Design And Strength Improvement Of Four Wheeler Rocker Panel

A PROJECT REPORT SUBMITTED IN PARTIAL FULFILLMENT OF THE DEGREE OF

Bachelor of Engineering
In
Mechanical Engineering

Submitted By

SHUBHAM KUMBHAR	**B150310992**
AMARTYA KANGANE	**B150310886**
PRANAV PATIL	**B150310958**
AJAY WARE	**B150311024**

Under the Guidance of

Prof. S. M. RAMNANI

Department of Mechanical Engineering,

PES's Modern College of Engineering

Pune – 411005

[2021-22]

Progressive Education Society's

Modern College of Engineering, Pune

CERTIFICATE

This is to certify that the project work (402051) report entitled

Design And Strength Improvement Of Four Wheeler Rocker Panel

Has Successfully Completed By

SHUBHAM KUMBHAR	**B150310992**
AMARTYA KANGANE	**B150310886**
PRANAV PATIL	**B150310958**
AJAY WARE	**B150311024**

In the partial fulfilment of degree of

BACHELOR OF MECHANICAL ENGINEERING

In

SAVITRIBAI PHULE PUNE UNIVERSITY

Under the guidance of

PROF. S. M. RAMNANI

During the Academic Year 2021-22

Prof. S. M. Ramnani	**Prof. S. Y. Bhosale**	**Prof. Dr. Mrs. K. R. Joshi**
Asst. Professor	Associate Professor & Head of	Principal,
(Guide)	Mechanical Engineering	PES's MCOE, Pune

Progressive Education Society's

Modern College of Engineering, Pune

CERTIFICATE

This is to certify that the project work (402051) report entitled

Design And Strength Improvement Of Four Wheeler Rocker Panel

Has Successfully Completed By

SHUBHAM KUMBHAR	**B150310992**
AMARTYA KANGANE	**B150310886**
PRANAV PATIL	**B150310958**
AJAY WARE	**B150311024**

In the partial fulfilment of degree of

BACHELOR OF MECHANICAL ENGINEERING

In

SAVITRIBAI PHULE PUNE UNIVERSITY

Under the guidance of

PROF. S. M. RAMNANI

During the Academic Year 2021-22

Internal Examiner **External Examiner**

Seal

ACKNOWLEDGEMENT

It gives us great pleasure in presenting the final project report on ‘Design And Strength Improvement Of Four Wheeler Rocker Panel’. We would like to thank our guide Prof. S.M.Ramnani for giving us all the help and guidance needed. We are really grateful to them for their kind support. Their valuable suggestions were very helpful. We are also grateful to Prof S. Y. Bhosale, Head of Mechanical Engineering Department, PES Modern College of Engineering Pune for his indispensable support, suggestions.

SHUBHAM KUMBHAR

AMARTYA KANGANE

PRANAV PATIL

AJAY WARE

ABSTRACT

Vehicle side crash is a critical crash event where the vehicle is crashed by a movable car or vehicle may hit a tree or pole. Minimising the intrusion into the occupant space is important to protect the occupant. In side pole crash, vehicle rocker (sill) plays an important role in resisting the load due to the crash. The objective is to study the functional performance and potential mass reduction in the vehicle sill/rocker area by use of carbon fibre reinforced polymer (CFRP) tubes.

In this project investigates the behaviour of CFRP square section tubes in a three-point quasi-static bending in comparison to conventional steel structure using finite element method. By keeping the resistance force offered by a steel section as the baseline resistance value, different combination of CFRP tubes and metal holding brackets are evaluated and compared with the baseline. Design and analysis of existing Rocker Panel specimen will be done using CATIA R5V20 and ANSYS 19 software. new design & weight optimization of rocker panel specimen will be done using CFRP . Experimental investigation will be done by three-point bending test on UTM.

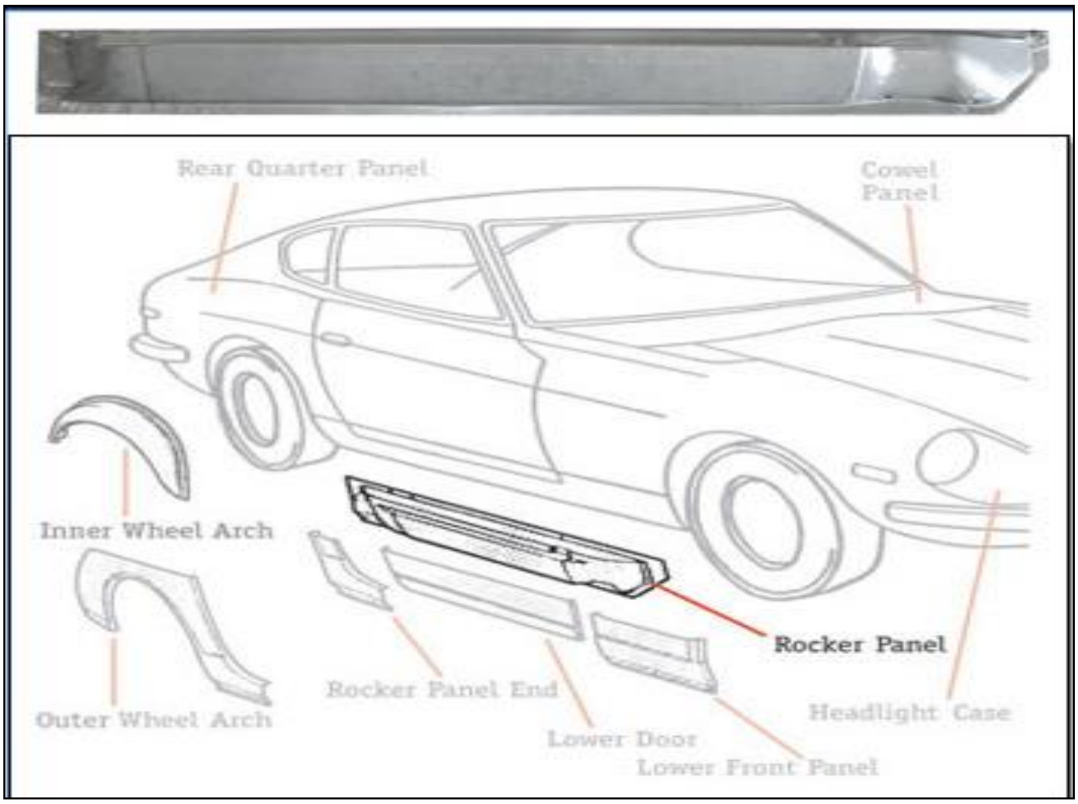
INDEX

LIST OF FIGURES

LIST OF TABLES

1. INTRODUCTION

In the twenty-first century, people are more oriented toward vehicles with better fuel economy and reduced emission levels. At the same time, due to an increase in awareness on safety and stringent crash test regulations, the automotive manufacturers are heading towards a smarter design of the occupant space by use of high strength materials for better crashworthiness. The term crashworthiness signifies the ability of the structure to protect the occupant in crash scenario. Crash performance requirements are focused on occupant injury parameters and structural deformation measurements like intrusion, acceleration and velocity of the deforming structure. Protecting people inside crash is challenging because the sides of vehicles have relatively little space to absorb energy and shield occupants, unlike the front and rear, which have substantial crumple zones.



What is rocker panel ?

Rocker panels are stamped pieces of strong metal that form part of the structural body of the car. They are an integral part that runs along the side of your car between the front and rear wheel wells. In other words, rocker panels keep the back of your car from separating from the front of your car.

What is purpose of providing rocker panels:

The rocker panels help to make sure the cabin of the car doesn't deform in a way that could harm the passengers. Essentially, in an accident your car's job is to turn into a smushed can protecting an interior capsule. The rocker panel forms the bottom edge of that capsule.

1.1 Organization of Report

1 Introduction: This chapter gives the overall introduction about the four wheeler rocker panel.

2 Theory & Literature review: This chapter gives the information about one of the most important components of the side structure of a vehicle, the rocker panel, with unidirectional carbon fibre composite material. The result of their experimentation is summarized.

3 Problem definition, objective and scope: After studying all the experimentation results from literature review the concept of reinforcement of rocker panel with the composite material which helps to reduce in weight and because of that it will give better fuel economy as well. The scope, problem definition and objective of this project are defined in this chapter.

4 Design: 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.

2.LITURATURE REVIEW

1. **“Bending Performance and Reinforcement of Rocker Panel Components with Unidirectional Carbon Fibre Composite”** by Huili Yu, Hui Zhao and Fangyuan Shi

This study aimed to redesign one of the most important components of the side structure of a vehicle, the rocker panel, with unidirectional carbon fiber composite material. Our results show that it is not easy to acquire the same bending performance as that of a steel rocker panel by merely replacing it with carbon fiber material and increasing the wall thickness. Therefore, reinforcements were employed to improve the bending performance of the carbon fiber rocker panel, and a polypropylene reinforcement method achieved a weight reduction of 40.7% compared with high-strength steel.

2. **“Design of crown pillar thickness using finite element method and multivariate regression analysis” By** Kumar Hemant , Deb Debasis, Chakravarty D.

This paper provides a methodology for the evaluation of the required thickness of crown pillars for safe operation at depth ranging from 600 m to 1000 m. Analyses are conducted with the results of 108 non-linear numerical models considering Drucker-Prager material model in plane strain condition. Material properties of ore body rock and thickness of crown pillars are varied and safety

factors of pillars estimated. Then, a generalized statistical relationship between the safety factors of crown pillars with the various input parameters is developed. The developed multivariate regression model is utilized for generating design/stability charts of pillars for different geo-mining conditions. These design charts can be used for the design of crown pillar thickness with the depth of the working, taking into account the changes of the rock mass conditions in underground metal mine.

3. **“Experimental and numerical crushing analysis of circular CFRP tubes under axial impact loading” By** Corin Reutera,, Kim-Henning Sauerlandb, Thomas Tröstera

In this paper, a prospective simulation method for composite crushing under axial crash loading is presented. To this end carbon fibre-reinforced plastic (CFRP) circular crash tubes are investigated in drop tower tests. Flat specimen tests are performed to determine calibration parameters and are used for efficient re-parameterization of a transversally isotropic material card used in finite element (FE) simulation. An existing material card for CFRP based on basic tension and compression tests is used as a starting point and only a small set of material parameters is numerically reasonable adjusted to account for crushing. Once calibrated by means of flat specimens the material model is able to cover a variety of different composite layups and specimen geometries, e.g. tube specimens. Therefore, numerical simulation of drop tower testing is carried out and results show good agreement between numerical and experimental results. In addition to these tests, it can be shown that the presented approach is leading to equally good results when the material and geometry of the specimens are changed to a glass fibre-reinforced plastic (GFRP) tube structure.

4. **“Evaluation of the survivability of CFRP honeycomb-cored panels in compression after impact tests” By** Oleg A. Staroverov, Elena M. Strungar, Valery E. Wildemann

This paper is oriented to the experimental research of the mechanics of the CFRP sandwich plates, glass and carbon fiber sample panels with a large-cell honeycomb core. The method for testing polymer composite sample plates in compression after impact (CAI) tests with joint use of a testing machine and a video system for deformation field registration was tested. Analysis of the experimental data obtained highlighted the impactive sensitivity zone for the test specimens. A quantitative assessment of the loadbearing capacity of glass and carbon fiber sample panels in CAI tests with the different levels of the drop weight impact energy was performed. Photos of samples after impact have been provided. Vic-3D non-contact three-dimensional digital optical system was used to register the displacement and deformation fields on the surface of the samples. The video system was used to evaluate various damage mechanisms, including matrix cracking, delaminations, and rupture of the damaged fibers. The paper studied the evolution of non-homogeneous deformation fields on the surface of the composite samples during the post-impact compression tests and analyzed the configuration of non-homogeneous deformation fields.

