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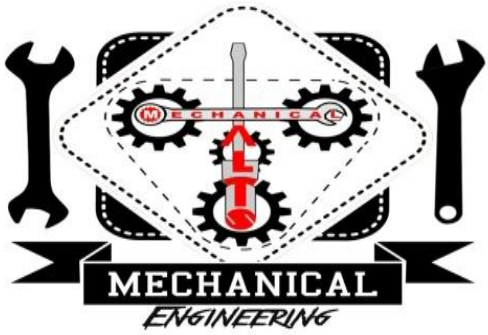

COMPUTER AIDED MODELING LABORATORY

(20A03506)

PREPARED

BY

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DEPARTMENT OF MECHANICAL ENGINEERING

PREFACE

CAM IS THE MOST EMERGING TECHNOLOGY IN THE FIELD OF MECHANICAL ENGINEERING

Nowadays as the technology improving more and more we no longer require the use of protractors and compasses to create drawings, instead there are several classes that focus on the use of various software.

This laboratory mainly deals with the detailed engineering of 3D models and/or 2D drawings of physical components using CATIA software, but it is also used throughout the engineering process from conceptual design and layout of products, through strength and dynamic analysis of assemblies to definition of manufacturing methods of components. It can also be used to design objects.

Computer Aided Iterative Application (CATIA) is the use of computer systems to assist in the creation, modification, analysis, or optimization of a design. CATIA software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing.

With this in mind, this manual is prepared as an introductory note for the laboratory experiments. Sufficient information has been included to emphasize Self-learning. Although the scope of the work is broad, the level of presentation is introductory.

We prepared with careful attention to the organization of the contents, and it is expected that this manual will be well received by the students. Any suggestions for improvement are always welcome

VISION OF THE DEPARTMENT:

To produce competent global mechanical engineers who can solve industrial and societal problems with technology, innovation, environment and ethical spirits.

MISSION OF THE DEPARTMENT:

The Vision will be attained by

- Providing quality teaching-learning methods combining with modern pedagogies.
- Developing the laboratories & infrastructure as practice labs with modern equipment and imparting skill based training.
- Encouraging and enriching the faculty to learn & teach advanced technologies through faculty development programs & international conferences.
- Motivating and encouraging the students to acquire the graduate attributes through workshops, seminars, conferences, innovative projects, internships and industrial trainings.

PROGRAMME EDUCATIONAL OBJECTIVES (PEO):

The following are the Program Educational objectives (PEOs) of the Mechanical Engineering Graduates: The Mechanical engineering program is designed to produce students for successful careers in manufacturing & service Industry, research & consultancy at the national and global level. Our Graduates are expected to:

PEO I: Apply their engineering knowledge combining with graduate attributes to develop technology-based solutions in professional engineering practice and also non-engineering fields such as agriculture, society, and environment.

PEO II: Develop their intellectual quotient through higher educations and online courses.

PEO III: Adapt leadership roles as Intrapreneur and entrepreneur.

PROGRAMME OUTCOMES (PO):

The following are the Program Outcomes (POs) of Mechanical Engineering Graduates:

PO1: Engineering knowledge - Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.

PO2: Problem analysis: Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.

PO3: Design/development of solutions: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.

PO4: Conduct investigations of complex problems: Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.

PO5: Modern tool usage: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modelling to complex engineering activities with an understanding of the limitations.

PO6: The engineer and society: Apply reasoning in formed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.

PO7: Environment and sustainability: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.

PO8: Ethics: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.

PO9: Individual and teamwork: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.

PO10: Communication: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as,being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.

PO11: Project management and finance: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.

PO12: Life-long learning: Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change

PROGRAMME SPECIFIC OUTCOMES (PSO):

The following are the Program Specific Outcomes (PSOs) of Mechanical Engineering Graduates:

PSO-1: Apply the knowledge of materials, product design, simulation and manufacturing to solve mechanical domain related problems.

PSO-2: Enable to work in innovative and product development themes collaborating with multidisciplinary teams.

PSO-3: Contribute to the society, providing solutions by considering ethical and environmental policies

COURSE OUTCOMES

CO1: Creation of part drawings and 3D models using CATIA techniques. (L5)

CO2: Execute steps required for modelling 3D objects by using Bezier-spline curve commands(L4)

CO3: Design different parts of mechanical equipment's(L6)

CO4: Apply their skills in various designing and Manufacturing Industries. (L3)

CO5: Apply dimensioning and tolerancing techniques in order to complete technical drawing**s**(L3)

CO6: Design a set of 3D solid / surface models of an E / M design, mechanical design. (L6)

JAWAHARLAL NEHRU TECHNOLOGICAL UNIVERSITY ANANTAPUR

B. Tech (ME)– III-I Sem

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(20A03506) COMPUTER AIDED MODELING LABORATORY

Course Objectives:

- To train the students with CAD packages.
- To impart the 2D and 3D modeling skills to the students.
- To import and export different IGES files from one software to another

Course Outcomes:

- Students will be able to design different parts of mechanical equipment's
- Students will be able to apply their skills in various designing and Manufacturing Industries.

List of Experiments:

1. Generation of the following curves using "C"/ Python language

a) Cubic Splines
b) Bezier curves
c) B-Splines.

