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Shape Optimization of Heavy Vehicles Chassis

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Abstract: Chassis is a skeletal flame on which various mechanical parts like engine, tires axle assemblies, brakes, steering etc. are bolted. It is the most crucial element that gives strength and stability to the vehicle under different conditions .Role of the chassis is to be rigid enough to withstand the shock, twist,vibration and other stress. Strength and Stress are two main criteria for the designof the chassis.

The main objective of this project is to modify the design to decrease the deformation of chassis. The Model has been done using CATIA, which is advanced modelling software. The design and material of chassis structure significantly affects it's strength and weight. The primary objective of this project is to minimize weight while maintaining or improving structural strength and stiffness, thereby increasing pay load capacity, fuel efficiency and overall vehicle performance. In this project CAD is used to design the chassis outlet and also Ansy's software is used for structural analysis, including, to optimize the shape of heavy vehicles chassis. The structural analysis and optimization of heavy vehicles chassis is crucial for ensuring safety and performance

Keywords: Exhaust Manifold, Solidworks, CAD Model, 3d Printing.

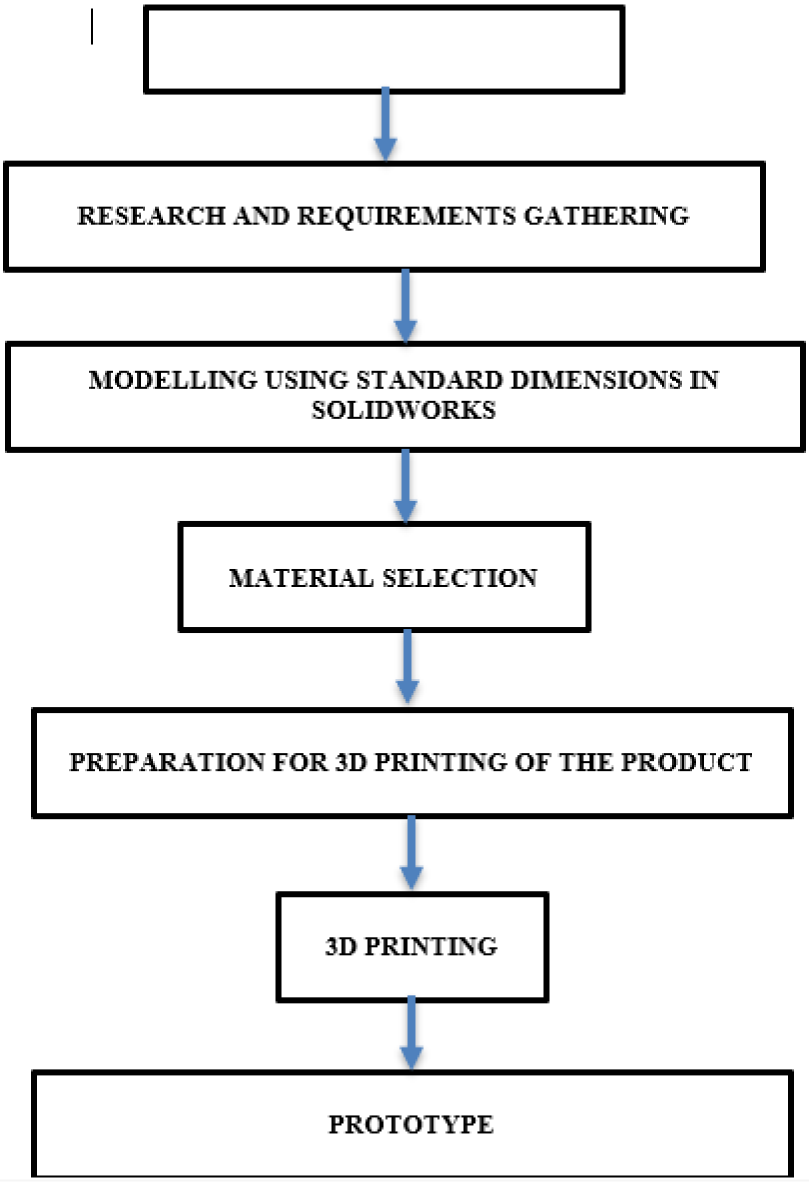
1. Introduction

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 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 wasdiscontinued in 1971. However, nearly all trucks, buses and pickups continue to use a separate frame as their chassis

convetional chassis

1. Methodology



3.1 INTRODUCTION TO CATIAV5R20

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 supportsproduct 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.

3.2 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.

3.21 SKETCHER:

Sketcher is used two-dimensional representations of profiles associated within the part. I 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.

3.22 CREATING A NEW SKETCH:

A new sketch- chose Start□Mechanical Design□Sketcher thenselect there refernce plane or sketch plane in which the sketch is to be created.

