International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 09 Issue: 05 | May - 2025 SJIF Rating: 8.586 ISSN: 2582-3930

Shape Optimization of Heavy Vehicle Chassis

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ABSTRACT

A chassis consists of an internal vehicle frame that supports a manmade object in its construction and use. An example of a chassis is the under part of a vehicle, consisting of the frame (on which the body is mounted). If the running gear such as wheels and transmission, and sometimes even the driver's seat, are included, then the assembly is described as a rolling chassis.

The main objective of this project was to modify the design to decrease the deformation of chassis. The design was done using CATIA, which is advanced modelling software.

In these two types of analysis were done one is model analysis and the other is deformation and stress analysis. In the model analysis deformations were finding with natural frequencies and in static analysis deformations were to find the shape deformation and high stress of component was finding before and after modification of chassis

CHAPTER 1
INTRODUCTION

1.1 INTRODUCTION

A chassis consists of an internal vehicle frame that supports a man-made object in its construction and use. An example of a chassis is the under part of a motor vehicle, consisting of the frame (on which the body is mounted). If the running gear such as wheels and transmission, and sometimes even the driver's seat, are included, then the assembly is described as a rolling chassis.

A vehicle frame, also known as its chassis, is the main supporting structure of a motor vehicle to which all other components are attached, comparable to the skeleton of an organism.

Until the 1930s, virtually every (motor) vehicle had a structural frame, separate from the car's body. This construction design is known as-body-on-frame. Over time, nearly all passenger cars have migrated to Uni-body construction, meaning their chassis and bodywork has been integrated into one another. The last UK mass-produced car with a separate chassis was the Triumph Herald, which was discontinued in 1971. However, nearly all trucks, buses and pickups continue to use a separate frame as their chassis.

1.2 FUNCTIONS

To support the vehicle's mechanical components and body

1. To deal with static and dynamic loads, without undue deflection or distortion. These include:
   - Weight of the body, passengers, and cargo loads.
   - Vertical and torsional twisting transmitted by going over uneven surfaces.
   - Transverse lateral forces caused by road conditions, side wind, and steering the vehicle.
   - Torque from the engine and transmission.
   - Longitudinal tensile forces from starting and acceleration, as well as compression from braking.
   - Sudden impacts from collisions.

1.3 TYPES OF CHASSIS

The word "chassis" is commonly used in the automobile industry. A chassis is the foundation or framework of a vehicle that supports and connects all the essential components, including the engine, transmission system, braking system, suspension, steering, cooling system, and wheels.

There are two types of chassis:

1.3.1 CONVENTIONAL CHASSIS

In this type of chassis, the body is made as a separate unit and then joined with ladder frame. It supports all the systems in a vehicle such as the Engine, Transmission system, Steering system, Suspension system.

Fig: 1. 1 Conventional chassis

Advantages

Durability and Strength

- Excellent for heavy loads and rough terrain.
- Ideal for off-road and utility vehicles.

Easier Repairs

- Frame and body can be repaired or replaced separately.
- Damaged body panels don't necessarily affect the structural integrity.

Better for Towing and Hauling

- Strong frame structure supports greater towing capacity.

Flexibility in Design

- Easier to build multiple body styles on a common frame.
- Suitable for custom modifications (e.g., ambulances, trucks, RVs).

Crash Safety (in some cases)

- Heavy frame can absorb impact in certain types of crashes.

Disadvantages

- The body tends to vibrate easily and the overall vehicle handling and refinement is lower.

Heavier Weight

- The separate frame adds extra weight, reducing fuel efficiency.

Reduced Handling and Stability

- Higher centre of gravity and more flex in the frame lead to less precise handling, especially on smooth roads.

Lower Fuel Efficiency

- Due to added weight and less aerodynamic design

1.3.2 NON-CONVENTIONAL CHASSIS

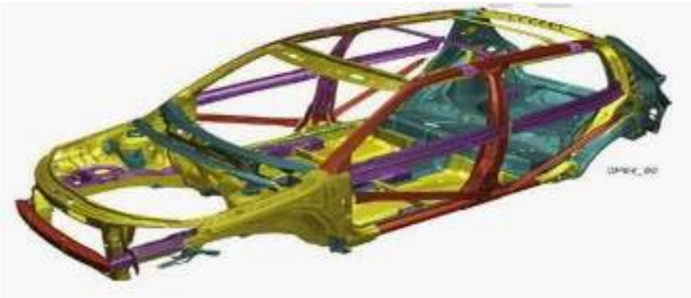


Fig: 1. 2 non-conventional chassis

In this type of chassis, the ladder frame is absent and the body itself acts as the frame. It supports all the systems in a vehicle such as the Engine, Transmission system, Steering system, Suspension system.

Advantages

1. **Lightweight**
   - No separate heavy frame, which improves fuel efficiency and performance.
2. **Better Fuel Economy**
   - Due to lower weight and better aerodynamics.
3. **Improved Handling and Stability**
   - More rigid structure results in better control, cornering, and overall ride quality.
4. **More Interior Space**
   - No bulky frame allows for more cabin and boot space.
5. **Safer in Some Crashes**
   - Crumple zones and rigid passenger cells enhance crash protection.
6. **Lower Manufacturing Cost for Mass Production**
   - Easier to produce in high volumes using modern stamping techniques

Disadvantages

1. **Less Suitable for Heavy Loads:** Not ideal for towing or off-road conditions.
2. **Difficult to Repair:** Structural damage usually means replacing large body sections or the entire shell.

3. **Lower Durability for Rugged Use:** Less tolerant to twisting and rough terrain compared to body-on-frame vehicles.

1.4 TYPES OF CHASSIS ACCORDING TO BODY

Body of a car decides the space available for passenger and language in the car. There is various type of body used in Indian market, and they are given below.

Hatchback: Hatchbacks are vehicles with a separate engine area, and passenger area (or two boxes), the luggage area is enclosed with the passenger area behind the rear seats. Example: Nano, Indica, Jazz, Punto etc.

Fig: 1. 3 Hatchback

Sedan/Notchback: Sedan are basically vehicles with an engine area, passenger area, and boot area (or three box), all separate. Example: Indigo Manza, Swift Dzire etc.

Fig: 1. 1 Sedan

Estate/Station Wagon: Estates or Station wagons are modified saloon vehicles by combining the boot with passenger area & extending it till the roof. The boot area is significantly larger and does not have third row seating. This makes it convenient to carry big objects. Example: Indigo Marina, Octavia Combi etc.

Fig: 1. 2 Estate/Station Wagon

Multi-Purpose Vehicle (MPV) / Multi Utility Vehicle (MUV): MPV (Multi-Purpose Vehicles) or MUV (Multi Utility Vehicles) can have the engine, passenger area and boot area enclosed together or they can have the engine area separate and the passenger and boot area enclosed. MUV/MPV can also have third row of seating. These vehicles are two-wheel drive. Example: Sumo Grande, Tata Tavera, Tata Innova etc.

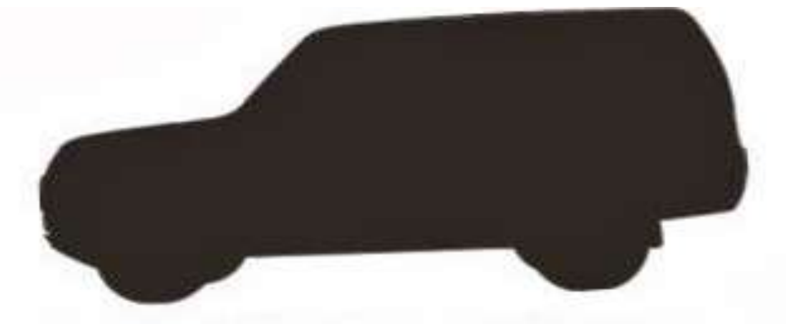


Fig 1. 3 multi-purpose vehicle

Sport Utility Vehicle (SUV): These vehicles have large tires, higher seating, and higher ground clearance. The engine area is separate and the passenger and boot area are enclosed together. These vehicles are either equipped with 4 wheels drive or has as an option of 4-wheel drive. Example: Safari, Scorpio, Gypsy, Fortuner.

Fig 1. 4 Sport Utility Vehicle

Pick-Up Truck: These vehicles have large tires, higher seating capacity, and higher ground clearance. The engine area is separate and the passenger compartment available in single or double cab configurations. Also, luggage loading bay is available behind the passenger compartment. These vehicles are either equipped with 4 wheels drive or has as an option of four-wheel drive. Example: Xenon, Scorpio Getaway etc.

Fig 1. 5 Pick-Up Truck

Van: The engine is placed below the passenger area. Vans can also have Third row of seating. They are also taller and generally more spacious. Example: Winger, Ace Magic, Omni etc.

Fig: 1. 4 Van

The Transit is known for its strong performance, ample cargo space, and comfortable driving experience, making it a favorite among businesses for deliveries, transport services, and even mobile workshops. Modern versions are equipped with advanced features such as EcoBoost engines, adaptive cruise control, lane-keeping assist, and Ford's SYNC infotainment system. Its combination of practicality, efficiency, and customization options makes the Ford Transit a top choice in the van segment worldwide.

CHAPTER 2

LITERATURE SURVEY

Ahmad et al., [i] Their study employed finite element analysis (FEA) to evaluate stress distribution and optimize the geometry for improved performance. The findings contribute to lightweight design strategies in automotive suspension systems while maintaining structural integrity.

Xiaocui Wang et al., [ii] Their study utilized computational methods to optimize strength, stiffness, and manufacturability. The findings support the development of lightweight, high-performance seat frames in the automotive industry.

Rajasekaret al., [iii] applied a genetic algorithm to optimize the box-cross section modulus of a heavy vehicle chassis for improved strength and weight efficiency. Their study utilized computational techniques to enhance structural performance while minimizing material usage. The findings contribute to the development of optimized chassis designs with improved load-bearing capacity and durability.

Guo, Zhijun, et al.,[iv] optimized the modal characteristics of an articulated frame using FEA, reducing resonance and improving structural stability. Their study enhanced the frame’s durability and dynamic performance. optimized an ATV steering knuckle using FEA, minimizing weight while maintaining strength. Their findings improved stress distribution, leading to a more durable and efficient design.

Bhusari, Ameya et al .,[v] conducted FEA-based optimization of an ATV steering knuckle to minimize weight while ensuring structural strength. Their study identified critical stress regions and proposed an improved design for enhanced durability. The results contribute to the development of lightweight yet robust steering components for off-road vehicles.

Randive, Sagar, et al., [vi] optimized the weight of a truck chassis using FEA while ensuring structural strength and durability. Their study reduced material usage without compromising load-bearing capacity. The findings support the development of lightweight and efficient truck chassis design.

Ece Yenilmez et al., [vii] optimized the topology of an anti-roll bar for a heavy commercial truck to improve vehicle dynamics and durability. Their study reduced weight while maintaining structural performance. The findings enhance suspension stability and efficiency in heavy-duty vehicles. performance was assessed under simulated road loads. Results demonstrate that simulation and optimization techniques effectively meet automotive industry requirements for strength, durability, and weight reduction.

Shrivastava et al.,[viii] reviewed various truck chassis design approaches, focusing on structural improvements and weight optimization. Their study analyzed material selection and advanced design techniques for enhanced durability. The findings provide insights into modern chassis development for improved performance and efficiency.

Mantovani et al.,[ix] optimized a steering column mounting bracket using additive manufacturing and topology optimization, considering overhang constraints. Their study reduced weight while ensuring manufacturability and structural integrity. The findings support the development of lightweight, efficient automotive components for advanced manufacturing techniques.

Mohajer, et al., [x] optimized the dynamic response of road vehicles using a computational multibody system approach to enhance ride quality. Their study balanced suspension parameters to improve both comfort and stability. The findings contribute to the advancement of vehicle ride dynamics through multi-objective optimization techniques.

CHAPTER 3
INTRODUCTION TO CATIAV5R20

3.1 INTRODUCTION

CATIA is the leading solution for product success. It addresses all manufacturing organizations. CATIA can be applied to a wide variety of industries- from aerospace- automotive- and industrial machinery- to electronics- shipbuilding- plant design- and consumer goods. Today- CATIA is used to design anything from an airplane to jewelry and clothing. With the power and functional range to address the complete product development process- CATIA supports product engineering- from initial specification to product-in-service in a fully-integrated manner. It facilitates reuse of product design knowledge and shortens development cycles- helping enterprises to accelerate their response to market needs.

CATIAV5R20 is an interactive Computer- Aided Design and Computer Aided Manufacturing system. The CAD functions automate the normal engineering- design and drafting capabilities found in today's manufacturing companies. The CAM functions provide NC programming for modern machine tools using the CatiaV5 R16 design model to describe the finished part. CATIAV5R20 functions are divided into "applications" of common capabilities. These applications are supported by a prerequisite application called "CATIAV5R20 gateway".

CATIAV5R20 is fully three dimensional- double precision system that allows to accurately describing almost any geometric shape. By combining these shapes- one can design- analyze- and create drawings of products.

3.2 METHODOLOGY:

Step 1: Designing of Model with Assembly

Step 2 Converting of Product File to Step File for Analysis

Step 3 Importing STP File to Ansys

Step 4 Analysis of Modal

Step 5 Process of Analysis of Modal

Step 6 Analysis of Static Structural

Step 7 Process of Analysis of Static Structural

Step 8 Results

Step 9 Graphs

3.2.1 BASIC PROCEDURE FOR CREATING A 3-D MODEL IN CATIAV5R20:

Creation of a 3-D model in CatiaV5R20 can be performed using three workbenches i.e.- sketcher- modeling and assembly.

3.2.2 SKETCHER:

Sketcher is used two-dimensional representations of profiles associated within the part. We can create a rough outline of curves- and then specify conditions called constraints to define the shapes more precisely and capture our design intent. Each curve is referred to as a sketch object.

3.2.3 CREATING A NEW SKETCH:

A new sketch- chose Start→Mechanical Design→Sketcher then select the reference plane or sketch plane in which the sketch is to be created.

3.2.4 SKETCH PLANE

The sketch plane is the plane that the sketch is located on. The sketch plane menu has the following options:

Face/Plane: With this option- we can use the attachment face/plane icon to select a planar face or existing datum plane. If we select a datum plane- we can use the reverse direction button to reverse the direction of the normal to the plane.

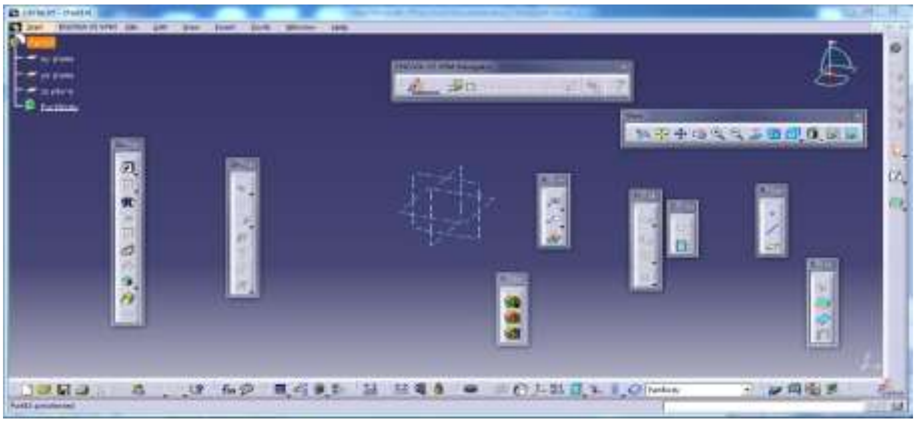


Fig: 3. 1 Creating a new sketch

XC-YC- YC-ZC- and ZC-XC: With these options- we can create a sketch on one of the WCS planes. If we use this method- a datum plane and two datum axes are created as above.

CHAPTER 4

INTRODUCTION TO FEM AND ANSYS

4.1 INTRODUCTION TO FEM

The finite element method represents an extension of the matrix methods for the analysis of framed structures to the analysis of the continuum structures. The basic philosophy of the method is to replace the structure of the continuum having an unlimited or infinite number of unknowns at certain chosen discrete points. The method is extremely powerful as it helps to accurately analyze structures with complex geometrical properties and loading conditions. In the infinite method, a structure or continuum is discretized and idealized by using a mathematical model which is an assembly of subdivisions or discrete elements, known as finite element, are assumed to be interconnected only at the joints called nodes.

The equations, which are obtained using the above conditions, are in the form of force-displacement relationship. Finally, the force-displacement equations are solved to obtain displacements at the nodes, which are the basic unknowns in the finite element method.

The basic idea in the finite element method is to find solution of a complicated problem by replacing it by simpler one. Since a simpler one in finding the solution replaced the actual problem, we will be able to find only an approximate solution rather than exact solution. In finite element method, it will often be possible to improve or refine the approximate solution by sending more computational effort.

This is a numerical solution for obtaining solutions to many of the problems encountered in engineering analysis. In this method, the body or the structure may be divided in to small elements of finite dimensions called finite elements. The original body or continuum is then considered as assemblage of these elements connected at a finite number of joints called nodes.

1. Equilibrium or steady state or time independent problem,
2. Eigen value problem,
3. Transient or propagation programs.

4.1.1 GENERAL DESCRIPITON OF FEM

The step-by-step procedure for static problem can be stated as follows:

Step1: discretization of continuum

Step2: selection of proper interpolation model

Step3: derivative of element stiffness matrices and load vectors

Step4: assemblage of element equation to obtain the overall equilibrium equations:

Step5: solution of systems equations to find nodal values of the displacement:

1. Types of elements
2. Number of elements
3. Size of elements
4. Convergence requirements
5. Nodal degrees of freedom

4.1.2 ADVANTAGES OF FEM:

1. Its ability to use various size and shape and to modal a structure of arbitrary geometry.
2. Its ability to accommodate arbitrary boundary conditions, loading, including thermal loading.
3. Its ability to modal composite structures involving different structural components such as stiffening member on a shell and combination of plates, bars and solids, etc.,
4. The finite element structure closely resembles the actual structure instead of being quite different obstruction that is hard to visualize.
5. The fem is proven successfully in representing various types of complicated material properties and material behavior (nonlinear, anisotropic, time dependent or temperature dependent material behavior).
6. It readily accounts for non-homogeneity of the material by assigning different properties to different elements or even it is possible to vary the properties within an element according to a pre-determined polynomial pattern.

4.2 INTRODUCTION TO ANSYS SOFTWARE

The purpose of a finite element analysis is to model the behavior of a structure under a system of loads. In order to do so, all influencing factors must be considered and determined whether their effects are considerable or negligible on the final result.

The ANSYS program is self-contained general purpose finite element program developed and maintained by analysis systems inc. The program contains many routines, all interrelated and all for main purpose of achieving a solution to an engineering problem by finite element method.

ANSYS provides a complete solution to design problems. It consists of powerful design capabilities like full parametric solid modeling, design optimization and auto meshing, which gives engineers full control over their analysis.

