MODELING AND 3D PRINTING OF VEHICLE CHASSIS

A major project(PROJECT II) report submitted to

Jawaharlal Nehru Techonological University Hyderabad

in partial fulfillment of the requirements for the reward of the degree of

BACHELOR OF TECHNOLOGY

in

MECHANICAL ENGINEERING

Submitted By

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GURU NANAK INSTITUTIONS TECHNICAL CAMPUS

(AUTONOMOUS)

(Affiliated to JNTU, Hyderabad),Approved by AICTE,New Delhi)

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2022-2023

CERTIFICATE

This is to certify that the Major-project entitled "**MODELING AND 3D PRINTING OF VEHICLE CHASSIS"** is being submitted by **L.SANTHOSH**(19WJ1A0399),**L.BHARATH**(19WJ1A03A0),**L.CHARAN**(19WJ1A03A1),in partial fulfillment for the award of the **Degree of Bachelor of Technology in Mechanical Engineering to the Jawaharlal Nehru Technological University** is a record of bonafide work carried out by them under my guidance and supervision.The results embodied in this Major-Project report have not been submitted to any other University or Institute for the award of any Degree or Diploma.

Internal Guide	**Head of the Department**
Mr.J.KISHORE	**(MECHANICALENGINEERING)**
(Assistant professor)	

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DECLARATION

We declare that this Minor-Project report titled "**MODELING AND 3D PRINTING OF VEHICLE CHASSIS"** submitted partial fulfillment for the award of the Degree of Bachelor of Technology in Mechanical Engineering to the Guru Nanak Institutions Technical Campus, Ibrahimpatnam is a record of original work carried out us under the guidance of Mr.J.KISHORE,Assistant Professor,Department of Mechanical Engineering, and has not formed the basis for the award of any other degree or diploma, in this or any other Institution or University. In keeping with the ethical practice in reporting scientific information, due acknowledgement has been made whenever the findings of others have been cited.

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ACKNOWLEDGEMENT

We wish to express our sincere thanks to Dr.H.S.SAINI, Managing Director, Guru Nanak Institutions and Dr.KODUGANTI VENKATA RAO, Director, Guru Nanak Institutions Technical Campus, School of Engineering and Technology, providing us with all the necessary facilities and their support.

We place on record,our sincere thanks to Dr.A.RAJ KUMAR, Professor and Head of the Department, Mechanical Engineering for their wholehearted co-operation, providing excellent lab facility ,constant encouragement and unfailing inspiration.

We would like to say sincere thanks to Mr.V.SHYAMU ,Assistant Professor, Department of Mechanical Engineering for coordinating Projects.

We would like to say sincere thanks to our guide Mr.J.KISHORE , Assistant Professor, Department of Mechanical Engineering for Coordinating Projects for the suggestions and constant guidance in every stage of the project, we also like to thank all our lecturers helping us in every possible way. On a more personal note, we thank our beloved parents and friends for their moral support during our project.

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ABSTRACT

Chassis is a major component of a vehicle system. It consists of internal framework that supports man-made object. It is the underpart of the vehicle which consists of frame and running gear like engine, transmission system, suspension system etc. The automotive chassis is tasked with keeping all components together while driving and transferring vertical and lateral loads, caused by acceleration, on the chassis through suspension and the wheels. The key to good chassis design is that further the mass is away from the neutral axis the more rigid it is.In this project, SolidWorks is the software used for the modelling of the chassis. It is an advanced CAD software and is used for analysis (static, dynamic, electro-magnetic, thermal) using Finite element method . Stress anaylsis is a key characteristics of a chassis . The design and analysis of chassis is done by identifying the location of high stress areas. The chassis design used in this project is the ladder frame chassis.

Ladder frame chassis is the simplest and oldest of the chassis design used in modern vehicular construction . It is originally adapted from the “horse and buggy” style carriages as it provides sufficient strength for holding the weight of the components. Ladder frame has several members that cross link to hold frame rails together. A simple design of two rails connected by a simple span and simulated provides a very good indication of how ladder frame is useful in regards to performance auto design.In this project the model is Designed with respect to all the available constraints using an advanced cad software's solid works, catia and solid edge .Later the product file is converted to ".stp" file format (standard exchange of product file) and imported to solidworks to design vehicle chassis.

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CHAPTER-1

1.INTRODUCTION OF 3D PRINTING

Prototyping or model making is one of the important steps to finalize a product design. It helps in conceptualization of a design. Before the start of full production a prototype is usually fabricated and tested. Manual prototyping by a skilled craftsman has been an age old practice for many centuries. Second phase of prototyping started around mid-1970s, when a soft prototype modeled by 3D curves and surfaces could be stressed in virtual environment, simulated and tested with exact material and other properties. Third and the latest trend of prototyping, i.e., Rapid Prototyping (RP) by layer-by-layer material deposition, started during early 1980s with the enormous growth in Computer Aided Design and Manufacturing (CAD/CAM) technologies when almost unambiguous solid models with knitted information of edges and surfaces could define a product and also manufacture it by CNC machining. The historical development of RP and related technologies is presented in below.

1.1 GENERAL EXPLANATION OF 3D PRINTING:

A method of manufacturing known as ‘Additive manufacturing’, due to the fact that instead of removing material to create a part, the process adds material in successive patterns to create the desired shape.

Main areas of use:

- Prototyping
- Specialized parts – aerospace, military, biomedical engineering, dental
- Future applications– medical (body parts), buildings and cars

3D Printing uses software that slices the 3D model into layers (0.01mm thick or less in most cases). Each layer is then traced onto the build plate by the printer, once the pattern is completed, the build plate is lowered and the next layer is added on top of the previous one. Typical manufacturing techniques are known as ‘Subtractive Manufacturing’ because the process is one of removing material from a preformed block. Processes such as Milling and Cutting are subtractive manufacturing techniques. This type of process creates a lot of waste since; the material that is cut off generally cannot be used for anything else and is simply sent out as scrap. 3D Printing eliminates such waste since the material is placed in the location that it is needed only, the rest will be left out as empty space.

1.1.1 ADVANTAGES AND LIMITATIONS:

Layer by layer production allows for much greater flexibility and creativity in the design process. No longer do designers have to design for manufacture, but instead they can create a part that is lighter and stronger by means of better design. Parts can be completely re-designed so that they are stronger in the areas that they need to be and lighter overall.

3D Printing significantly speeds up the design and prototyping process. There is no problem with creating one part at a time, and changing the design each time it is produced. Parts can be created within hours. Bringing the design cycle down to a matter of days or weeks compared to months. Also, since the price of 3D printers has decreased over the years, some 3D printers are now within financial reach of the ordinary consumer or small company.

The limitations of 3D printing in general include expensive hardware and expensive materials. This leads to expensive parts, thus making it hard if you were to compete with mass production. It also requires a CAD designer to create what the customer has in mind, and can be expensive if the part is very intricate.

3D Printing is not the answer to every type of production method; however its advancement is helping accelerate design and engineering more than ever before. Through the use of 3D printers designers are able to create one of a kind piece of art, intricate building and product designs and also make parts while in space!

We are beginning to see the impact of 3D printing many industries. There have been articles saying that 3D printing will bring about the next industrial revolution, by returning a means of production back within reach of the designer or the consumer.

1.2 TYPES OF 3D PRINTING:

1.2.1 FDM – Fused Deposition Modeling

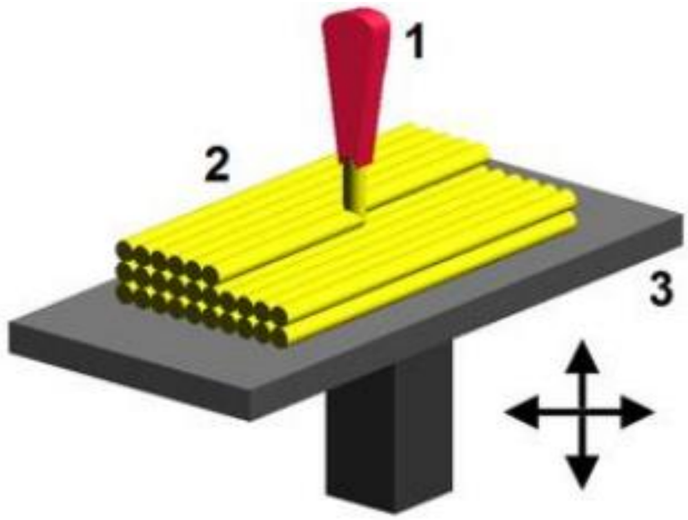

Fig.1.1 Fused deposition modeling

Fused Deposition Modeling, is an additive manufacturing technology commonly used for modeling, prototyping, and production applications. FDM works on an "additive" principle by laying down material in layers. A plastic filament or metal wire is unwound from a coil and supplies material to an extrusion nozzle which can turn the flow on and off. The nozzle is heated to melt the material and can be moved in both horizontal and vertical directions by a numerically controlled mechanism, directly controlled by a computer-aided manufacturing (CAM) software package. The model or part is produced by extruding small beads of thermoplastic material to form layers as the material hardens immediately after extrusion from the nozzle. Stepper motors or servo motors are typically employed to move the extrusion head.

FDM, a prominent form of rapid prototyping, is used for prototyping and rapid manufacturing. Rapid prototyping facilitates iterative testing, and for very short runs, rapid manufacturing can be a relatively inexpensive alternative.

Advantages: Cheaper since uses plastic, more expensive models use a different (water soluble) material to remove supports completely. Even cheap 3D printers have enough resolution for many applications.

Disadvantages: Supports leave marks that require removing and sanding. Warping, limited testing allowed due to Thermo plastic material.

1.2.2 SLA – Stereolithography :

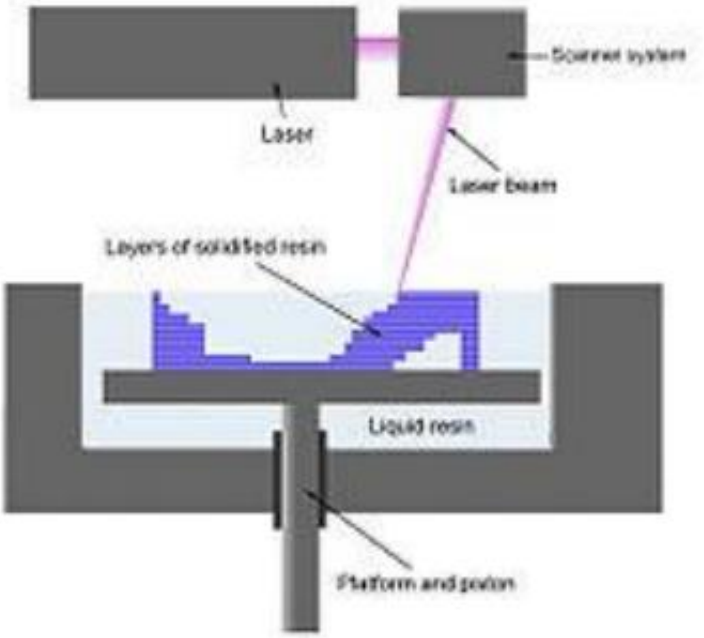

Fig.1.2 Stereolithography

Stereolithography is an additive manufacturing process which employs a vat of liquid ultraviolet curable photopolymer "resin" and an ultraviolet laser to build parts' layers one at a time. For each layer, the laser beam traces a cross-section of the part pattern on the surface of the liquid resin. Exposure to the ultraviolet laser light cures and solidifies the pattern traced on the resin and joins it to the layer below. After the pattern has been traced, the SLA's elevator platform descends by a distance equal to the thickness of a single layer, typically 0.05 mm to 0.15 mm (0.002" to 0.006"). Then, a resinfilled blade sweeps across the cross section of the part, re-coating it with fresh material. On this new liquid surface, the subsequent layer pattern is traced, joining the previous layer. A complete 3-D part is formed by this process. After being built, parts are immersed in a chemical bath in order to be cleaned of excess resin and are subsequently cured in an ultraviolet oven. Stereolithography requires the use of supporting structures which serve to attach the part to the elevator platform, prevent deflection due to gravity and hold the cross sections in place so that they resist lateral pressure from the re-coater blade. Supports are generated automatically during the preparation of 3D Computer Aided Design models for use on the stereolithography machine, although they may be manipulated manually. Supports must be removed from the finished product manually, unlike in other, less costly, rapid prototyping technologies.