5. **"Design, Testing, Analysis, and Material Properties of Carbon Fiber Reinforced Polymers" By** Andrew Miner, Simon Jones

This research covers many of the learnings I found in the structures of fiber reinforced polymers, manufacturing processes and controls, testing procedures and standards, important considerations in the design process of CFRPs, and analysis capabilities and methods. This research is organized as a series of short guides to assist students with the individual subject matters at hand. While many of these are bolstered by the knowledge provided in other parts of the document, due to the multifaceted approach to the problem, this is a much easier way to communicate the information to undergraduate science and engineering students trying to work reliably with CFRPs or other composite materials. Special thanks to the Rose-Hulman Institute of Technology Mechanical Engineering Department for supporting this project.

6. **"Predicting the axial crush response of CFRP tubes using three damage-based constitutive models" By** Aleksandr Cherniaev, Clifford Butcher, John Montesano

In this study, predictive capabilities of three widely used LS-DYNA composite material models MAT054, MAT058 and MAT262 – were investigated and compared with respect to modeling of axial crushing of CFRP energy absorbers. Results of crush simulations with non-calibrated material models were compared with available experimental data, and then parameter tuning was conducted to improve correlation with experiments. Furthermore, calibrated material models were used to conduct independent crash simulations with distinct composite layups. As a result, advantages and shortcomings of the considered material models, as well as directions for future developments, were identified.

3. Problem Statement, Objectives and Scope

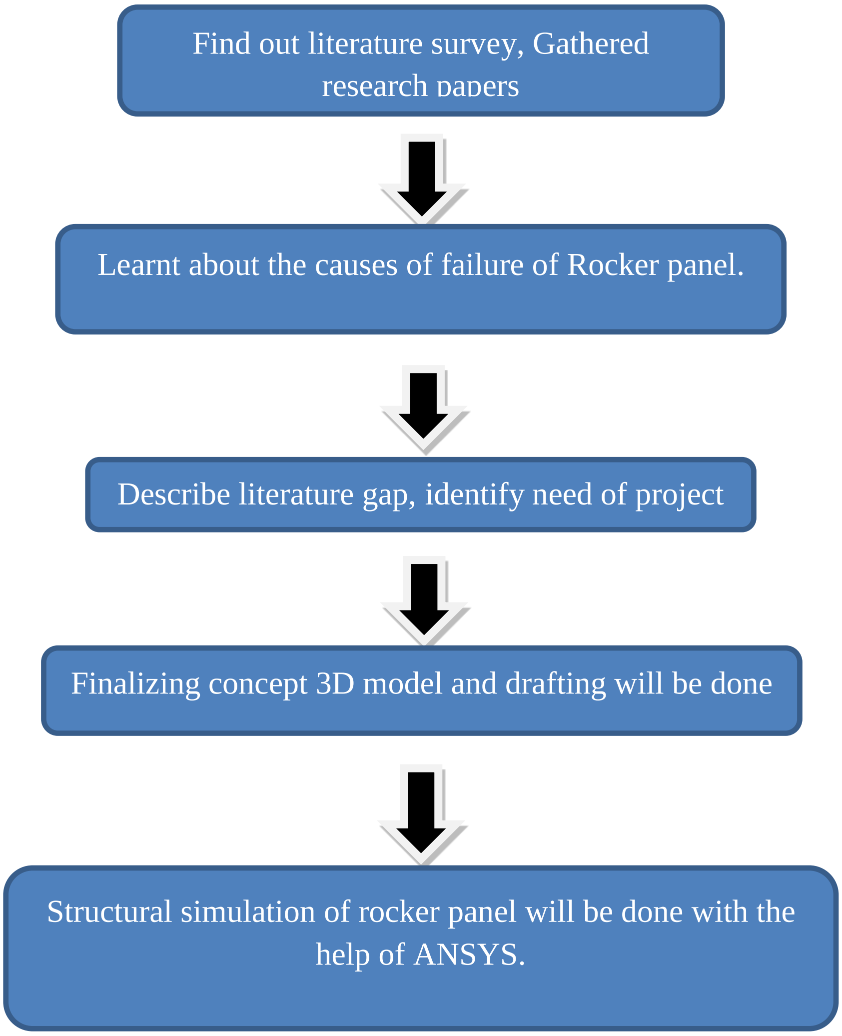
3.1 Problem statement

In order to meet the challenges of making a lighter vehicle to achieve fuel economy and good structure in crashworthiness, use of the carbon fibre reinforced polymer (CFRP) materials can be one of the solutions.

3.2 Objectives

- To study and perform static analysis on 4-wheeler rocker panel specimen under loading condition.
- To propose an optimized model this will have better or same performance and reduced weight.
- CAD modeling of 4-wheeler rocker panel specimen in Catia V5R20 software.
- To perform static structural Analysis of optimized 4-wheeler rocker panel specimen in ANSYS 19 workbench.
- Experimental investigation of CFRP rocker panel specimen will be done by three-point bending test on UTM.
- Comparative Analysis between Experimental & Analysis results.

3.3 Methodology



4.CASE STUDY

4.1.DESIGN:

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 graphics showing 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 analysed 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.

4.2TYPES:

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.

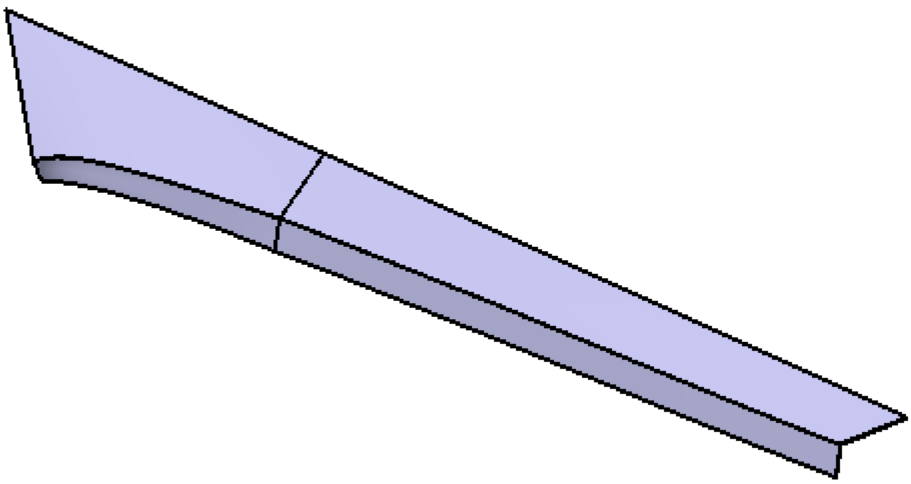
There are two types of 3D Solid Modeling

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.



.

4.3ANALYSYS:

The finite element method (FEM), is a numerical method for solving problems of engineering and mathematical physics. Typical problem areas of interest include structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential. The analytical solution of these problems generally require the solution to boundary value problems for partial differential equations. The finite element method formulation of the problem results in a system of algebraic equations. The method yields approximate values of the unknowns at discrete number of points over the domain.[1] To solve the problem, it subdivides a large problem into smaller, simpler parts that are called finite elements. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem. FEM then uses variational methods from the calculus of variations to approximate a solution by minimizing an associated error function.

Studying or analyzing a phenomenon with FEM is often referred to as finite element analysis (FEA).

4.4 BASIC CONCEPTS:

The subdivision of a whole domain into simpler parts has several advantages:[2]

- Accurate representation of complex geometry
- Inclusion of dissimilar material properties
- Easy representation of the total solution
- Capture of local effects.

A typical work out of the method involves (1) dividing the domain of the problem into a collection of sub domains, with each sub domain represented by a set of element equations to the original problem, followed by (2) systematically recombining all sets of element equations into a global system of equations for the final calculation. The global system of equations has known solution techniques, and can be calculated from the initial values of the original problem to obtain a numerical answer.

In the first step above, the element equations are simple equations that locally approximate the original complex equations to be studied, where the original equations are often partial differential equations (PDE). To explain the approximation in this process, FEM is commonly introduced as a special case of Galerkin method. The process, in mathematical language, is to construct an integral of the inner product of the residual and the weight functions and set the integral to zero. In simple terms, it is a procedure that minimizes the error of approximation by fitting trial functions into the PDE. The residual is the error caused by the trial functions, and the weight functions are polynomial approximation functions that project the residual. The process eliminates all the spatial derivatives from the PDE, thus approximating the PDE locally with

- a set of algebraic equations for steady state problems,
- a set of ordinary differential equations for transient problems.

These equation sets are the element equations. They are linear if the underlying PDE is linear, and vice versa. Algebraic equation sets that arise in the steady state problems are solved using numerical linear algebra methods, while ordinary differential equation sets that arise in the transient problems are solved by numerical integration using standard techniques such as Euler's method or the Runge-Kutta method.

In next step above, a global system of equations is generated from the element equations through a transformation of coordinates from the sub domains' local nodes to the domain's global nodes. This spatial transformation includes appropriate orientation adjustments as applied in relation to the reference coordinate system. The process is often carried out by FEM software using coordinate data generated from the sub domains.

FEM is best understood from its practical application, known as finite element analysis (FEA). FEA as applied in engineering is a computational tool for performing engineering analysis. It includes the use of mesh generation techniques for dividing a complex problem into small elements, as well as the use of software program coded with FEM algorithm. In applying FEA, the complex problem is usually a physical system with the underlying physics such as the Euler-Bernoulli beam equation, the heat equation, or the Navier-Stokes equations expressed in either PDE or integral equations, while the divided small elements of the complex problem represent different areas in the physical system.

FEA is a good choice for analyzing problems over complicated domains (like cars and oil pipelines), when the domain changes (as during a solid state reaction with a moving boundary), when the desired precision varies over the entire domain, or when the solution lacks smoothness. FEA simulations provide a valuable resource as they remove multiple instances of creation and testing of hard prototypes for various high fidelity situations. For instance, in a frontal crash simulation it is possible to increase prediction accuracy in "important" areas like the front of the car and reduce it in its rear (thus reducing cost of the simulation). Another example would be in numerical weather prediction, where it is more important to have accurate predictions over developing highly nonlinear phenomena (such as tropical cyclones in the atmosphere, or eddies in the ocean) rather than relatively calm areas.

4.5 MESH

ANSYS Meshing is a general-purpose, intelligent, automated high-performance product. It produces the most appropriate mesh for accurate, efficient multiphysics solutions. A mesh well suited for a specific analysis can be generated with a single mouse click for all parts in a model. Full controls over the options used to generate the mesh are available for the expert user who wants to fine-tune it. The power of parallel processing is automatically used to reduce the time you have to wait for mesh generation. Creating the most appropriate mesh is the foundation of engineering simulations. ANSYS Meshing is aware of the type of solutions that will be used in the project and has the appropriate criteria to create the best suited mesh. ANSYS Meshing is automatically integrated with each solver within the ANSYS Workbench environment. For a quick analysis or for the new and infrequent user, a usable mesh can be created with one click of the mouse. ANSYS Meshing chooses the most appropriate options based on the analysis type and the geometry of the model. Especially convenient is the ability of ANSYS Meshing to automatically take advantage of the available cores in the computer to use parallel processing and thus significantly reduce the time to create a mesh. Parallel meshing is available without any additional cost or license requirements