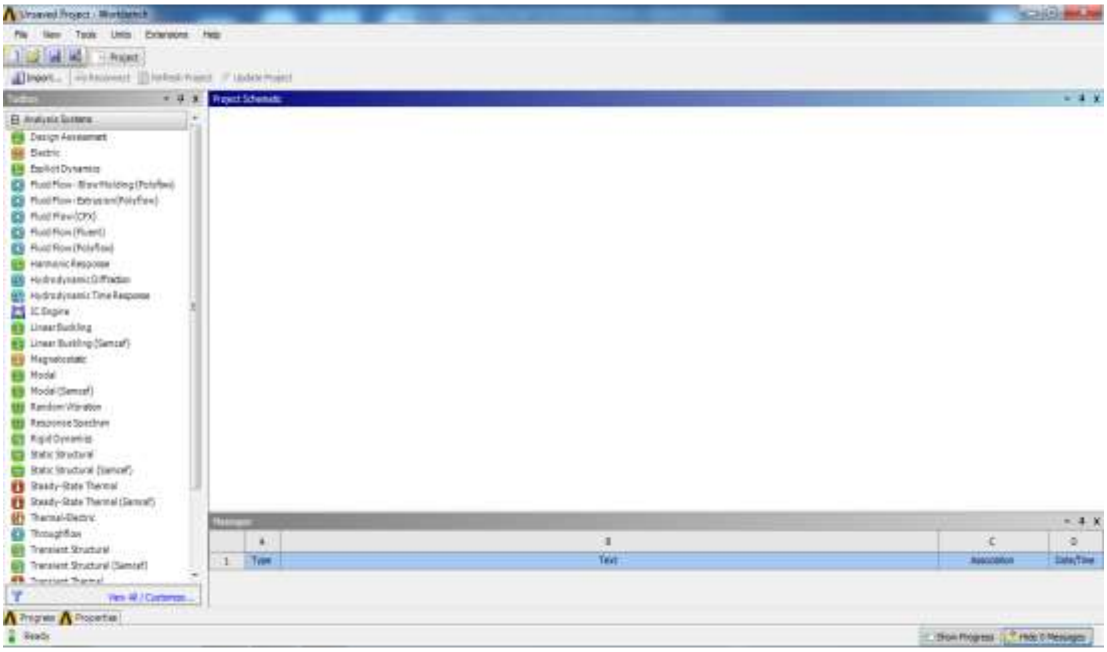


Fig: 4. 1 Introduction to Ansys software

The following are the special features of ANSYS software:

- It includes bilinear elements.
- Heat flow analysis, fluid flow and element flow analysis can be done.
- Graphic package and extensive preprocessing and post processing.

4.2.1 MESHING:

The default meshing controls that the program uses may produce a mesh that is adequate for the model we are analyzing. In this case, we need not specify any meshing controls. However, if we do use meshing controls, we must set them before meshing the solid model.

Meshing controls allow us to establish the element shape, midsize node placement and element size to be used in meshing the solid model, this step is one of the most important of the entire analysis for the decisions we make at this stage in the model development will profoundly affect the accuracy and economy of the analysis.

Smart element sizing (smart sizing) is a meshing feature that creates initial element sizes for free meshing operations. Smart sizing gives the meshing a better chance of creating reasonably shaped elements during automatic mesh generation.

4.1.2 PRE-PROCESSOR

The pre-processor stage in ANSYS package involves the following:

- Specify the title, which is the name of the problem.
- Set the type of the analysis to be used, i.e., structural, thermal, fluid, or electro-magnetic, etc.,
- The model may be created in pre-processor or it can be imported from another cad drafting package through a neutral file format like IGES file.
- Apply mesh – mesh generation is the process of dividing the analysis continuum in to number of discrete parts or finite elements. The finer the mesh, the better the result, but the longer the analysis time. Therefore, the compromise between accuracy and solution speed is usually made.
- Assign the properties – material properties (young’s modulus, Poisson’s ratio, density, and if applicable coefficient of expansion, friction, thermal conductivity, damping effect, specific heat, etc.,) have to be defined.

Solution:

- Apply the loads. Some type of load is actually applied to the analysis model. The loading may be in the form of a point load, pressure or a displacement in a stress analysis, a temperature or heat flux in a thermal analysis and a fluid pressure or velocity in a fluid analysis.
- Applying the boundary conditions. After applying load to the model in order to stop it accelerating infinitely through the computer virtually either at least one boundary condition must be applied

Fem solver can be logically divided in to three main parts, the pre-solver, the mathematical-engine and the post-solver. The pre-solver reads the model created by the pre-processor and formulates the mathematical representation of the model and calls the mathematical-engine, which calculates the results.

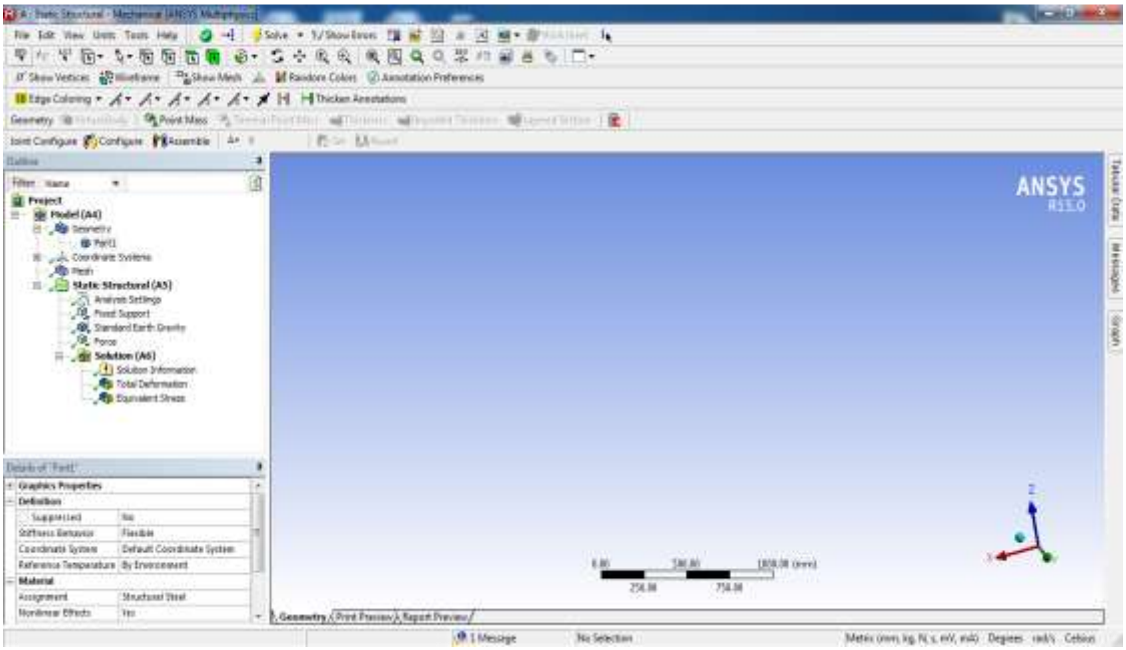


Fig: 4. 2 Preprocessor

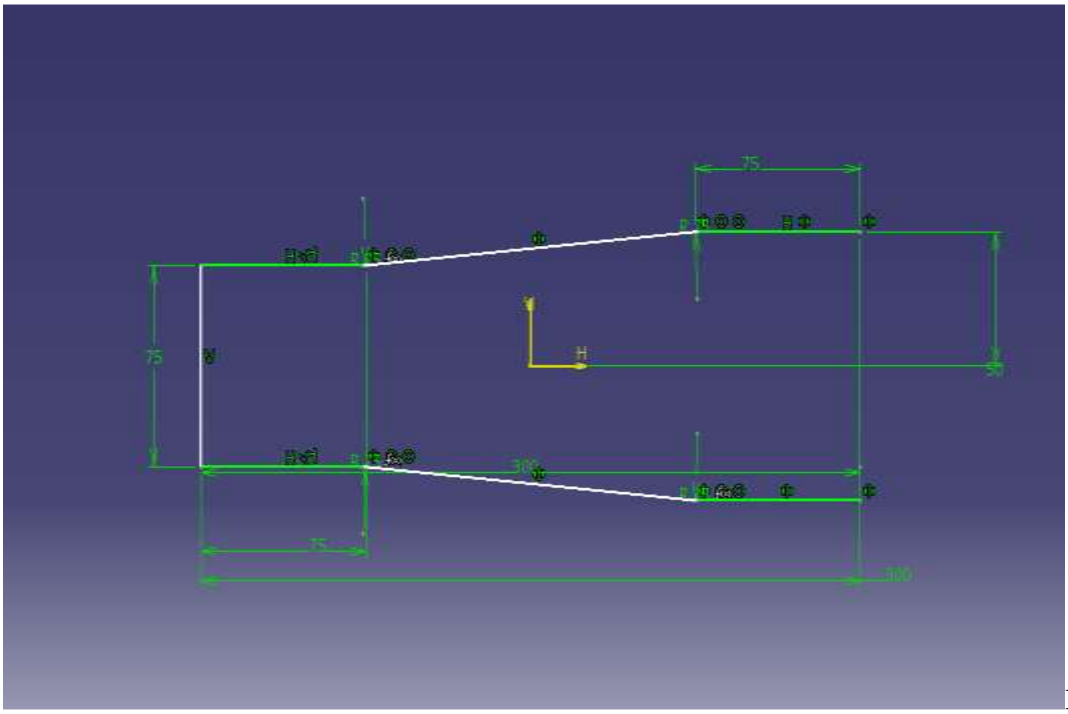
The result returned to the solver and the post-solver is used to calculate the strains, stresses, etc., for each within the component or continuum.

CHAPTER 5
MODELING OF CHASSIS

Launch CATIA, go to Start >

Creating a profile

After completing the analysis in ANSYS Workbench, exit the Workbench and switch to the Wireframe and Surface Design workbench. Open the Sketcher tool and select the YZ plane to enter the Sketcher module. From the Profile section, create the desired sketch as shown in Figure 5.2.



Mechanical Design, and select the Wireframe and Surface Design module. Then, open the Sketcher tool and choose the XY plane to enter the Sketcher environment. Using the Profile tool, draw the sketch as shown in Figure 5.1

Fig: 5. 1 Creating a profile

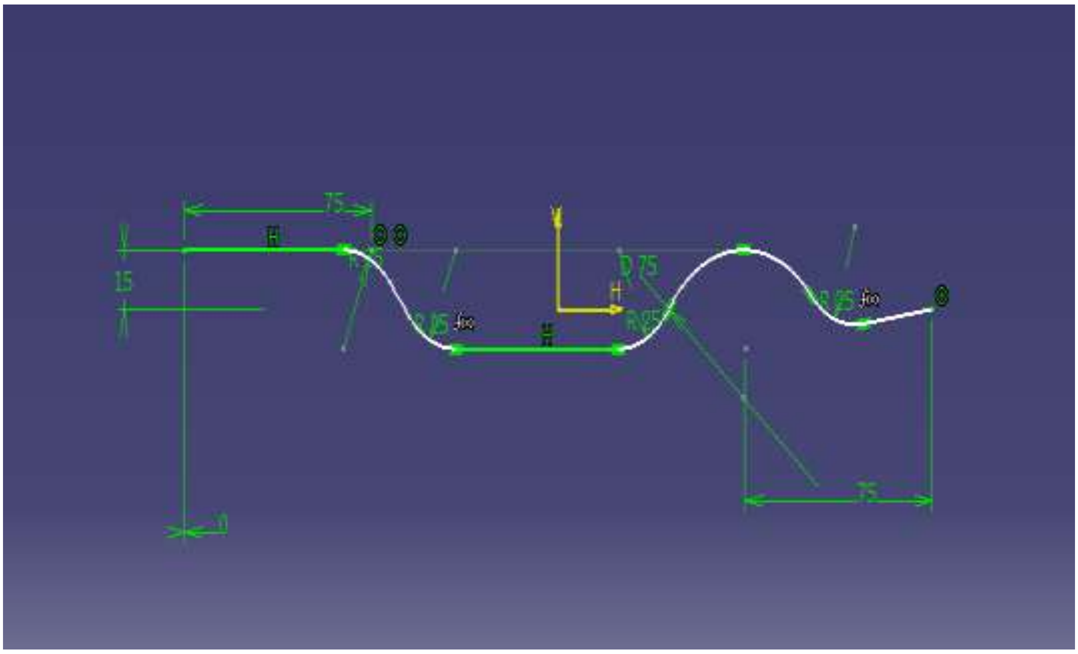


Fig: 5. 2 Creating a base of a chassis

Creating a base of a chassis

After exiting the Sketcher Workbench, switch to the Wireframe and Surface Design module. Go to the Surface-Based Features toolbar and select the Extrude tool.

In the Extrude Surface Definition window, select the second sketch as the profile and enter the dimensions as shown in Figure 5.3

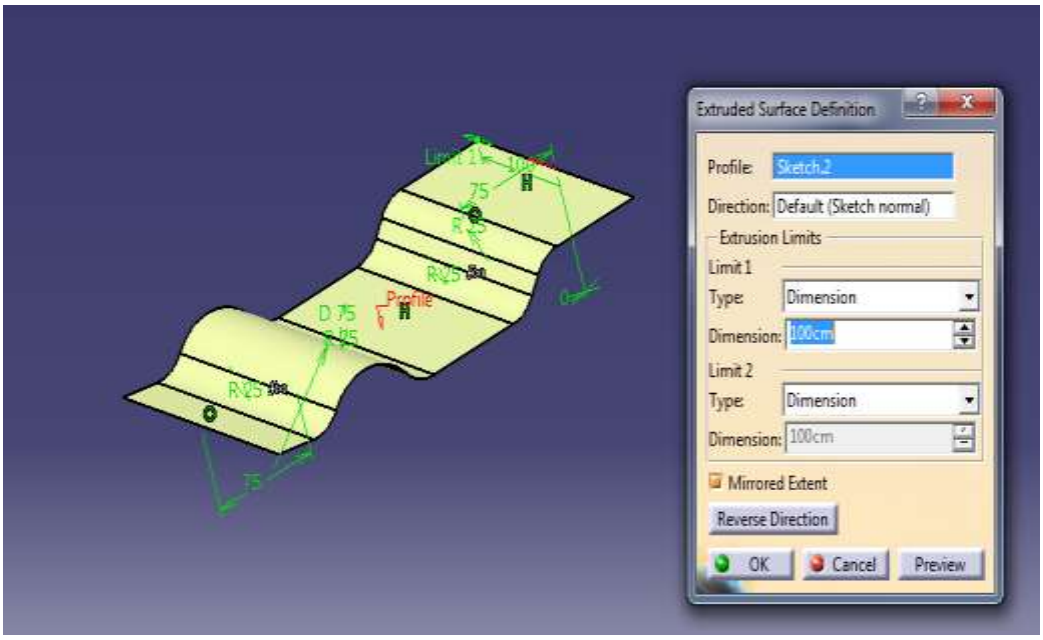


Fig: 5. 3 Extrude the surface

Next, exit the Workbench and enter the Wireframe and Surface Design module. Go to the Surface-Based Tools and select the Extrude feature. In the Extrude Surface Definition window, select the second sketch as the profile and set the dimensions as shown in Figure 5.4.

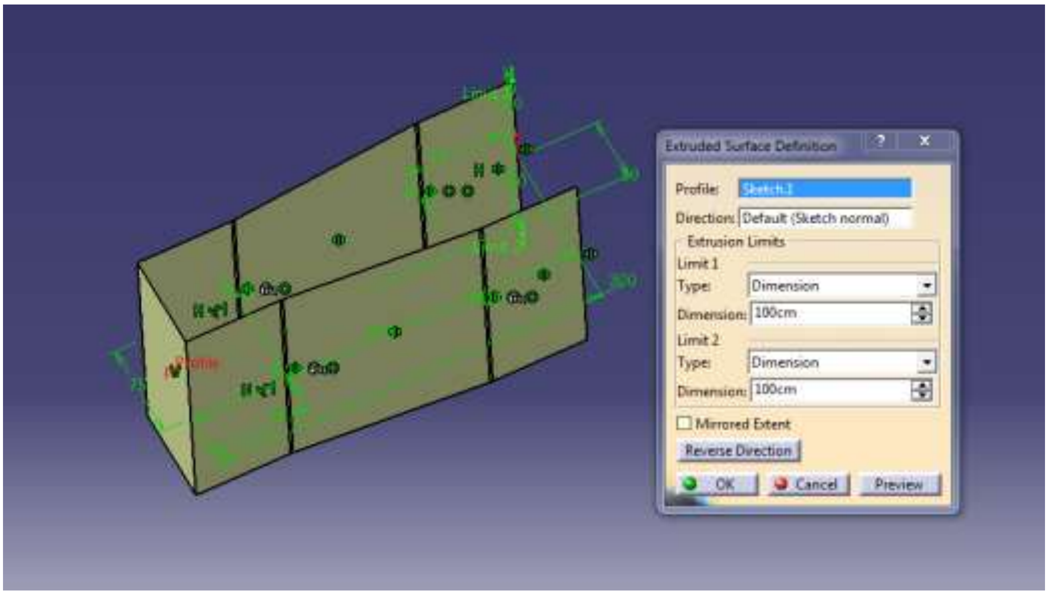


Fig: 5. 4 Specify the dimension

After that now go to wireframe-based tools and select intersection. On intersection definition select second and first extrude as shown in Figure 5.5.

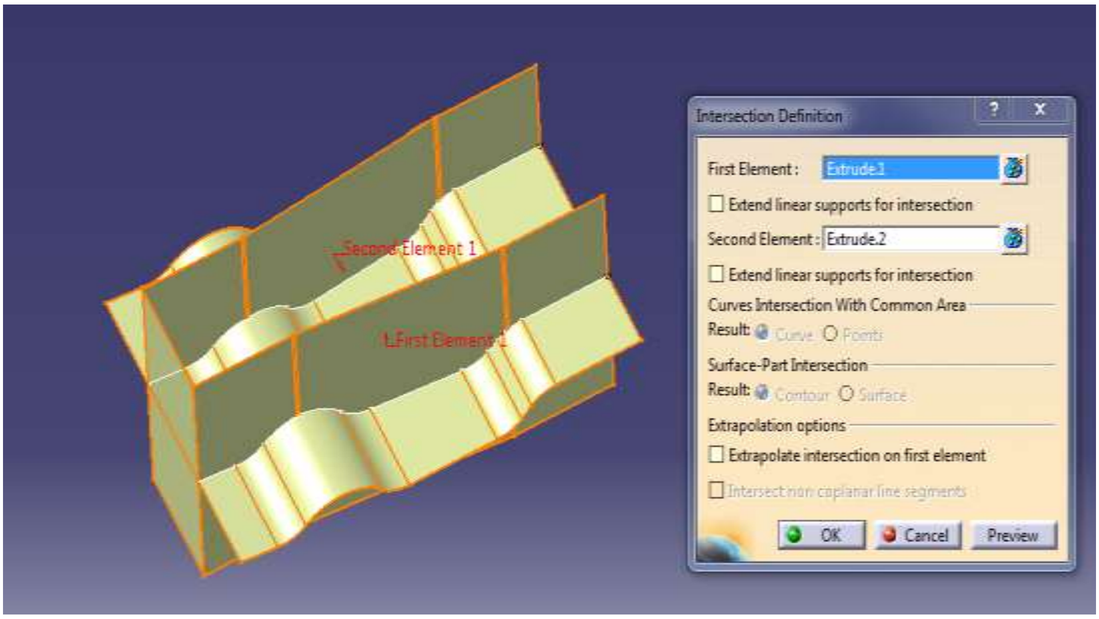


Fig: 5. 5 Extrude the second element

Next, select the extruded surface, right-click, and choose Hide/Show. Open CATIA, go to Start > Mechanical Design, and select the Part Design module.