1.2.3 SLS - Selective laser sintering:

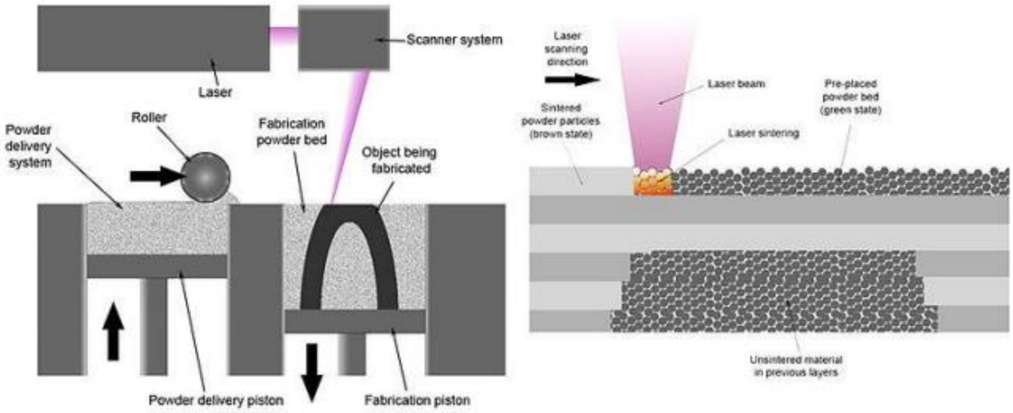

Fig.1.3 Selective laser sintering

Selective laser sintering is an additive manufacturing technique that uses a high power laser (for example, a carbon dioxide laser) to fuse small particles of plastic, metal (direct metal laser sintering), ceramic, or glass powders into a mass that has a desired three-dimensional shape. The laser selectively fuses powdered material by scanning cross-sections generated from a 3-D digital description of the part (for example from a CAD file or scan data) on the surface of a powder bed. After each cross-section is scanned, the powder bed is lowered by one layer thickness, a new layer of material is applied on top, and the process is repeated until the part is completed. Because finished part density depends on peak laser power, rather than laser duration, a SLS machine typically uses a pulsed laser. The SLS machine preheats the bulk powder material in the powder bed somewhat below its melting point, to make it easier for the laser to raise the temperature of the selected regions the rest of the way to the melting point. Some SLS machines use single-component powder, such as direct metal laser sintering. However, most SLS machines use two-component powders, typically either coated powder or a powder mixture. In single-component powders, the laser melts only the outer surface of the particles (surface melting), fusing the solid non-melted cores to each other and to the previous layer. Compared with other methods of additive manufacturing, SLS can produce parts from a relatively wide range of commercially available powder materials. These include polymers such as nylon (neat, glass-filled, or with other fillers) or polystyrene, metals including steel, titanium, alloy mixtures, and composites and green sand. The physical process can be full melting, partial melting, or liquid-phase sintering. Depending on the material, up to 100% density can be achieved with material properties comparable to those from conventional manufacturing methods. In many cases large numbers of parts can be packed within the powder bed, allowing very high productivity.

SLS is performed by machines called SLS systems. SLS technology is in wide use around the world due to its ability to easily make very complex geometries directly from digital CAD data. While it began as a way to build prototype parts early in the design cycle, it is increasingly being used in limited-run manufacturing to produce end-use parts. One less expected and rapidly growing application of SLS is its use in art.

1.3 CURRENT AND FUTURE APPLICATIONS OF 3D PRINTING

1.3.1 Biomedical Engineering

In recent years scientists and engineers have already been able to use 3D printing technology to create body parts and parts of organs. The first entire organ created through 3D Printing is expected to be done in the coming years. The process of creating the organ or body part is exactly the same as if you were to create a plastic or metal part, however, instead the raw material used are biological cells created in a lab. By creating the cells specifically for a particular patient, one can be certain that the patient's body will not reject the organ.Another application of 3D printing in the biomedical field is that of creating limbs and other body parts out of metal or other materials to replace lost or damaged limbs. Prosthetic limbs are required in many parts of the world due to injuries sustained during war or by disease. Currently prosthetic limbs are very expensive and generally are not customized for the patient's needs. 3D printing is being used to design and produce custom prosthetic limbs to meet the patient's exact requirements. By scanning the patient's body and existing bone structure, designers and engineers are able to re-create the lost part of that limb.

1.3.2Aerospace and Automobile Manufacturing :

High technology companies such as aerospace and automobile manufacturers have been using 3D printing as a prototyping tool for some time now. However, in recently years, with further advancement in 3D printing technology, they have been able to create functional parts that can be used for testing. This process of design and 3D printing has allowed these companies to advance their designs faster than ever before due to the large decrease in the design cycle. From what used to take months between design and the physical prototype, now within hours the design team can have a prototype in their hands for checks and testing.

The future of 3D printing in these industries lies with creating working parts directly from a 3D printer for use in the final product, not just for testing purposes. This process is already underway for future cars and aircraft. The way in which 3D printing works (creating a part layer by layer) allows the designer to create the part exactly the way is needs to be to accomplish the task at hand.

1.3.3 Construction and Architecture

Architects and city planners have been using 3D printers to create a model of the layout or shape of a building for many years. Now they are looking for ways of employing the 3D printing concept to create entire buildings. There are already prototype printer systems that use concrete and other more specialized materials to create a structure similar to a small house. The goal is the replace many cranes and even construction workers with these printing systems. They would work by using the 3D design model created on CAD software, to create a layer by layer pattern on the building just as a normal 3D printer works today. Most of the innovation in this area will have to come from the creation of the appropriate materials.

1.3.4 Product Prototyping

The creation of a new product is always one of that involves many iterations of the same design. 3D Printing revolutionized the industry by allows designers to create and the next day see and touch their design. No longer did it take several meetings for everyone to agree on one design to create, and then wait months for the actual part to arrive. Nowadays a version of each idea is created and the next day, all are reviewed together, thus giving the ability to compare and contrast each one's features.Plastic parts for example require molds and tooling to be created, these custom parts are expensive to create, therefore one must be certain the part designed meets the requirements. With 3D printing you can create a part that will look and feel exactly like the finished product. Some parts can also be tested just as the real injection molded part would.

CHAPTER 2

INTRODUCTION OF CHASSIS

2.1 Introduction Of Chassis Frame

Chassis frame is the fundamental casing work of the vehicle. It bolsters every one of the parts of the car appended to it. It is made of drop manufactured steel. Every one of the parts identified with vehicles are joined to it as it were. Every one of the frameworks identified with car like power plant, transmission, controlling, suspension, slowing mechanism and so forth.., are appended to and bolstered by it as it were.

A skeleton comprises of an inner vehicle outline that backings an artificial question in its development and utilize. A case of a skeleton is the under piece of an engine vehicle, comprising of the casing (on which the body is mounted). In the event that the running apparatus, for example, haggles, and here and there even the driver's seat, are incorporated, at that point the get together is depicted as a moving undercarriage.

A vehicle outline, otherwise called its frame, is the fundamental supporting structure of an engine vehicle to which every other part are appended, practically identical to the skeleton of a living being.

Until the 1930s, essentially every (engine) vehicle had a basic edge, isolate from the auto's body. This development configuration is known as-body-on-outline. After some time, about all traveler autos have moved to unibody development, which means their case and bodywork has been incorporated into each other. The last UK mass-created auto with a different skeleton was the Triumph Herald, which was suspended in 1971. In any case, about all trucks, transports and pickups keep on using a different casing as their body.

2.2 Layout Of Chassis And Its Main Components

"Skeleton" a French expression which implies the total Automobiles without Body and it incorporates every one of the frameworks like power plant, transmission, controlling, suspension , wheels tires , auto electric framework and so on without body. On the off chance that Body is likewise appended to it them it is referred to as the specific vehicle according to the shape and outline of the body

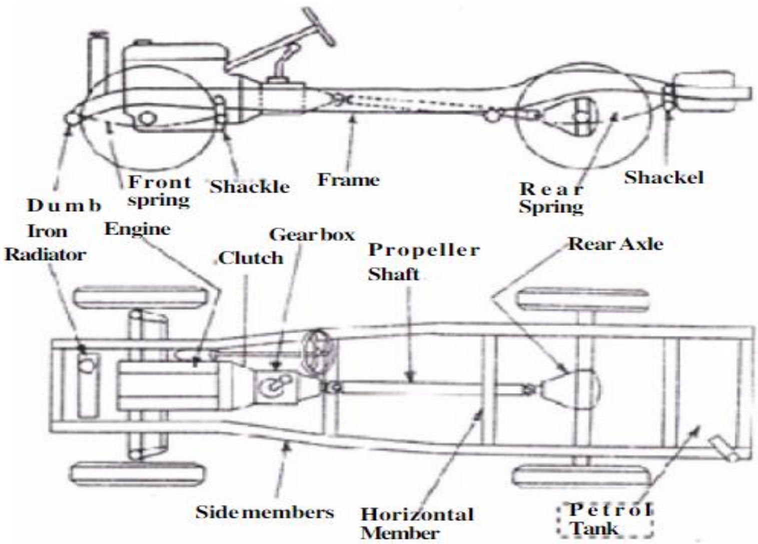

Figure 2. 1 Chassis

2.3 The Functions Of The Chassis Frame

1. To carryall the stationary loads attached to it and loads of passenger and cargo carried in it .
2. To withstand torsion vibration caused by the movement of the vehicle
3. To withstand the centrifugal force caused by cornering of the vehicle
4. To control the vibration caused by the running of the vehicle
5. To withstand bending stresses due to rise and fall of the front and rear axles.

2.3.1.TYPE OF CHASSSIS:

Chassis is considered to be one of the significant structures of an automobile. Itis usually made of a steel frame, which holds the body and motor of an automotive vehicle. To be precise, car chassis or automobile chassis is a skeletal frame which bolts various mechanical parts like engine, tires, brakes, steering and axle assemblies. Chassis usually made of light a metal or composite plastic which provides strength needed for supporting vehicle components and load into it. Here I listed several different types of automotive chassis which include **ladder chassis, backbone chassis, monocoque chassis and tubular space frame chassis** (Wakeham, 2009). 8 Ladder chassis is considered to be one of the oldest forms of automotive chassis or automobile chassis that is still been used by most of the SUVs till today. It is also resembles a shape of a ladder which having two longitudinal rails inter linked by several lateral and cross braces. The lateral and cross members provide rigidity to the structure (Wakeham, 2009).

Ladder frame chassis

Ladder frame chassis is one of the oldest types of chassis. It is still used today for some off-road and commercial vehicles. Such a structure consists of two long beams that are supported by further shorter beams. It is used for example in the Ford Transit model. This solution can be encountered less and less often due to its considerable weight,among other reasons. The biggest advantages of this type of chassis are the wide possibilities of modification, cost-effective production and relatively simple manufacture.

Figure 2.2 Ladder Frame chassis

2.4 Types Of Chassis Frame

There are different types of chassis frame sections

1. Channel section
2. Box section
3. Tubular section

The conventional frame is also known as Non-load carrying frame. In this types of frame, the loads on the vehicle are transferred to the suspension by the frame which is the main skeleton of the vehicle. The channel section is used in long members and box section in short members. Tubular section is used now-a-days is three wheelers, scooters, matadors and pickup vans. The frames should be strong enough to bear load while sudden brakes and accidents.

2.4.1. Conventional Chassis Or Frame-Full 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.

Figure 2. 2 Convectional Chassis

Advantage

- Higher stack limit and quality

Impediment

- The body has a tendency to vibrate effectively and the general vehicle taking care of and refinement is lower.

It is utilized as a part of truck, transport and in SUV autos and greater vehicles.

2.4.2. Non Conventional Or Frameless Chassis

In this kind of suspension the stepping stool outline is missing and the body itself goes about as the edge. It bolsters every one of the frameworks in a vehicle, for example, the Engine, Transmission framework, Steering framework, Suspension framework.

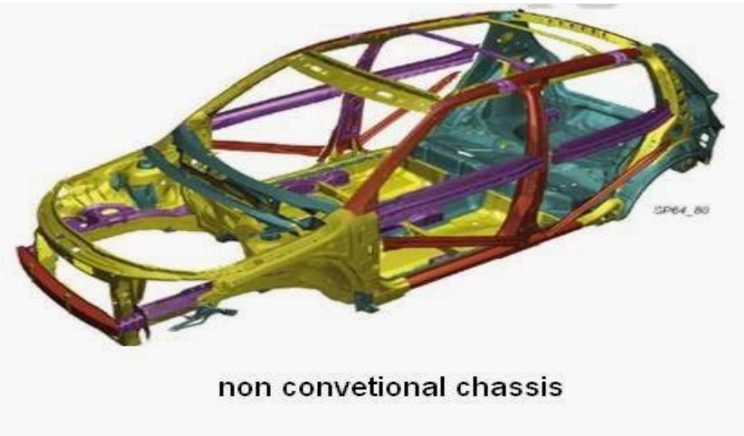

Figure 2. 3 Non Convectional Chassis

Advantages

- Less rattles and squeaks are created.
- Handling is better because of the higher body unbending nature and weight.

Drawback

- The stack conveying limit is lower.
- It is not sheltered in coincidental condition.

2.5. Diverse Types Of Cars According To Body Style

Body of an auto chooses the space accessible for traveler and dialect in the auto. There are different sort of body utilized as a part of Indian market. What's more, they are given beneath.

Figure 2. 4 Type Of Bodies Of Vehicles

- Hatchback: Hatchbacks are vehicles with a different motor region, and traveler territory (or two boxes), the gear zone is encased with the traveler range behind the back seats. Illustration: Nano, Indica, Jazz, Punto and so forth.
- Sedan/Notchback: Sedan are essentially vehicles with a motor territory, traveler zone, and boot zone (or three box), all different. Illustration: Indigo Manza, Swift Dzire and so forth.

- Estate/Station Wagon: Estates or Station wagons are changed cantina vehicles by consolidating the boot with traveler range and broadening it till the rooftop. The boot range is fundamentally bigger and does not have third column seating. This makes it helpful to convey enormous articles. Case: Indigo Marina, Octavia Combi and so forth.
- Multi Purpose Vehicle (MPV)/Multi Utility Vehicle (MUV): MPV (Multi-Purpose Vehicles) or MUV (Multi Utility Vehicles) can have the motor, traveler territory and boot region encased together or they can have the motor zone isolated and the traveler and boot range encased. MUV/MPV can likewise have third line of seating. These vehicles are two wheel drive. Illustration: Sumo Grande, Tata Tavera, Tata Innova and so forth.

- Sport Utility Vehicle (SUV): These vehicles have extensive tires, higher seating, and higher ground freedom. The motor range is independent and the traveler and boot region are encased together. These vehicles are either furnished with 4 wheels drive or has as an alternative of 4 wheel drive. Illustration: Safari, Scorpio, Gypsy, Fortuner.
- Pick-Up Truck: These vehicles have extensive tires, higher seating limit, and higher ground freedom. The motor range is independent and the traveler compartment accessible in single or twofold taxi arrangements. Additionally, gear stacking inlet is accessible behind the traveler compartment. These vehicles are either furnished with 4 wheels drive or has as an alternative of 4 wheel drive. Case: Xenon, Scorpio Getaway and so on.
- Van: The motor is put underneath the traveler range. Vans can likewise have Third column of seating. They are additionally taller and for the most part more open. Illustration: Winger, Ace Magic, Omni and so on.