4.6 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 result

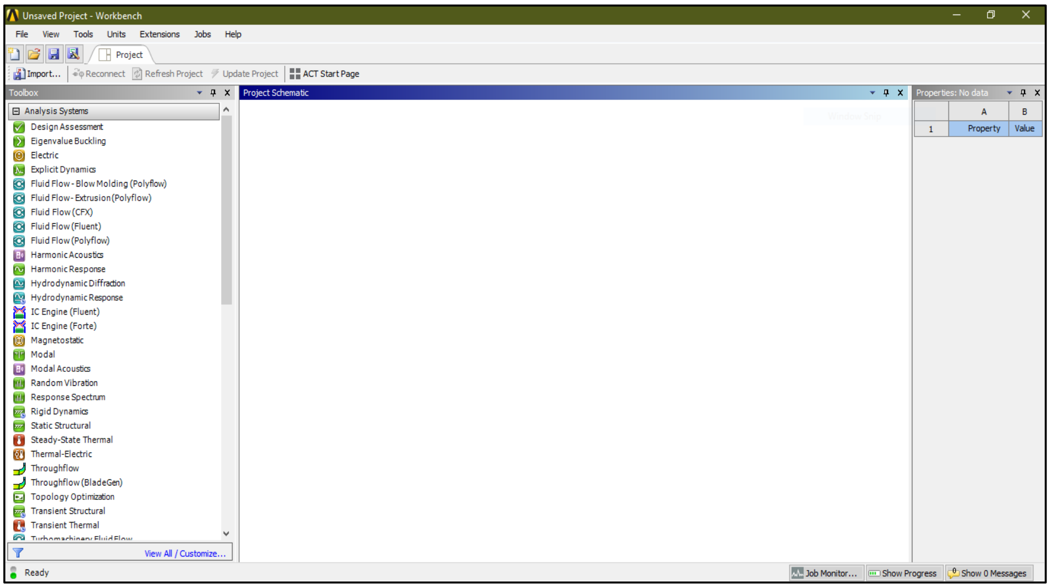


Fig.4.1 Ansys Workbench Slide

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

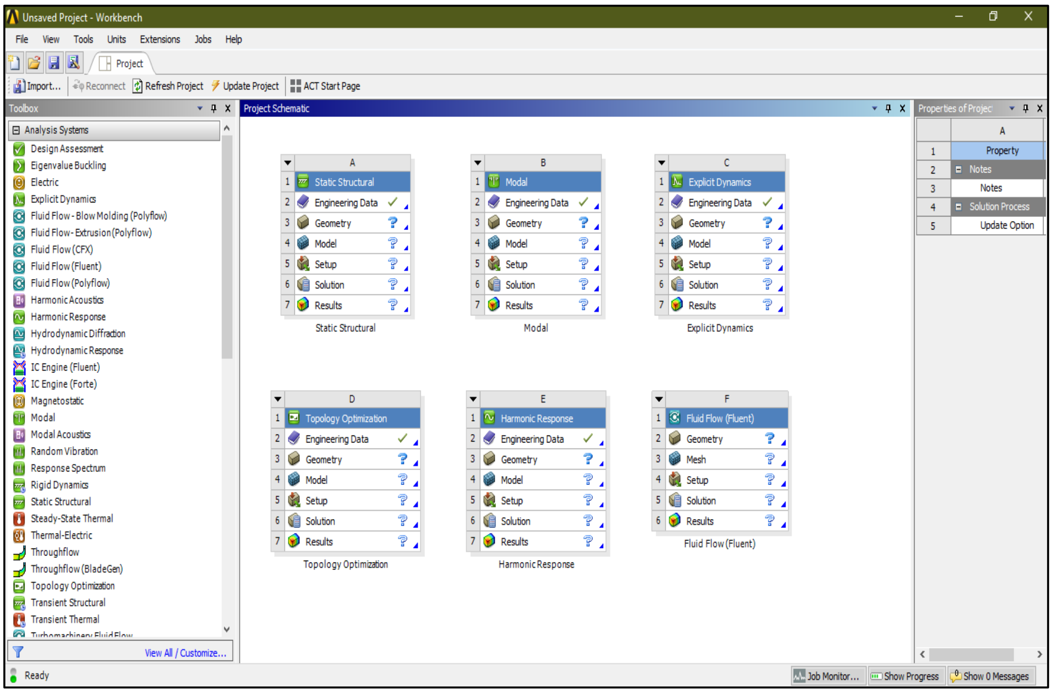
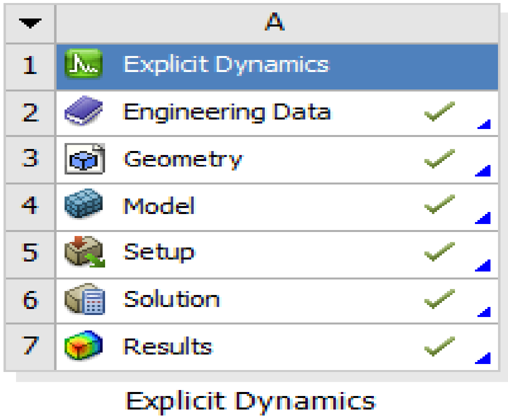


Fig.4.2 Ansys Workbench Slide



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.

5. STRUCTURAL ANALYSIS OF ROCKER PANEL

5.1 DESIGN AND ANALYSIS OF ROCKER PANEL USING REVERSE ENGINEERING METHOD

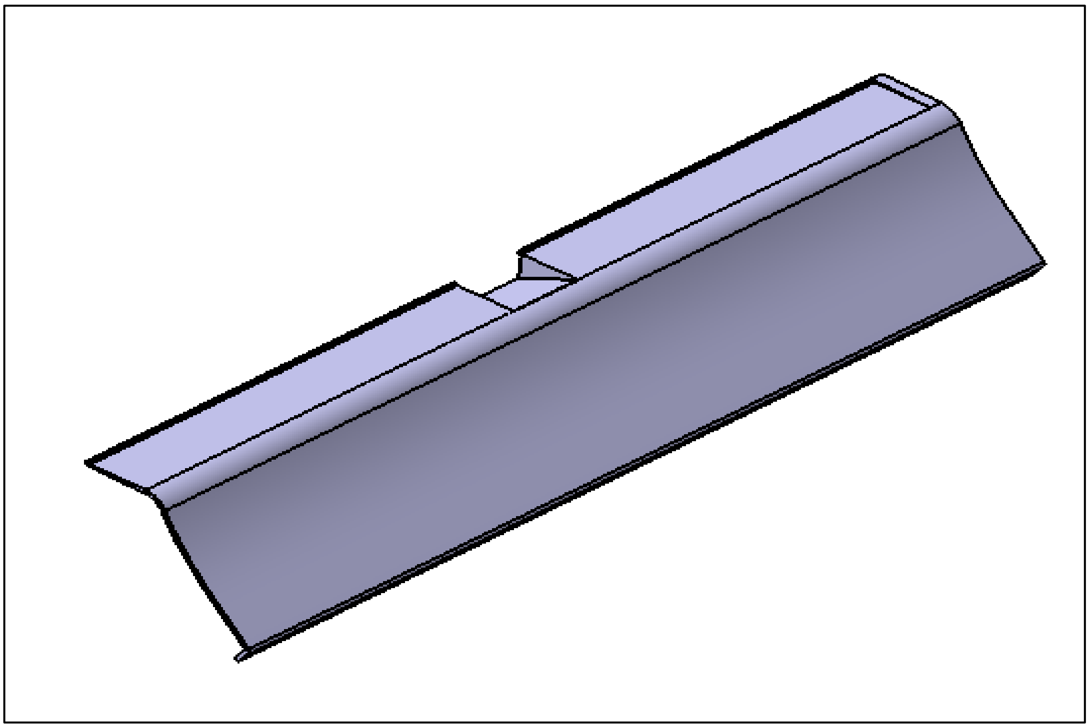


Fig.5.1 Rocker panel

Properties of Outline Row 4: PLASTIC

	A	B	C
1	Property	Value	Unit
2	Material Field Variables	Table	
3	Density	1.2	g cm^-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...	
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
12	Tensile Yield Strength	2.5E+07	Pa
13	Compressive Yield Strength	0	Pa
14	Tensile Ultimate Strength	3.3E+07	Pa
15	Compressive Ultimate Strength	0	Pa