SKETCH PLANE

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

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

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

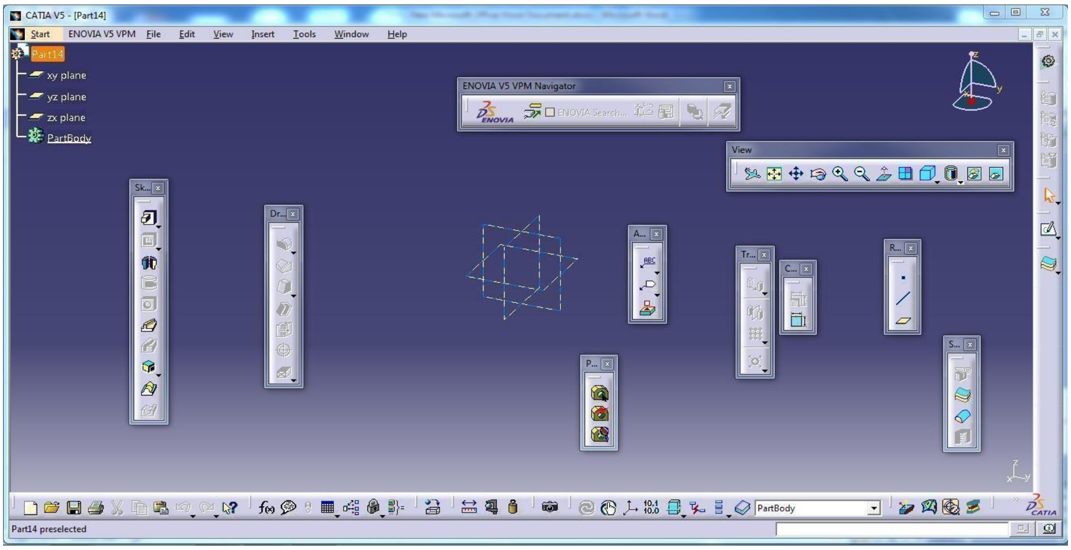


Fig3.23 solid works

4.1 Introduction To Fem and ansys

The finite element method represents an extension of the matrix methodsfor 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

Advantages of fem:

1. Its ability to use various size and shape and to modal a structure ofarbitrary 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 ofbeing quite different obstruction that is hard to visualize.
5. The fem is proven successfully in representing various types of complicated material properties and material behaviour (nonlinear, anisotropic, time dependent or temperature dependent material behaviour).
6. It readily account for non-homogeneity of the material by assigning differentproperties to different elements or even it is possible to vary the properties within an element according to a pre-determined polynomial pattern.

4.2Introduction of 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 programdeveloped and maintained by swans on analysis systems inc. The program contains many routines, all interrelated and all for main purpose of achieving a solution to anengineering problem by finite element method. Ansys provides a complete solution to design problems. It consists ofpowerful design capabilities like full parametric solid modeling, design optimization and auto meshing, which gives engineers full control over their analysis.

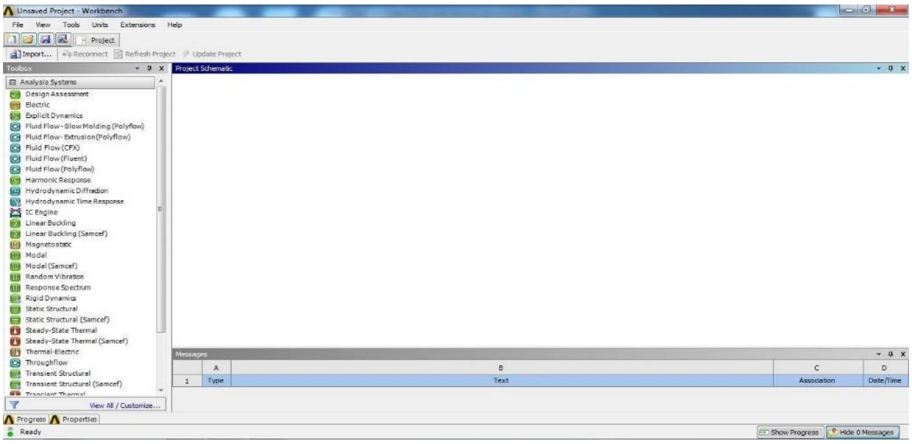


Fig 4.21 Windows of solid work

5. Meshing

The default meshing controls that the program uses may produce a mesh thatis adequate for the model i am analyzing. In this case, i need not specify any meshing controls. However if i do use meshing controls i 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 oneof the most important of the entire analysis for the decisions i 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 mesher a better chance of creating reasonably shaped elements during automatic mesh generation.

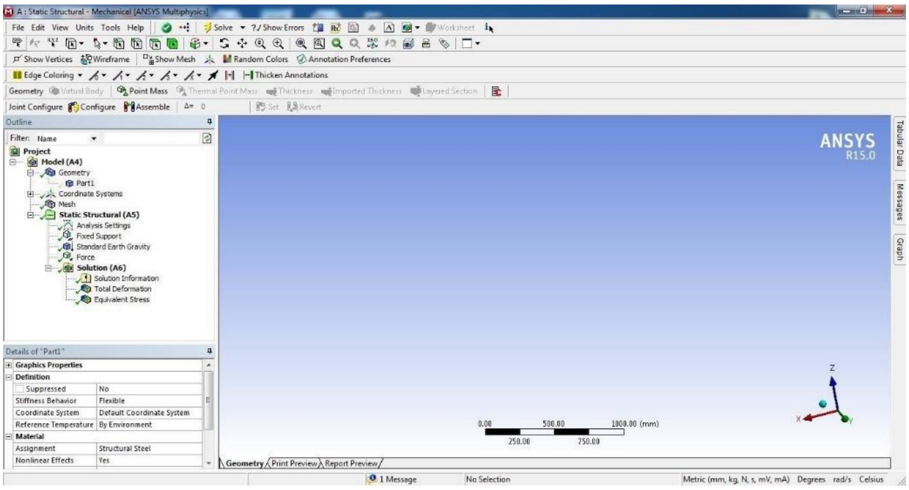


Fig 5. 1 Meshing process

6. Modeling of chassis frames.

Go to catia software and click on start – mechanical design and select wireframe and surface design module. In wireframe and surface design modulethen go to sketcher tool and select sketcher tool and select xy plane. Then we enter into sketcher module and go to profile and create a profile. After that go to workbench and select exit workbench. Then i enter into wireframe and surface design. Now go to sketcher tool and select sketcher tool and select yz plane. .