2.6 Various Loads Acting On The Chassis Frame

The heaps following up on the skeleton outline are as take after

1. Stationary loads in particular the heaps of lasting connection like every one of the parts of the suspension, body and so on.
2. Short span loads while turning , braking and so forth.
3. Momentary burdens while brisk speeding up , sudden braking and so forth.
4. Loads connected while crossing streets of unpredictable and uneven surfaces
5. Loads caused by sudden mishaps, head on intrigues and so on.
6. Loads caused by unpredictable and over-burdening of vehicle.

2.7 Different Bodies Used In Automobiles

The cars bodies are outlined by the necessity of the vehicle. As indicated by outline and necessity of the vehicle , there are distinctive sorts of Automobiles bodies. Some of them are recorded as underneath :

(i) Car

(ii) Straight truck or Punjab truck body

(iii) Truck with half body

(iv) Platform sort truck

(v) Tractor

(vi) Tractor with verbalized trailer

(vii) Tanker

(viii) Bus

(ix) Dumper truck

(x) Delivery van

(xi) Station wagon

(xii) Pick up van

(xiii) Jeep

(xiiv) Long wheel base truck and so on

CHAPTER 3

LITERATURE SURVEY

1) Kenji KARITA, Yoichiro KOHIYAMA, Toshihiko KOBIKI, Kiyoshi OOSHIMA, Mamoru HASHIMOTO (2003) had built up a case made by Aluminum. The material chose for the edge is6061-T6. They utilized the Variable segment expulsion strategy for making the undercarriage. It's created with the assistance of PC Aided Engineering. Aluminum material gives favorable position of weight lessening.

From this examination creators found that the Aluminum suspension meets the objective of weight lessening, quality and inflexibility. Likewise they reasoned that the staying specialized issues will be routed to empower business selection of the aluminum frame.Alireza Arab Solghar, Zeinab Arsalanloo (2013) contemplated and dissected the undercarriage of Hyundai Cruz Minibus. ABAQUS Software was utilized for displaying and reproduction. Self weight of the case is considered for static examination and Acceleration, Braking and Road Roughness were considered for dynamic investigation. It's watched that the weights on frame caused by braking were more contrasted and speeding up.

2)M. Ravichandra, S. Srinivasalu, Syed altaf Hussain (2012) considered the substitute material for skeleton. They considered and investigated Carbon/Epoxy, E glass/Epoxy and S-glass/Epoxy as skeleton material invarious cross areas like C, I and Box Section. Goodbye 2515 EX case was taken for think about. Genius Eand Ansys programming were utilized for this work. Study uncovers that the Carbon/Epoxy I segment undercarriage has predominant quality, firmness and lesser weightcompared to different materials and cross segment. Roslan Abd Rahman, Mohd Nasir Tamin,Ojo Kurdi (2008) utilized FEM stretch investigation as a preparatory information for weakness life forecast. They utilized ABAQUS programming for recreation and examination and furthermore taken ASTM Low Alloy steel A710 (C) for ponder. Essential target was to locate the high focused on region where the Fatigue Failure will begin. It's discovered that the frame opening region having contact with jolt encounters high anxiety.

3)N.V.Dhandapani, G Mohan kumar, K.K.Debnath (2012) have utilized Finite component strategies to contemplate the impact of different anxiety conveyance utilizing Ansys programming. To examine the field disappointment of 100Ton dumper they presented gussets in disappointment territory. After adjustment the frame structure was approved by straight static investigation and found that the changed body was protected.

4)Teo Han Fui, Roslan Abd. Rahman (2007)have concentrated the 4.5 Ton truck skeleton against street harshness and excitations. Vibration prompted by street Roughness and excitation by the vibrating parts mounted on case were contemplated. Skeleton reactions were analyzed by push dispersion and relocations. Mode shape comes about decide the appropriate mounting areas of parts like motor and suspension frameworks. Investigation comes about uncover that the street excitation was a primary aggravation to the body.

5)S.S Sane, Ghanashyam Jadhav, H.Ananadaraj (2008) investigated the light Commercial Vehicle case utilizing FEM and mimicked the disappointment amid testing. Hyper work and Opti-struct programming were utilized for investigation and reenactment. Amid the investigation they acquainted nearby stiffeners with diminish the greatness of the anxiety. The changed undercarriage stretch esteems were lessened by 44%.

6)Vijay kumar V. Patel, R. I. Patel (2012) have contemplated the Ladder undercarriage edge of Eicher E2 by static basic investigation. For this examination case was expected as essentially upheld shaft with overhang. Professional E and Ansys programming were utilized for this work. The examination additionally included the diagnostic estimation of case. Both programming examination and investigative figuring comes about were thought about and found that the anxiety esteem acquired from programming investigation is 10%more and furthermore removal was 5.92% more.

7)Kutay Yilmazcoban, Yasar Kahraman (2011) have contemplated and upgraded the thickness of a center tonnage truck case by utilizing Finite Element strategy. The principle target of this work was to diminish the material utilization through that picking up decrease in material cost. They had examined three sorts of thickness material to case and looked at the outcomes by stress and relocation. Study uncovers that the 4mm thickness is sufficiently sheltered to convey 15tonload.

8)N.K.Ingole, D.V. Bhope (2011) investigated the Tractor Trailer made by Awachat Industries, Wardhato decrease the assembling cost. Four distinctive adjusted outlines of trailer were made in Pro-E programming and broke down utilizing Ansys programming. Looking at the aftereffects of four distinctive composed trailer undercarriage, fourth plan was an ideal outline in view of weight. It's proposed that the fourth plan trailer frame was reasonable for large scale manufacturing and furthermore financially savvy.

9)V. Veloso, H.S. Magalhaes, G.I. Bicalho, E.S. Palma (2009) considered the Failure of longitudinal stringer of vehicle. Disappointment saw at close guards obsession purposes of the vehicle suspension amid sturdiness test. Starting Crack was made and has developed causing break of the segment. To conquer this issue they explored six distinct sorts of fortification. Each of the six sorts of support techniques were examined utilizing hyper work programming and results were affirmed that the 6th kind of fortification gave better outcomes. In light of programming comes about research center test had directed and the disappointment had not watched. Utilizing the product examination they disposed of no of research center tests and accomplished better outcomes in shorter time traverse. In this way takes out the real level of testing costs.

10)Yongjie Lu, Shaopu Yang, Shaohua Li,Liqun Chen (2009) have created virtual model of overwhelming obligation vehicle (DFL1250A9) usingmulti body flow. The geometric auxiliary parameters and nonlinear qualities of safeguard and leaf springs were unequivocally characterized. The dynamic model was approved by looking at the testing information. The examination uncovers that the virtual model vehicle demonstrate and the testing information's were close and furthermore Increase of running velocity may cause harm.

11)Ojo Kurdi, Roslan Abdul Rahman (2010)studied the street unpleasantness consequences for push dissemination of overwhelming vehicle suspension. They had dissected Static and Dynamic conditions utilizing Finite Element Analysis programming to diminish the cost and get ideal plan. The heap was expected as a uniform pressure got from the greatest stacked weight partitioned by the aggregate contact zone amongst payload and upper surface of skeleton. Keeping in mind the end goal to show signs of improvement result, locally better lattice was connected in the district which was suspected to have most astounding anxiety. From ponder it's comprehended that the predominant stacking on the truck body originates from payload and its substance as static stacking. The street harshness has not given a huge impact to the worry of part.

12)Alireza Arab Solghar, Zeinab Arsalanloo(2013) contemplated and Analyzed the suspension of Hyundai Cruz Minibus. ABAQUS Software was accustomed to displaying and recreation. Self weight of the suspension was considered for static examination and Acceleration, Braking and Road Roughness were considered for dynamic investigation. It's watched that the weights on frame caused by braking were more contrasted and increasing speed.

13)Cicek Karao, glu, N. Sefa Kuralay (2001),have investigated truck skeleton with bolted joints utilizing FEM. The business limited component bundle ANSYS variant 11.0 was utilized for the arrangement of the issue. Keeping in mind the end goal to accomplish a lessening in the greatness of worry close to the bolted joint of the undercarriage outline, side part thickness, association plate thickness and association plate length were shifted. Numerical outcomes demonstrated that weights as an afterthought part can be decreased by expanding the side part thickness.

-the auto body in a games utility vehicle (SUV). The anxiety circulation under unit dislodging excitation was acquired by the limited component (FE) strategy. A two-sided track display was embraced to get stack spectra as per the vehicle unwavering quality test. The aggregate existence of the auto body was assessed by the ostensible anxiety technique with the supposition of a uni-pivotal anxiety state, and therefore the basic locales were resolved. The life of parts with basic locales was additionally examined on the premise of multi hub weariness hypothesis. The outcomes demonstrated that segments close suspension were harmed because of effect loads from the street. Topological enhancement of the spot weld area in the basic district was done by the homogenization technique to enhance its weariness life.

15)K. Chinnaraj ,M. Sathya Prasad, Lakshmana Rao (2008) have streamlined the heaviness of skeleton outline get together of a substantial truck utilized for long separation products pulling application. Dynamic anxiety strain reaction of the part because of braking and cornering moves were tentatively measured and announced. A semi static approach that approximates the dynamic moves into number of little procedures having static harmonies was taken after to do the numerical reproduction, approximating the dynamic conduct of edge rail get together amid cornering and braking. Utilizing the business limited component bundle

CHAPTER-4

INTRODUCTION OF SOLIDWORKS SOFTWARE

Solid Works is a 3D solid modeling package which allows users to develop full solid models in a simulated environment for both design and analysis. In Solid Works; you sketch ideas and experiment with different designs to create 3D models. Solid Works is used by students, designers, engineers, and other professionals to produce simple and complex parts, assemblies, and drawings. Designing in a modeling package such as Solid Works is beneficial because it saves time, effort, and money that would otherwise be spent prototyping the design.

4.1 Solid Works Components - Parts

Before we begin looking at the software, it is important to understand the different components that make up a Solid Works model.

Part:

- The first and most basic element of a Solid Works model is a Part.
- Parts consist of primitive geometry and feature such as extrudes, revolutions, lofts, sweeps, etc.
- Parts will be the building blocks for all of the models that you will create

Assembly:

- The second component is the assembly. Assemblies are collections of parts which are assembled in a particular fashion using mates (constraints).
- Any complex model will usually consist of one, or many assemblies.

Drawing:

- The third and final component in Solid Works is the Drawing.
- A drawing is the typical way to represent a 3D model such that any engineer (or manufacturer) can recreate your part.
- Drawings are important because they provide a standard way of sharing your design.

4.2 Solid Works Software – Let's Begin

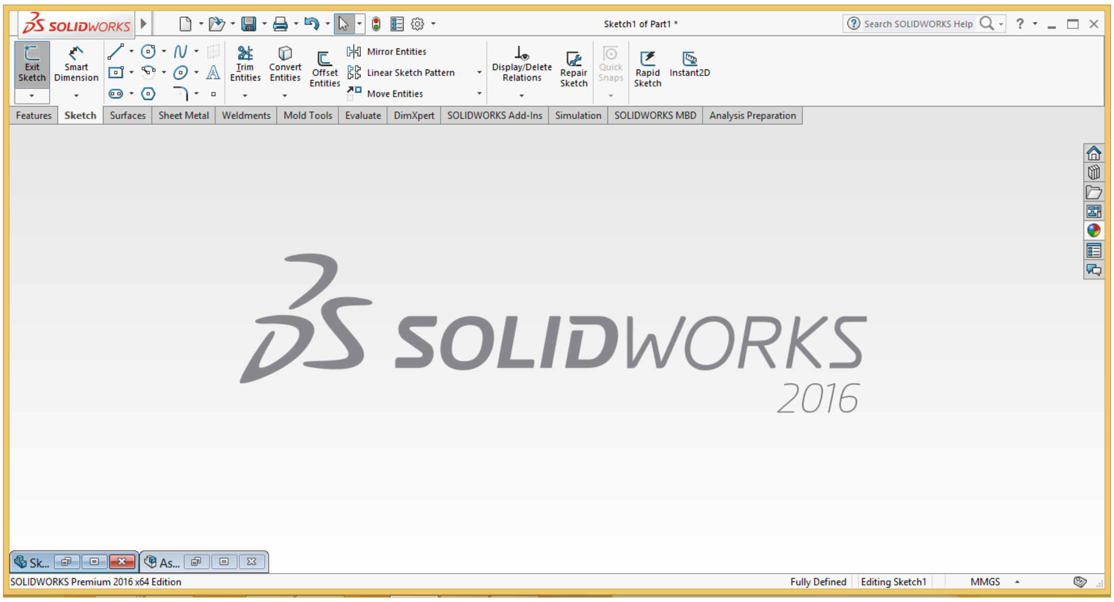

Figure 4.1 Solid Works Default Page

- By default, no file is opened automatically when you start the program.
- To create a new file, click on File - New or click the New File icon in the main toolbar.
- This will open the New Solid Works document wizard.