Table 5.1. Rocker arm material properties

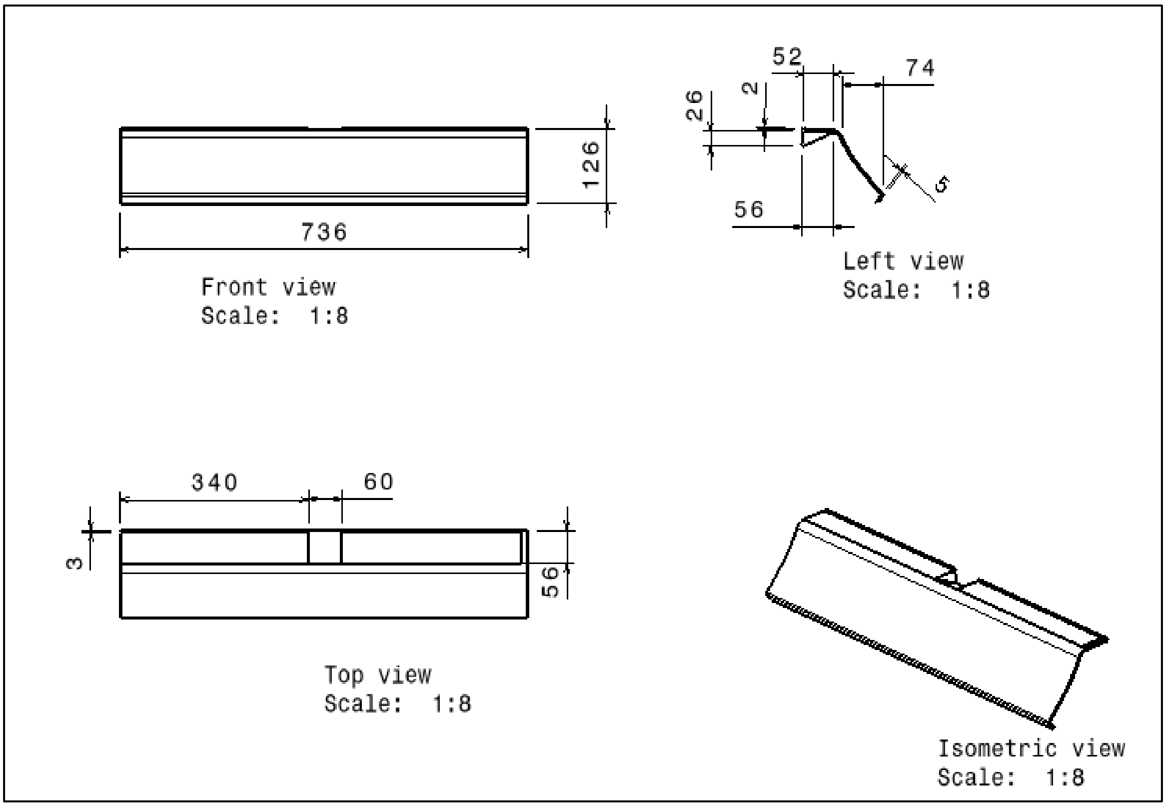


Fig.5.2 Drafting

5.2 ANALYSIS OF EXISTING ROCKER PANEL USING PLASTIC MATERIAL

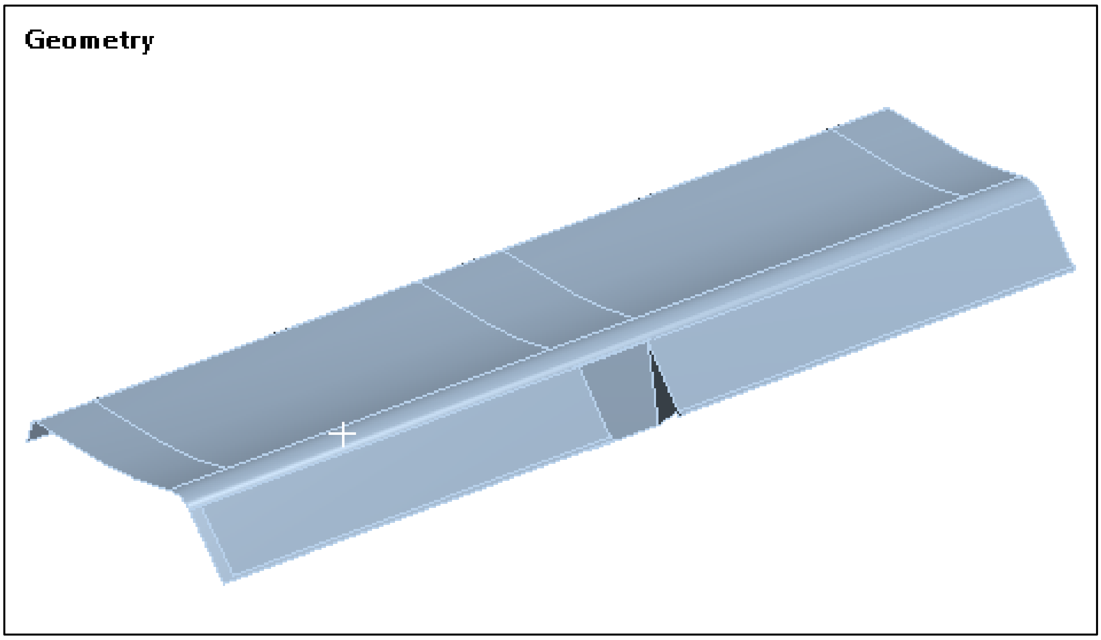
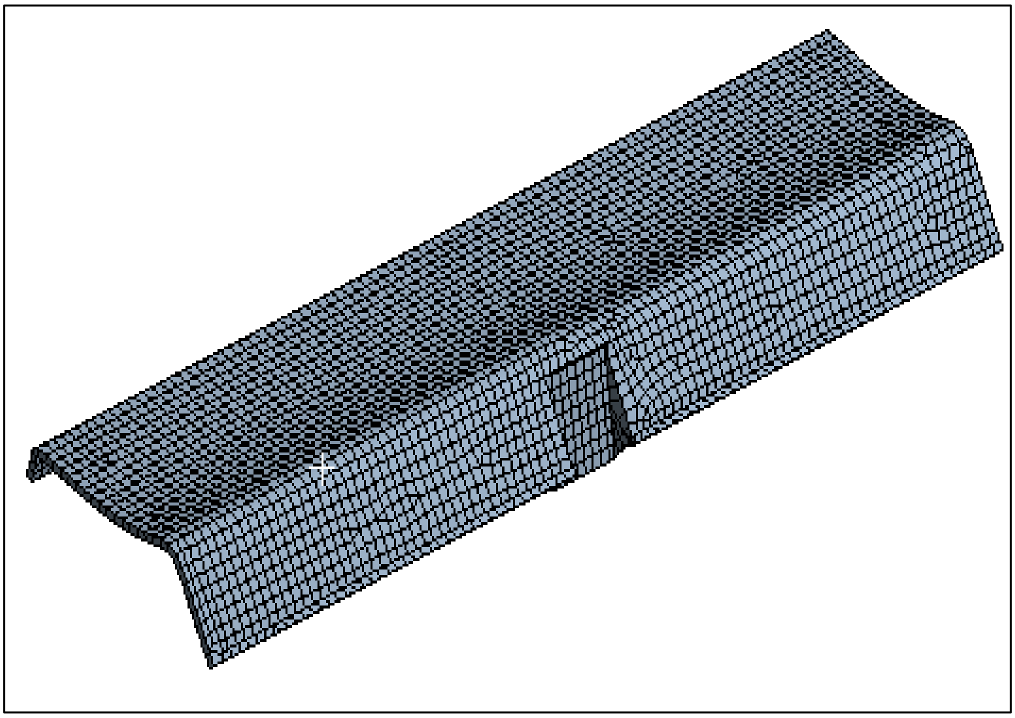


Fig.5.3 Geometry



Details of "Body Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	8.0 mm
Advanced	
Defeature Size	Default
Behavior	Soft

Statistics	
Nodes	3060
Elements	2934

Properties	
Volume	8.2448e+005 mm^3
Mass	0.98938 kg
Surface Area(approx.)	1.649e+005 mm^2