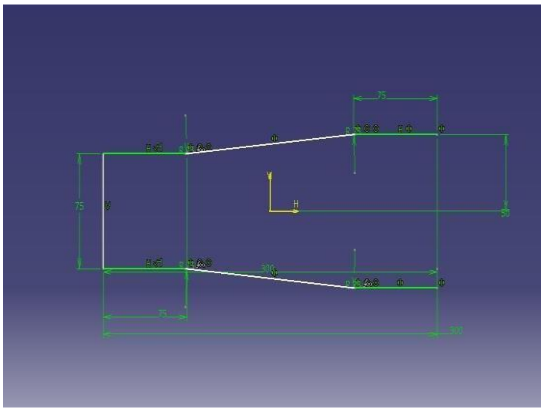


Fig 6.1 sketcher board

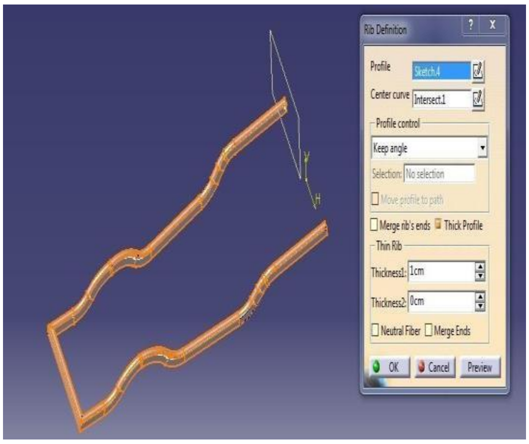


Fig 6.2 Extrude the profile

Now again, in part design module then go to sketcher tool and selectsketcher tool and select above plane. Then I enter into sketcher module and go to profile and create a profiles After completing sketcher go to workbench and select exit work bench. Then i enter into part module again. In part module, go to sketcher based feature and select pad tool. In definition, select above sketch as selection profile and specify up to next on first and second In part module, go to sketcher based feature and select stiffener tool.

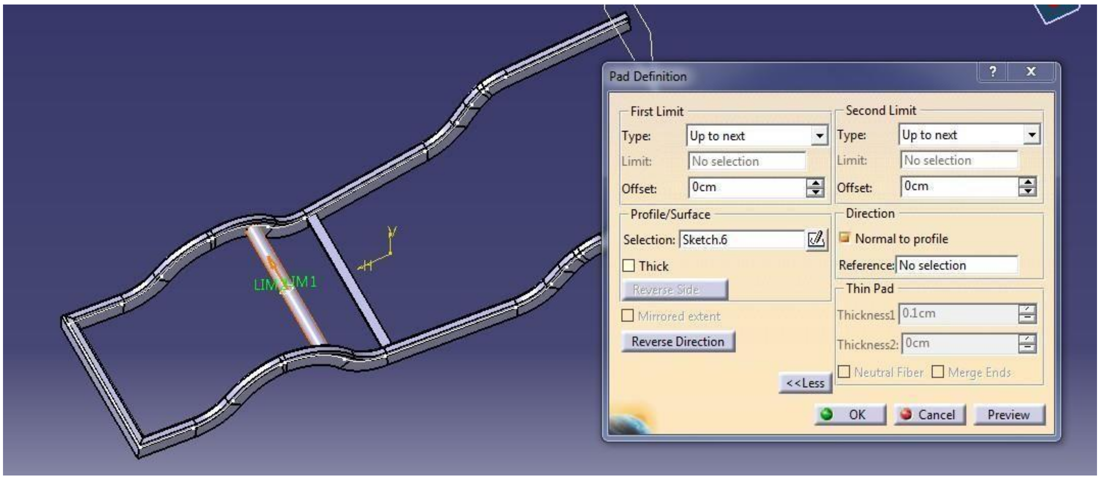


Fig 6.3 Extrude part

sAfter completing sketcher go to workbench and select exit work bench. Then i enter into part module again. In part module, go to sketcher based feature andselect pad tool. In pad definition, select above sketch as selection profile and specifyup to plane and select bottom In part module, go to dress up based feature and select edge fillet tool. In edge fillet definition, select the corners where i need fillet and specify fillet radius as a 2 cm as shown in fig