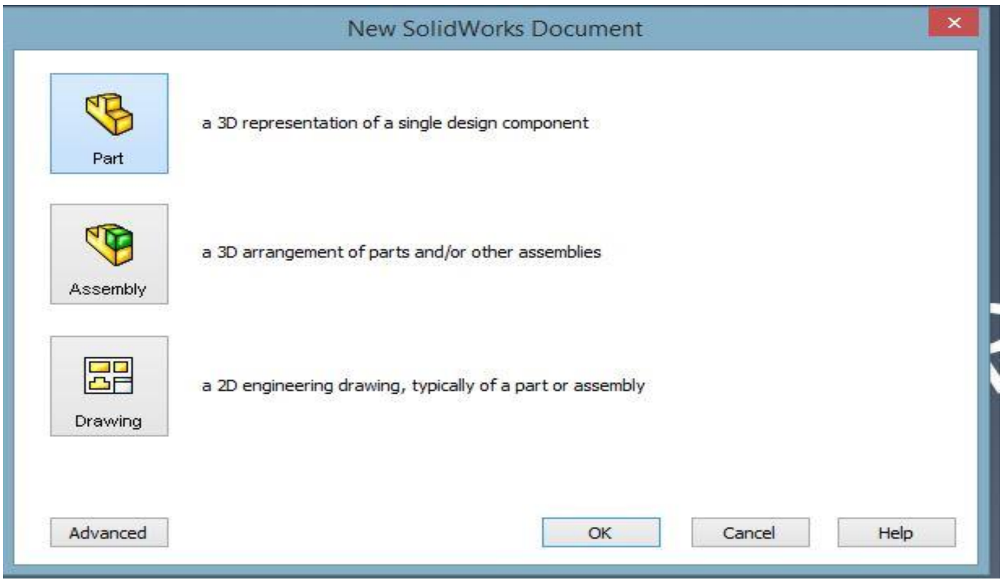

Figure.4.2 Parts Of Solid Works

- Let's begin by creating a new part.

- To do this, click on Part, then OK. Once you do this, you will be brought into the modeling view which should open several toolbars and panes.

4.3. Terminology

These terms appear throughout the Solid Works software and documentation.

- **Origin:** Appears as two blue arrows and represents the (0, 0, 0) coordinate of the model. When a sketch is active, a sketch origin appears in red and represents the (0, 0, 0) coordinate of the sketch. You can add dimensions and relations to a model origin, but not to a sketch origin.
- **Plane:** Flat construction geometry. You can use planes for adding a 2D sketch, section view of a model, or a neutral plane in a draft feature, for example.
- **Axis:** Straight line used to create model geometry, features, or patterns. You can create an axis in different ways, including intersecting two planes. The Solid Works application creates temporary axes implicitly for every conical or cylindrical face in a model.
- **Face:** Boundaries that help define the shape of a model or a surface. A face is a selectable area (planar or non planar) of a model or surface. For example, a rectangular solid has six faces.
- **Edge**: Location where two or more faces intersect and are joined together. You can select edges for sketching and dimensioning.
- **Vertex**: Point at which two or more lines or edges intersect. You can select vertices for sketching and dimensioning.

4.4. User Interface

The Solid Works application includes user interface tools and capabilities to help you create and edit models efficiently, including:

- **Windows Functions:** The Solid Works application includes familiar Windows functions, such as dragging and resizing windows. Many of the same icons, such as print, open, save, cut, and paste are also part of the Solid Works application.

4.4.1. Solid Works Document Windows:

Solid Works document windows have two panels. The left panel, or Manager Pane, contains:

4.4.2. Feature Manager /Design Tree

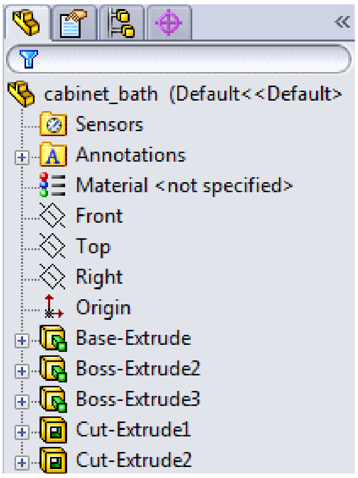

Figure 4.3 Specification Tree

Displays the structure of the part, assembly, or drawing. Select an item from the Feature Manager Design tree to edit the underlying sketch, edit the feature, and suppress and un suppress the feature or component, for example.

- **Property Manager:** Provides settings for many functions such as sketches, fillet features, and assembly mates.

Figure 4.4 Extrude Tool Box

- **Configuration Manager:** Lets you create, select, and view multiple configurations of parts and assemblies in a document. Configurations are variations of a part or assembly within a single document. For example, you can use configurations of a bolt to specify different lengths and diameters.

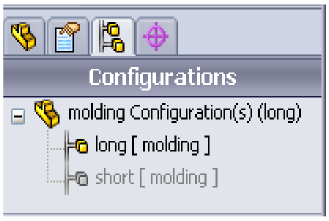

Figure 4.5 Configuration Box

4.4.3. Mouse Buttons:

Mouse buttons operate in the following ways:

- **Left:** Selects menu items, entities in the graphics area, and objects in the Feature Manager Design tree.
- **Right:** Displays the context-sensitive shortcut menus.
- **Middle:** Rotates, pans, and zooms a part or an assembly, and pans in a drawing.
- **Mouse gestures:** You can use a mouse gesture as a shortcut to execute a command, similar to a keyboard shortcut. Once you learn command mappings, you can use mouse gestures to invoke mapped commands quickly. To activate a mouse gesture, from the graphics area, right-drag in the gesture direction that corresponds to the command. When you right-drag, a guide appears, showing the command mappings for the gesture directions.

An assembly is a combination of two or more parts, also called components, within one SOLIDWORKS document. You position and orient components using mates that form relations between components.

This lesson discusses the following:

- Adding parts to an assembly
- Moving and rotating components in an assembly
- Creating display states in an assembly

4.5. Modeling

To create this part body go to solid works software and click on file new and select part design module. In part design module then go to sketcher tool and select sketcher tool and select required plane.

In Solid works, the extrude tool is used to create 3D objects from 2D shapes. To use the extrude tool, you first need to create a 2D sketch of the shape you want to extrude. Then, you can select the extrude tool from the toolbar and use it to specify the height and direction of the extrusion. The resulting 3D object will be a solid body with the same shape as the original 2D sketch.

4.5.1 Step for Extrude in Solid Works

1. Create a new document in Solid works and switch to the sketch mode.
2. Use the drawing tools to create a 2D sketch of the shape you want to extrude.
3. Once your sketch is complete, select the extrude tool from the toolbar.
4. In the extrude dialogue box, specify the height and direction of the extrusion.
5. Click "OK" to create the extruded 3D object.
6. Note: It is important to make sure your sketch is fully defined and closed before using the extrude tool, as this will ensure that the resulting 3D object is a solid body.

4.5.2 Step for Revolve in Solid Works

Open SolidWorks and create a new part or open an existing part.

1. Select the "Revolve" tool from the "Feature" menu or from the "Features" tab in the "Command Manager."
2. In the "Property Manager," select the "Profile" to revolve. This can be a sketch or a pre-existing feature such as a curve or a surface.
3. Define the axis of revolution by selecting a sketch line, a reference plane, or a 3D axis.
4. Set the "Angle of Revolution" by entering a value or by selecting two points to define the angle.
5. Set any additional options, such as "Reverse Direction," "Midplane," or "Thin Feature."
6. Click "OK" to create the revolved feature.
7. Use the "Rebuild" tool to update the model and ensure that the revolved feature has been created correctly.
8. Save the part.

4.5.3 Step for Fillet and Chamfer in Solid Works

To create a fillet or chamfer feature in SolidWorks, follow these steps:

1. Open SolidWorks and create a new part or open an existing part.
2. Select the "Fillet" or "Chamfer" tool from the "Feature" menu or from the "Features" tab in the "Command Manager."
3. In the "Property Manager," select the edges or faces to be filleted or chamfered.
4. Set the fillet or chamfer dimensions by entering a value or using the handles in the graphics window.
5. Set any additional options, such as "Type," "Trim," or "Propagation."
6. Click "OK" to create the fillet or chamfer feature.
7. Use the "Rebuild" tool to update the model and ensure that the fillet or chamfer has been created correctly.
8. Save the part.

4.5.4 Step for Pattern in Solid Works

Open SolidWorks and create a new part or open an existing part.

1. Select the "Pattern" tool from the "Feature" menu or from the "Features" tab in the "Command Manager."
2. In the "Property Manager," select the entities to be patterned.
3. Set the pattern options, such as the "Type," "Direction," and "Number of Instances."
4. Set any additional options, such as "Spacing" or "Alignment."
5. Click "OK" to create the pattern feature.
6. Use the "Rebuild" tool to update the model and ensure that the pattern has been created correctly.
7. Save the part.

CHAPTER 5.

DESIGN OF CHASSIS FOR HEAVY WHEELER

CATIA is the main answer for item achievement. It tends to all assembling associations. CATIA can be connected to a wide assortment of businesses from aviation car and modern hardware to gadgets shipbuilding-plant outline and shopper products. Today-CATIA is utilized to plan anything from a plane to gems and apparel. With the power and useful range to address the entire item advancement process-CATIA underpins item building from beginning determination to item in-benefit in a completely incorporated way. It encourages reuse of item outline information and abbreviates advancement cycles-helping undertakings to quicken their reaction to advertise needs.

CatiaV5R20 is an intuitive Computer-Aided Design and Computer Aided Manufacturing framework. The CAD capacities computerize the typical building outline and drafting abilities found in the present assembling organizations. The CAM capacities give NC programming to present day machine instruments utilizing the CatiaV5 R16 configuration model to portray the completed part. CatiaV5R20 capacities are separated into "applications" of regular abilities. These applications are bolstered by an essential application called "CatiaV5R20 Gateway".

CatiaV5R20 is completely three dimensional-twofold exactness framework that permits to precisely portraying any geometric shape. By consolidating these shapes-one can configuration examine and make illustrations of items.

5.1.1. Fundamental Procedure For Creating A 3-D Model In Catiav5r20:

Production of a 3-D demonstrate in CatiaV5R20 can be performed utilizing three workbenches i.e.- sketcher-displaying and get together.

5.1.2.Sketcher:

Sketcher is utilized two-dimensional portrayals of profiles related inside the part. We can make an unpleasant framework of bends and afterward determine conditions called imperatives to characterize the shapes all the more decisively and catch our plan goal. Each bend is alluded to as a draw question.

5.1.3. Making A New Sketch:

Another portray picked StartàMechanical DesignàSketcher at that point select the reference plane or outline plane in which the draw is to be made.

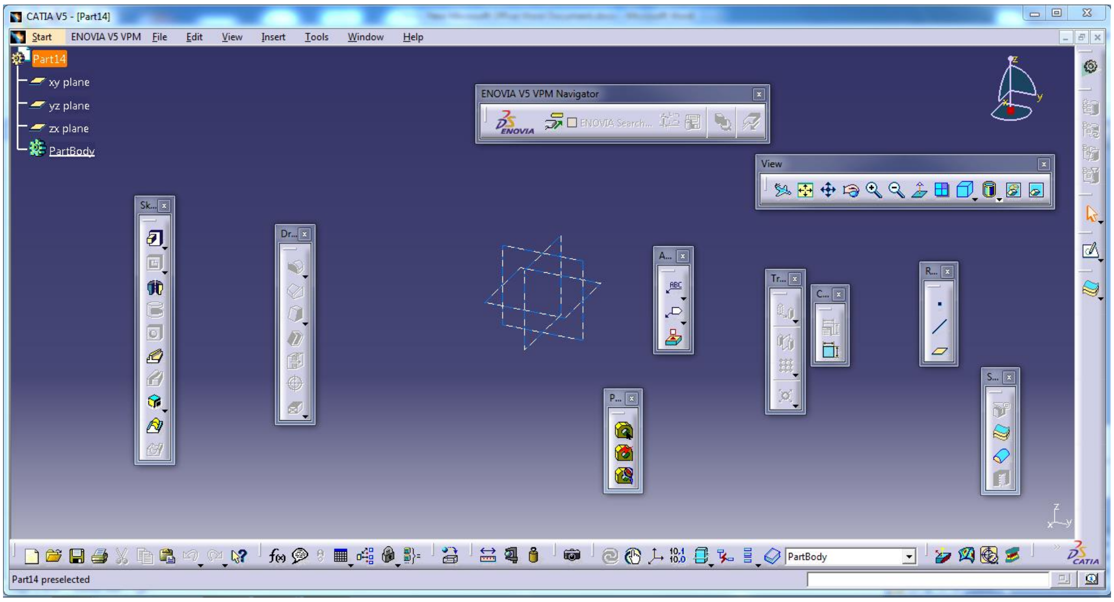

Figure 5.6 Part Design Module

5.2. Modeling Of Chassis

Go to catia software and click on start – mechanical design and select wireframe and surface design module. In wireframe and surface design module then 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 as shown in fig.

After that go to workbench and select exit workbench. Then we enter into wireframe and surface design. Now go to sketcher tool and select sketcher tool and select yz plane. Then we enter into sketcher module and go to profile and create a profile as shown in fig.

After that go to workbench and select exit workbench. Then we enter into wireframe and surface design. Now go to surface based tools and select extrude. On extrude surface definition select second sketch as a profile and specify dimension as shown in fig.

After that go to workbench and select exit workbench. Then we enter into wireframe and surface design. Now go to surface based tools and select extrude. On extrude surface definition select second sketch as a profile and specify dimension as shown in fig.