Table 5.2. Meshing details

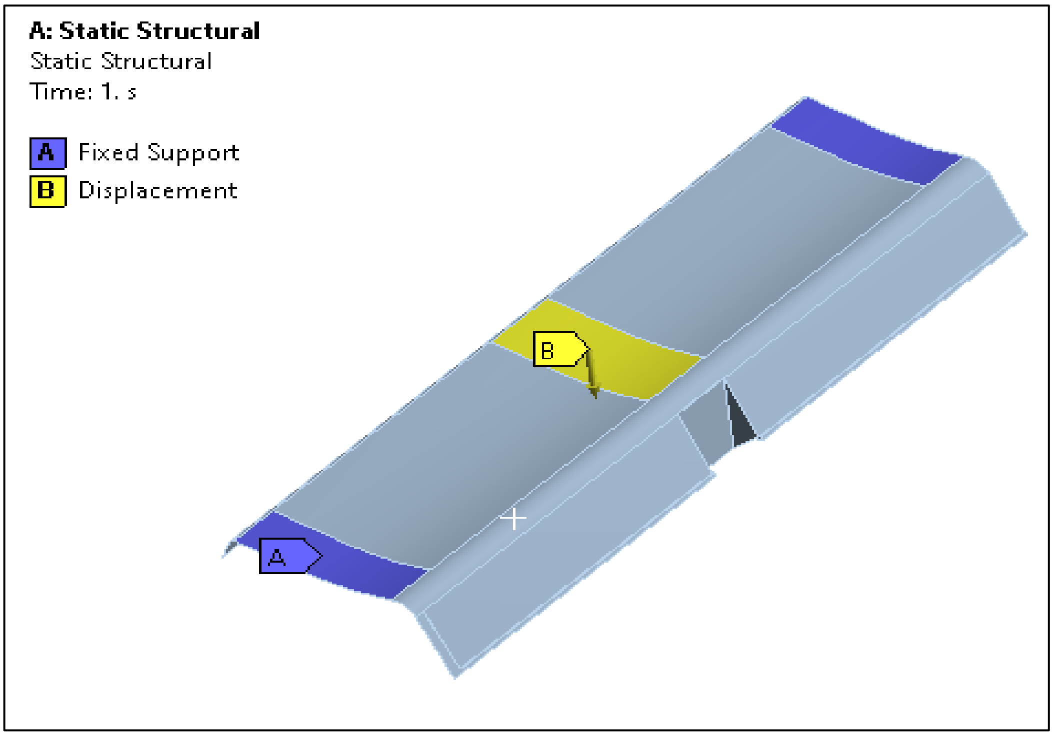


Fig.5.4 Boundary condition

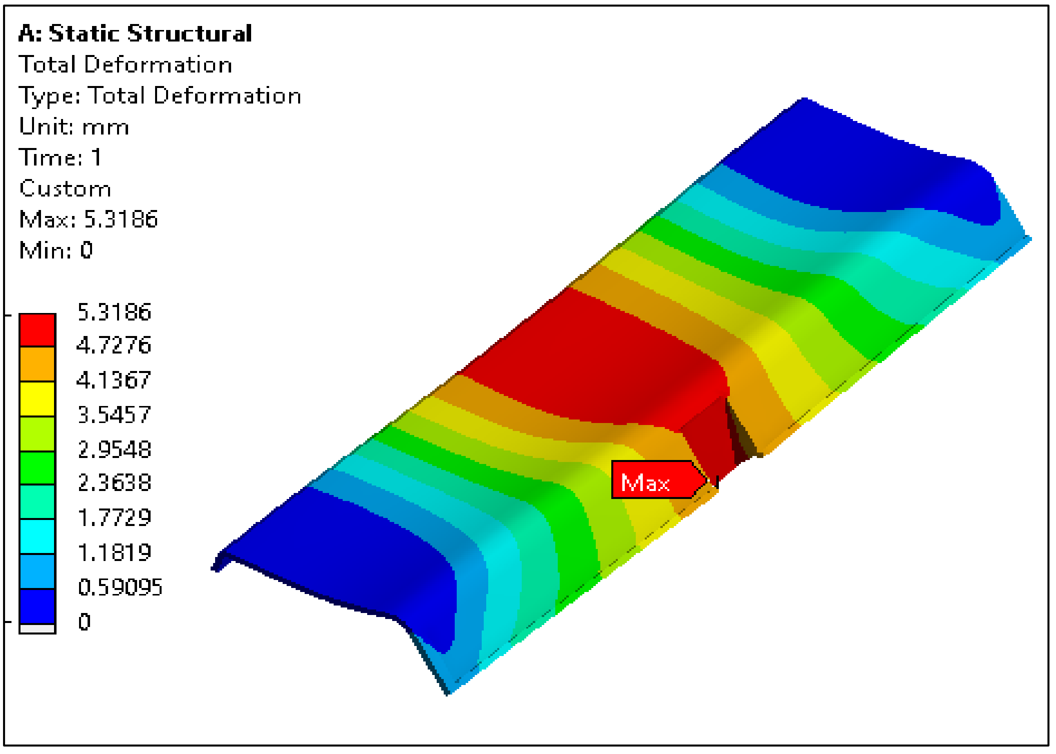


Fig.5.5 Total deformation

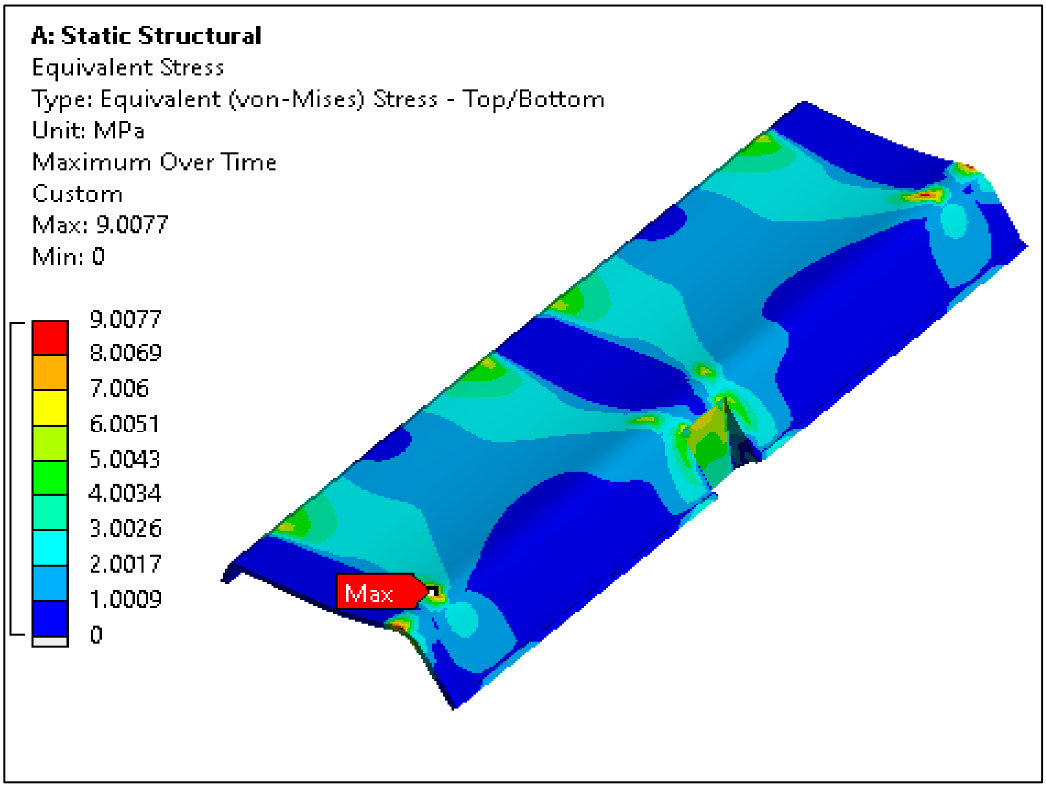


Fig.5.6 Equivalent stress

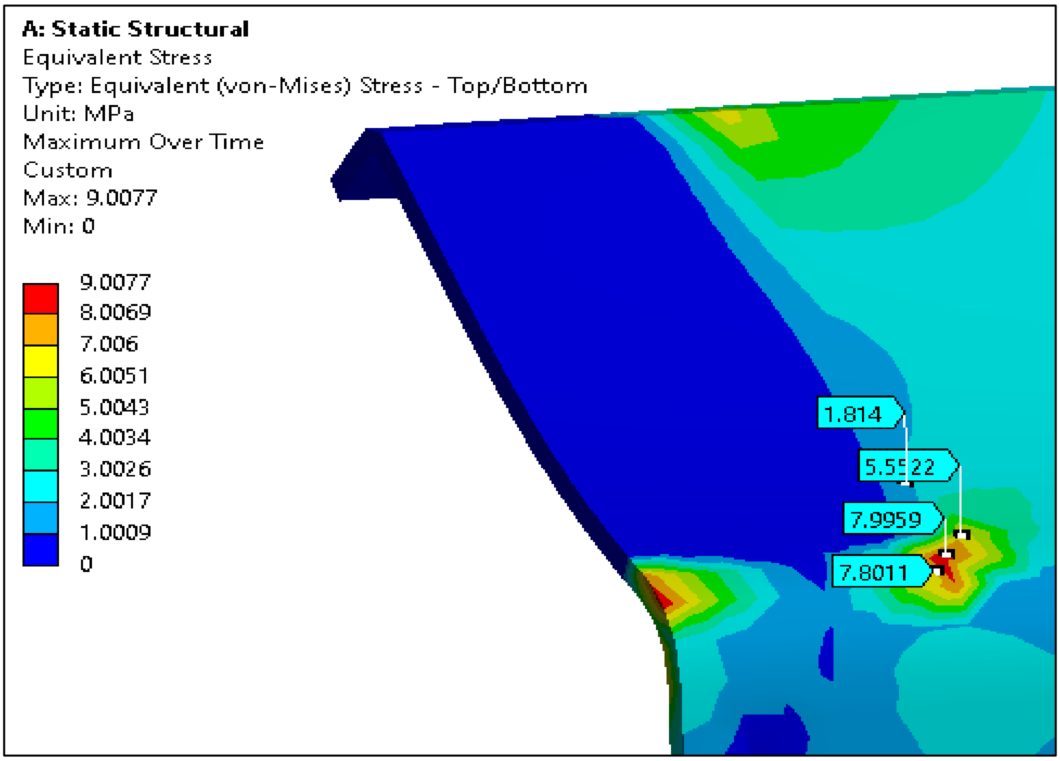


Fig.5.7 Equivalent stress

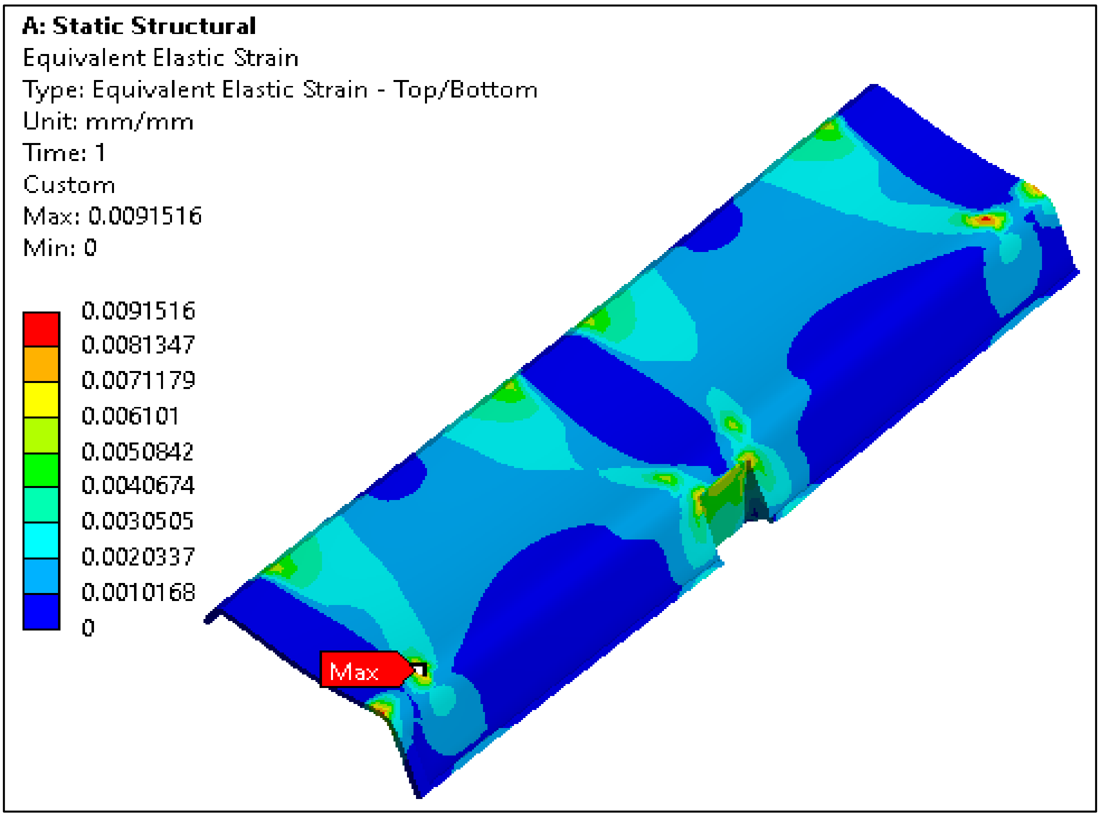


Fig.5.8 .Equivalent elastic strain

Details of "Force Reaction"	
Maximum Value Over Time	
X Axis	4.5453e-003 N
Y Axis	-430.87 N
Z Axis	1156.9 N
Total	1234.5 N

Table.5.1 Force reaction

6. Manufacturing of composite materials-

6.1 Open Molding

Composite materials (resin and fibers) are placed in an open mold, where they cure or harden while exposed to the air. Tooling cost for open molds is often inexpensive, making it possible to use this technique for prototype and short production runs.

Hand Lay-Up

Hand lay-up is an open molding method suitable for making a wide variety of composites products from very small to very large. Production volume per mold is low; however, it is feasible to produce substantial production quantities using multiple molds. Hand lay-up is the simplest composites molding method, offering low cost tooling, simple processing, and a wide range of part sizes. Design changes are readily made. There is a minimum investment in equipment. With skilled operators, good production rates and consistent quality are obtainable.

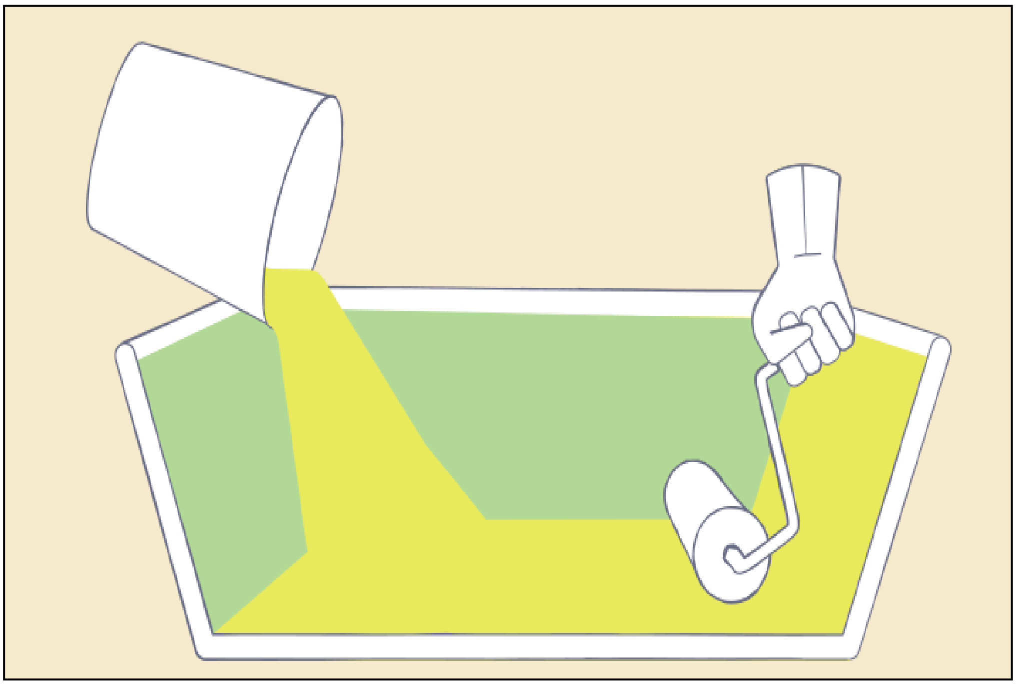


Fig.6.1 Hand lay-up

Process

Gel coat is first applied to the mold using a spray gun for a high-quality surface. When the gel coat has cured sufficiently, roll stock fiberglass reinforcement is manually placed on the mold. The laminating resin is applied by pouring, brushing, spraying, or using a paint roller. FRP rollers, paint rollers, or squeegees are used to consolidate the laminate, thoroughly wetting the reinforcement and removing entrapped air. Subsequent layers of fiberglass reinforcement are added to build laminate thickness. Low density core materials such as end-grain balsa, foam, and honeycomb, are commonly used to stiffen the laminate. This is known as sandwich construction.

6.2 GLASS FIBER INFORMATION

Glass fibers are formed from melts and manufactured in various compositions by changing the number of raw materials like sand for silica, clay for alumina, calcite for calcium oxide, and colemanite for boron oxide. Therefore, different types of glass fibers show different performances like alkali resistance or high mechanical properties using various amounts of silica or other sources. Glass fiber products are classified according to the type of composite at which they are utilized. Moreover, chopped strands, direct draw roving's, assembled roving, and mats are the most important products that are used in the injection molding, filament winding, pultrusion, sheet molding, and hand layup processes to form glass fiber-reinforced composites. Protection of the glass fiber filaments from breakage or disintegration is an important issue either during manufacturing of glass fiber or during composite production.

Applying sizing agent to the glass fiber during manufacturing of fibers causes lubrication of the glass fiber filaments in addition to inhibit static electricity accumulation, adhesion of the fiber filaments together, and adhesion between fiber filaments and polymer matrix of the composites. During manufacturing of composites, an interphase layer, at which interpenetration of the sizing to the matrix or diffusion of the matrix polymer to the sizing, is formed. The resultant interphase layer can either increase or decrease the performance of the composite considering harmony between sizing components and matrix polymer. Compatibility between sizing and matrix polymer enhances high mechanical properties and on the contrary incompatible sizing results poor mechanical properties. From energy point of view, reduction in the weight of vehicles is the main reason to save energy in the transportation industry, and in this regard growth in the production of lightweight cars to about 50% indicates importance of the glass fiber-reinforced composites. Consequently, growth in the glass fiber production is what that happened and will be continued in the future.

Glass fiber is formed when thin strands of silica-based or other formulation glass are extruded into many fibers with small diameters suitable for textile processing. The technique of heating and drawing glass into fine fibers has been known for millennia; however, the use of these fibers for textile applications is more recent. Until this time, all glass fiber had been manufactured as staple (that is, clusters of short lengths of fiber).

The modern method for producing glass wool is the invention of Games Slayter working at the Owens-Illinois Glass Company (Toledo, Ohio). He first applied for a patent for a new process to make glass wool in 1933. The first commercial production of glass fiber was in 1936. In 1938 Owens-Illinois Glass Company and Corning Glass Works joined to form the Owens-Corning Fiberglas Corporation. When the two companies joined to produce and promote glass fiber, they introduced continuous filament glass fibers.[2] Owens-Corning is still the major glass-fiber producer in the market today

Fig.6.2 Glass fibre composite material

EXPERIMENTAL TESTING

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.