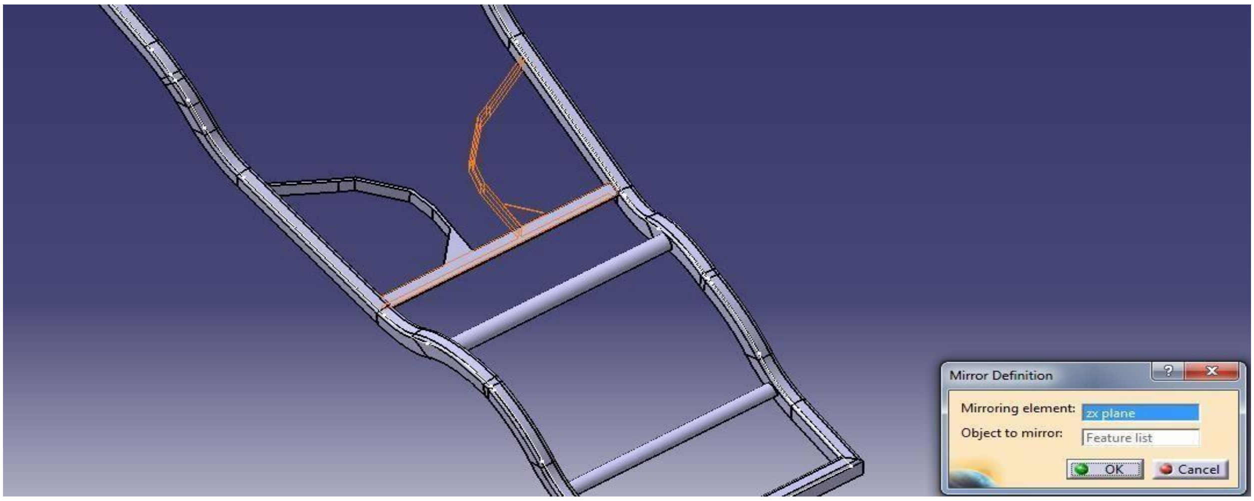


Fig 6.4 sketch the supports bed

After completing sketcher go to workbench and select exit work bench. Then i enter into part module again. In part module, go to sketcher based feature and select pad tool. In pad definition, select above sketch as selection profile and specify up to next on first and second In part module, go to transformation based feature and select mirror tool. Inmirror definition, select zx plane as a mirroring element and select above pad as object to mirror as shown fig.

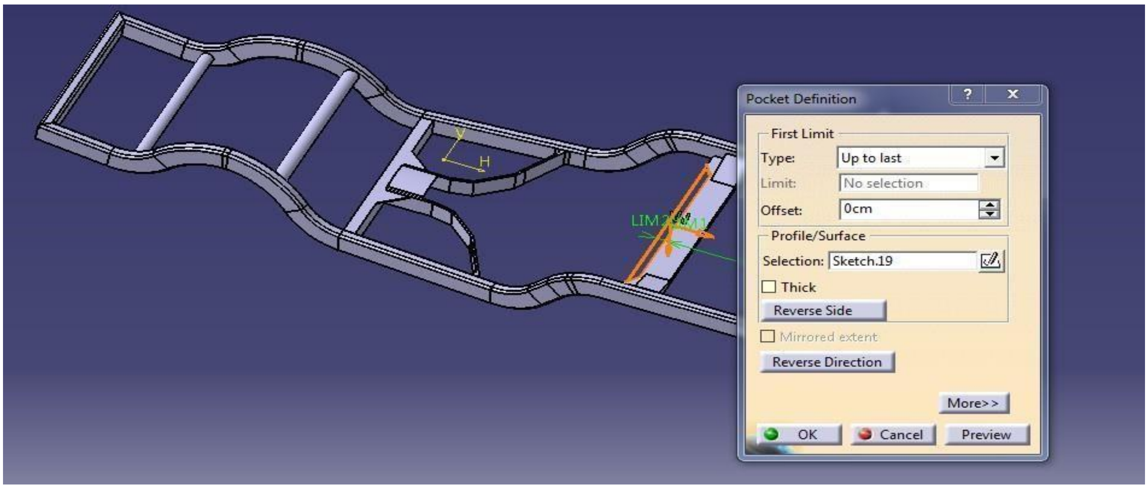
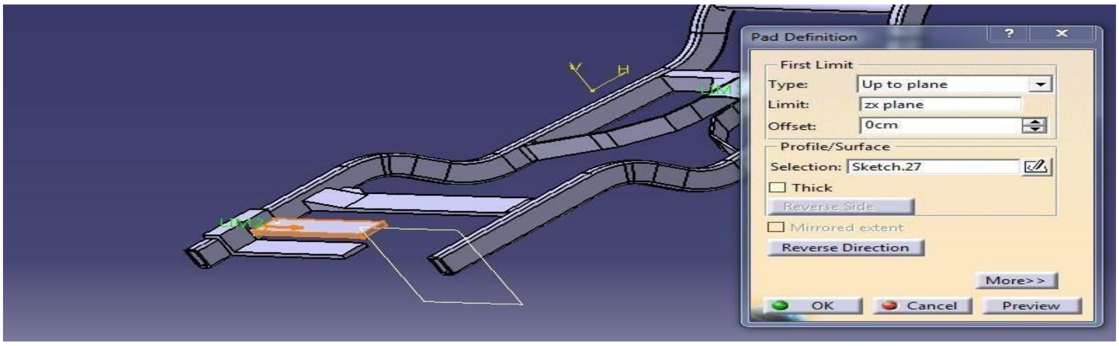


Fig 6.5 Extrude the supporting beds

After completing sketcher go to workbench and select exit work bench. Then i enter into part module again. In part module, go to sketcher based feature and select pad tool. In pad definition, select above sketch as selection profile and specify up to the plane and select zx plane as a limit as shown in fig 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 fig.