In the Part Design environment, navigate to Reference Elements and select Plane.

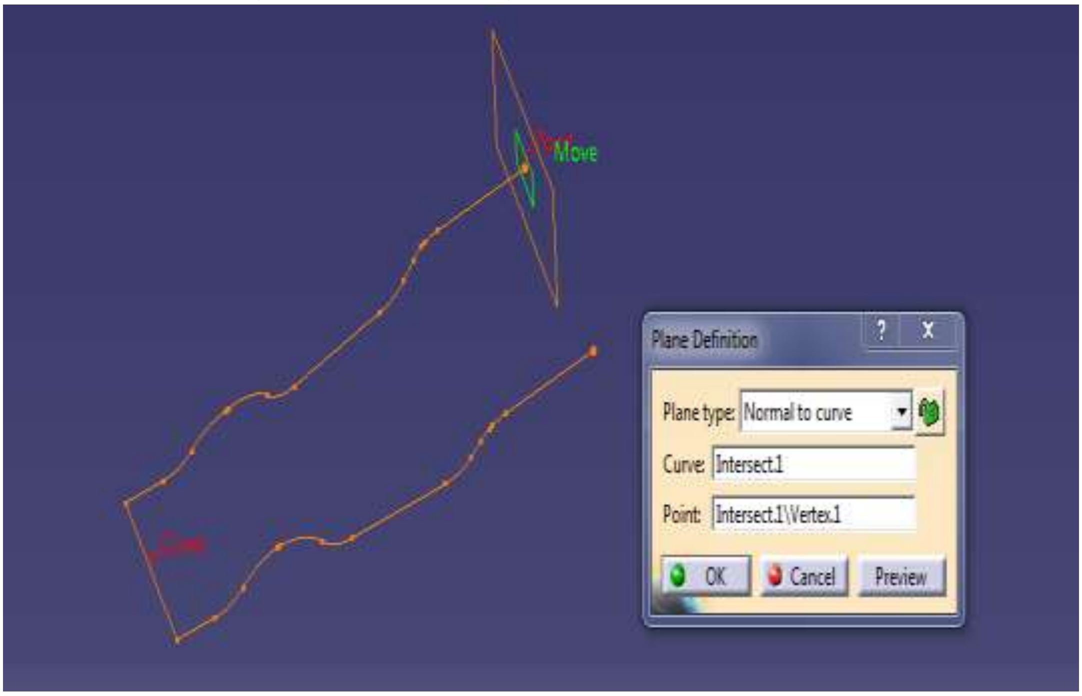


Fig: 5. 6 Select plane definition

"Now, enter the Part Design module, then select the Sketcher tool and choose the desired plane above. Once in the Sketcher module, create the profile structure as shown in Figure 5.7."

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 09 Issue: 05 | May - 2025 SJIF Rating: 8.586 ISSN: 2582-3930

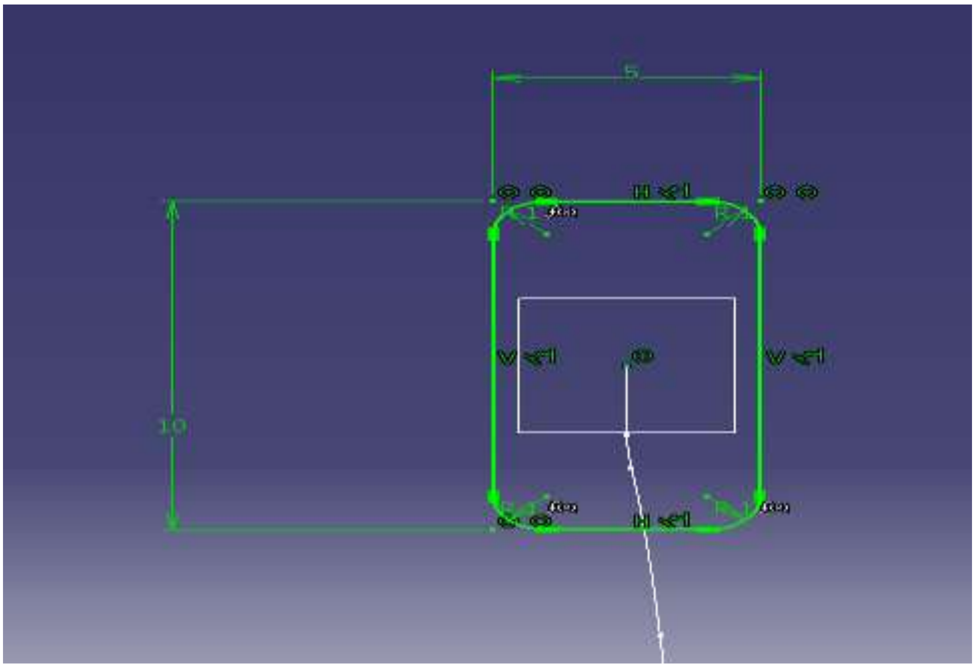


Fig: 5. 7 Create a profile structure

Once the sketch is complete, go to the workbench and select 'Exit Workbench' to return to the Part module. In the Part module, navigate to the Sketcher-based features and choose the 'Rib' tool. In the Rib Definition, select the sketch you created earlier as the profile and choose the intersection above as the curve path, as shown in Figure 5.8.

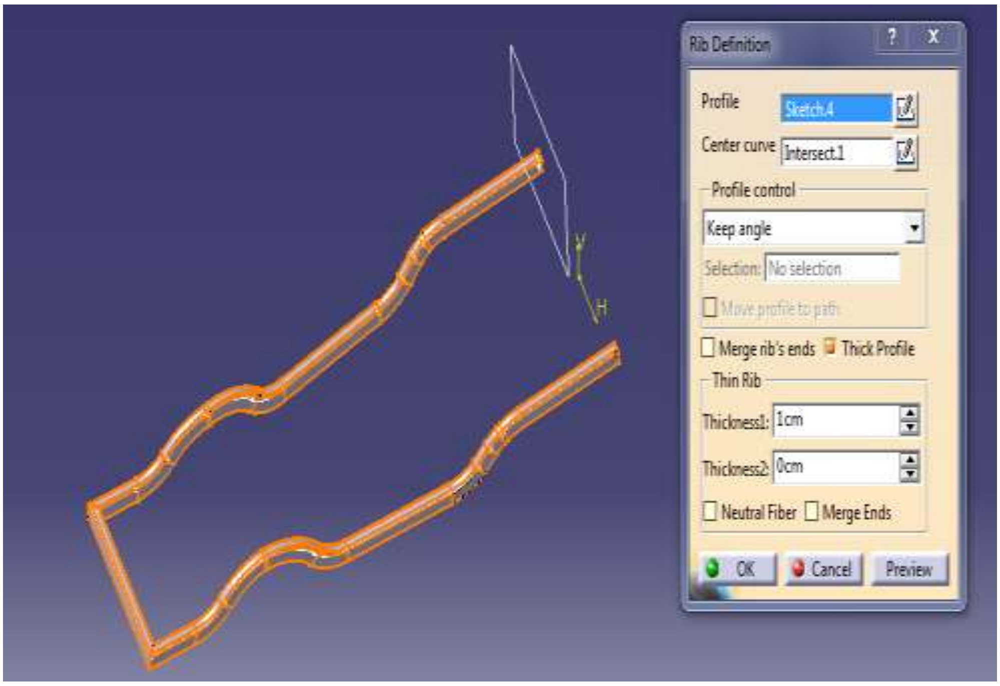


Fig: 5. 8 Profile Structure

Next, go to the Part Design module, select the Sketcher tool, and choose the desired plane. Once inside the Sketcher module, create the profile structure as shown in Figure 5.9.

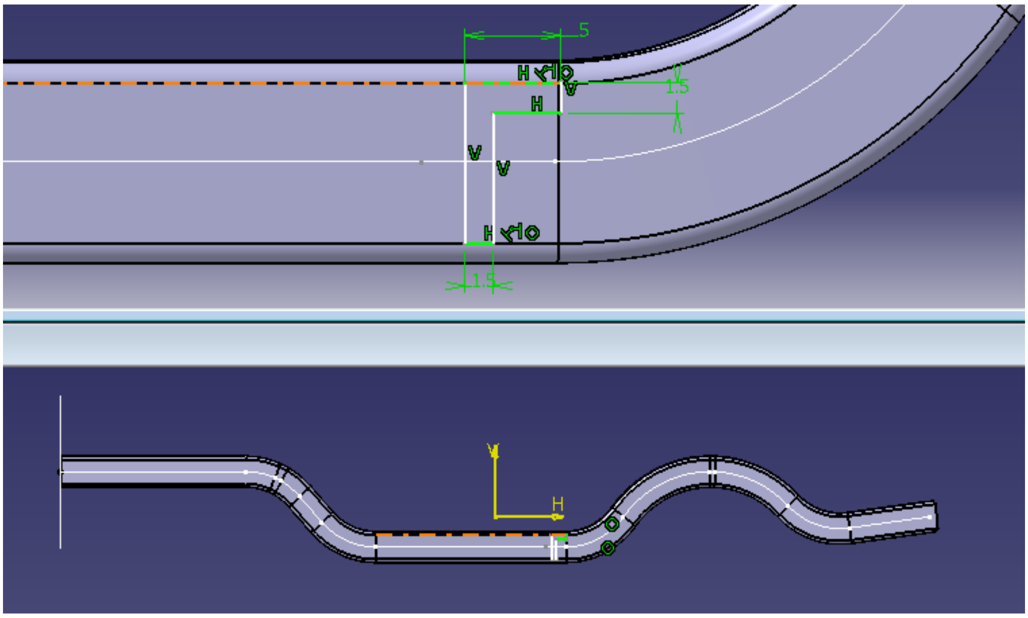


Fig: 5. 9 Part design

Once the sketch is complete, exit the Sketcher module and return to the Part module. In the Part module, go to the Sketcher-based features and select the 'Pad' tool. In the Pad Definition, select the sketch as the profile and set both the first and second limits to 'Up to Next,' as shown in Figure 5.10.

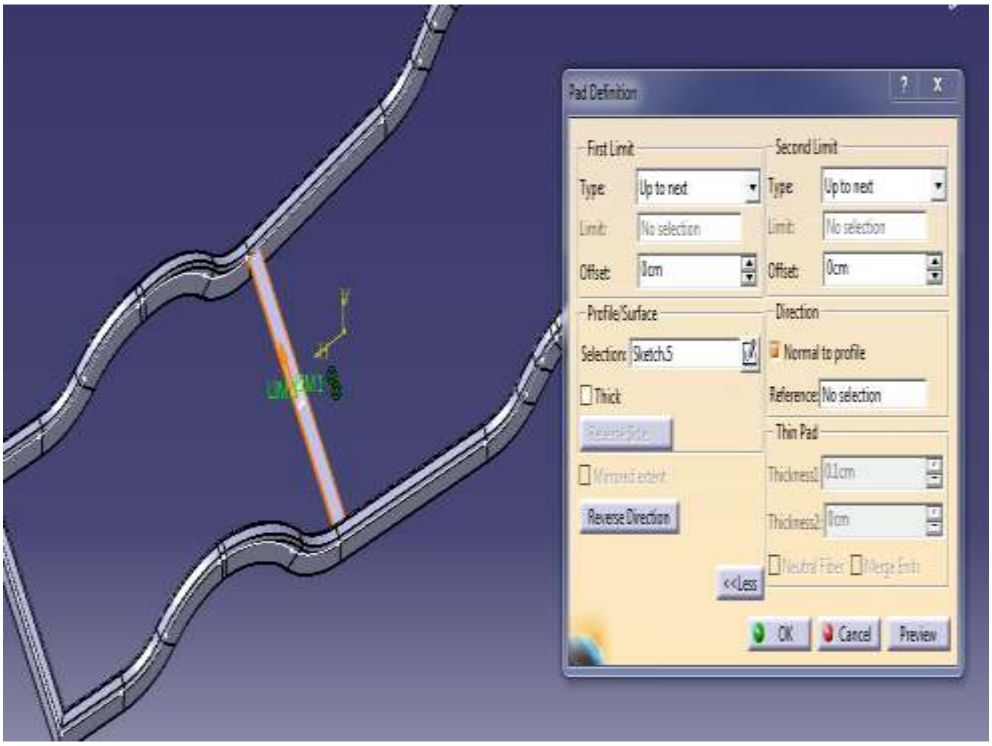


Fig: 5. 10 Selection profile

Next, enter the Part Design module, select the Sketcher tool, and choose the appropriate plane. Then, in the Sketcher module, create the profile structure as shown in Figure 5.11.

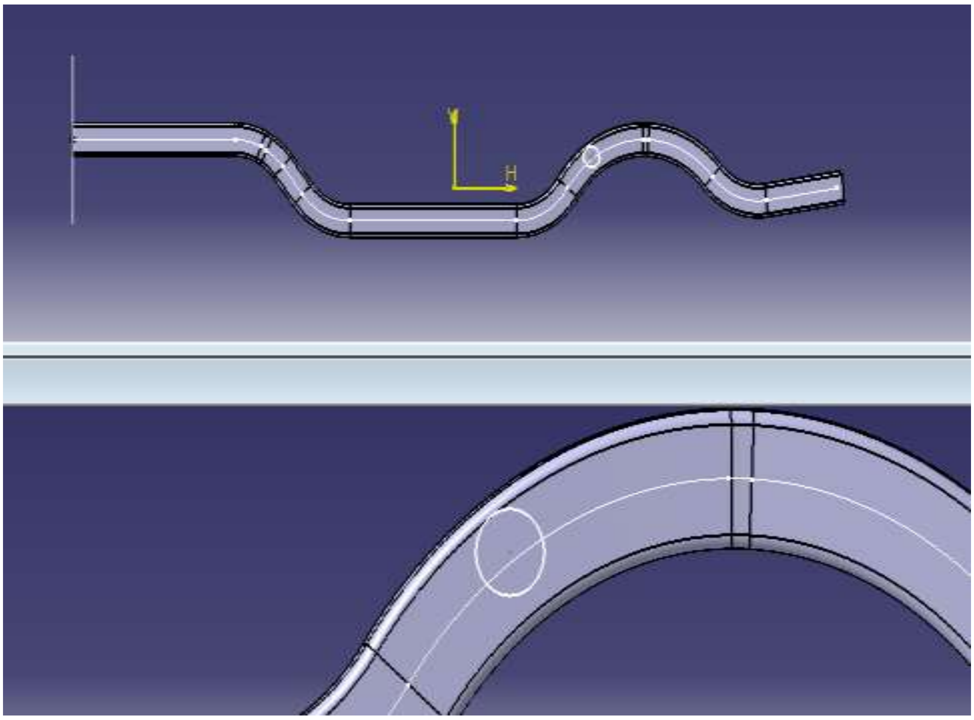


Fig: 5. 11 Select sketcher tool

Once the sketch is complete, exit the Sketcher workbench to return to the Part module. In the Part module, navigate to the Sketcher-based features and select the 'Pad' tool. In the Pad Definition, choose the sketch as the profile and set both the first and second limits to 'Up to Next,' as shown in Figure 5.12.

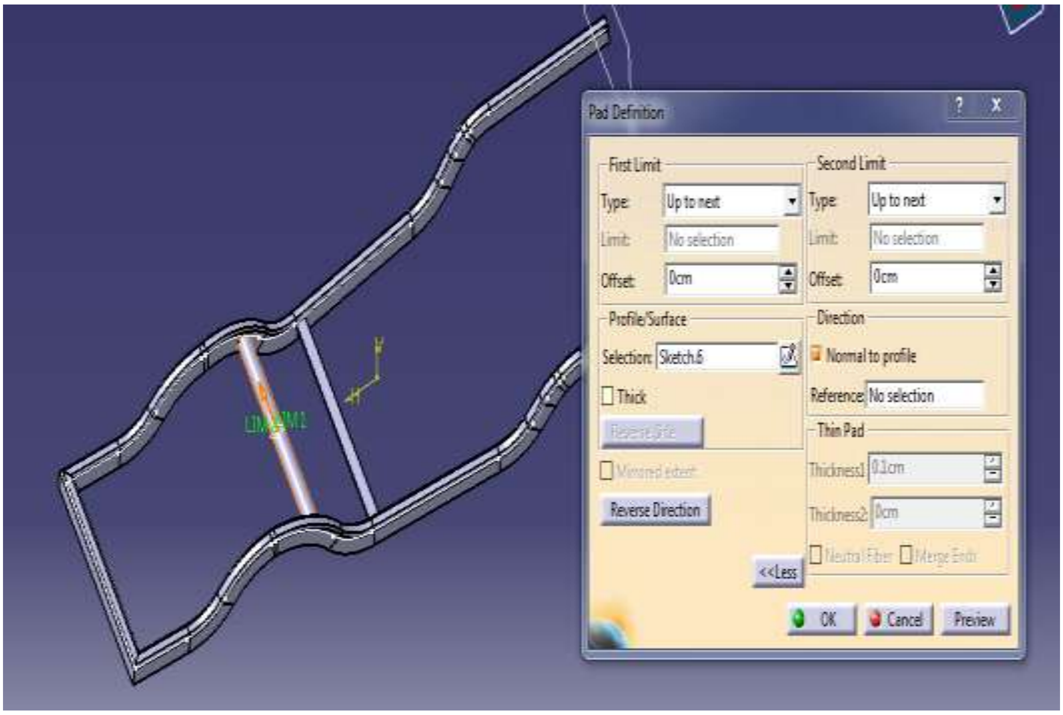


Fig: 5. 12 Select pad tool

Now, enter the Part Design module, select the Sketcher tool, and choose the appropriate plane. Once in the Sketcher module, create the profile structure as shown in Figure 5.13

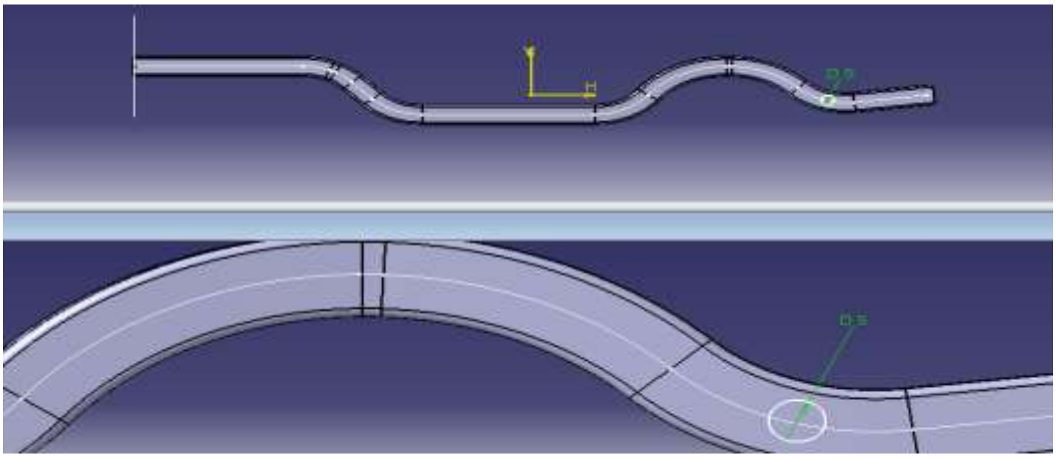


Fig: 5. 13 Select plane

Once the sketch is complete, exit the Sketcher workbench and return to the Part module. In the Part module, navigate to the Sketcher-based features and select the 'Pad' tool. In the Pad Definition, select the sketch as the profile and set both the first and second limits to 'Up to Next,' as shown in Figure 5.14.