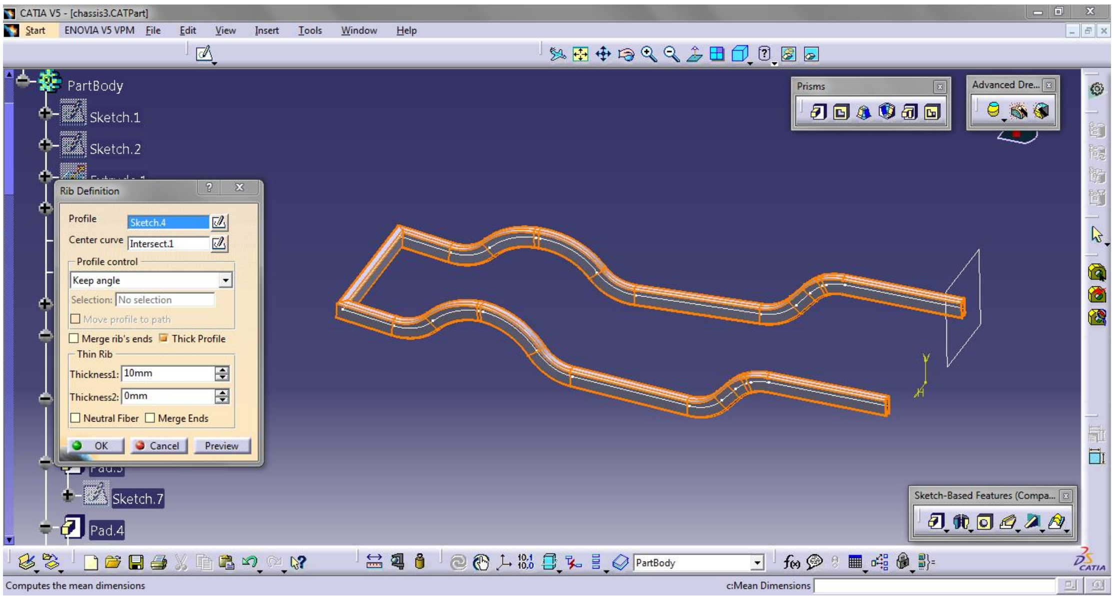

Figure 5.7 Used Rib Tool For The Side Rails In Chassis

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

After that, select above extrudes and right click-click hide/shown. Go to catia software and click on start – mechanical design and select part design module. In part design module then go to reference element and select plane on plane definition, select below intersection as a curve and select one of end point as a point tool as shown in fig.

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

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 intersection as a curve path as shown in fig

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

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 pad tool. In pad definition, select above sketch as selection profile and specify up to next on first and second limit as shown in fig.

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

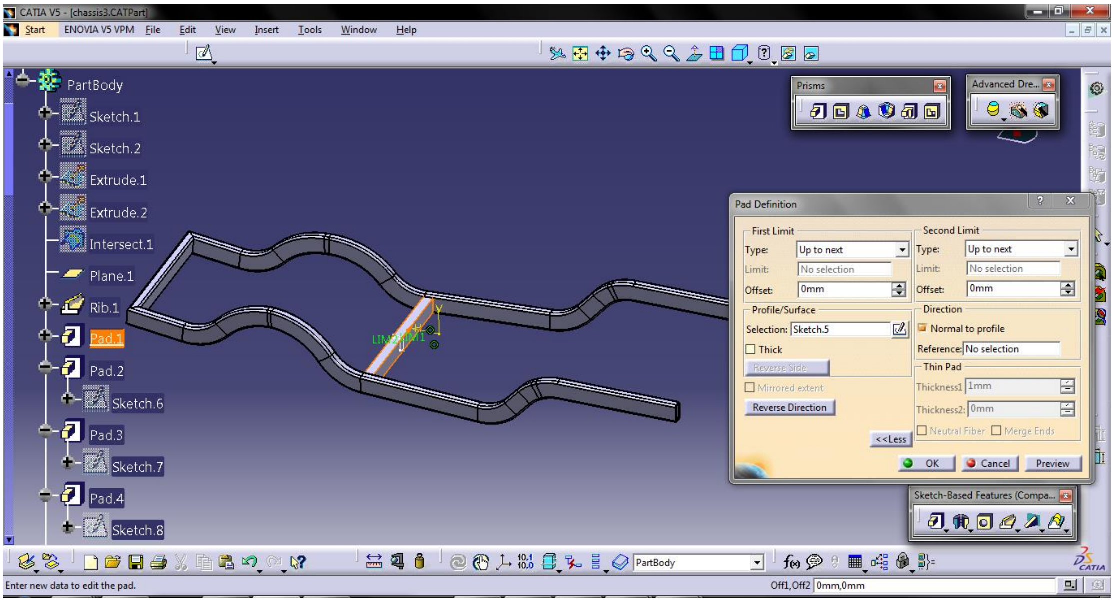

Figure 5.8 Used Pad Tool For The Side Rails In Chassis

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 pad tool. In pad definition, select above sketch as selection profile and specify up to next on first and second limit as shown in fig.

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

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 pad tool. In pad definition, select above sketch as selection profile and specify up to next on first and second limit as shown in fig.

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

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 pad tool. In pad definition, select above sketch as selection profile and specify up to plane and select bottom surface as shown in fig.

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 fig .

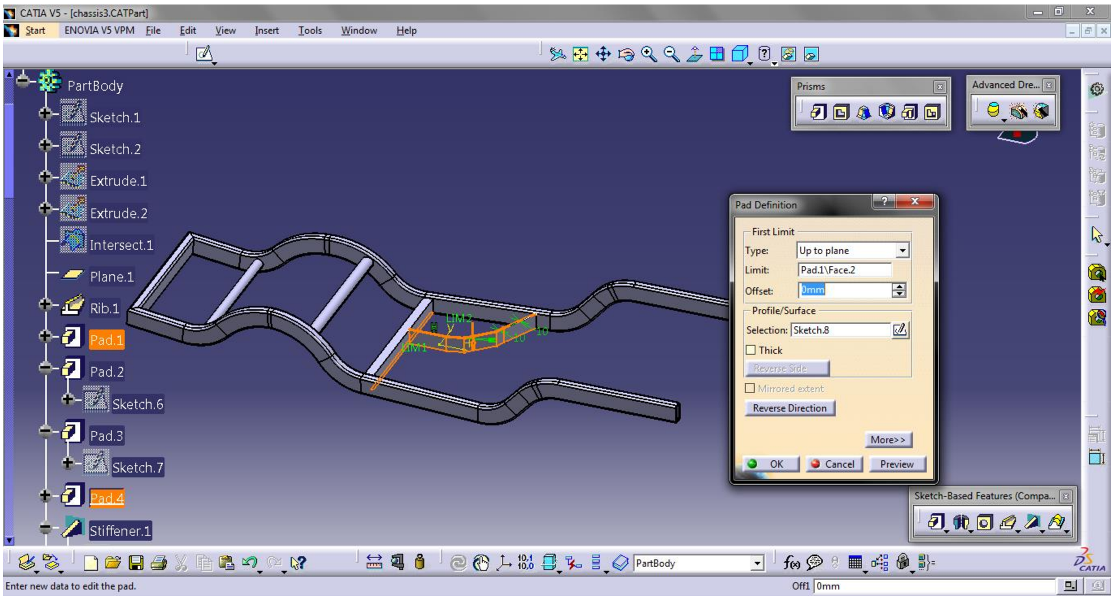

Figure 5.9 Used Pad And Pocket Tool For The Side Rails In Chassis

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.

In part module, go to dress up based feature and select edge fillet tool. In edge fillet definition, select the corners where we need fillet and specify fillet radius as a 2 cm as shown in fig.

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 fig.

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 fig.

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 fig

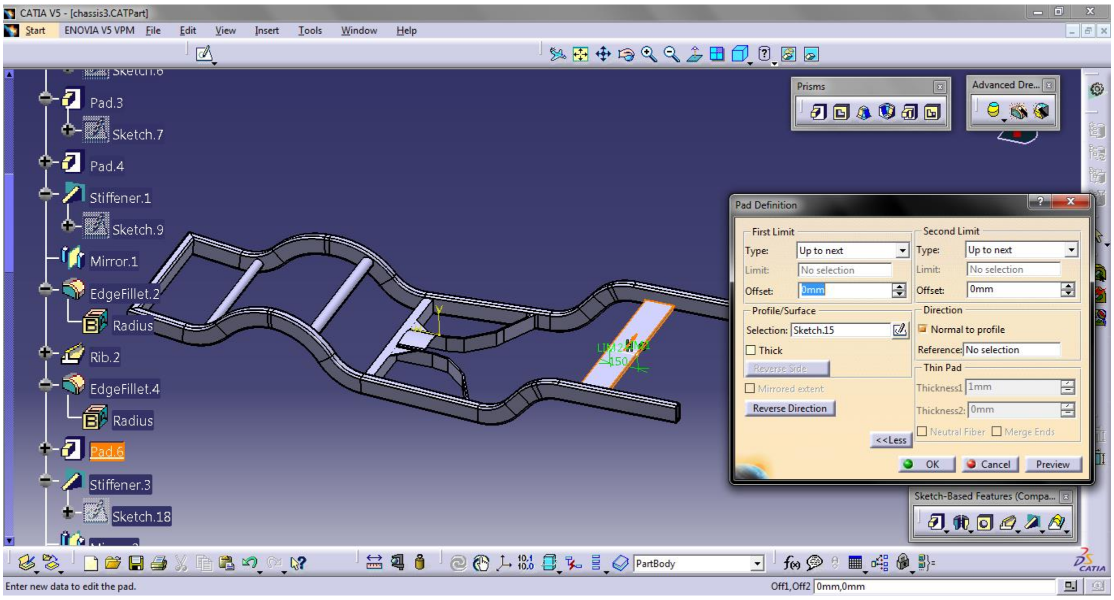

Figure 5.10 Used Pad Tool For The Side Rails In Chassis

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 fig

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 pad tool.

In pad definition, select above sketch as selection profile and specify up to next on first and second limit as shown in fig.

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 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.

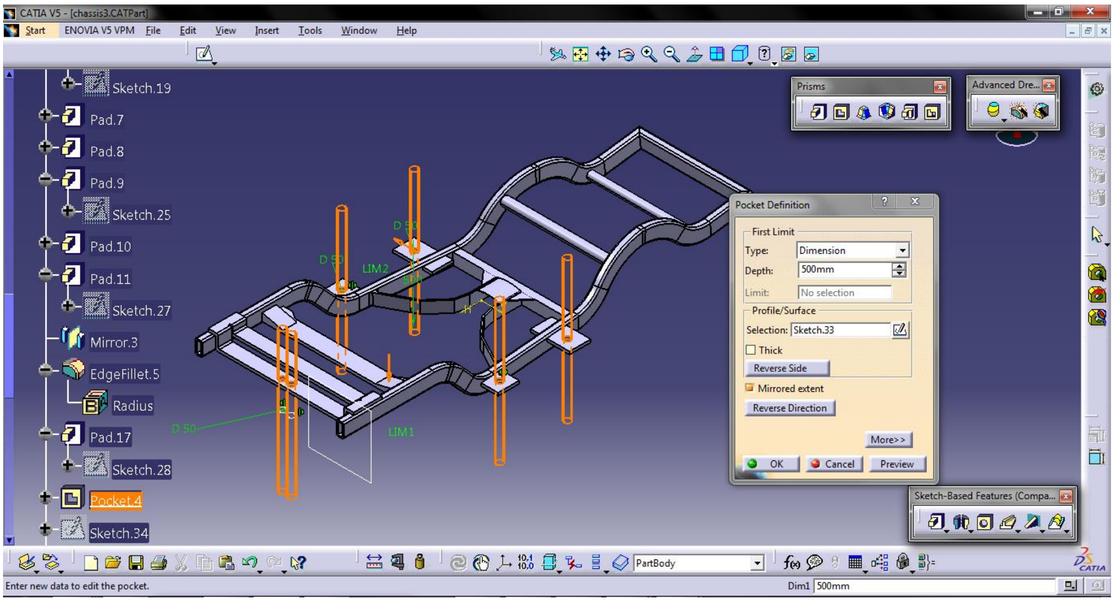

Figure 5.11 Used Pocket Tool For The Side Rails In Chassis

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 fig

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 pocket tool. In pocket definition, select above sketch as selection profile and specify up to last on first as shown in fig.

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 fig

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 pad tool. In pad definition, select above sketch as selection profile and specify 2 mm in length as shown in fig.

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 fig

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 pad tool. In pad definition, select above sketch as selection profile and specify 2 mm in length as shown in fig.

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 fig

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 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.