After completing sketcher go to workbench and select exit work bench. Then i enter into part module again. In part module, go to sketcher based feature and select pad tool. In pad definition, select above sketch as selection profile and specify length 10 cm as In part module, go to transformation based feature and select mirror tool. Inmirror definition, select zx plane as a mirroring element and select above pad as object to mirror as shown in fig.

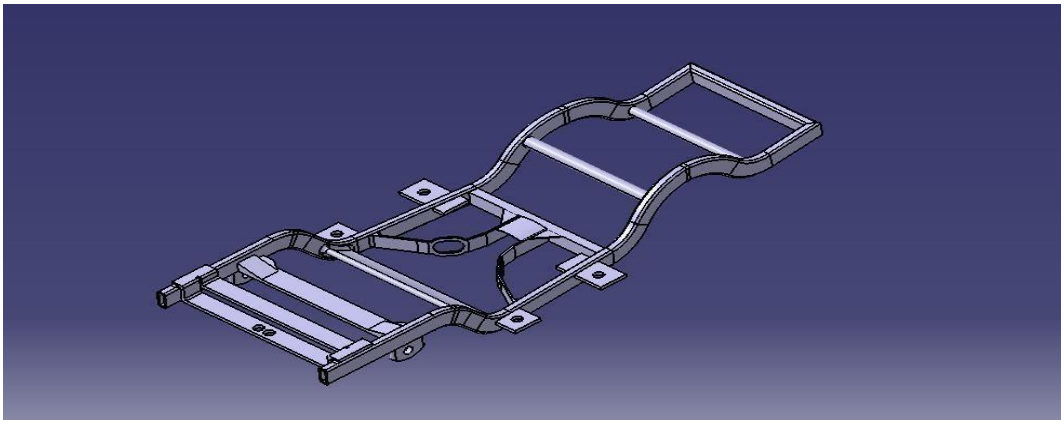


Fig 6.6 final chassis

Process 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 material properties are the important factor which will be considered asthe 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 i am 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.

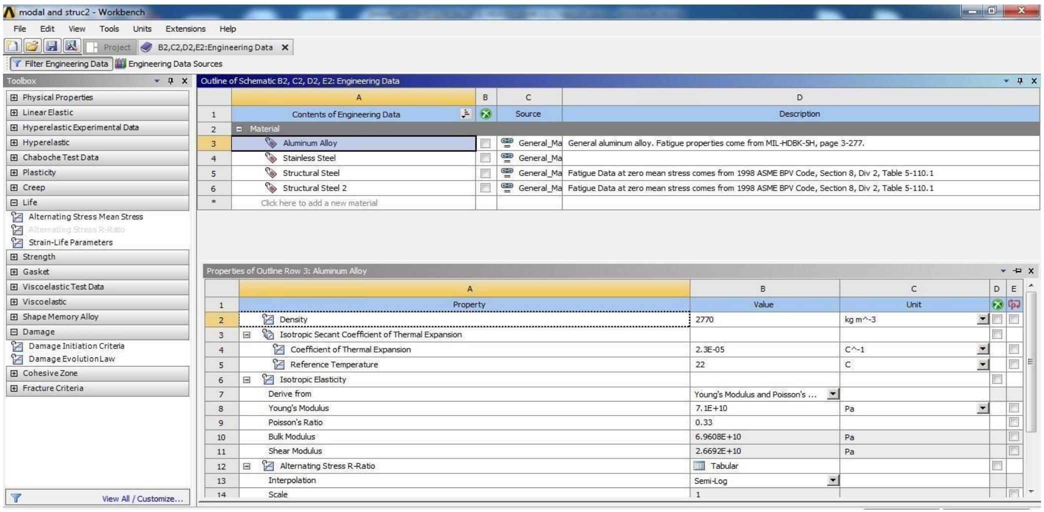


Fig 6.1.1 Modal and structure work bench scree

After importing the model into project schematic window drag anddrop 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.