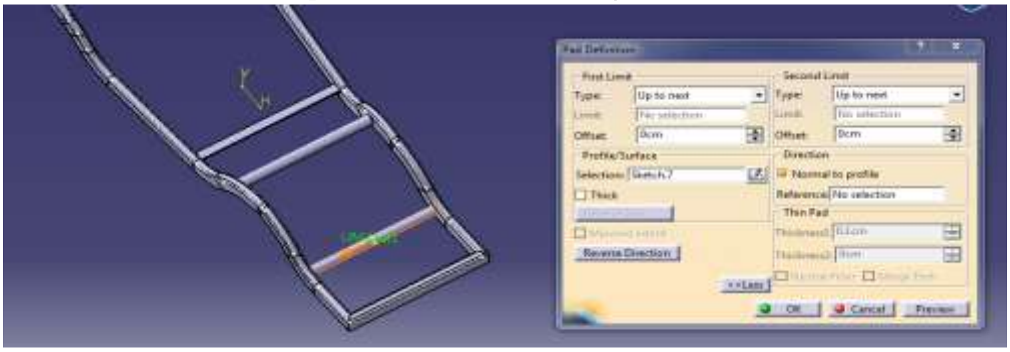


Fig: 5. 14 Specify limit

Next, enter the Part Design module, select the Sketcher tool, and choose the appropriate plane. Then, in the Sketcher module, create the profile structure as shown in Figure 5.15.

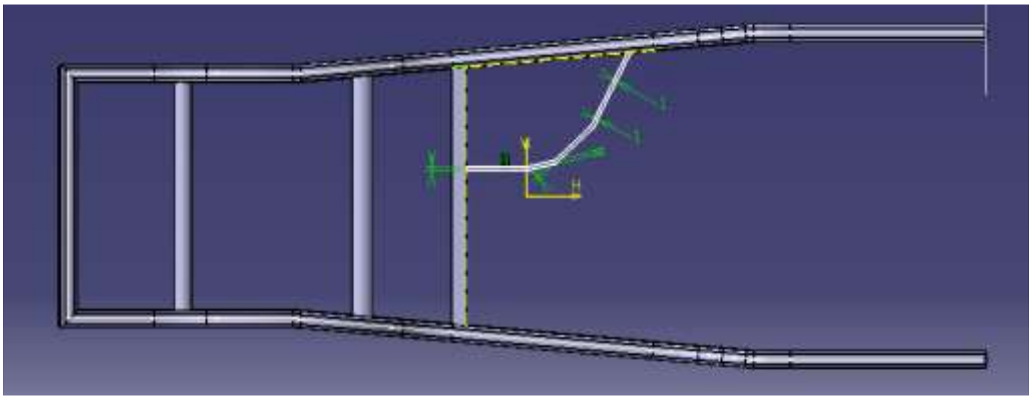


Fig: 5. 15 Create a profile structure

Once the sketch is complete, exit the Sketcher workbench to return to the Part module. In the Part module, go to the Sketcher-based features and select the 'Pad' tool. In the Pad Definition, choose the sketch as the profile, set the limit to

'Up to Plane,' and select the bottom surface, as shown in Figure 5.16.

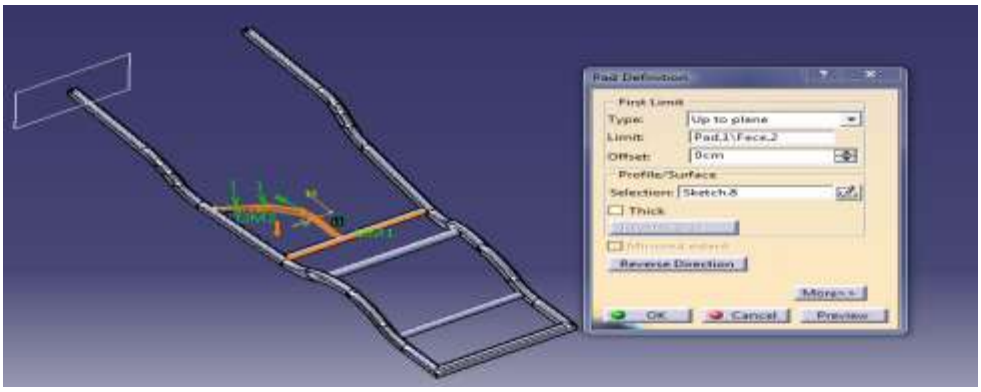


Fig: 5. 16 Select bottom surface

In part module, go to sketcher-based feature and select stiffener tool. In stiffener definition, create a line between L shapes then specify thickness as shown in Figure 5.17.

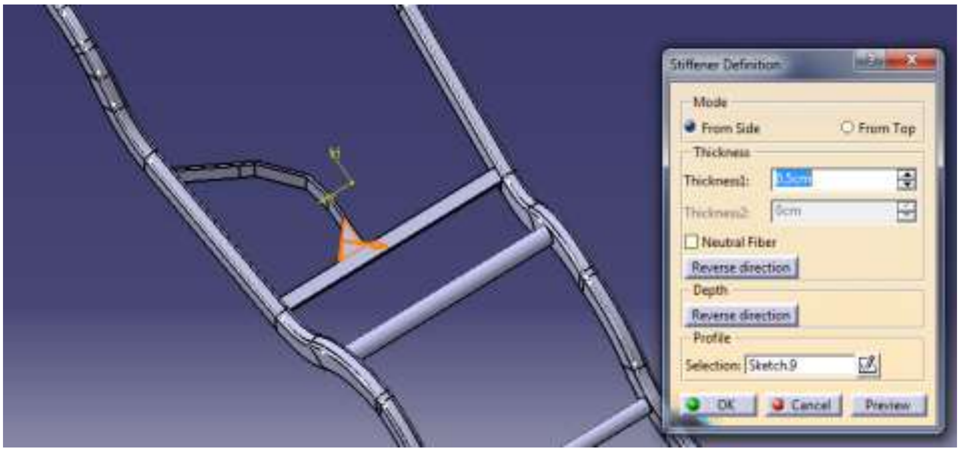


Fig: 5. 17 Specify thickness

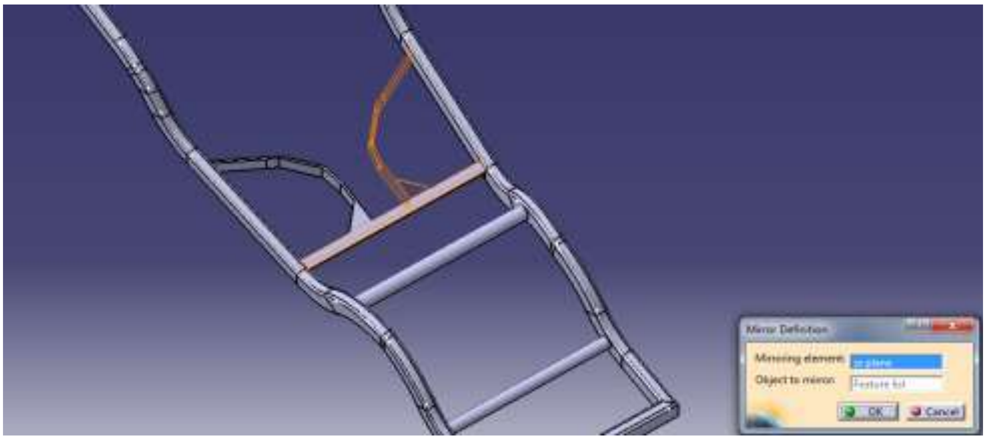


Fig: 5. 18 Mirror the object

In the Part module, navigate to the Dress-up-based features and select the 'Edge Fillet' tool. In the Edge Fillet Definition, choose the corners where the fillet is required and set the fillet radius to 2 cm, as shown in Figure 5.19

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Fig: 5. 19 Specifies fillet radius

Now, go to the Part Design module, select the Sketcher tool, and choose the required plane. Once in the Sketcher module, go to the profile section and create the desired profile.

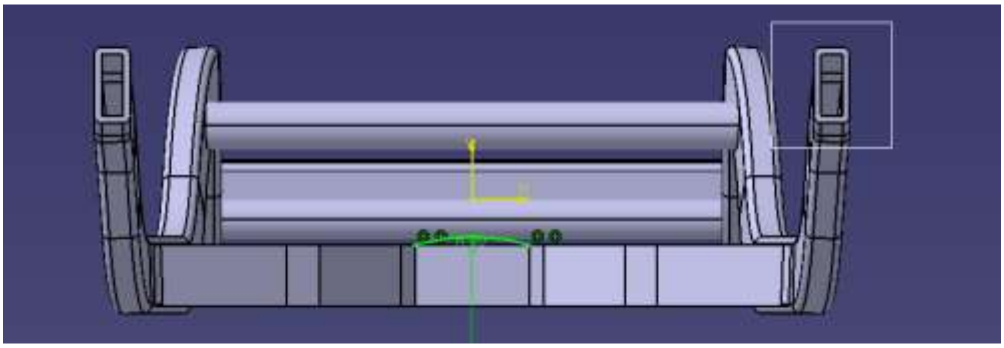

Fig: 5. 20 Creating a profile structure

Next, enter the Part Design module, select the Sketcher tool, and choose the required plane. Then, in the Sketcher module, go to the profile section and create the profile structure as shown in Figure 5.21.

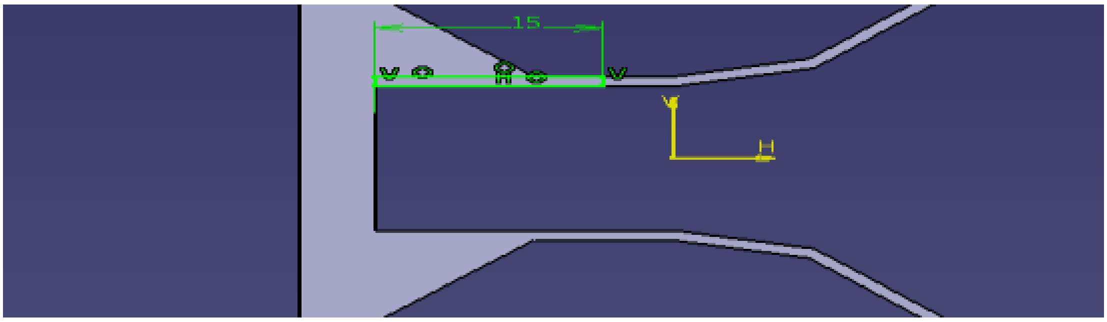

Fig: 5. 21 Profile structure

After completing sketcher go to workbench and select exit work bench. Then we enter into part module again. In part module, go to sketcher-based feature and select rib tool. In rib definition, select above sketch as selection profile and select above curve as a curve path as shown in Figure 5.22.

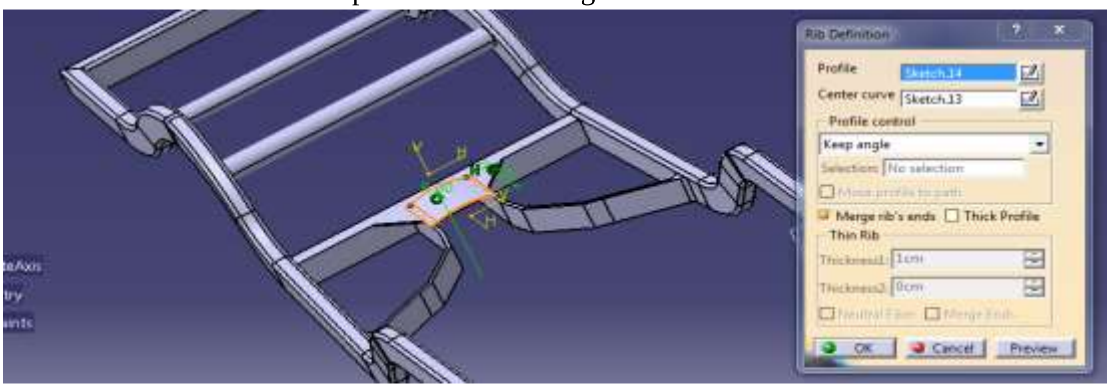

Fig: 5. 22 Selecting above curve as a curve path

Now again, in part design module then go to sketcher tool and select sketcher tool and select required plane. Then we enter into sketcher module and go to profile and create a profile structure as shown in Figure 5.23.

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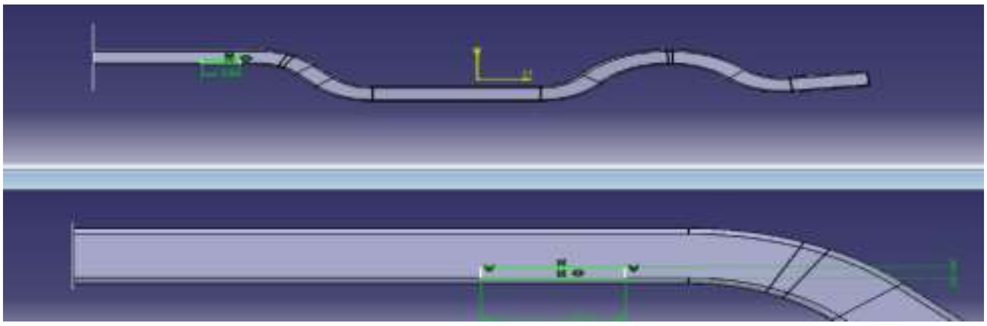

Fig: 5. 23 Design the process

Once the sketch is complete, exit the Sketcher workbench to return to the Part module. In the Part module, go to the Sketcher-based features and select the 'Pad' tool. In the Pad Definition, choose the sketch as the profile and set both the first and second limits to 'Up to Next,' as shown in Figure 5.24.

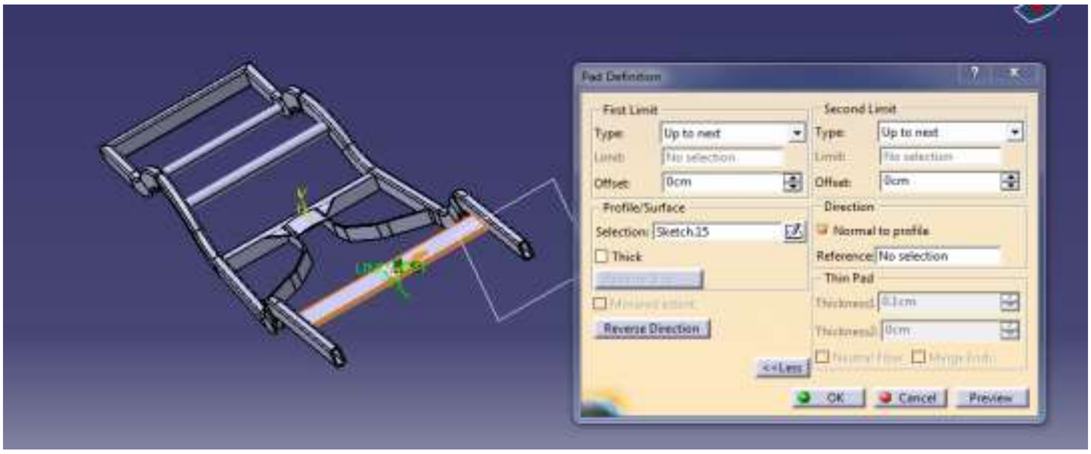

Fig: 5. 24 Specify up to next on first and second limit

In the Part module, go to the Sketcher-based features and select the 'Stiffener' tool. In the Stiffener Definition, create a line between the L-shaped elements and specify the thickness Once the sketch is complete, exit the Sketcher workbench to return to the Part module. In the Part module, navigate to the Sketcher-based features and select the 'Pad' tool. In the Pad Definition, choose the sketch as the profile and set both the first and second limits to 'Up to Next shown in figure 5.25.

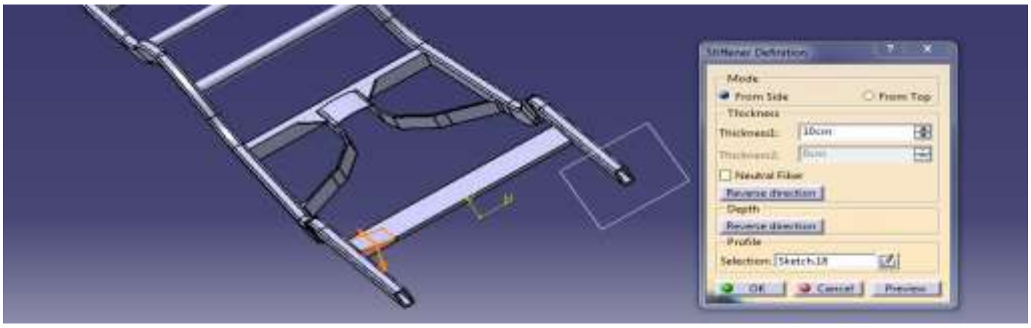

Fig: 5. 25 Specify thickness

In part module, go to transformation-based feature and select mirror tool. In mirror definition, select ZX plane as a mirroring element and select above pad as object to mirrors shown in Figure 5.26.