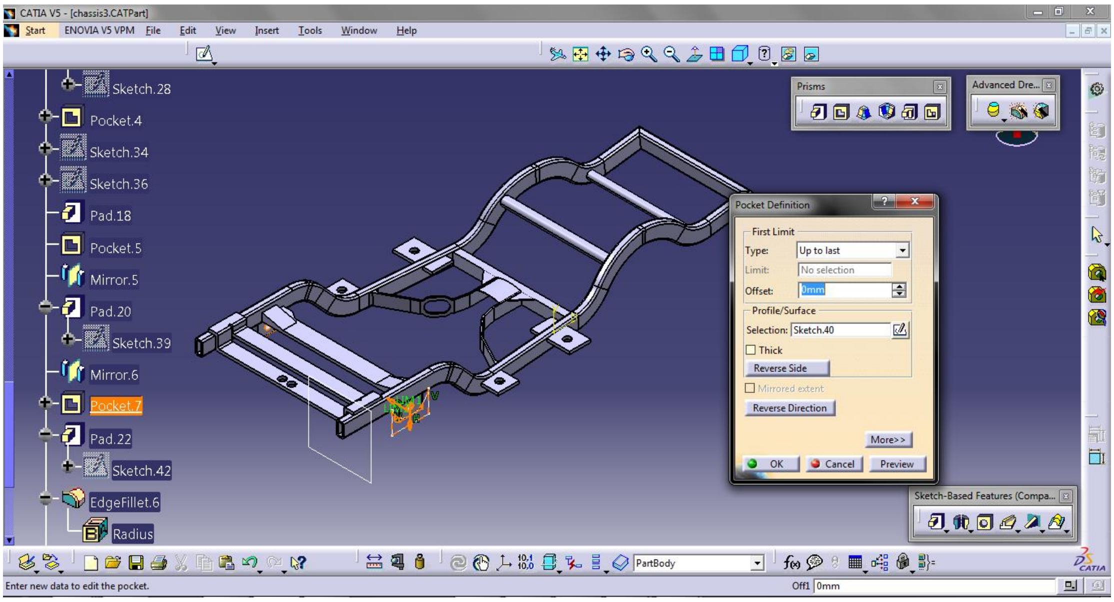

Figure 5.12 Used Pocket Tool For The Side Rails In Chassis

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 fig

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 pad tool. In pad definition, select above sketch as selection profile and specify 2mm in length as shown in fig

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 fig

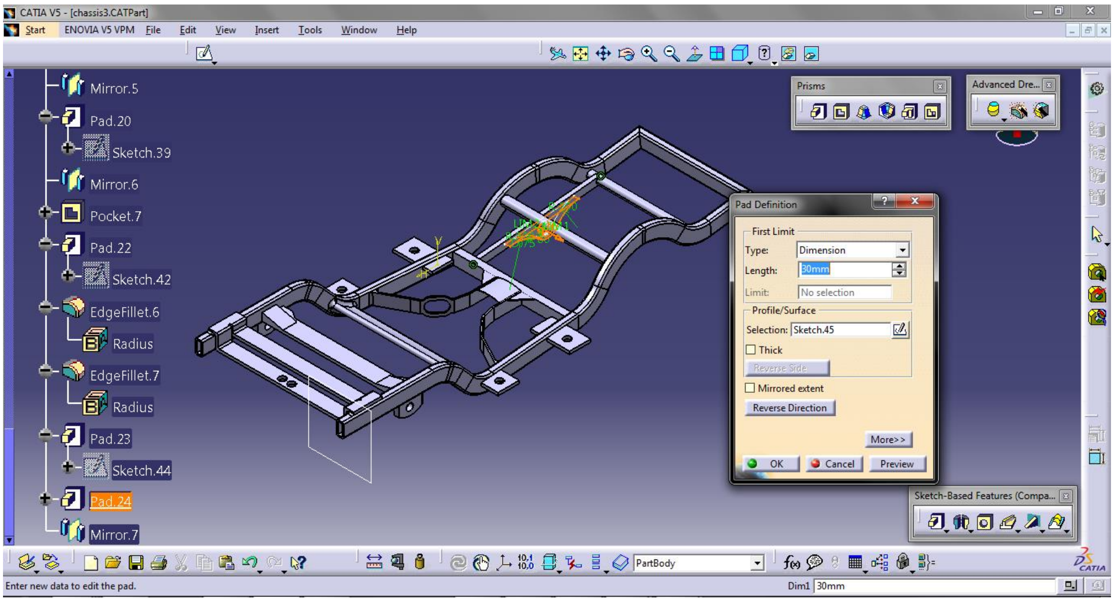

Figure 5.13 Used Pockets Tool For The Side Rails In Chassis

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 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.

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

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 pocket tool. In pocket definition, select above sketch as selection profile and specify depth 50 cm as shown in fig.

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

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

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 pad tool. In pad definition, select above sketch as selection profile and specify length 1cm as shown in fig.

Now in part module, go to sketcher based feature and select pocket tool. In pocket definition, select above sketch as selection profile and specify depth 1cm 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.

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

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 pad tool. In pad definition, select above sketch as selection profile and specify length 10 cm 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.

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

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 pocket tool. In pocket definition, select above sketch as selection profile and specify up to last in type cm as shown in fig.

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

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 pad tool. In pad definition, select above sketch as selection profile and specify up to next on first and second limit as shown in fig.

In part module, go to dress up based feature and select edge fillet tool. In edge fillet definition, select the corners where we need fillet and specify fillet radius as a 2 cm as shown in fig.

In part module, go to dress up based feature and select edge fillet tool. In edge fillet definition, select the corners where we need fillet and specify fillet radius as a 2 cm as shown in fig.

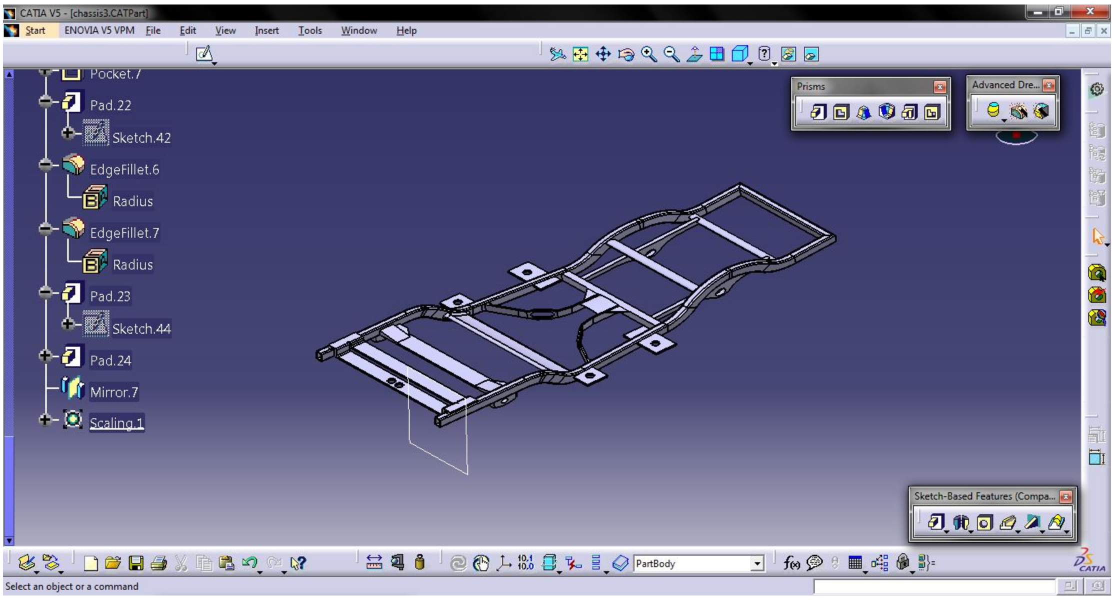

Figure 5.14 Final Shape Of Chassis On Catia

5.3 3D Printing Slicer

- A 3D slicer is a 3D printer software program that you can import models into — for example in .STL, .GCODE or .OBJ file format. These 3D printer models form the basis of your print; they contain the edges, heights and all the information about your print. For all software for 3D printers beyond just 3D slicers, check out our ranking of the best 3D printing software. To be considered one of the best 3D slicers, a slicer should offer:
- Speed: Any great 3D slicer should save you time by importing STL files (or any other 3D printer file) in record time, no matter how large or complex the model is.
- Repairing and warning features: An advanced 3D printer slicer will alert you if your file there are glaring errors in your file that could cause errors in your print, and fix them for you if necessary. This saves you time in preventing wasteful failed prints, and money in wasted 3D printer filament.
- Easy to use: There is no point having infinitely advanced slicing tools (or any other great features) if nobody can find them, or work out how to use them. The interface should be simple and easy to learn, with appropriate guides for professional 3D slicing tools.

- 3D printing data: Most good slicers now, during printing, inform you of the expected remaining time, and how much required filament remains (so you can estimate if you have enough left).
- 3D slicer price: many tools are free, and even open source, so if a program if paid, it needs to offer notable advantages of its free slicer peers.
- Good slicer supports: one of the main advantages of a good 3D slicer is automatically generated supports which make sure your finished print is as precise and accurate as possible, especially over bridges and wide angle 3D printing.

5.4. Creality Slicer

Creality Slicer is a program released by the software company Creality3D. Some people want to remove this application. Sometimes this can be troublesome because uninstalling this manually requires some skill regarding Windows internal functioning. One of the best QUICK procedures to remove Creality Slicer is to use Advanced Uninstaller PRO.

To generate 3d print, go to Creality slicer software. On the top we can see folder option, the click on it a browser will pop up – select the modal “pelton”. Once the modal is placed on Creality slicer, and then select download option at the top. Show that modal can be sliced by using layer method.