6.3 ANALYSIS OF OPTIMIZED ROCKER PANEL USING PLASTIC AND GLASS FIBER COMPOSITE MATERIAL

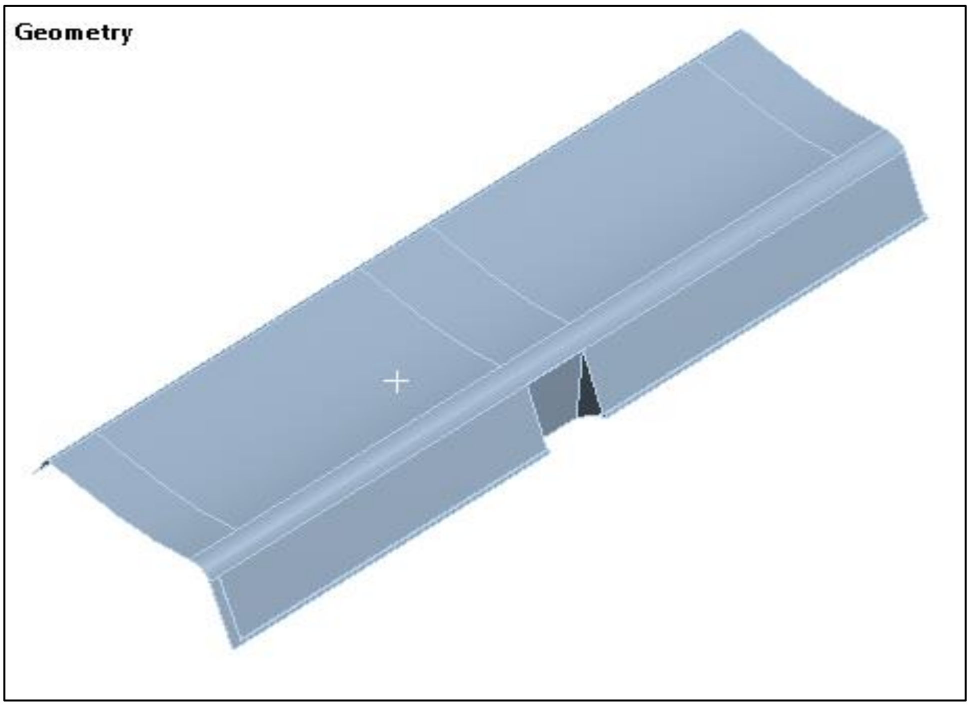
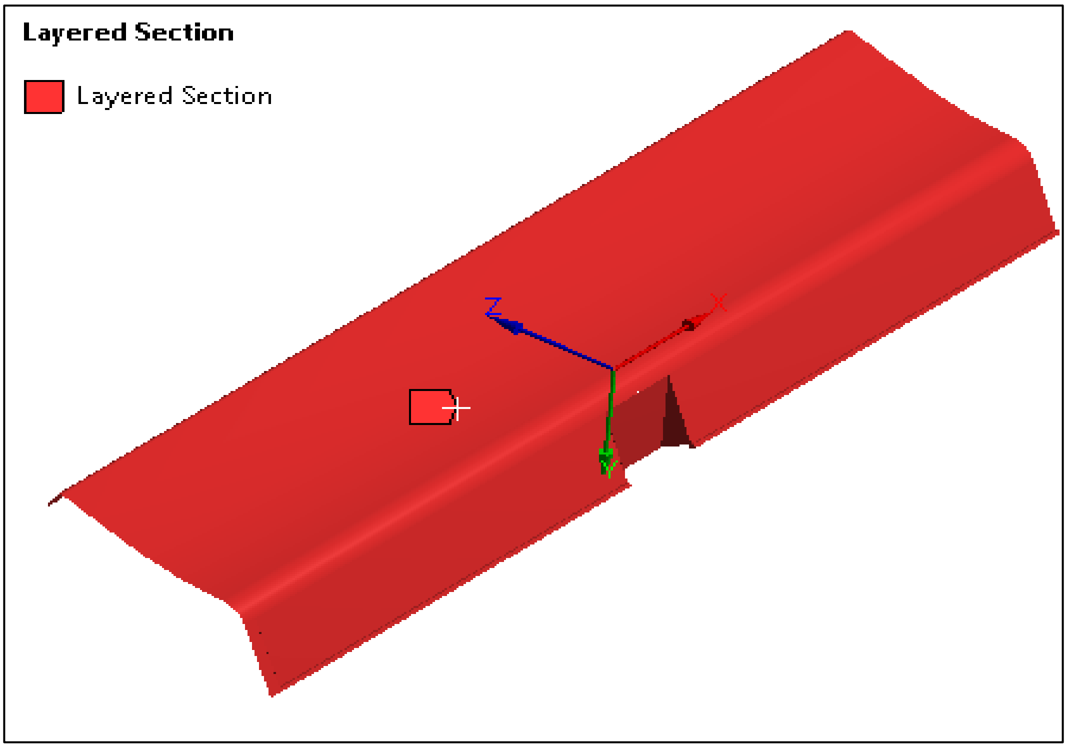


Fig.6.3 Geometry

Properties of Outline Row 3: Epoxy E-Glass UD

	A	B	C
1	Property	Value	Unit
2	Density	2000	kg m^-3
3	Orthotropic Elasticity		
4	Young's Modulus X direction	45000	MPa
5	Young's Modulus Y direction	10000	MPa
6	Young's Modulus Z direction	10000	MPa
7	Poisson's Ratio XY	0.3	
8	Poisson's Ratio YZ	0.4	
9	Poisson's Ratio XZ	0.3	
10	Shear Modulus XY	5000	MPa
11	Shear Modulus YZ	3846.2	MPa
12	Shear Modulus XZ	5000	MPa

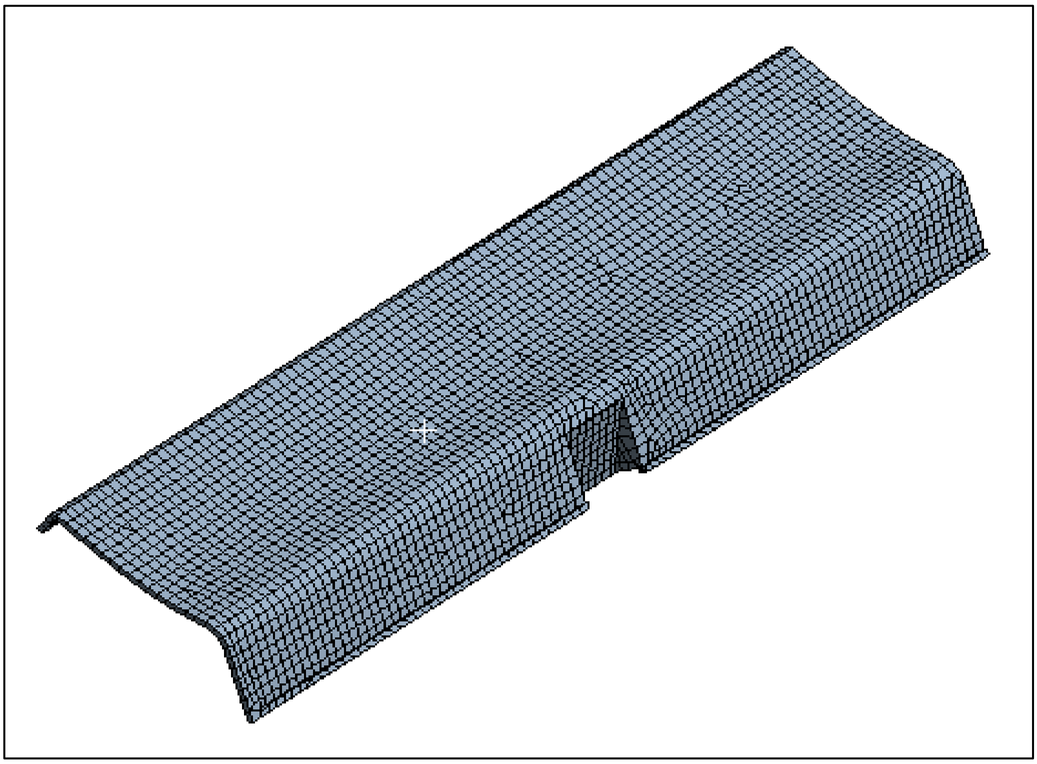
Table.6.1. Material properties of glass fiber



Layer	Material	Thickness (mm)	Angle (°)
(+Z)			
2	Epoxy E-Glass UD	1	0
1	PLASTIC	3	0
(-Z)			

Properties	
Total Thickness	4. mm
Total Mass	0.92342 kg

Fig.6.4 Layered selection for composite material



Details of "Body Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	8.0 mm
Advanced	
Defeature Size	Default
Behavior	Soft