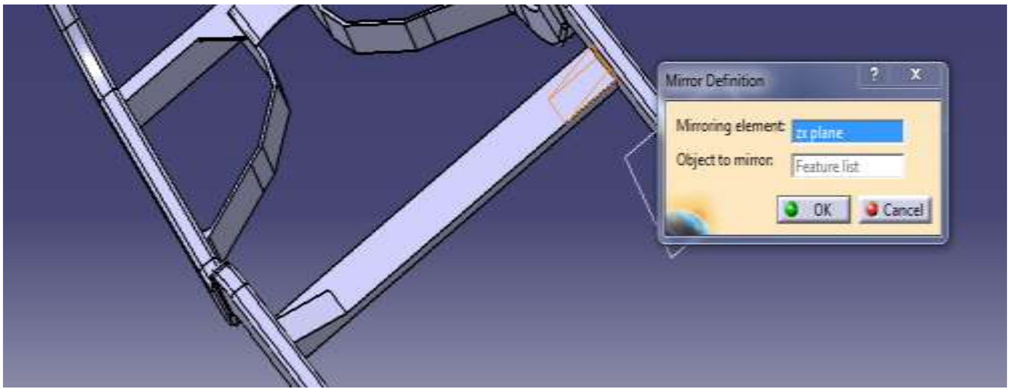

Fig: 5. 26 Mirroring element

Now again, in part design module then go to sketcher tool and select sketcher tool and select required plane. Then we enter into sketcher module and go to profile and create a profile structure as shown in Figure 5.27.

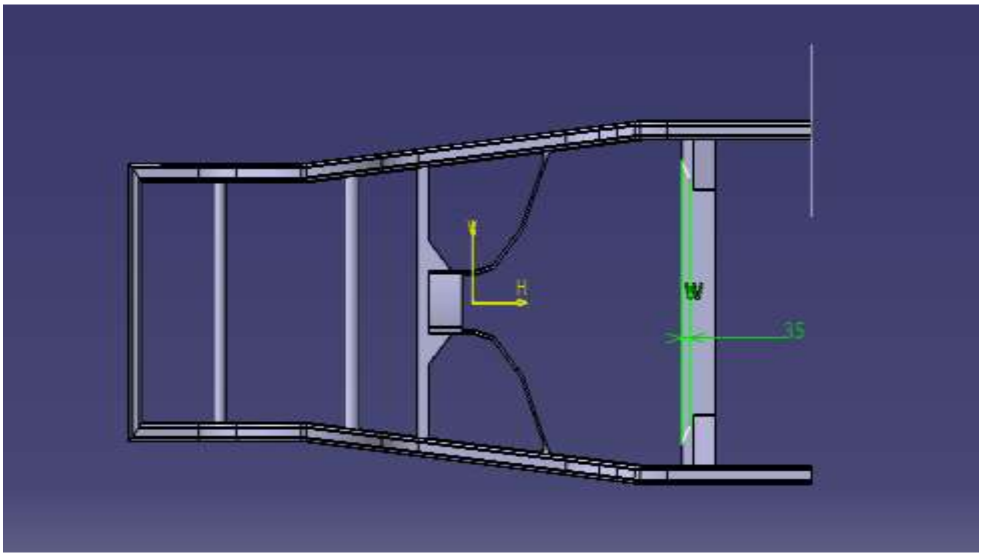

Fig: 5. 27 Sketcher module

After completing the sketch in the Sketcher module, exit the Sketcher workbench to return to the Part module. In the Part module, navigate to the Sketcher-based features and select the 'Pocket' tool. In the Pocket Definition, choose the sketch as the profile and set the first limit to 'Up to Last,' as shown in Figure 5.28

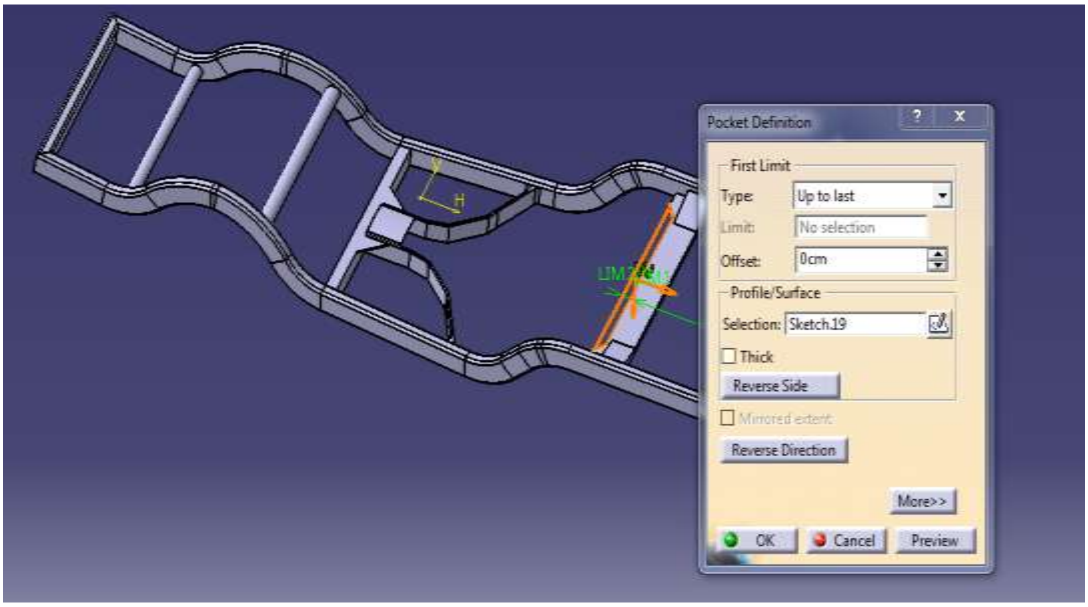

Fig: 5. 28 Select above sketch

Now, go to the Part Design module, select the Sketcher tool, and choose the required plane. Once in the Sketcher module, go to the profile section and create the profile structure as shown in Figure 5.29.

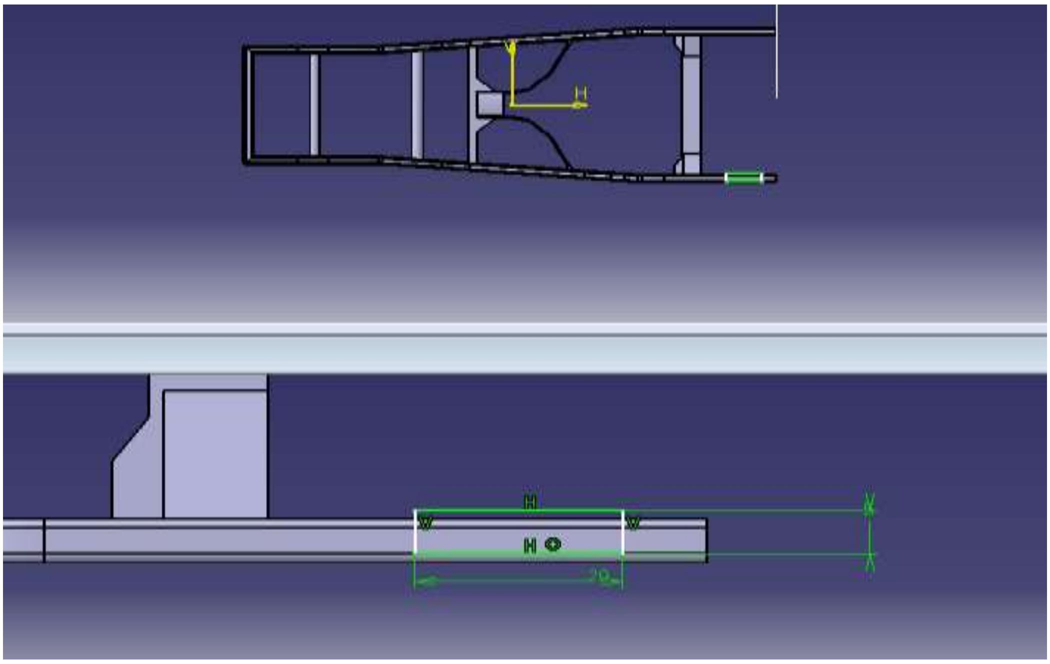

Fig: 5. 29 Go to profile

Once the sketch is complete, exit the Sketcher workbench to return to the Part module. In the Part module, navigate to the Sketcher-based features and select the 'Pad' tool. In the Pad Definition, select the sketch as the profile and set the length to 2 mm, as shown in Figure 5.30.

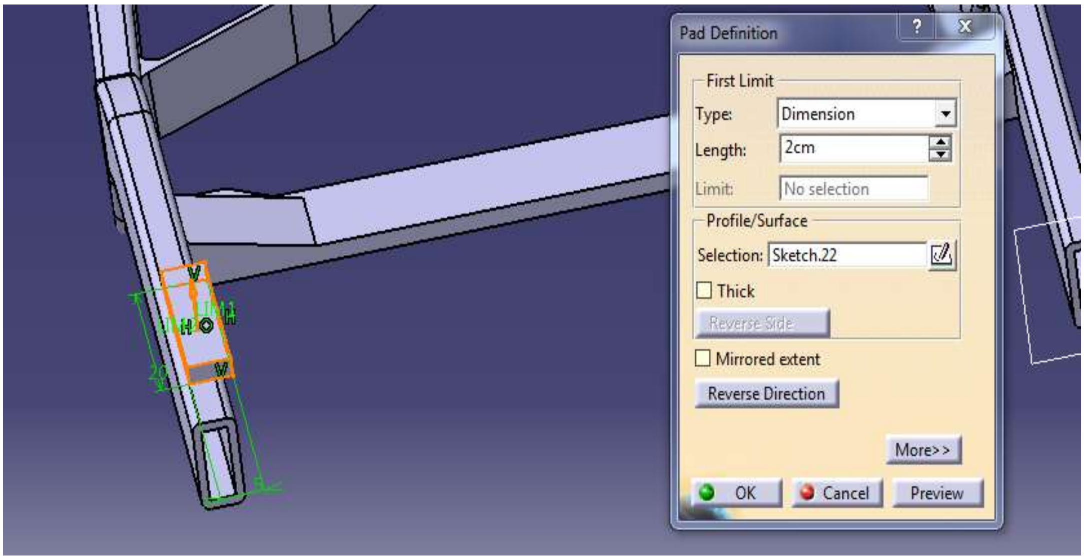

Fig: 5. 30 specify 2 mm in length

Next, enter the Part Design module, select the Sketcher tool, and choose the required plane. Then, in the Sketcher module, go to the profile section and create the profile structure as shown in Figure 5.31.

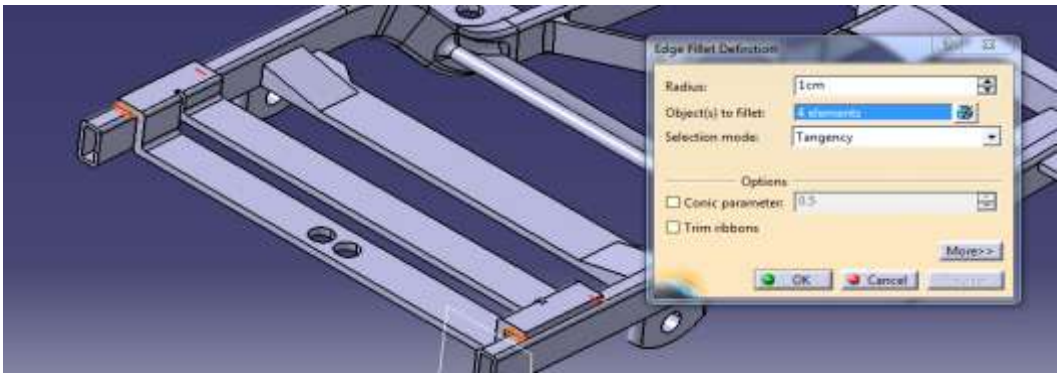

Fig: 5. 31 fillet radius as a 2 cm

In the Part module, navigate to the Dress-up-based features and select the 'Edge Fillet' tool. In the Edge Fillet Definition, choose the corners where the fillet is needed and specify the fillet radius as 2 cm, as shown in Figure 5.32

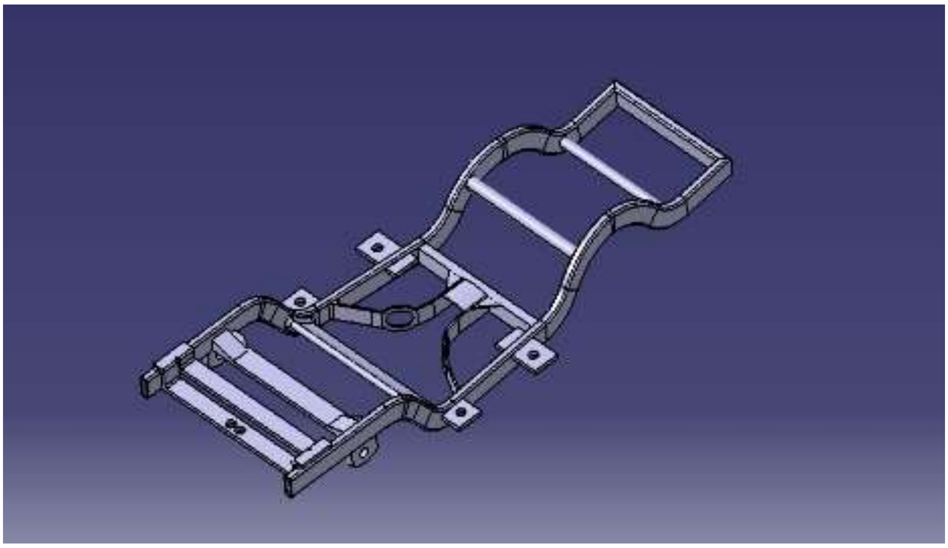

Fig: 5. 32 Final component

Final component is ready to export from the CATIA after the frame includes longitudinal side members with multiple bends and elevations, indicating suspension mounts or accommodation for drive-train components.

CHAPTER 6

PROCESS OF ANALYSIS

6.1 ANALYSIS

Importing of the chassis will be done after opening the workbench. For the supporting purpose of the geometry, the file format of CATIA will be changed to step format. This is to match up the graphical properties of the CATIA V5 to ANSYS workbench.

The full form of the step is standard for the exchange of product model data which itself states that will exchange the graphical properties of models.

The material properties are the important factor which will be considered as the second preference after importing or creating the geometry. The procedure of material application, double click on the engineering data which will appear on the top of the analysis system. The analysis system which we are using in this project is transient thermal analysis. After opening the window of engineering data, the material application will be done by selecting the add symbol in the general materials. These materials are available in thermal materials from engineering data source and select the above-mentioned material, and reset layout from view menu and update project.

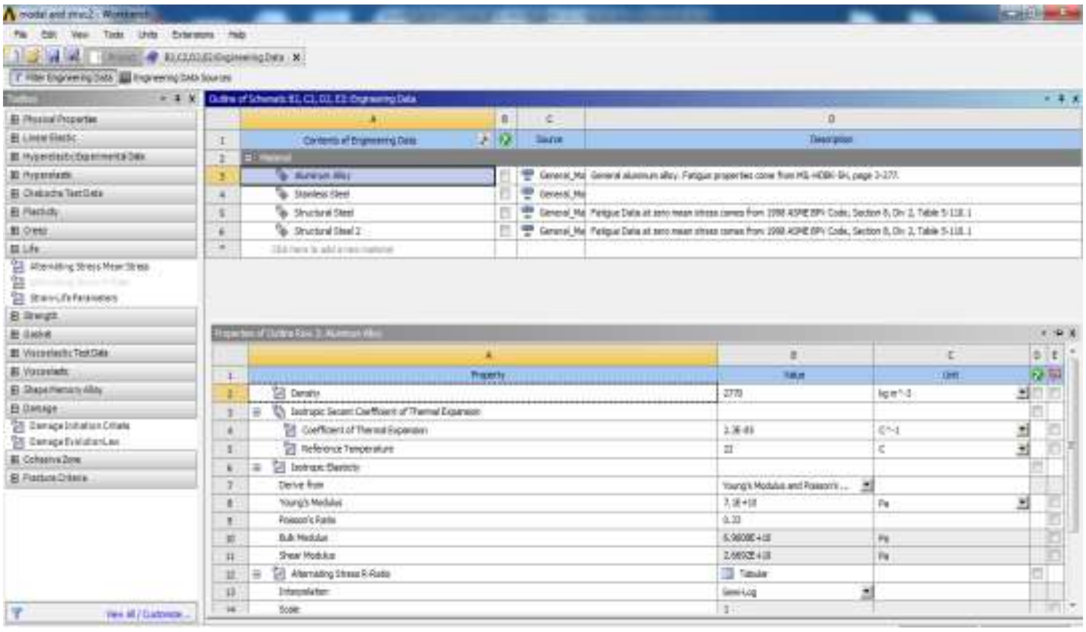

Fig: 6. 1 Material selection

6.1.1 MODAL AND STATIC STRUCTURAL ANALSYS

Work bench: After importing the model into project schematic window drag and drop the static structural tab on to the screen from the toolbox window and link the geometry by right and browser to geometry step or IGES file. Double click on the model it opens the mechanical window with object.

