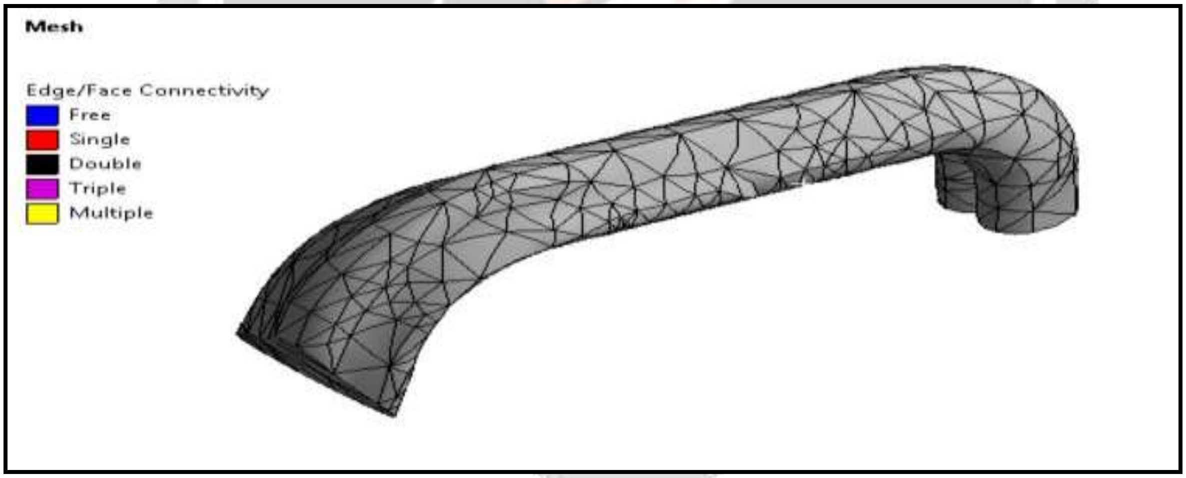


Properties of Outline Row 3: ABS

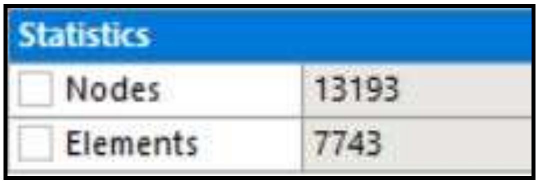
	A	B	
1	Property	Value	
2	Material Field Variables	Table	
3	Density	1120	kg m^-3
4	⊟ Isotropic Elasticity		
5	Derive from	Young's Modulus and Poisson's Ratio ▼	
6	Young's Modulus	2174	MPa
7	Poisson's Ratio	0.35	
8	Bulk Modulus	2.4156E+09	Pa
9	Shear Modulus	8.0519E+08	Pa
10	Tensile Yield Strength	45	MPa

We are using ABS material for 3D printing of interior car door handle.

Meshing:



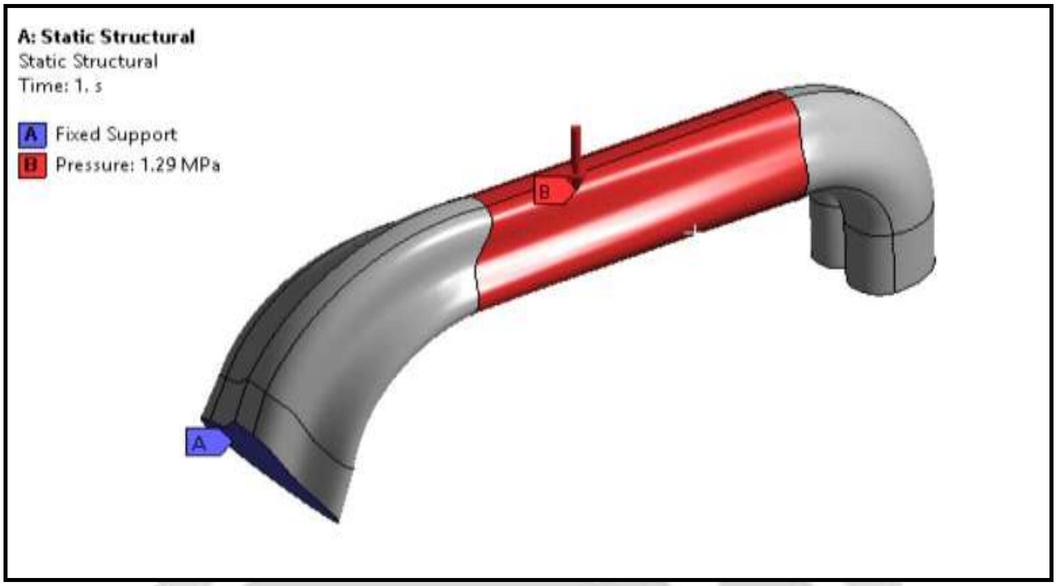
Meshing plot



Statistics	
☐ Nodes	13193
☐ Elements	7743

Nodes and Elements

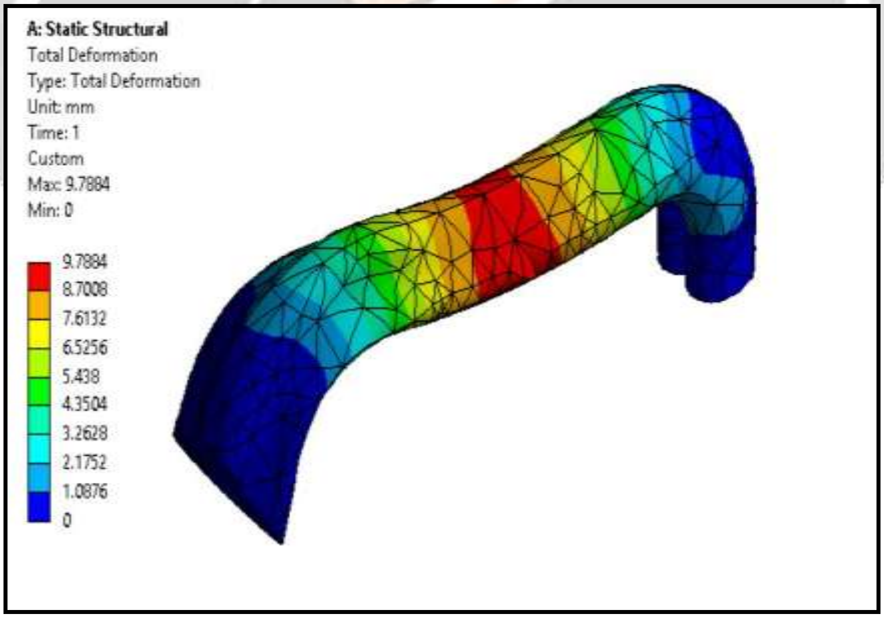
Boundary condition for experimental analysis:



Boundary condition

As we have non-uniform surface for the application of force we have applied gradual pressure on the interior car door handle and find out the results. We have applied 1.29 Mpa pressure.

Deformation plot:



Max. Deformation of the car door handle is 9.78mm

UTM:

A Universal Testing Machine (UTM) is used to test both the tensile and compressive strength of materials. Universal Testing Machines are named as such because they can perform many different varieties of tests on an equally diverse range of materials, components, and structures.

Universal Testing Machines can accommodate many kinds of materials, ranging from hard samples, such as metals and concrete, to flexible samples, such as rubber and textiles. This diversity makes the Universal Testing Machine equally applicable to virtually any manufacturing industry.

The UTM is a versatile and valuable piece of testing equipment that can evaluate materials properties such as tensile strength, elasticity, compression, yield strength, elastic and plastic deformation, bend compression, and strain hardening. Different models of Universal Testing Machines have different load capacities, some as low as 5kN and others as high as 2,000kN.