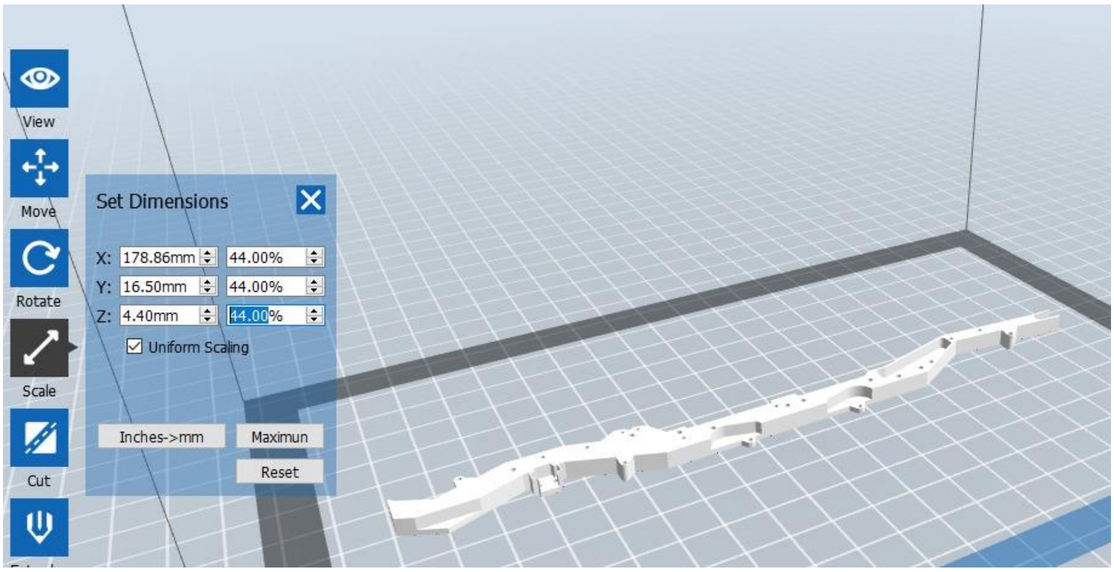

Figure 5.10 Dimensions of Overall Slicing

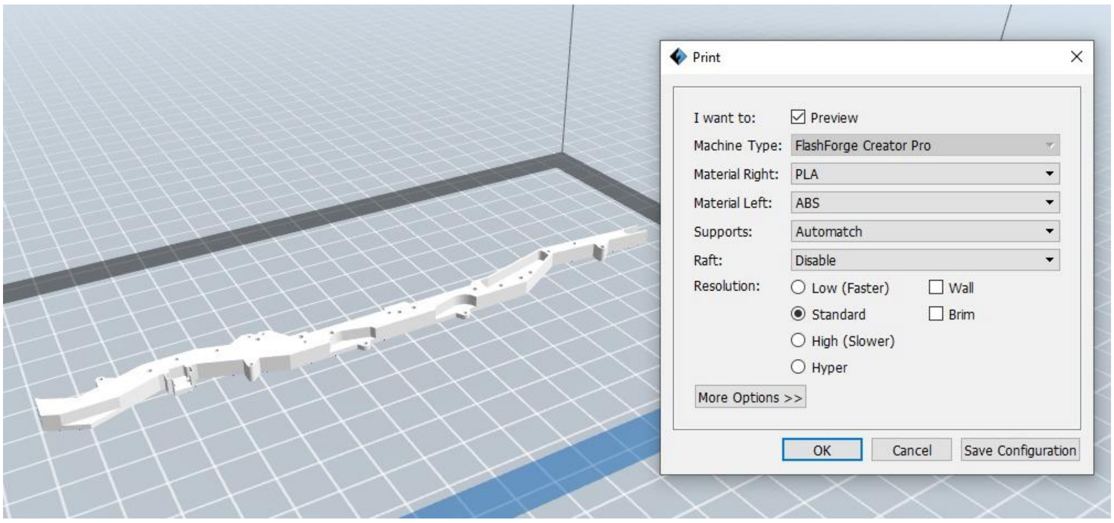

Figure 5.11.PLA material with standard quality printing

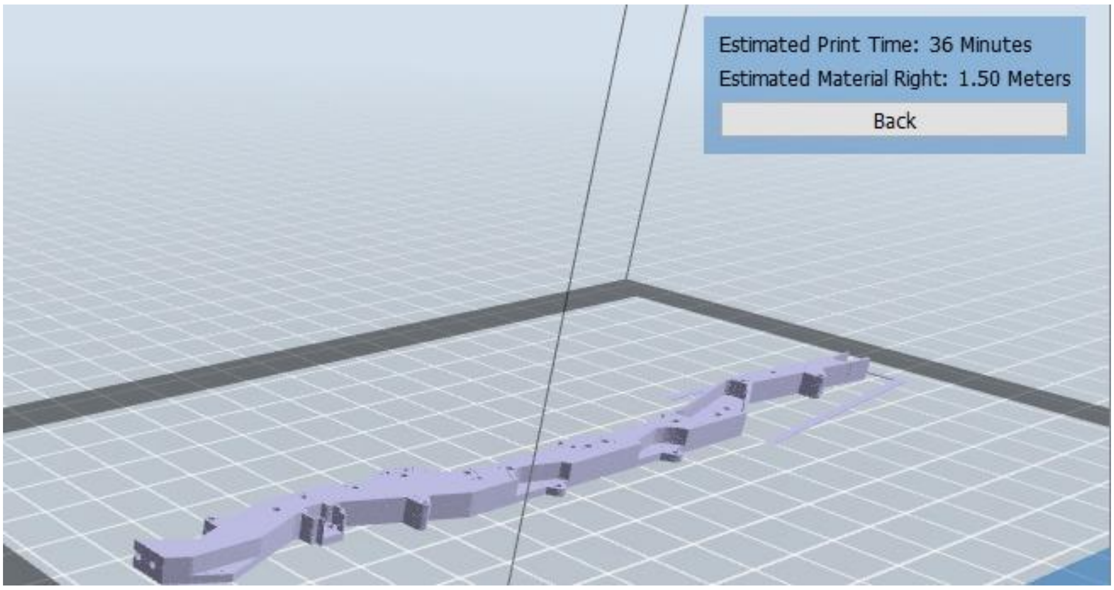

Figure 5.12.Shell 1

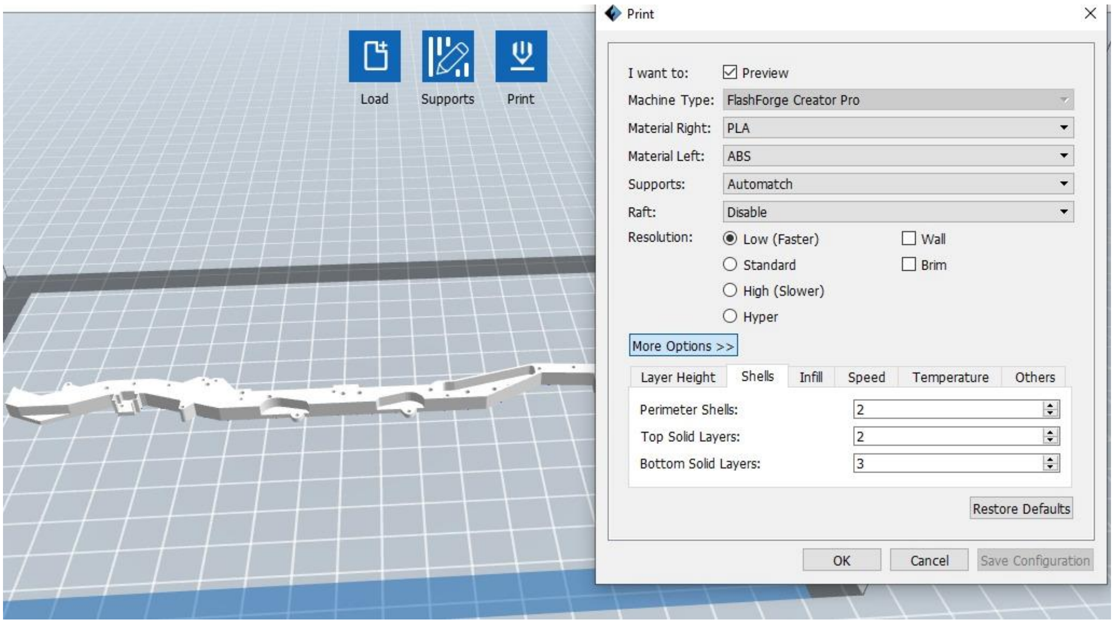

Figure 5.13.Shell-2

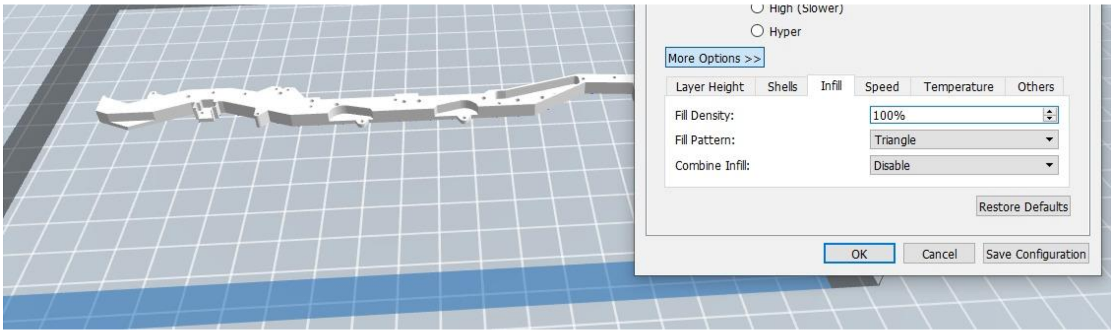

Figure 5.14.Fill density is 100%

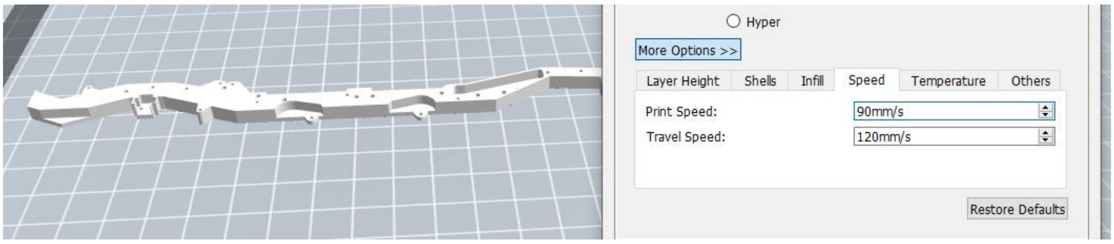

Figure.5.15.Speed

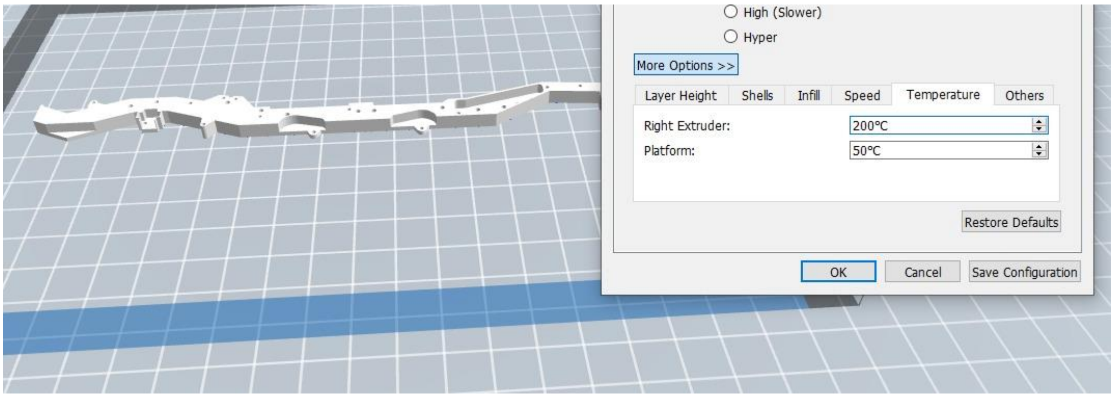

Figure.5.16.Temperature

CHAPTER-6

DESIGN PROCEDURE FOR VEHICLE CHASIS

- Select front View and draw front sketch with dimension and angle

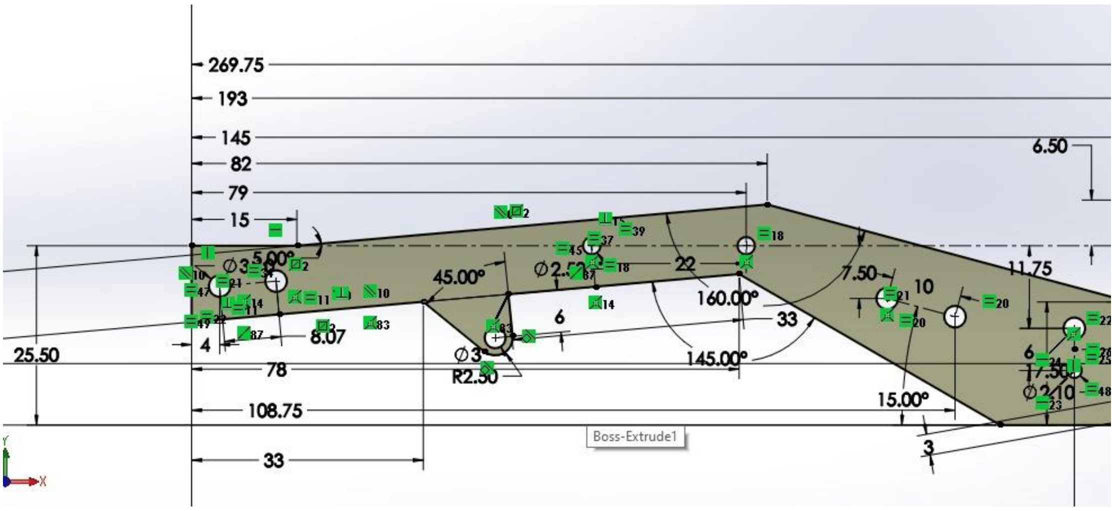

Figure 6.1.Dimension and Angle

- draw support bottom part with leaf suspension shaft holder

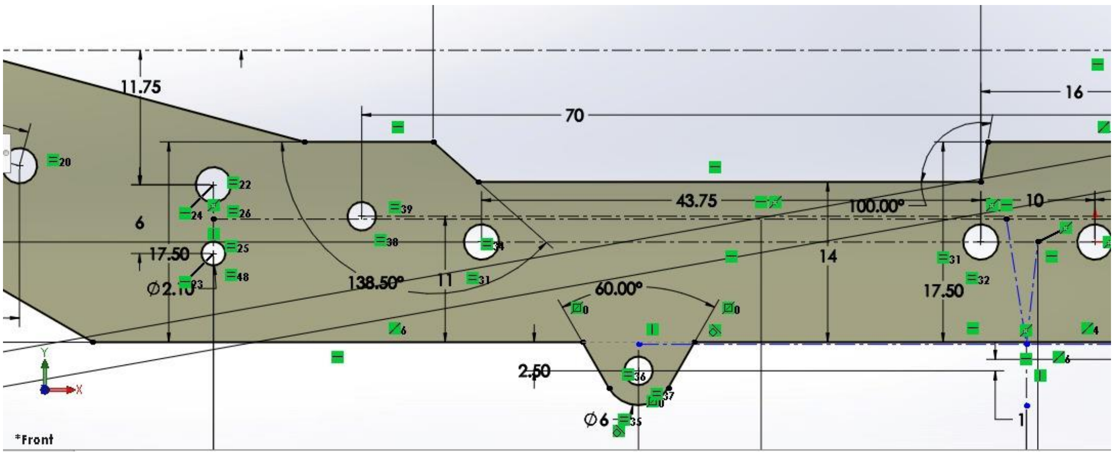

Figure 6.2. Bottom part with leaf suspension shaft holder

- Draw bodyframe holder fixture to main frame.draw circle of 2.5 dia with area of 45 degres for suspension holder for leaf spring

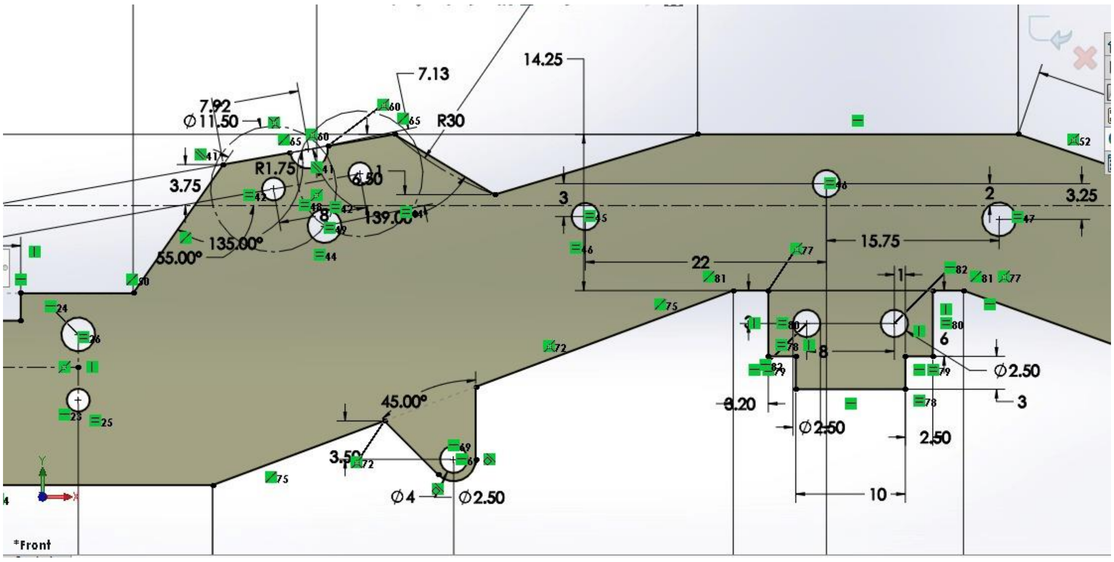

Figure 6.3.Suspension holder for leaf spring

- Select inner frame surface for main chassis holder for weight reduction with angle and length