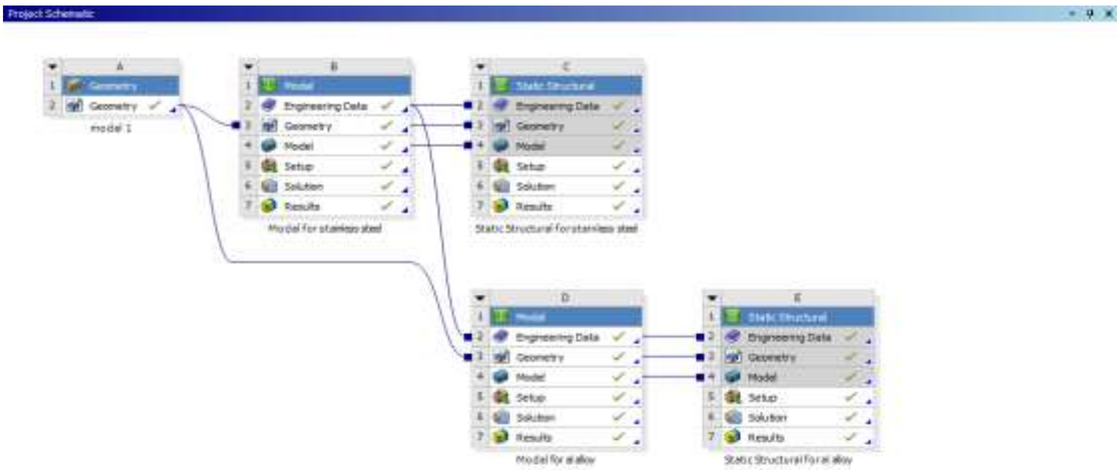

Fig: 6. 2 before modification of chassis

Modal tree

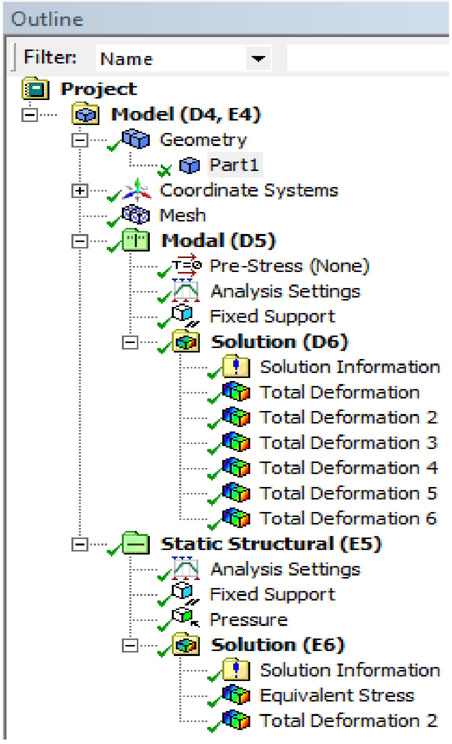

Fig: 6. 3 Modal tree

Geometry and meshing:

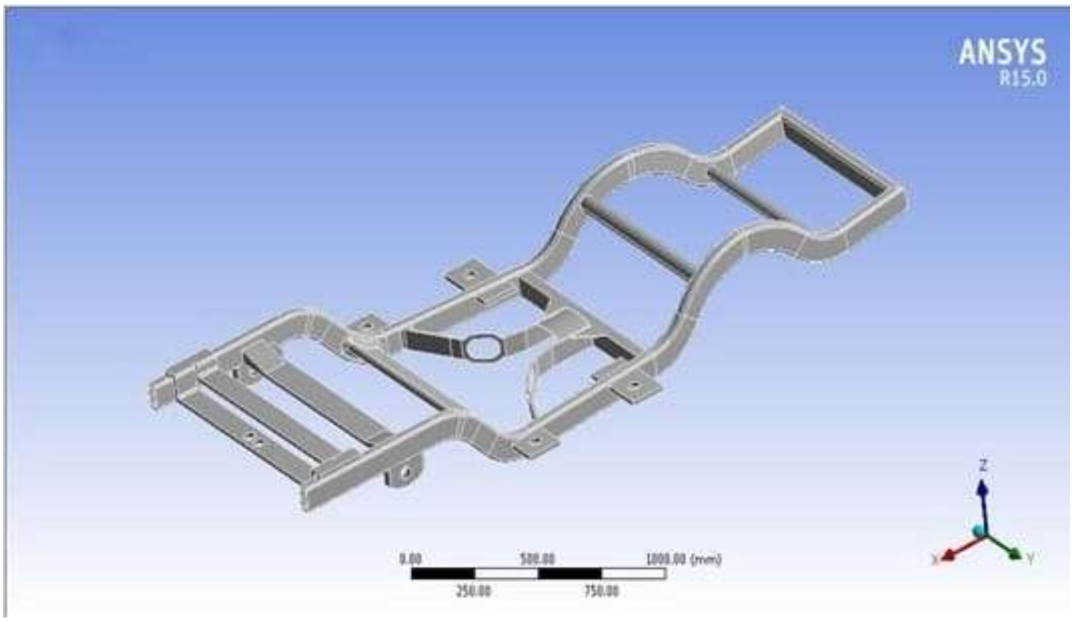

Fig: 6. 4 Geometry

6.1.2 GEOMETRY

From the outline tab – select the geometry – part – from the bottom detailed window- material –assignment – select required material.

Mesh: To generate the meshing, there are two methods one is automatic mesh generation and the other is with required size meshing. In this we used auto meshing with medium meshing.

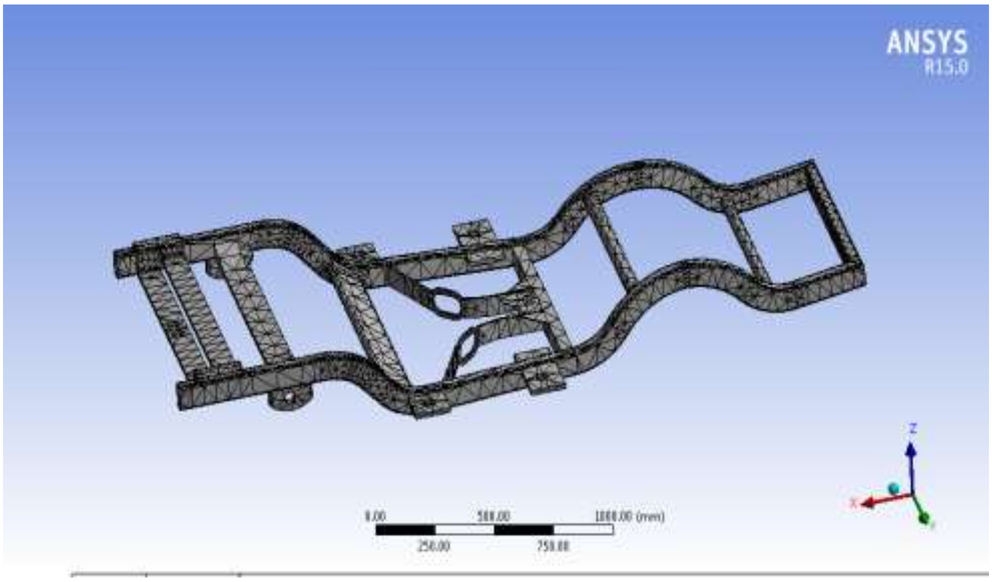

Fig: 6. 5 Meshing

For Material: stainless steel
A: modal analysis
Fixed support:

To fix the component: right click on the analysis settings – insert – fixed support – select all the bushes or bearing's location and apply, as shown above.

Fig: 6. 6 modal analysis

Solution

Right click on solution – insert total deformation

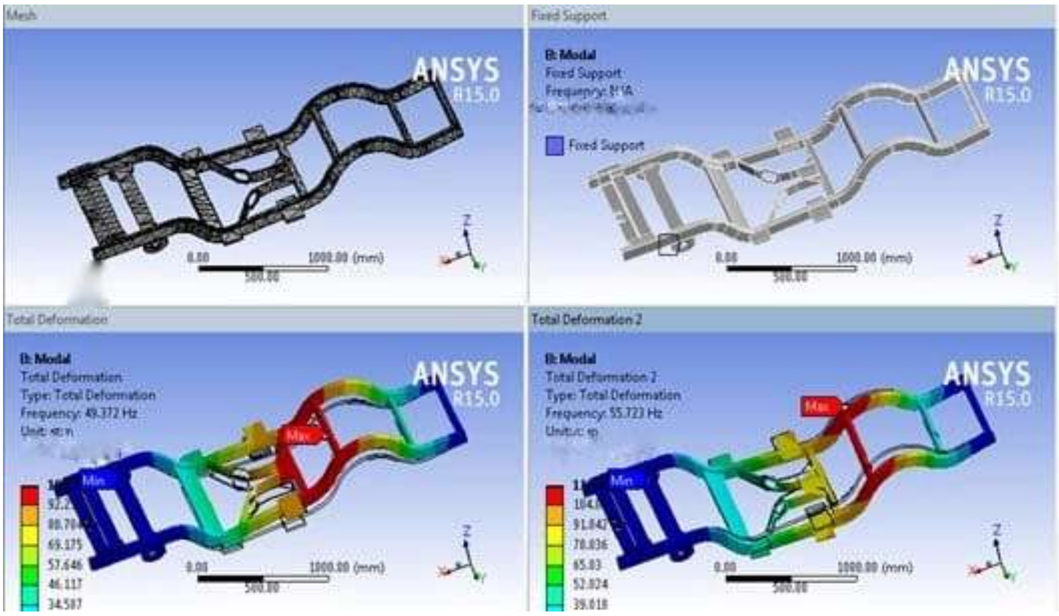

Fig: 6. 7 Total deformation

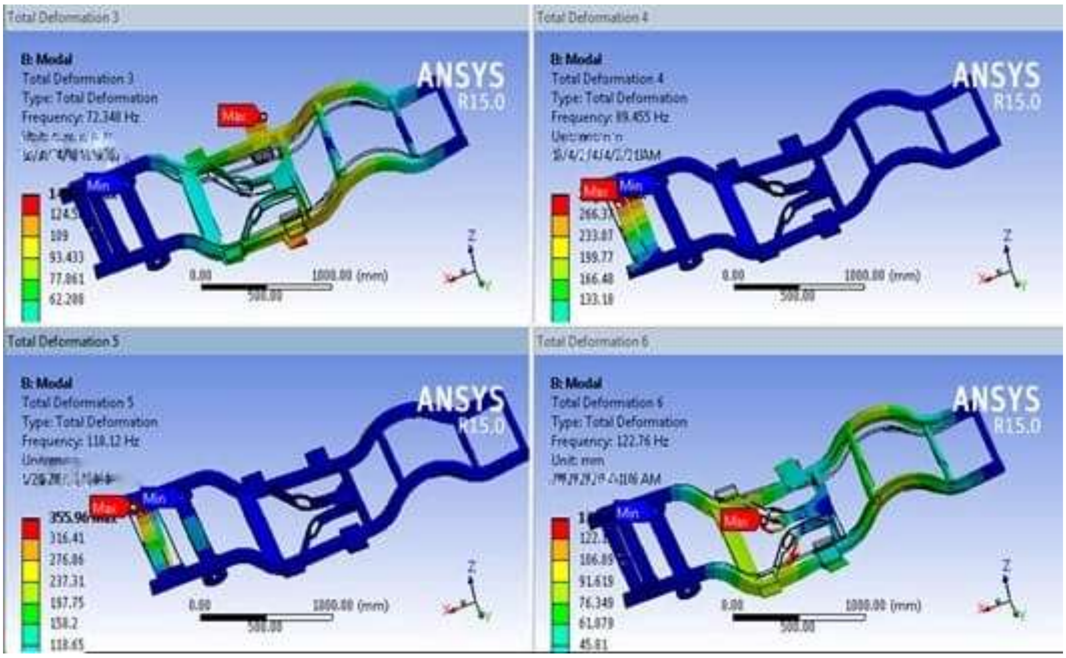

Fig: 6. 8 Total deformation

6.2 STATICSTRUCTURAL ANALYSIS

6.2.1 FIXED SUPPORT AND LOAD

To fix the component: right click on the analysis settings – insert – fixed support – select all the bushes or bearing's location and apply, as shown above.

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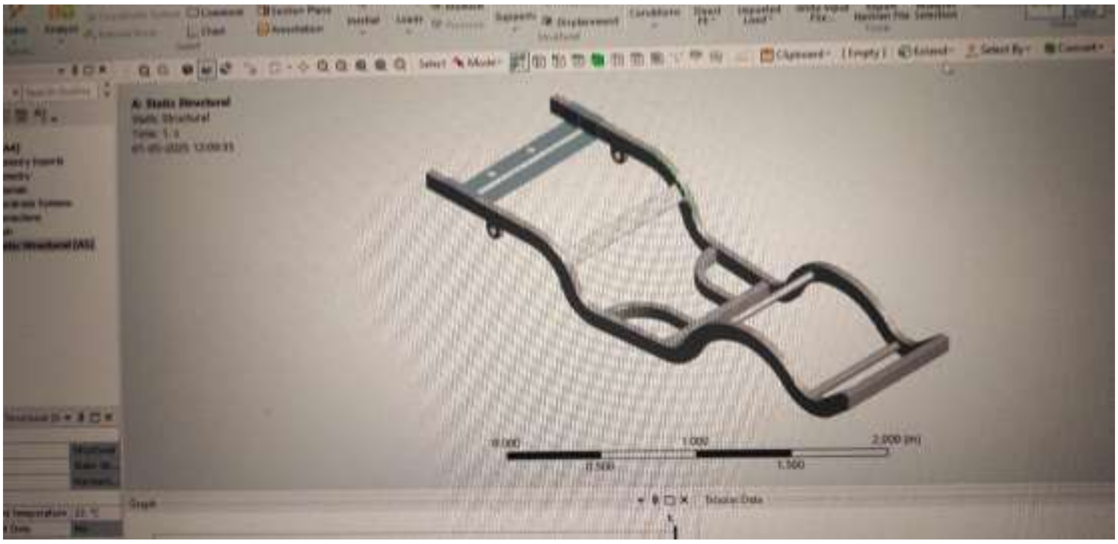

Fig: 6. 9 Static structural analysis

To apply force the component: right click on the analysis settings – insert – pressure – on pressure definitions specify 2 MPA N (20 tones) and selects faces is applied because of body.

Fig: 6. 10 Structural deformation after applying force

Solution

Right click on solution – insert total deformation

Right click on solution – insert equivalent stress

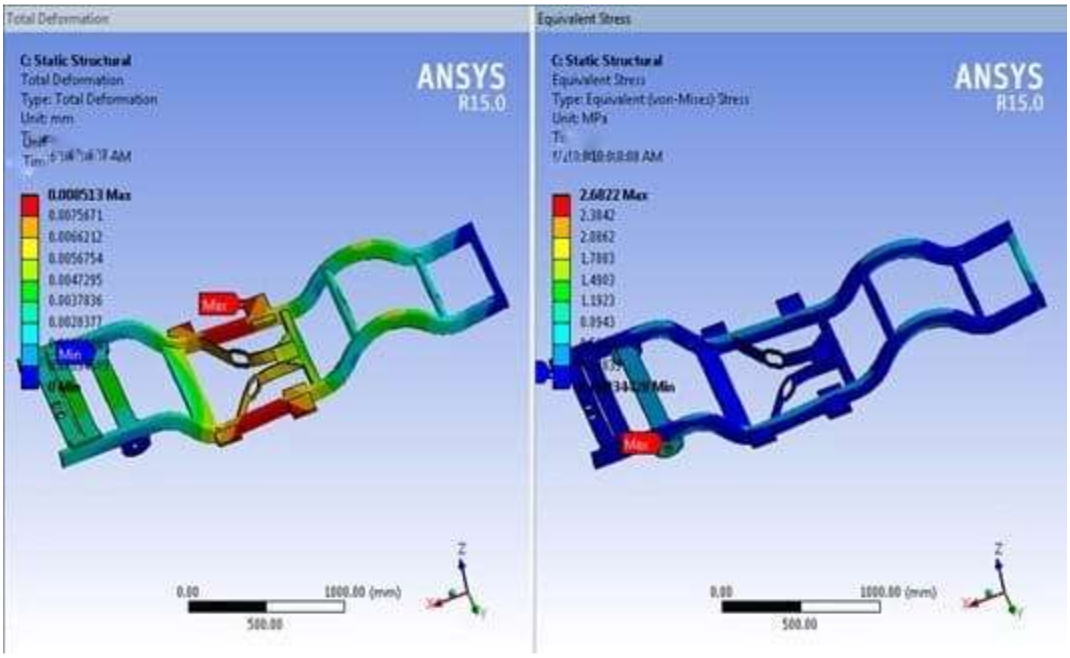

Fig: 6. 11 After deformation

For Material: Aluminum Alloy

A: modal analysis

6.2.2 FIXED SUPPORT:

To fix the component: right click on the analysis settings – insert – fixed support – select all the bushes or bearing's location and apply, as shown above.