SPECIFICATION OF UTM

1	**Max Capacity**	**400KN**
2	**Measuring range**	**0-400KN**
3	**Least Count**	**0.04KN**
4	**Clearance for Tensile Test**	**50-700 mm**
5	**Clearance for Compression Test**	**0- 700 mm**
6	**Clearance Between column**	**500 mm**
7	**Ram stroke**	**200 mm**
8	**Power supply**	**3 Phase , 440Volts , 50 cycle. A.C**
9	**Overall dimension of machine (L*W*H)**	**2100*800*2060**
10	**Weight**	**2300Kg**

Experimental results:

- **As we have simply supported the car door handle on two ends.**
- **The gradual pressure is applied at the center portion.**

Experimental setup for 3 point bending test

8.CONCLUSION:

As the aim of this project is to reduce the weight and find strength of the car door handle. By using 3D printing technique, materials use for 3D printing is ABS. Car door handle is analyzed ANSYS software which utilizes finite element method technologies.

- In this project we develop existing car door handle is selected to perform FEA analysis to determine maximum deformation and respective equivalent stress.
- Static structural analysis of car door handle is performed to determine deformation and equivalent stress. It is observed that around maximum deformation is 0.86 mm and equivalent stress is 7.0 MPa.
- We optimized existing model using honeycomb structure and ABS material to reduce the weight. Hence, we develop 3D cad model of optimized door handle.
- Static structural analysis of 3d printing car door handle using ABS material is performed to determine deformation and equivalent stress. It is observed that around maximum deformation is 0.05 mm and equivalent stress 9.52 MPa.
- We develop model of car door handle using 3D printing technique and use honeycomb structure inside the door handle to reduce the weight of the model compare to exist model.
- The weight of existing car door handle is 240.09 g and the weight of optimized door handle is 200.94 g. So, the weight optimization in this project is 16.30 %.
- After experimental analysis of car door handle the deformation is noted as 9.8mm.
- After applying pressure on door handle, after deformation of 9.8 mm, it regains it’s original shape.

9.REFERENCE:

1] “Experimental Analysis of Dryer Door Handle to Predict Fatigue Life Compare with FEA Results” by Sachin Khade, G. D. Korwar. 2017 IJESC, Volume 7 Issue No.8.

2] “Finite element analysis and Topology optimization design for automobile door-handle” by Enguang Zhang and Xin Zhang 2018 IOP Conf. Ser.: Mater. Sci. Eng. 392 062072.

3] Investigation of ergonomics design for the vehicle door handle for PROTON (BLM) and PERODUA (VIVA) KA Shamsuddin, NI Mokhtar, MMAMM Aris, TA Abdul Razak. IJESRT September, 2015.

4] Optimizations of outer door handle in automobiles using composite material. Girish Naresh Raut, Prof V. P. Mali. Volume: 04 Issue: 05 | May -2017.

5] “design of a vehicle door structure based on finite element method” by Tawanda Mushiri and Charles Mbohwa.

6] “an overview on 3d printing technology: technological, materials printing and application. N. Shahrubudina, T.C. Lee, R. Ramlan. 2351-9789 © 2019 The Authors. Published by Elsevier B.V. Peer-review under responsibility of the organizing committee of SMPM 2019.

7] “COMPARISON OF MECHANICAL BEHAVIOUR OF METALLIC AND COMPOSITE FRONT DOOR OF A STANDARD AUTOMOBILE CAR BY FEA” by ANIKET CHANDU THOSAR THE UNIVERSITY OF TEXAS AT ARLINGTON Spring 2017

8]“3D printing technologies and composite materials for structural applications.” by Rajkumar Velu, Felix Raspall, Sarat Singamneni.

9] “Additive Manufacturing of Honeycomb Structure and Analysis of Infill and Material Characteristics.” By B.Vishnu Vardhana Naidu , G. Dileep Kumar

10] “An experimental and numerical investigation of a Novel 3D printed sandwich material for motorsport application.” By Danial Harland, Ahmed W. Alshaer, Hardley brooks.