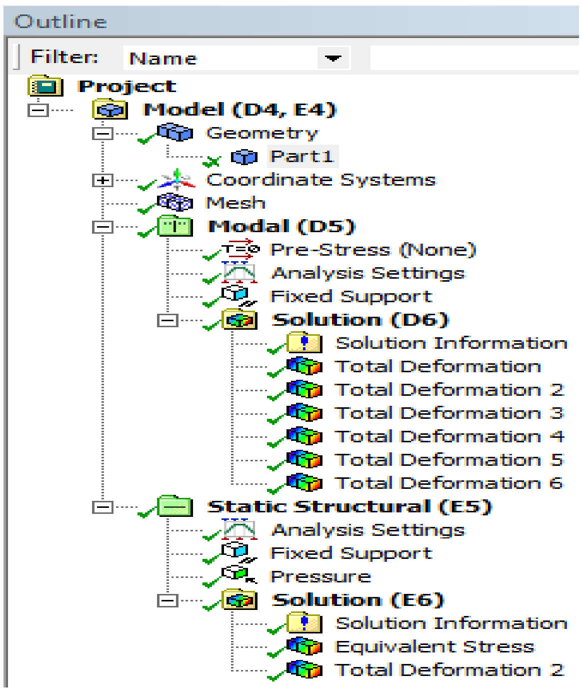
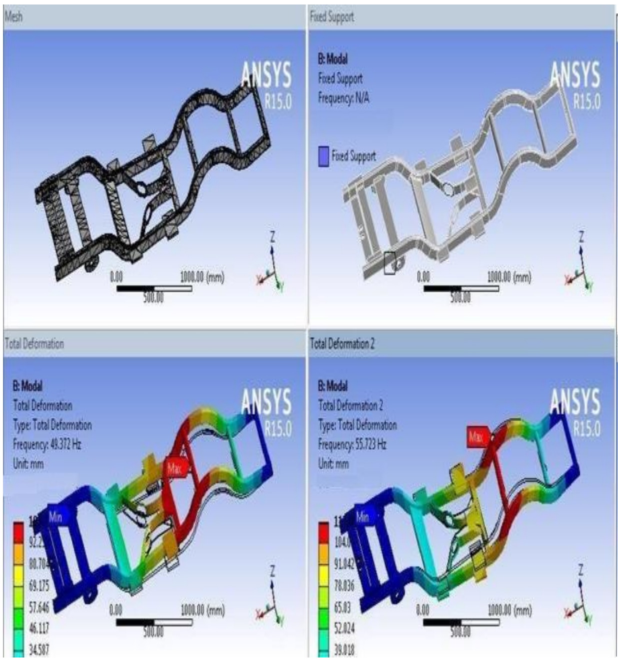
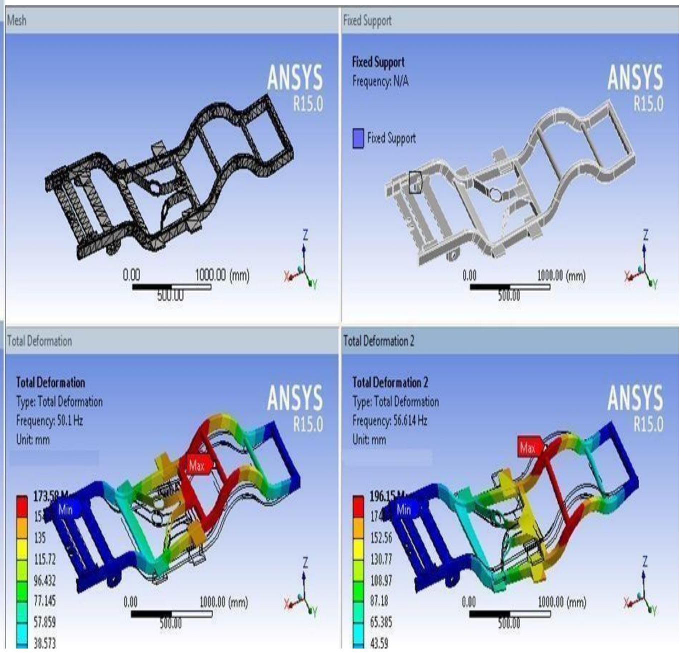