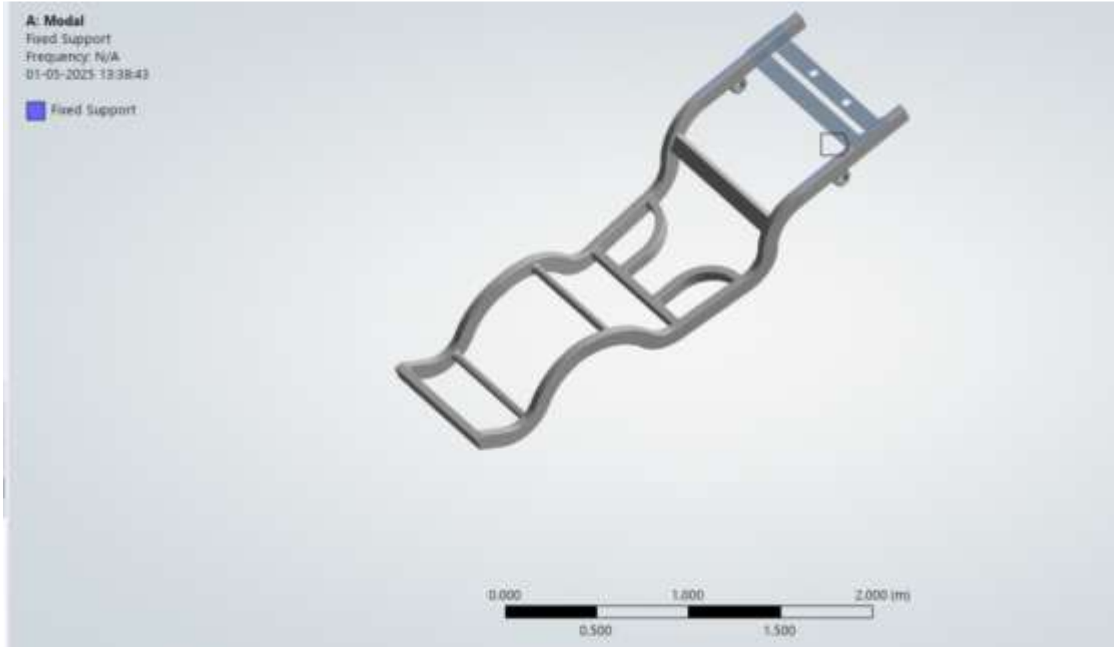

Fig: 6. 12 Material aluminum alloy

Solution

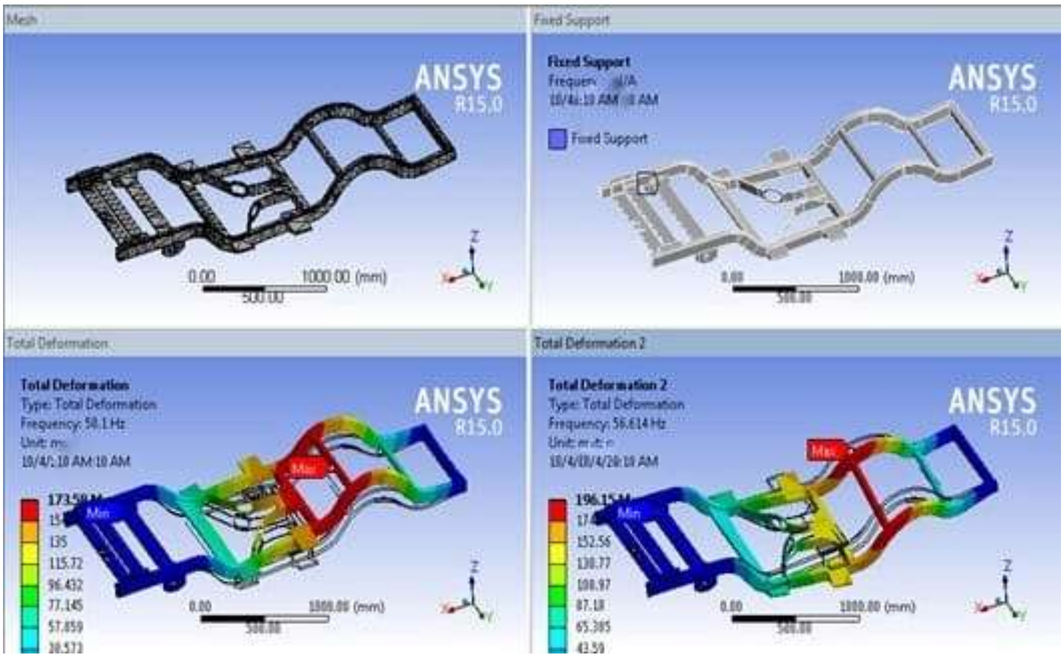

Fig: 6. 13 Solution

Right click on solution – insert total deformation

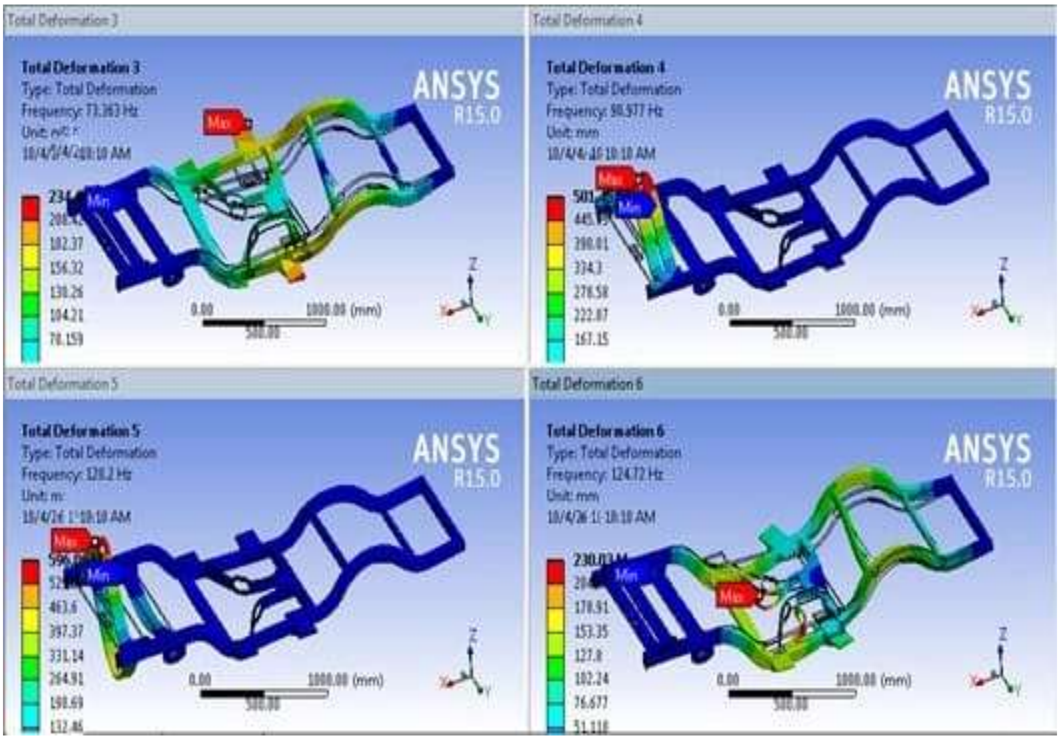

Fig: 6. 14 After inserting total deformation

The image displays results from a modal analysis conducted using ANSYS R15.0, showing the total deformation of a mechanical structure at different natural frequencies. Modal analysis is used to determine the natural vibration characteristics of a structure, identifying both the frequencies at which it naturally vibrates and the corresponding deformation patterns (mode shapes).

In this analysis, four different modes (3rd to 6th) are shown, with frequencies ranging from 73.193 Hz to 124.172 Hz. Each plot visualizes how the structure deforms at a specific frequency, using a color scale where red indicates regions of maximum deformation and blue indicates minimum deformation.

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CHAPTER 7

MODIFICATION

7.1 ADDING SUPPORTS:

Launch CATIA software, click on 'Start,' then select 'File' and choose 'Open.' Browse to the location where the chassis file is saved. In the Part Design module, select the Sketcher tool and choose the required surface plane. Once in the Sketcher module, go to the profile section and create the profile as shown in Figure 7.1.

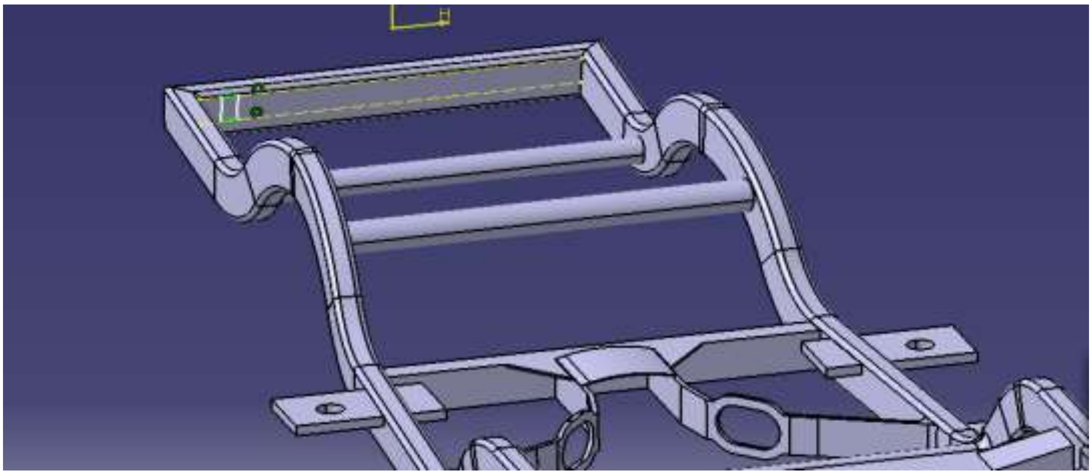


Fig: 7. 1 Create a profile

Once the sketch is complete, exit the Sketcher workbench to return to the Part module. In the Part module, go to the Sketcher-based features and select the 'Pad' tool. In the Pad Definition, select the sketch as the profile, set the first limit to 'Up to Plane' and choose the surface, then set the second limit to 'Up to Surface' and select the surface, as shown in Figure 7.2.

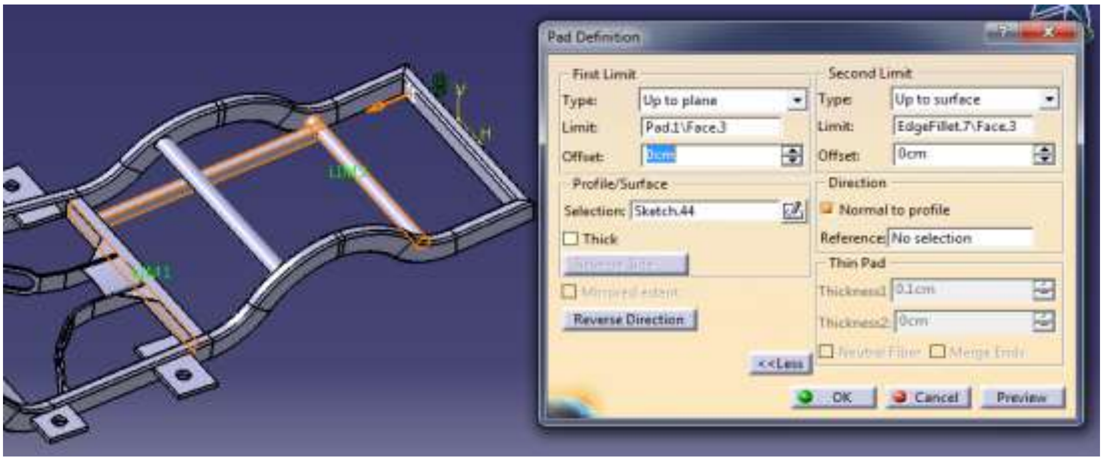


Fig: 7. 2 Select surface on second limit.

In the Part Design module, select the Sketcher tool and choose the required surface plane. Then, enter the Sketcher module, go to the profile section, and create the profile as shown in Figure 7.3.

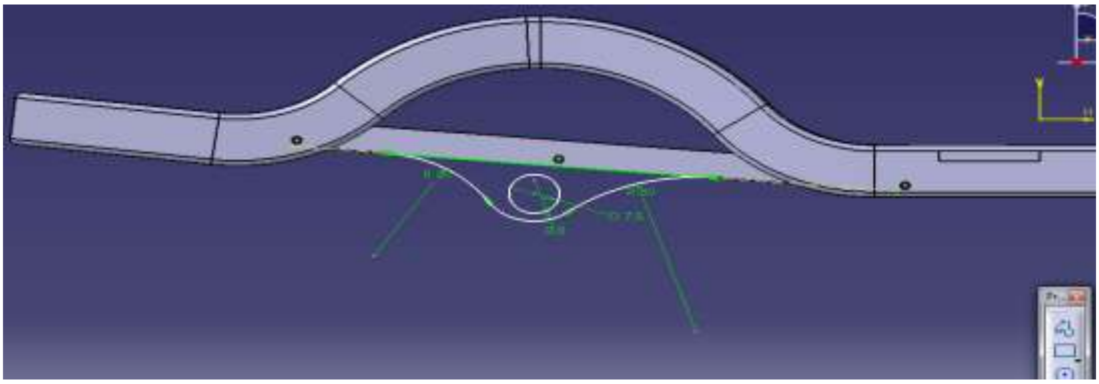


Fig: 7. 3 Add dimensions

Once the sketch is complete, exit the Sketcher workbench to return to the Part module. In the Part module, navigate to the Sketcher-based features and select the 'Pad' tool. In the Pad Definition, select the sketch as the profile and set the length to 3 cm, as shown in Figure 7.4.

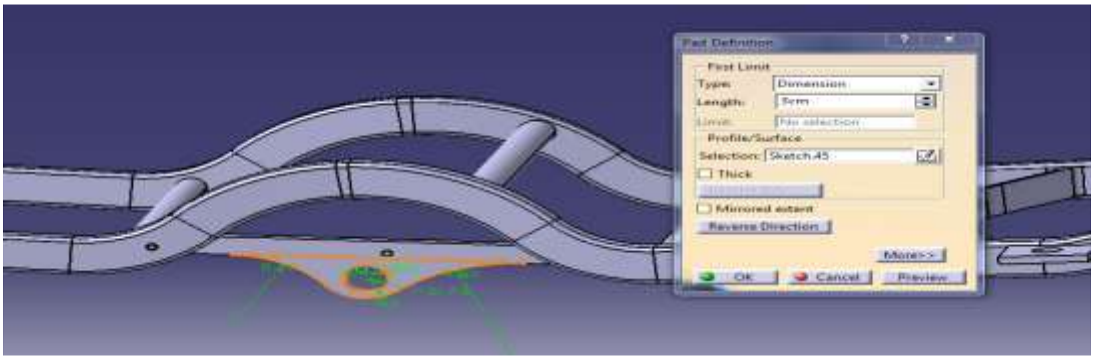


Fig: 7. 4 specify 3 cm in length

In part module, go to transformation-based feature and select mirror tool. In mirror definition, select ZX plane as a mirroring element and select above pad as object to mirror as shown in Figure 7.5.

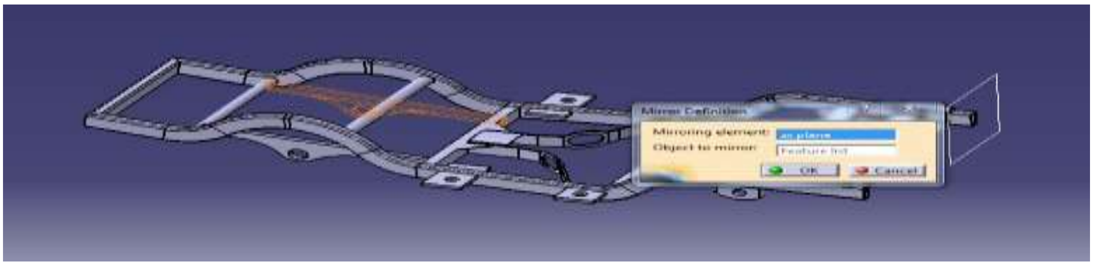


Fig: 7. 5 mirroring element

7.2 CHASSIS AFTER MODIFICATION

Work bench: After importing the model into project schematic window drag and drop the static structural tab on to the screen from the toolbox window and link the geometry by right and browser to geometry step or IGES file. Double click on the model it opens the mechanical window with object**.**

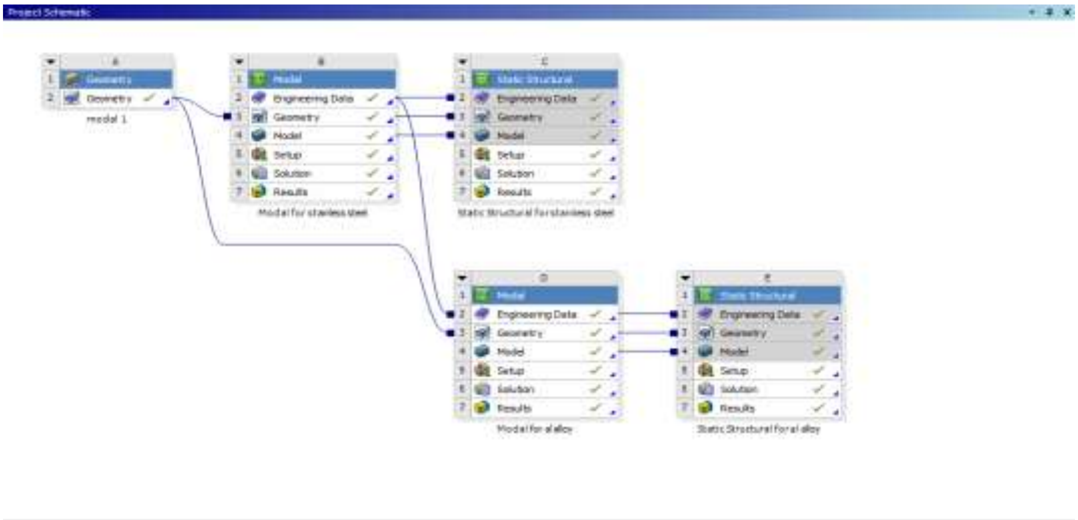


Fig: 7. 6 Work bench

Modal tree

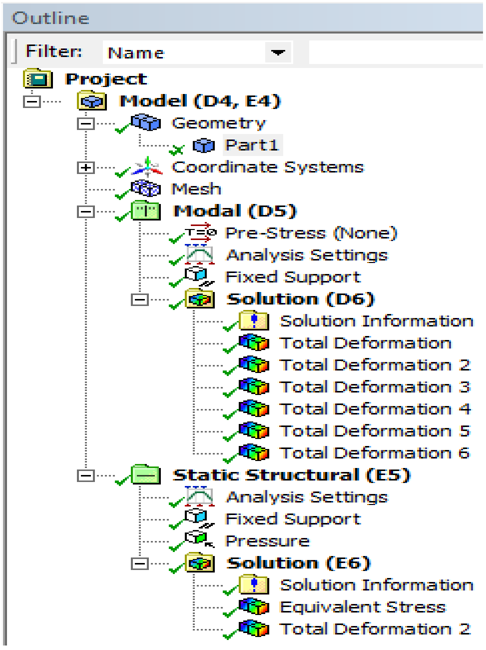


Fig: 7. 7 Modal tree

Geometry and meshing:

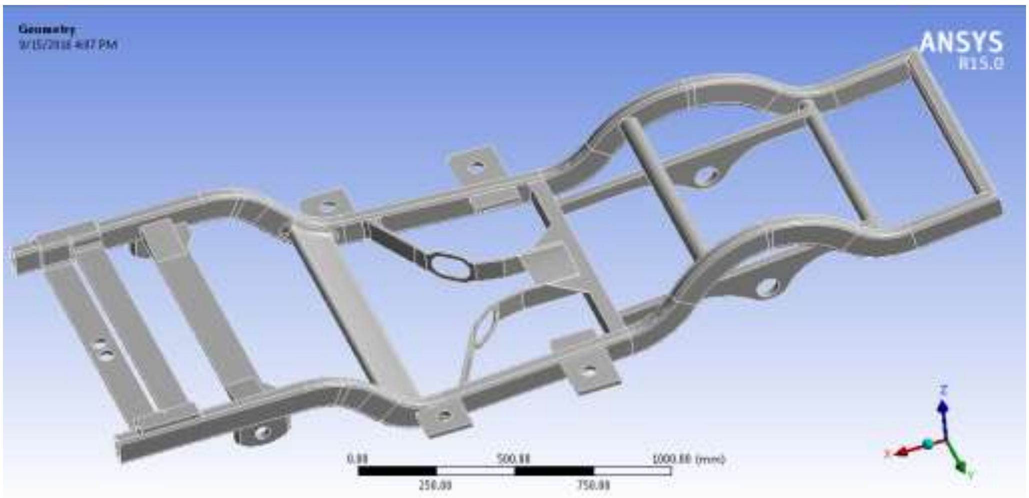


Fig: 7. 8 Geometry after modification

Geometry

From the outline tab – select the geometry – part – from the bottom detailed window- material –assignment – select required material.

Mesh: to generate the meshing, there are two methods one is automatic mesh generation and the other is with required size meshing. In this we used auto meshing with medium meshing.

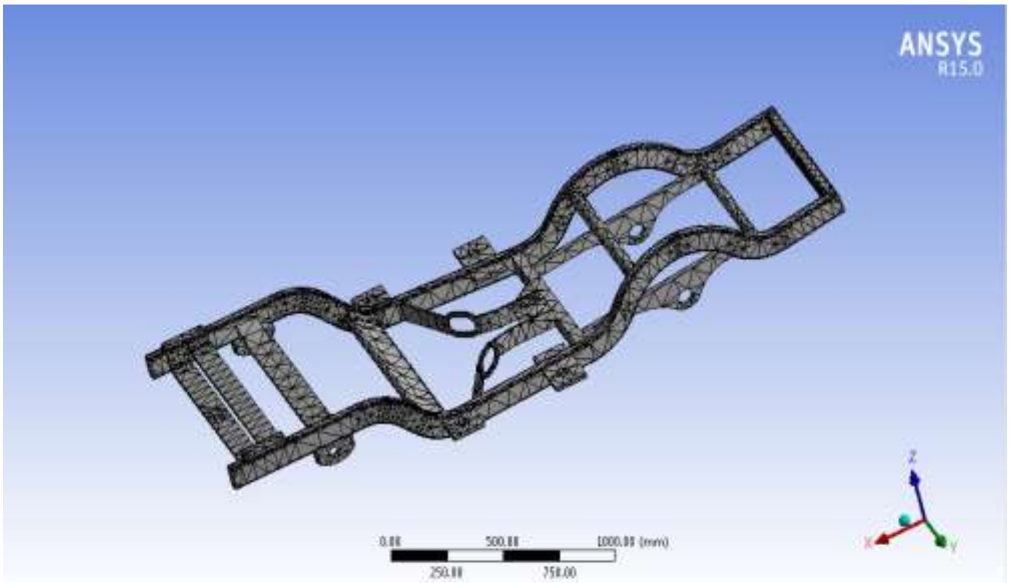


Fig: 7. 9 Meshing after modification

For Material: stainless steel

A: modal analysis

Fixed support:

To fix the component: right click on the analysis settings – insert – fixed support – select all the bushes or bearing's location and apply, as shown above.