Statistics	
Nodes	3060
Elements	2934

Fig.6.5 Meshing details

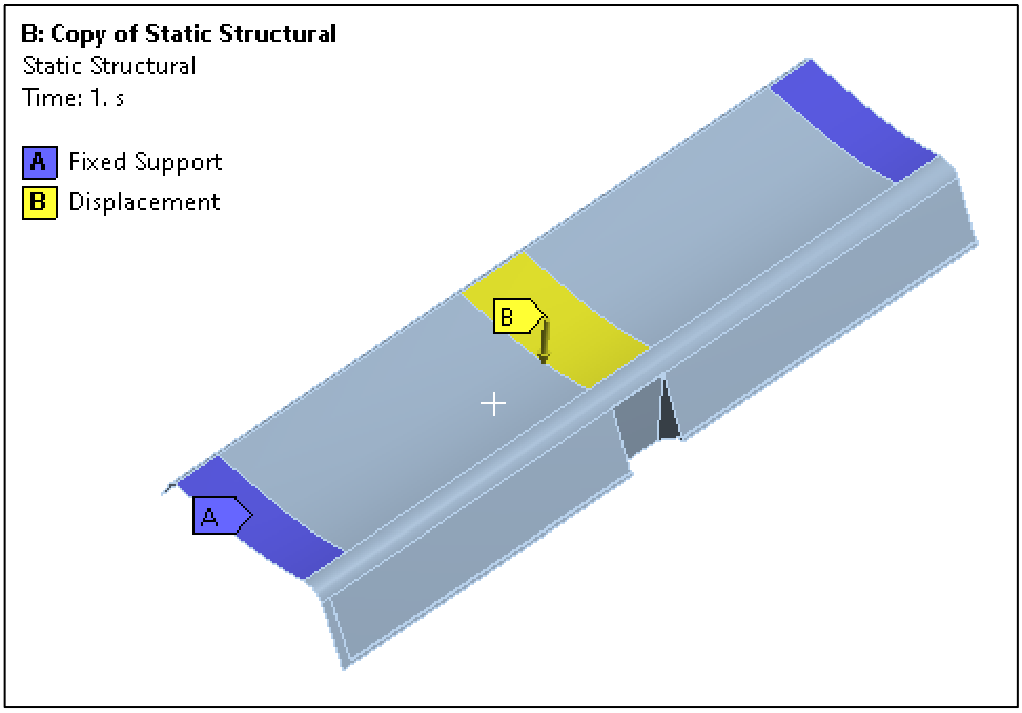


Fig.6.6 Rocker panel boundary condition

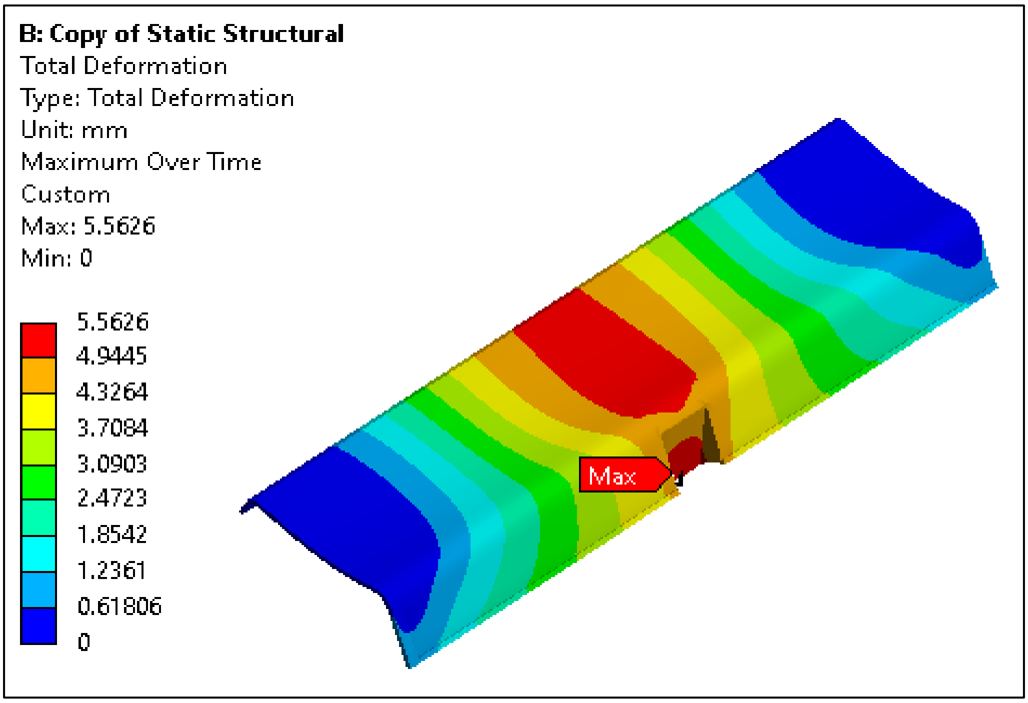


Fig.6.7 Total deformation

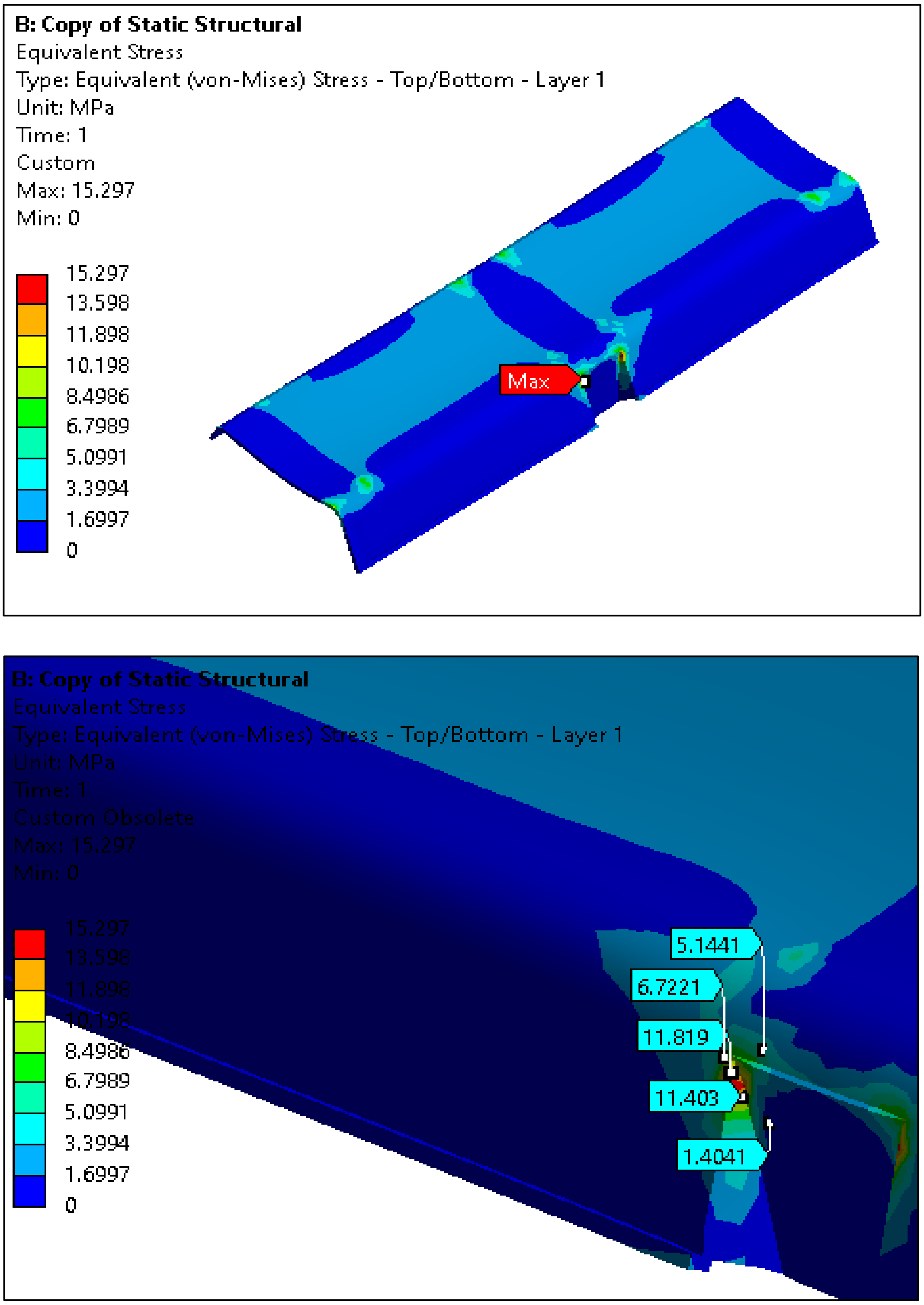


Fig.6.8 Equivalent stress

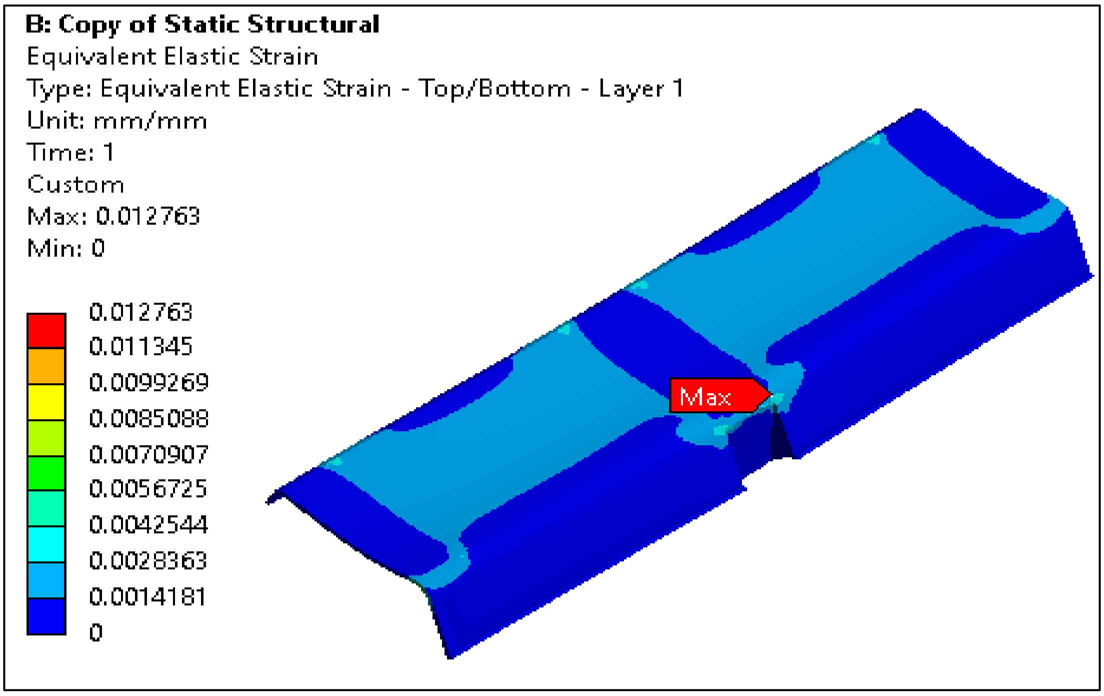


Fig.6.9 Equivalent elastic strain

Details of "Force Reaction"

Maximum Value Over Time	
X Axis	-8.3707 N
Y Axis	-1588.1 N
Z Axis	4634.8 N
Total	4899.4 N

Fig. Force reaction

7. 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

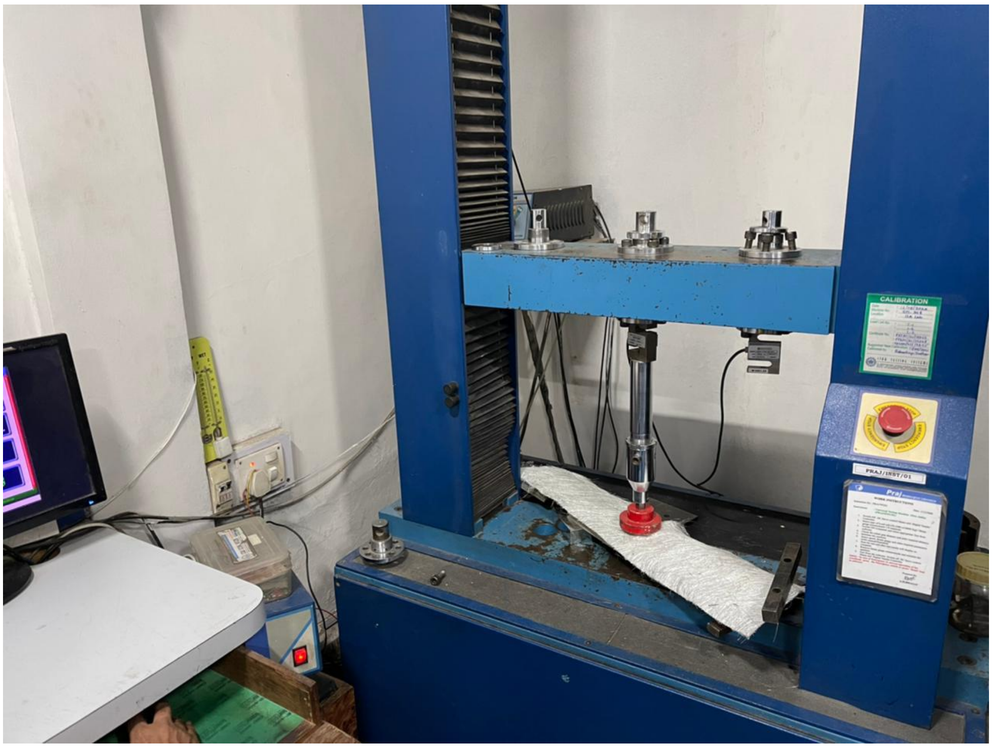
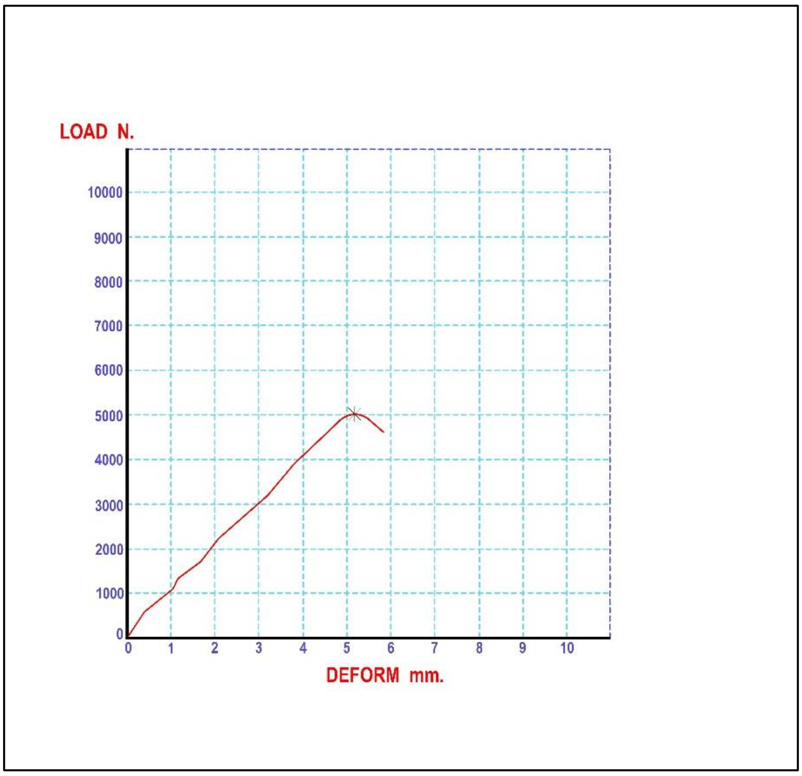


Fig.7.1 Experimental testing



As per FEA result the reaction force for optimized rocker panel is 4899 N. And for experimental testing the reaction force for 5 mm displacement is 4950 N.