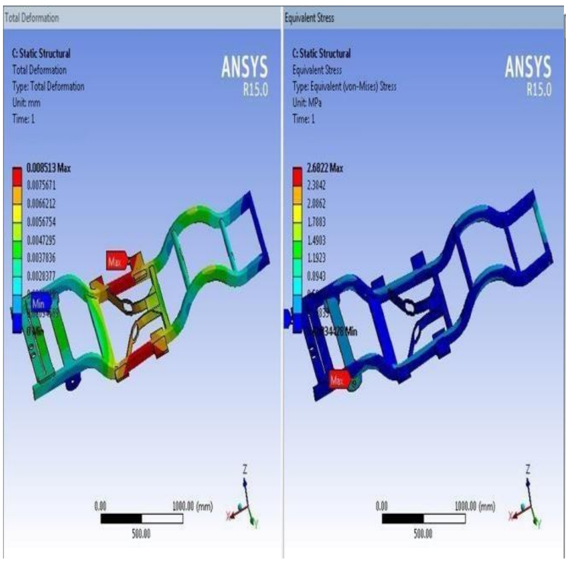
Fig 7.2 Work bench



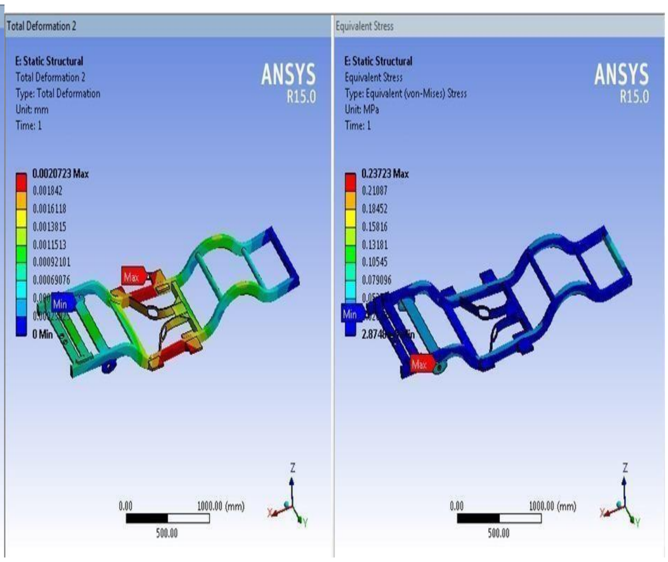
Stain less steel material



Aluminum alloy



Static and structural analysis of stain less materials



Static and structural analysis of aluminum materials

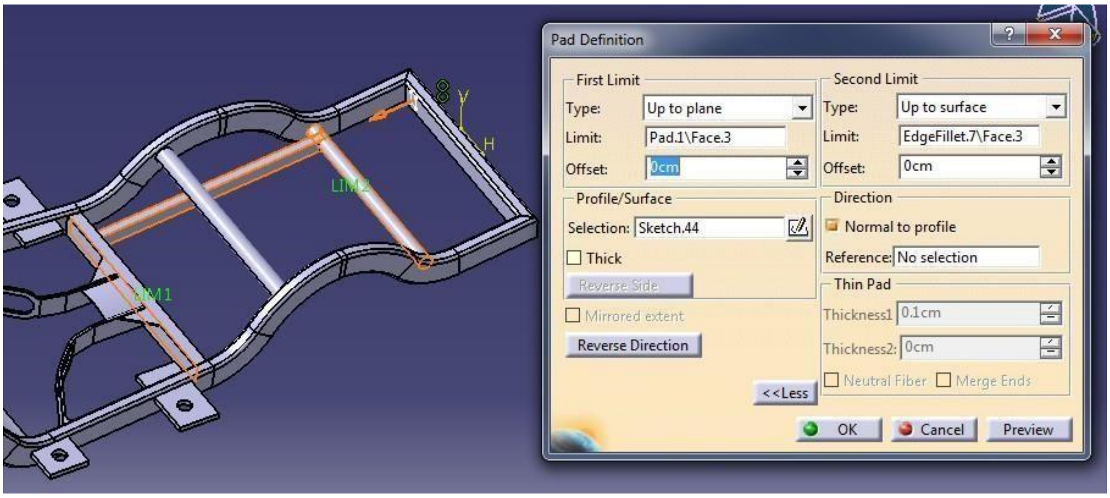
8. Remodeling to reduce deformation

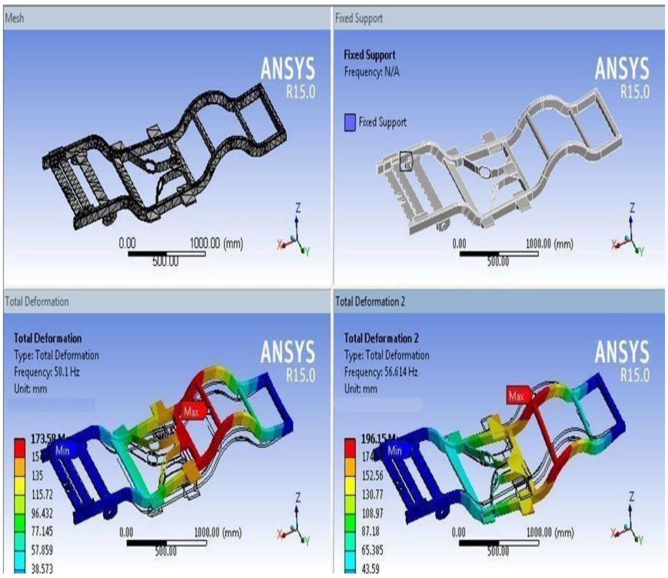
8.1 Adding of supports

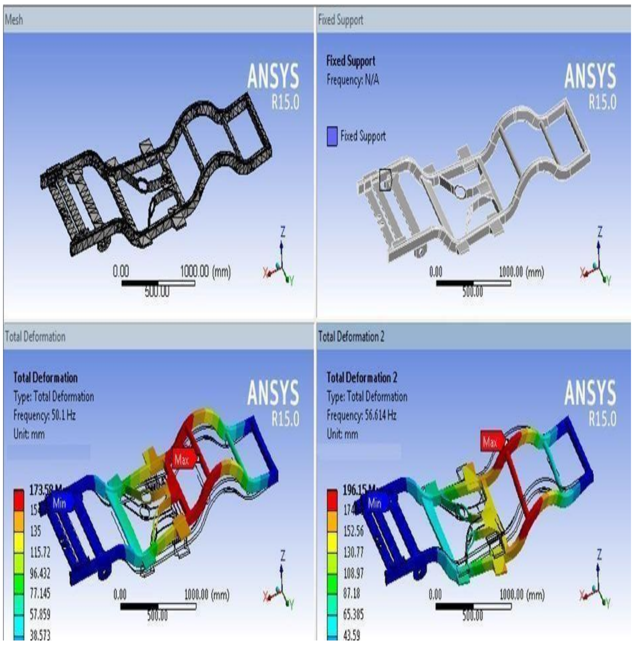
Go to catia software and click on start –click on file then select open. Browser to the file location where i save the chassis. part file. Then part design module then go to sketcher tool and select sketcher tool and select required surface plane. Then i enter into sketcher module and go to profile and create a profile as shown in fig. After completing sketcher go to workbench and select exit work bench. Then i enter into part module again. In part module, go to sketcher based feature and select pad tool. In pad definition, select above sketch as selection profile and specify up to plane and select surface.