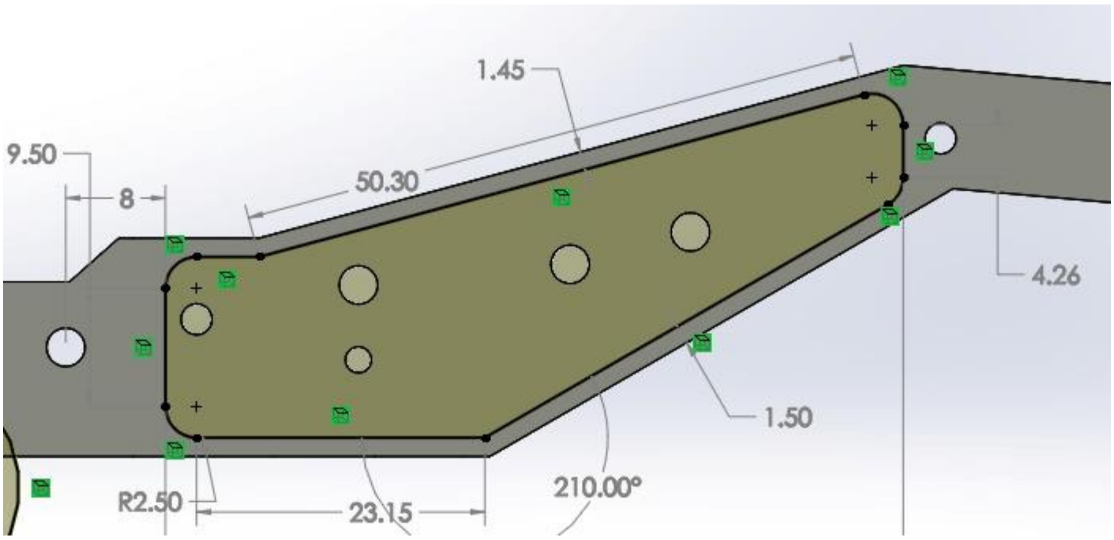

Figure 6.4. Main chassis holder

- Draw area for Leaf spring support shaft area / fix area

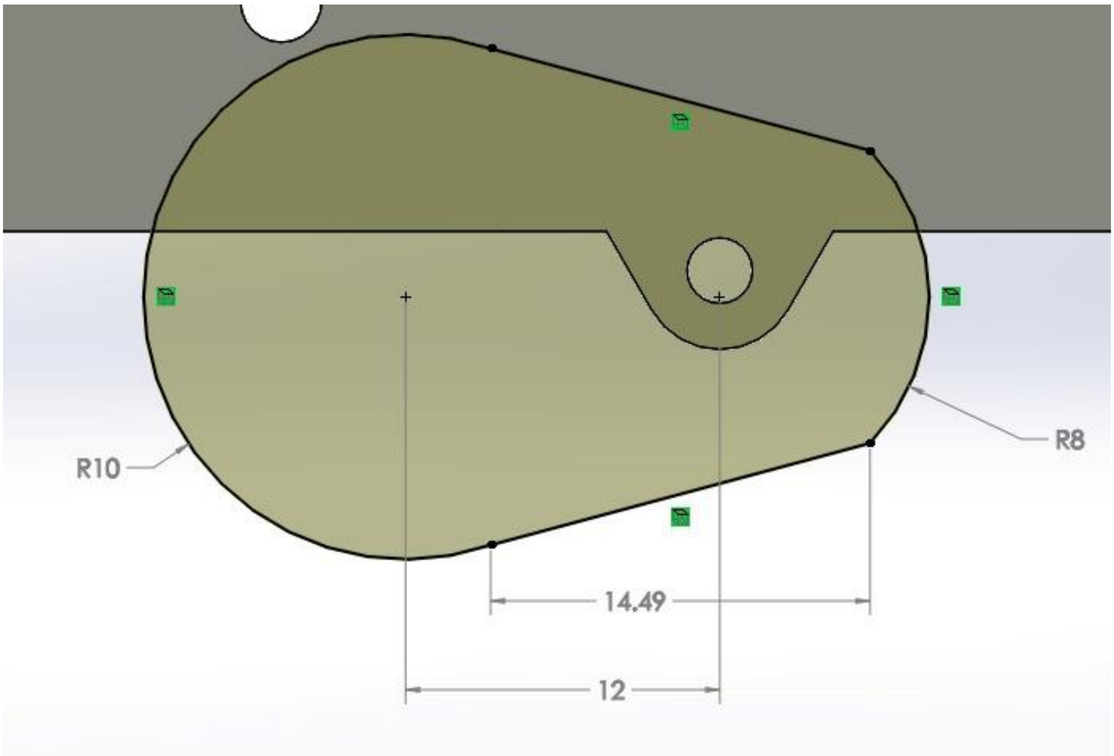

Figure 6.5. Leaf spring support shaft area

- Front frame fix area removed by Extrude cut

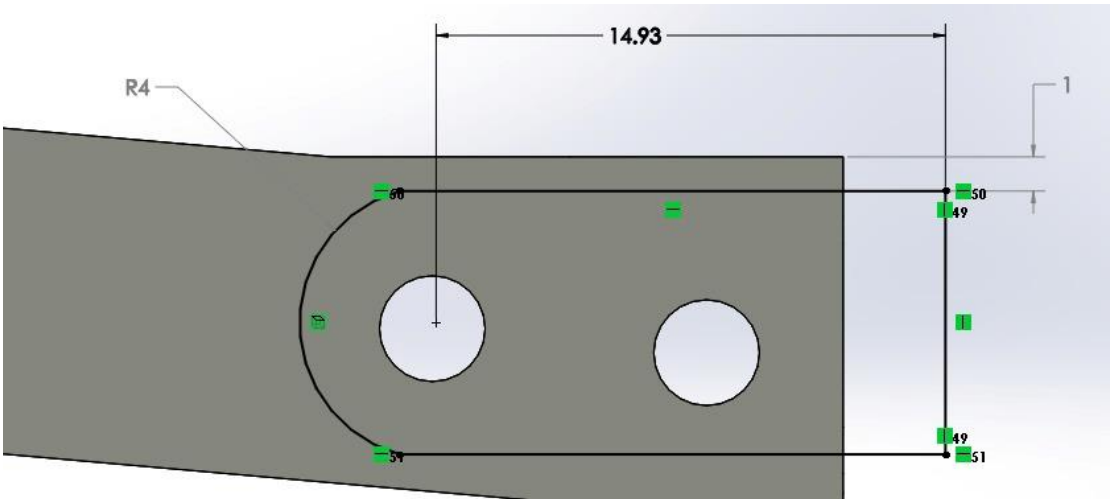

Figure 6.6.Front frame fix area removed by Extrude cut

CHAPTER 7

RESULT

Thus on Modeling and Designing of Vehicle Chassis in Solid Works,the final design of vehicle chassis as shown below.The 2D Design was Extruded by means of EXTRUDE BOSS part wise frame was assembled lastly

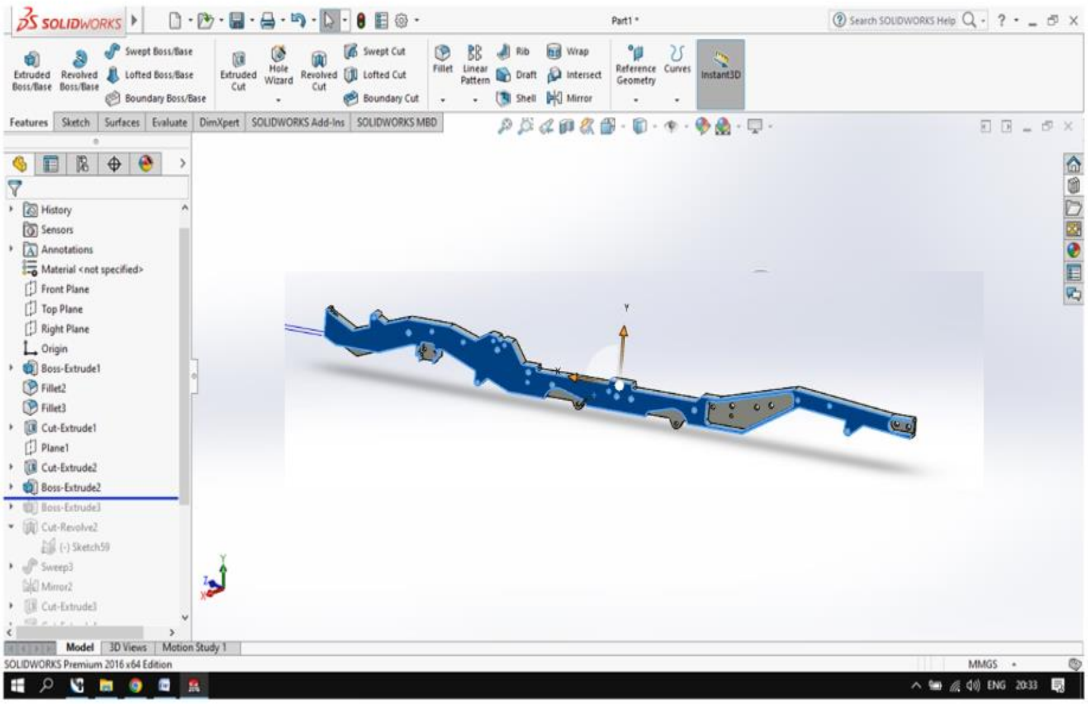

Figure 7.1.Design of frame of chassis

- Thus upon Conversion of the above CAD File,We receive a File in STL File Format upon Slicing the Vehicle Chassis.

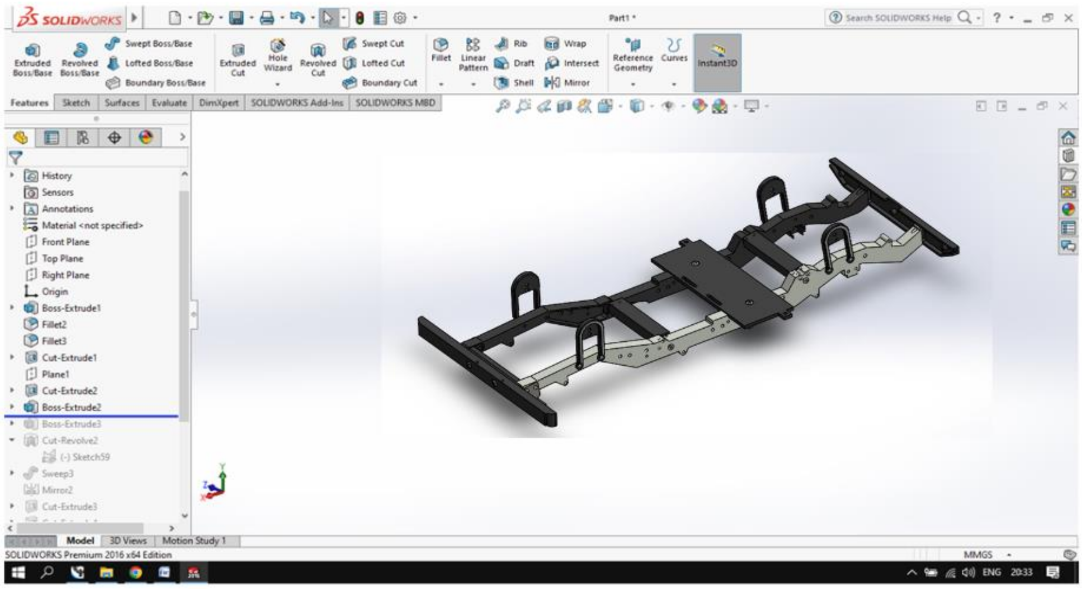

Figure 7.2.Final Design of vehicle chassis in Solid Works

7.1.Final Output Component of Vehicle chassis:

Thus,it can be seen from resultss that the vehicle chassis can be reduced by doing some changes in the geometry of vehicle chassis.It can be seen that the Chassis is reduced in the proposed model ,and the length of frame and overall height of the structure is slightly reduced.

Thus the fabrication of product is done with the help of 3D printing by using Fused Deposition Modeling and PLA material.

The final 3D printed object is a prototype of **VEHICLE CHASSIS** made by 3D printing to overcome the small errors and solve them to manufacture a perfect model.

Figure7.3 Final Output Component of Vehicle chassis

Figure7.4 Final Top View of Vehicle Chassis

8.CONCLUSION

In this project the model is Designed with respect to all the available constraints using an advanced cad software like, solid works, Later the product file is converted to "STL" file format (standard exchange of product file).

After conducting a modeling and 3D printing process for a Vehicle Chassis, several conclusions can be drawn:

Design Validation: The use of SolidWorks software enables the creation of a detailed and accurate model of the Vehicle Chassis. Through the modeling process,the design can be validated, ensuring that it meets the required specifications and functions as intended.

Prototyoping Efficiency:3D printing provides a rapid and cost-effective method for creating prototypes of the Vehicle Chassis. Traditional manufacturirng techniques, such as casting or machining can be time consuming and expensive. With 3D printing, multiple iterations of the Vehicle Chassis design can be prodcued quickly, allowing for efficient testing and refinement.

Material Selection: The choice of material for 3D printing the Vehicle chassis is crucial.Depending on the specific requirements of the application, different materials can be utilized,including various types of plastics , metals, or composites when selecting the material for vehicle chassis. The materials general using in manufacturing of the chassis in industries like steel, cast iron and aluminum alloy.

Post-Processing Consideratuions : After 3D printing ,the vehicle chassis may require post-processing steps such as sanding, polishing ,or heat treatment to achieve the desired surface finish, dimensional accuracy and mechanical properties. These additional steps should be taken into account when planning the overall manufacturing process.

Performance Evaluation: Once the 3D-printed connecting rod is fabricated, it should undergo rigorous testing and evaluation to assess its performance. this may involve subjecting vehicle chassis to simulated operating conditions or conducting physical tests to measure its strength, durability, and overall functionality.

In conclusion the combination of modeling and 3D printing offers numerous advantages for the production of vehicle chassis.it enables efficient design validation , rapid prototying, material flexibility, and customization possibilities , However, through testing and post-processing steps are essential to ensure the quality ,reliability, performance of the 3D-printed vehicle chassis in practical applications.

9.FUTURE SCOPE

The market is anticipated to attain a CAGR of ~6% over the forecast period,i.e,2023-2023.Chassis being the structural skeleton for every vehicle including truck.Chassis is the platform on which the truck is laid on as it contains the axles,fuel tank , engine , cab and , batteries.

The rising dependancy on trucks is expected to boost the automotive chassis market in the projected period. For instance , 70% of all freight transported United States depends on trucks for deliveries.

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