RESULT AND DISCUSSION

The rocker panel used to protect vehicle from external damage. The material used for rocker panel is plastic. To increase the strength of rocker panel without increasing the glass fiber material is selected.

Applied the glass fiber layer on the rocker panel using hand layup method. As per result the reaction force of rocker panel using composite material is increased.

SR NO	***COMPONENT***	***TOTAL DEFORMATION (mm)***	***EQUIVALENT STRESS (MPa)***	***WEIGHT (kg)***	***REACTION FORCE (N)***
1	EXISTING MODEL	5.3	9.00	0.989	1234
2	OPTIMIZED MODEL	5.5	15.29	0.92	4899

Table 7.1 Result Table

8.CONCLUSION

CASE STUDY

In this project we have performed the structural analysis of Rocker panel made of steel and ploted the result of total deflection and Force reaction of Rocker panel. After the optimization of rocker panel by reducing the thickness of steel plate and layering of carbon fiber the overall weight of the optimized rocker panel is observed. The optimized rocker panel is also gone through the process of structural analysis and from the plots it is concluded that the optimized rocker panel has best reaction force than the original one. As the reaction force for the original rocker panel is having force reaction of 9754.4 N and the optimized model has the force reaction of 9969.2 N.

ROCKER PANEL

Perform static analysis using ANSYS software and find out optimized model for the rocker panel. The material used for rocker panel for existing material is plastic. The equivalent stress and reaction force generated by the existing rocker panel is 9.00 MPa and 1234 N.

For optimized model layer of glass fiber used for improving strength of model. The equivalent stress and reaction force generated in optimized model is 15.29 MPa and 4899 N. As per analysis result the optimized model perform better in three point bending test.

The weight of existing rocker panel is 0.98 kg and weight of optimized model with glass fiber reinforcement is 0.92 kg. Hence total weight optimized in rocker panel is 6.97%.

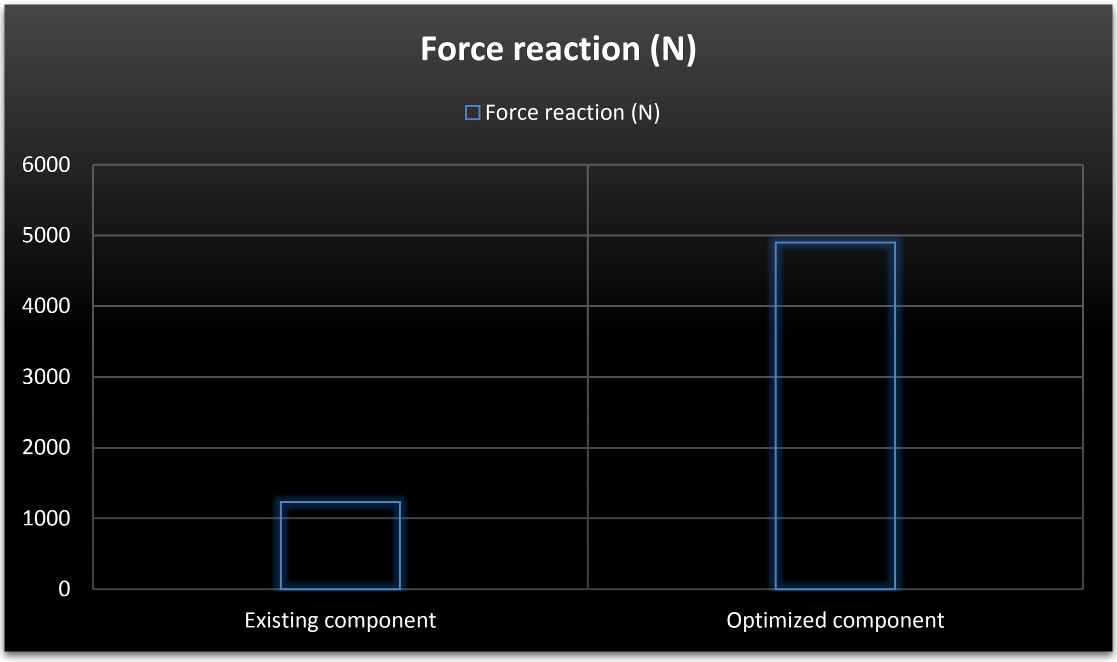


Fig.8.1 Force reaction comparison

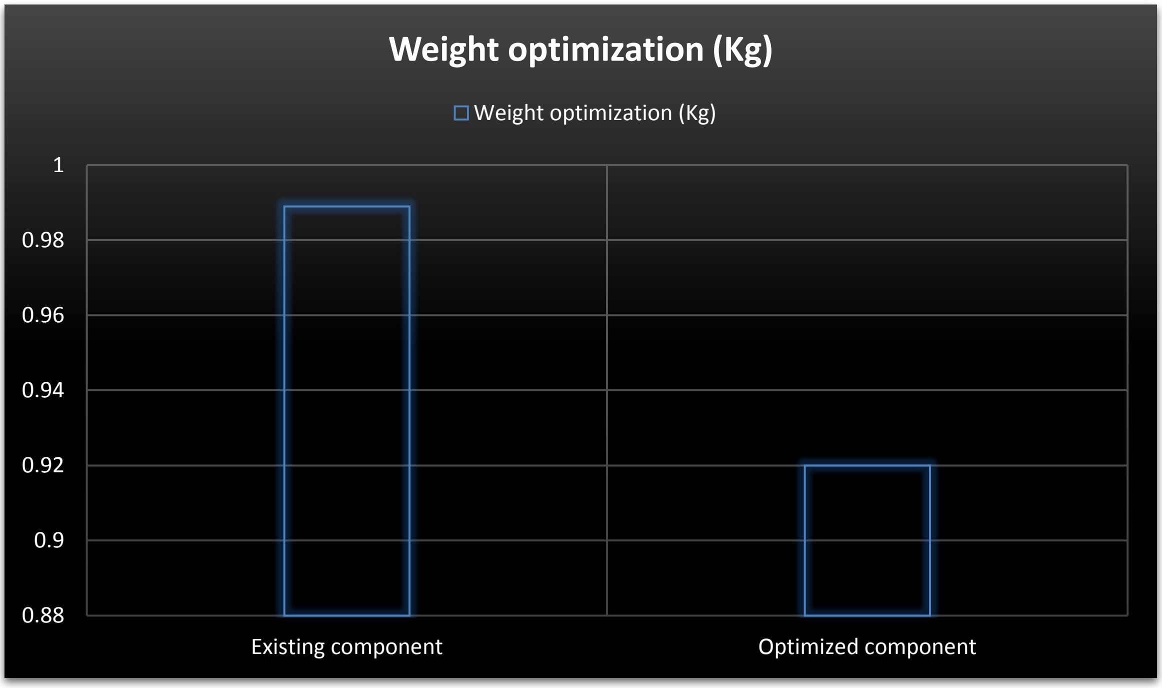


Fig.8.2 Weight Optimization Comparison

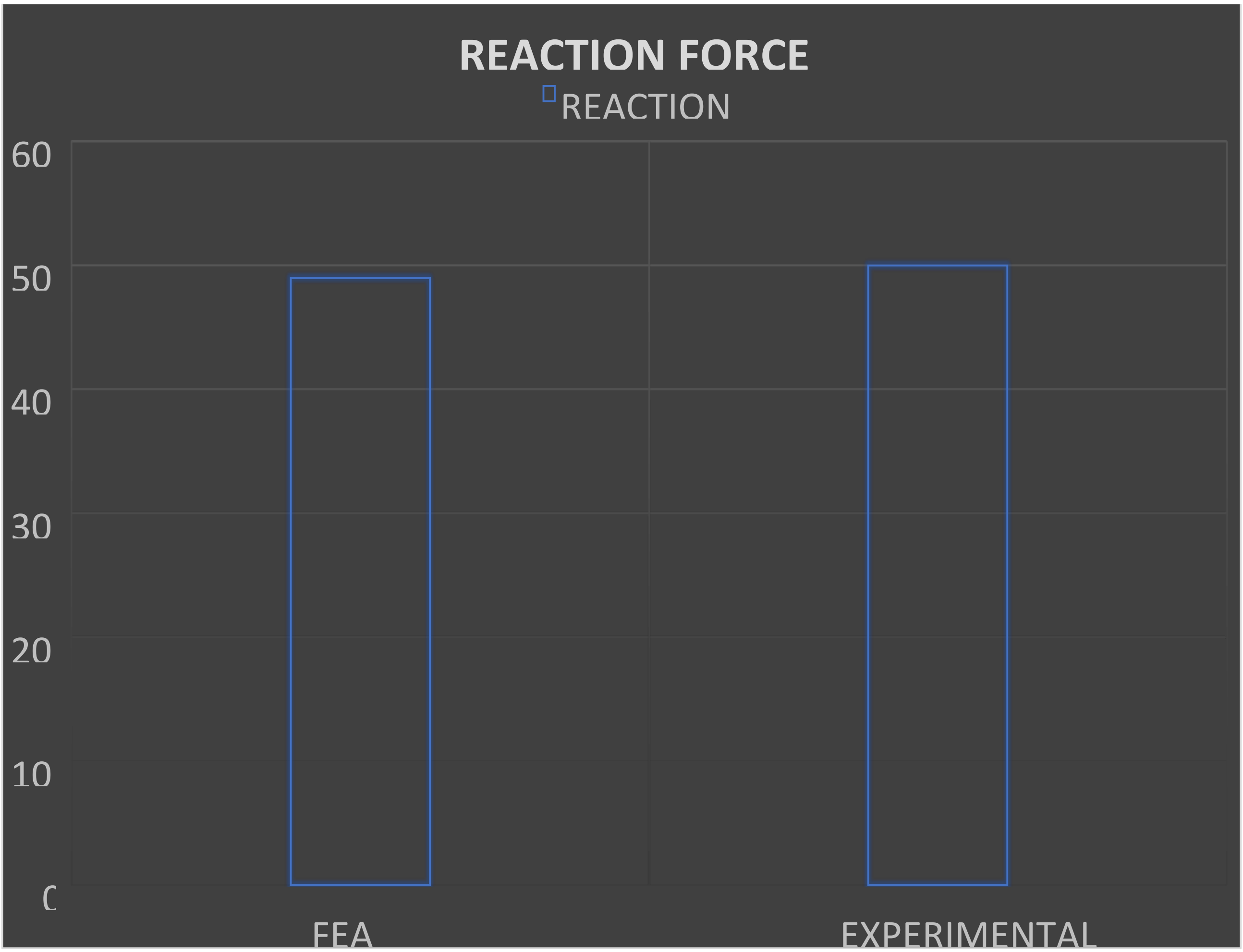


Fig.8.3 Weight optimization comparison

9.REFERANCE

1. Huili Yu, Hui Zhao and Fangyuan Shi “Bending Performance and Reinforcement of Rocker Panel Components with Unidirectional Carbon Fiber Composite”.
2. Kumar Hemant, Deb Debasis, Chakravarty D. “Design of crown pillar thickness using finite element method and multivariate regression analysis”.
3. Corin Reutera,, Kim-Henning Sauerlandb, Thomas Tröstera “Experimental and numerical crushing analysis of circular CFRP tubes under axial impact loading”.
4. Oleg A. Staroverov, Elena M. Strungar, Valery E. Wildemann “Evaluation of the survivability of CFRP honeycomb-cored panels in compression after impact tests”.
5. Andrew Miner, Simon Jones “Design, Testing, Analysis, and Material Properties of Carbon Fiber Reinforced Polymers”.
6. Aleksandr Cherniaev, Clifford Butcher, John Montesano “Predicting the axial crush response of CFRP tubes using three damage-based constitutive models”.