For stain less material

For aluminum materials

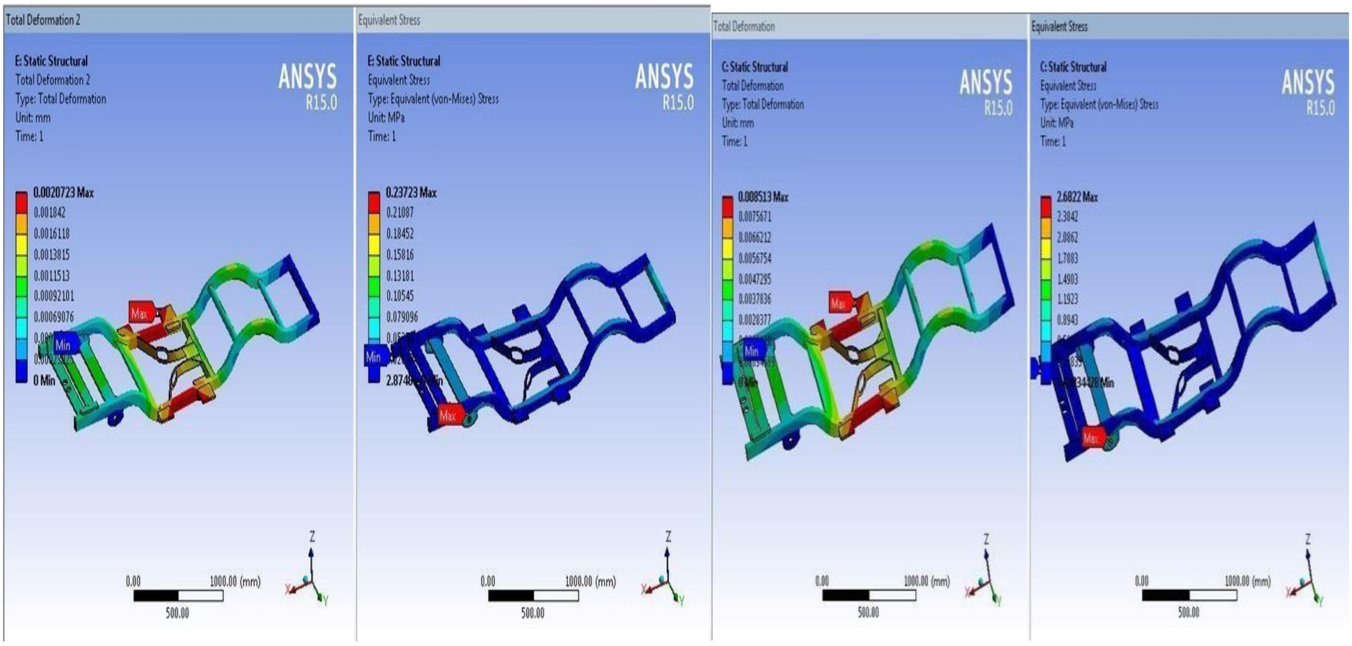






Static and structural analysis of stain less material

Static and structural analysis of aluminum materials



9 Results

Before modification

	Stainless Steel		Aluminum Alloy	
	Maxi mum (M m)	Freque ncy(HZ)	aximum(Mm)	equency(HZ)
Total	103	49.	173.58	50.1
Deformation	.76	372		
Total	117	55.	196.15	56.614
Deformation2	.05	723		
Total	140	72.	234.48	73.363
Deformation3	.15	348		
Total	299	89.	501.45	90.977
Deformation4	.66	455		
Total	355	118	596.06	120.2
Deformation5	.96	.12		
Total	137	122	230.03	124.72
Deformation6	.43	.76		

After modification

	Stainless Steel		Aluminum Alloy	
	aximum (Mm)	Freque ncy(HZ)	aximum(Mm)	equency(HZ)
Total	3.1	64.	5.3558	65.19
Deformation	98	128		
Total	4.9	76.	8.3643	78.08

Deformation2	953	843		
Total	5.0	79.	8.4649	80.277
Deformation3	61	156		
Total	9.4	87.	15.826	88.892
Deformation4	552	467		
Total	4.1	100	7.0247	101.45
Deformation5	966	.13		
Total	3.7	103	6.3334	104.92
Deformation6	912	.53		

Conclusion

In this project modeling of a chassis is prepared with the help of Ca v5 software by using dimension. The von-mises stresses and deformation have reduced after changing the shape model for Mild steel. similarly, have done after changing the shape model for Aluminum alloy but, deformation is little bithigh. As i see the stress is almost same for both materials i.e for stainless steel andaluminum alloy. While coming to the strain stainless steel has better value compare to aluminum alloy, but while coming to the deformation aluminum alloy material has best deformation than stainless steel material.So here I can conclude that aluminum alloy material is best for shape optimization design of chassis.

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29. **Robert L. Norton:** Robert L. Norton is an American engineer, academic and author. He is the President of Norton Associates and the Milton P. Higgins II Distinguished Professor Emeritus in Mechanical Engineering at the Worcester Polytechnic Institute. https://en.m.wikipedia.org/wiki/Robert_L._Norton.
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