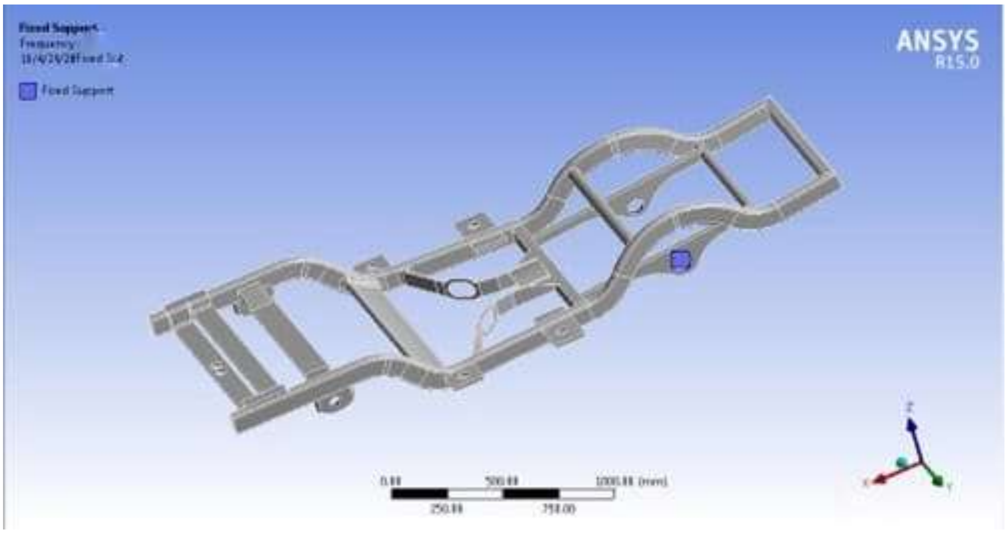


Fig: 7. 10 modal analysis

Solution

Right click on solution – insert total deformation

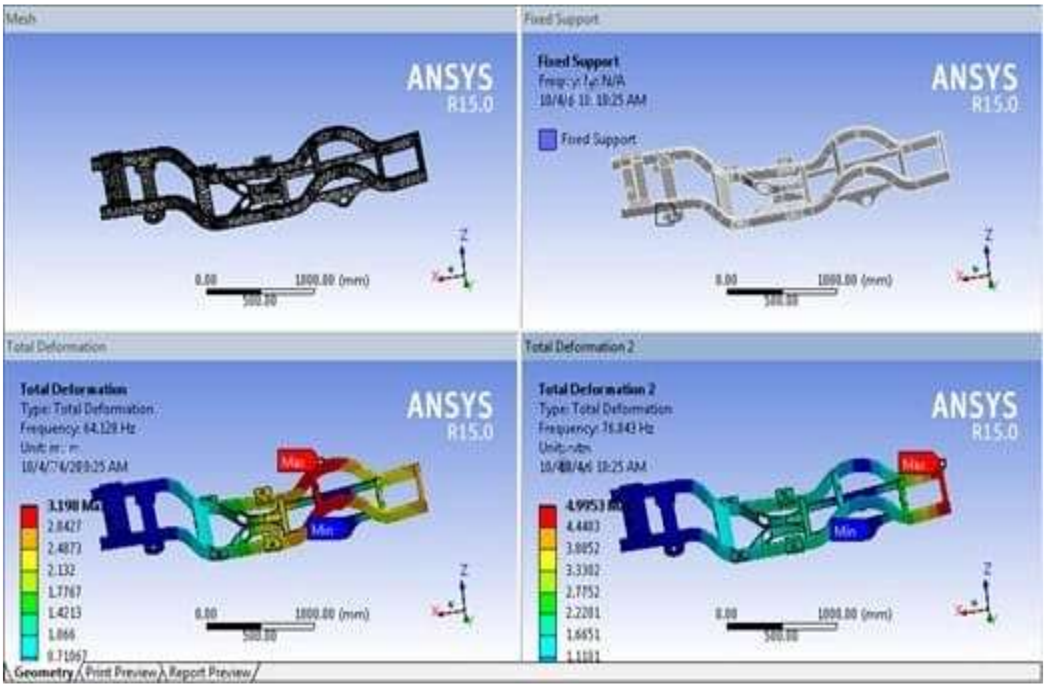


Fig: 7. 11 solution

Static structural analysis

Fixed support and load

To fix the component: right click on the analysis settings – insert – fixed support – select all the bushes or bearing's location and apply, as shown above.

To apply force the component: right click on the analysis settings – insert – pressure – on pressure definitions specify 2 MPA (20 tones) and selects faces is applied because of body.

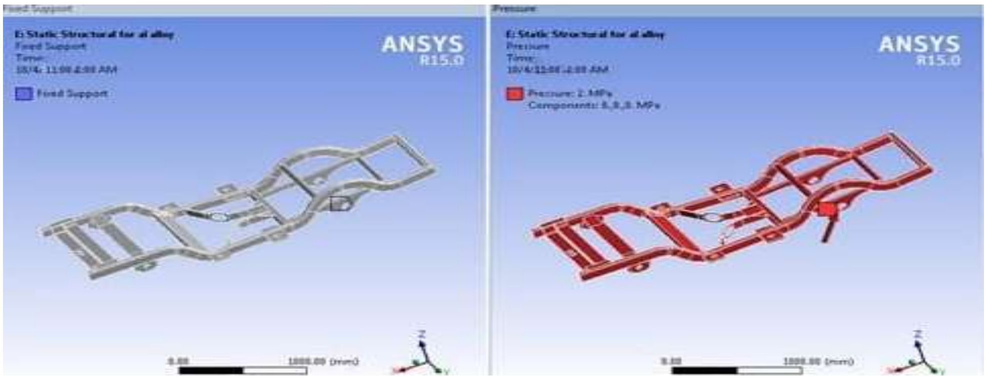


Fig: 7. 12 Applying load

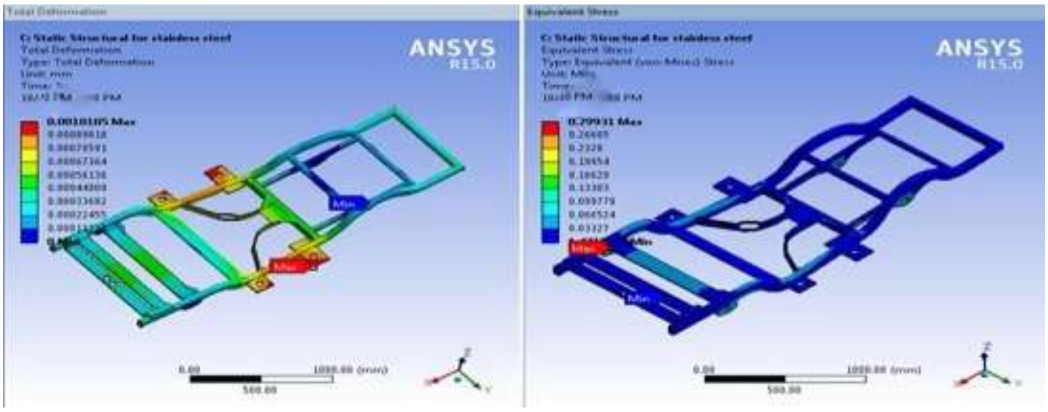


Fig: 7. 13 Total deformation

Solution

A. Right click on solution – insert total deformation
B. Right click on solution – insert equivalent stress

CHAPTER 8

RESULTS
Modal analysis
Before modification
Table 1 Before modification**:**

	Stainless Steel		Aluminum Alloy	
	Maximum (Mm)	Frequency (HZ)	Maximum (Mm)	Frequency (HZ)
Total Deformation1	103.76	49.372	173.58	50.1
Total Deformation2	117.05	55.723	196.15	56.614
Total Deformation3	140.15	72.348	234.48	73.363
Total Deformation4	299.66	89.455	501.45	90.977
Total Deformation5	355.96	118.12	596.06	120.2
Total Deformation6	137.43	122.76	230.03	124.72

Table 2 After modification:

	Stainless Steel		Aluminum Alloy	
	Maximum (Mm)	Frequency (HZ)	Maximum (Mm)	Frequency (HZ)
Total Deformation1	3.198	64.128	5.3558	65.19
Total Deformation2	4.9953	76.843	8.3643	78.08
Total Deformation3	5.061	79.156	8.4649	80.277
Total Deformation4	9.4552	87.467	15.826	88.892
Total Deformation5	4.1966	100.13	7.0247	101.45
Total Deformation6	3.7912	103.53	6.3334	104.92

II. Static structural analysis

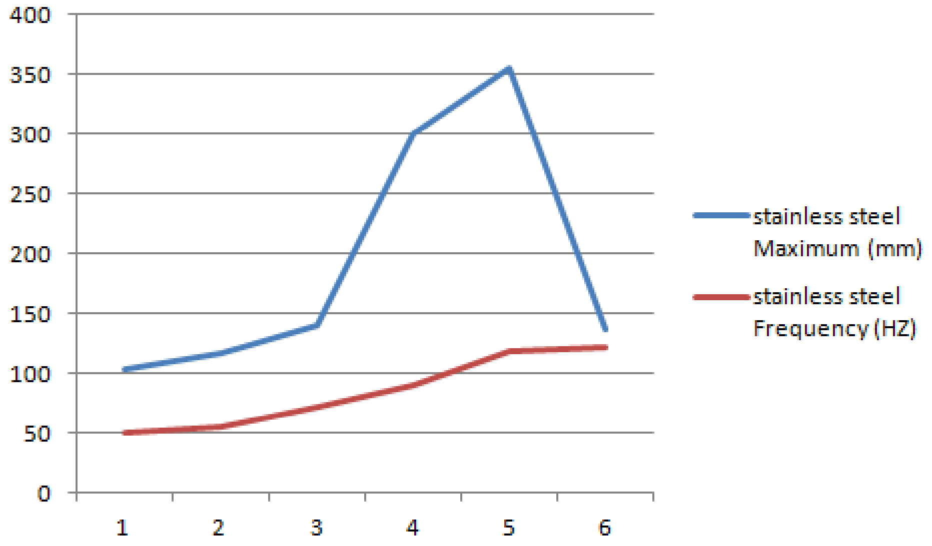
Graph for deformation and stress for stainless steel and aluminum alloy for before and after modification

Table 3 Static structural analysis

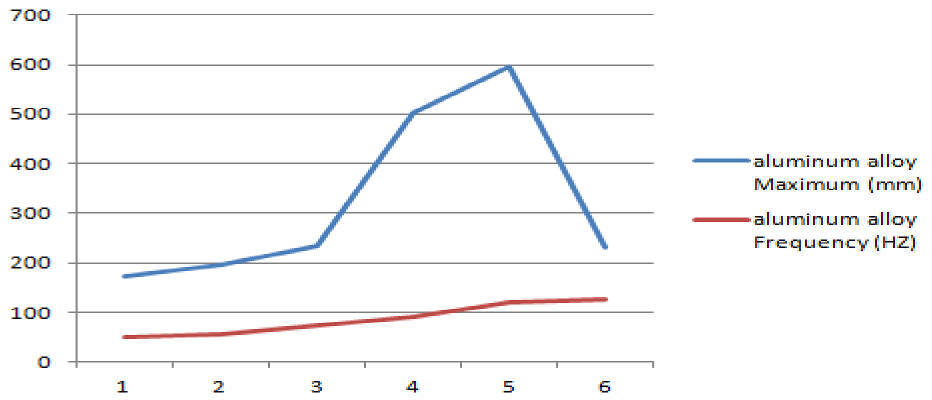
S.No		material	total deformation (mm)	Equivalent stress (MPA)
1	stainless steel	before	0.008513	2.6822
		after	0.0010105	0.29931
2	aluminum alloy	before	0.0020723	0.23723
		after	0.0024596	0.263

GRAPH:

Before modification



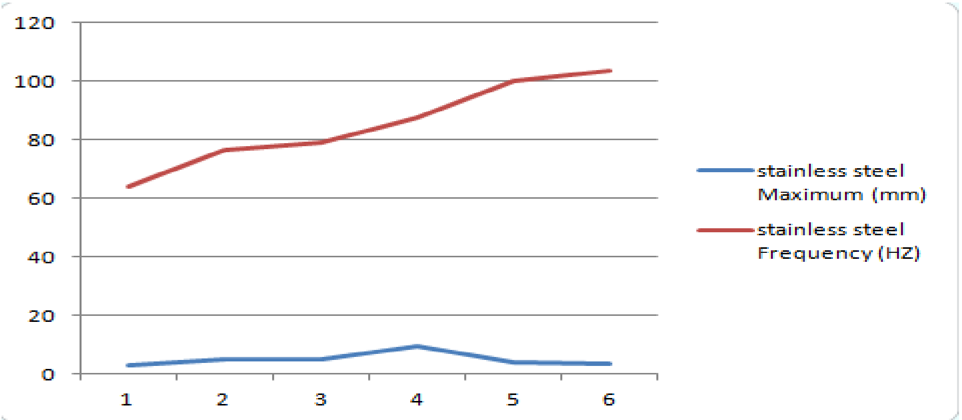
Graph 1 frequency and deformation for stainless steel



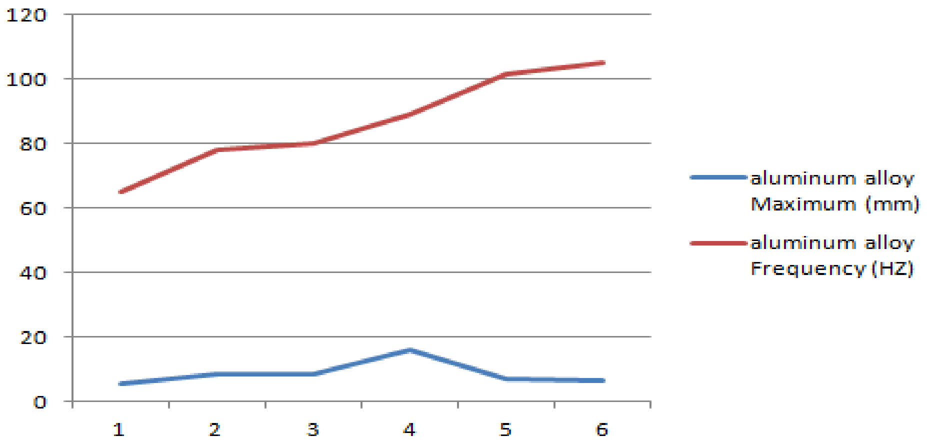
Graph 2 frequency and deformation for aluminum alloy

After modification

Graph for frequency and deformation for stainless steel

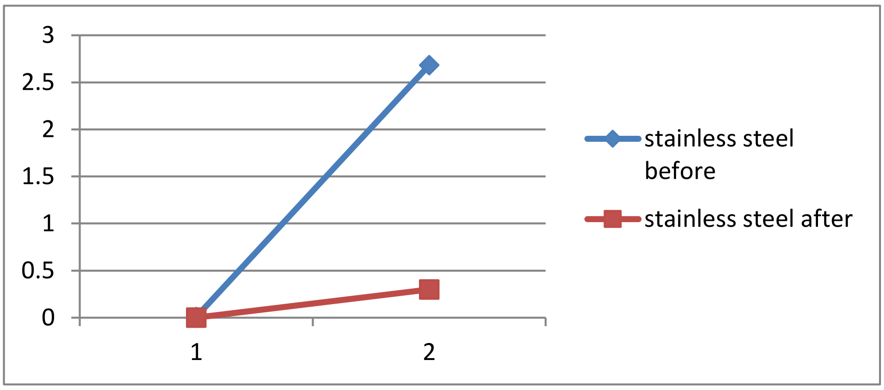


Graph for frequency and deformation for aluminum alloy



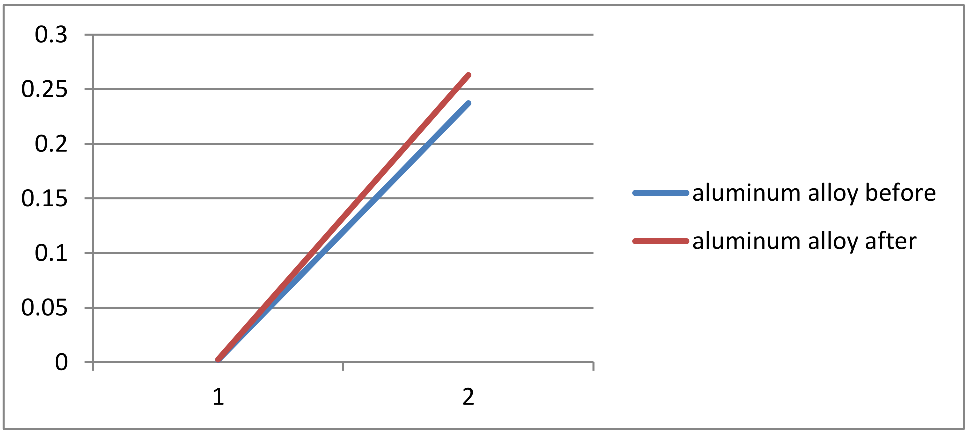
Graph 3 deformation for aluminum alloy

Graph for deformation and stress for stainless steel for before and after modification



Graph 4 stainless steel for before and after modification

Graph for deformation and stress for aluminum alloy for before and after modification



Graph 5 aluminum alloy for before and after modification

CONCLUSION

1. In this project modeling of a chassis is carried out. The chassis is modeled with the help of CATIAV5 software by using dimension.
2. Later the file is saved in the format as a STP or IGES file to do analysis on the component.
3. The analysis is done by ANSYS software one of the most practical meshing accurate analysis software to find out the results over the component.
4. The Model weight was increased after modification from 313.1 kg to 331.15 kg at a density of 7.75e-006 kg mm^-3 for stainless steel.
5. The Model weight was increased after modification from 111.91 kg to 118.36 kg at a density of 2.77e-006 kg mm^-3 for aluminum alloy.
6. The stress value was decrease from 2.6822 MPA to 0.29931 MPA for stainless steel.
7. The deformation was decrease from 0.008 mm to 0.001 mm for stainless steel
8. The stress value was increase from 0.23723Mpa to 0.263 MPA for aluminum alloy.
9. The deformation was increase from 0.0020723 mm to 0.0024596 mm for aluminum alloy
10. From the above results it is suggest that the design modification was acceptable.

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