2. Generation of the following surfaces using "C"/Python language

a) Bezier surfaces
a) B-Spline surfaces

3. Typical tasks of Modeling using any solid modeling packages such as PRO/E, IDEAS, CATIA, etc.,

a) Solid Boolean algebra - 1 Exercise
b) Wireframe &Surface Modelling – 3 Exercises
c) 3D – Drafting in detail – 1 Exercise
d) Production Drawing with Geometric Dimensioning and Tolerances– 3 Exercises

(Preferably for the assembly drawings drawn in Computer Aided Machine Drawing in previous semester)

Production drawings: (Any four of the following using solid model software)

Lathe tool post, tool head of shaping machine, tail stock, machine vice, gate valve, carburetor, piston, connecting rod, eccentric, screw jack, plumber block, axle bearing, pipe vice, clamping device, Geneva cam, universal coupling,

References:

1. James D Meadows "Geometric Dimensioning and Tolerancing-Applications, Analysis & Measurement ASME Y14.5-2018.
2. KL Narayana, P Kannaiah and K. Venkat Reddy, Production Drawing, New Age publishers, 2014.
3. Ibrahim Zeid, Tata Mc Graw hill, CAD/CAM Theory and Practice, 2012.

Online Learning Resources/Virtual Labs:

- https://www.youtube.com/watch?v=er7xJFKv5k&list=PL5w7L_xR0pu2wLbJtOuK49WxJJVjiyKks&index=2
- https://www.youtube.com/watch?v=Gy0MKabzDa8&list=PLrOFa8sDv6jccqLnN7UDa1YW4s_hR6YX0
- https://www.youtube.com/watch?v=k3kFC9uTdUk&list=PLM5xm8DJKViImdv5ZXxQ2NyIdSlid_jCB

LIST OF EXPERIMENTS

COMPUTER AIDED MODELING LABORATORY
(20A03506)

Course Objectives:

- To write program for CAD modelling.
- To learn part programming and path generation from a CAD model.
- To train on machining of various parts in CNC machines.

GEOMETRIC MODELING

Introduction to 3D Modelling (4 or 5 exercises).

1. Generation of the B-Spline Curves Using Catia.

2. Generation of the B-Spline Surfaces Using Catia.

3. Exercise On 3d Solid Model Using Boolean Algebra Operations.

4. Exercise on Wireframe &Surface Modelling in Bracket

5. Exercise on Wireframe &Surface Model in Angular Bracket

6. Exercise on Wireframe &Surface Modelling In V-block With Circular Rod

7. Exercise On 3d Drafting in Drum with Mounting

8. Production Drawing with Geometric Dimensioning and Tolerances on Lathe Tool Post.

9. Production Drawing with Geometric Dimensioning and Tolerances on Screw Jack Post.

10. Production Drawing with Geometric Dimensioning and Tolerances on Universal Coupling.

INSTRUCTIONS TO STUDENTS

1.Students are required to remove their footwear outside the centre and keep it in the box provided for the same.

2.Students should leave their belongings outside the lab except their observation note book, the concerned books/manuals and calculators.

3.Students are requested not to place their legs on the wall or on the table.

4.Students should refrain from leaning on the table and sitting on it.

5. Before logging in to a particular terminal, if there is something wrong in the terminal, the student should report the same immediately to the concerned staff.

6.Students should not use any disks brought from outside without prior permission from the concerned staff.

7.Students can get the required manual or disks from the staff after signing in the appropriate register.

8.Students should collect their printouts before leaving the lab for that particular session.

9.Before leaving the Terminal, the students should logout properly and leave their chairs in position.

10.Students are not allowed to take any manual outside the centre.

11.Edibles are strictly prohibited in the centre. No internet browsing allowed during the lab hours

INTRODUCTION TO CATIA

Computer Aided Design (CAD) is the use of computer software to design a product or an object.
Computer Aided Manufacturing (CAM) is the use of computer software and hardware to plan, manage and control the operations of a manufacturing plant.
Computer Aided Engineering is the use of computer software to solve engineering problems and analyze products created using CAD.

CATIA is an acronym for Computer Aided Three-dimensional Interactive Application. It is one of the leading 3D software used by organizations in multiple industries ranging from aerospace, automobile to consumer products.

CATIA is a multi platform 3D software suite developed by Dassault Systèmes, encompassing CAD, CAM as well as CAE. Dassault is a French engineering giant active in the field of aviation, 3D design, 3D digital mock-ups, and product lifecycle management (PLM) software. CATIA is a solid modelling tool that unites the 3D parametric features with 2D tools and also addresses every design-to-manufacturing process. In addition to creating solid models and assemblies, CATIA also provides generating orthographic, section, auxiliary, isometric or detailed 2D drawing views. It is also possible to generate model dimensions and create reference dimensions in the drawing views. The bi-directionally associative property of CATIA ensures that the modifications made in the model are reflected in the drawing views and vice-versa.

The first release of CATIA was way back in 1977, and the software suite is still going strong more than 30 years later. While CATIA V6 is just being released, the most popular version of CATIA is V5 which was introduced in 1998. That said, it is important to note that each version of CATIA introduces considerable additional functionality. For example, V4 (introduced in 1992) offered enhancements to the Assembly Modelling Product including easy-to-use graphical tree-based assembly management. V5 and V6 saw changes in the way data is handled. Dassault Systems typically offers new updates, releases and bug fixes for each version. The CATIA software is written in C++. It runs on both Unix and Windows.

What does CATIA do?
CATIA provides the capability to visualize designs in 3D. When it was introduced, this concept was innovative. Since Dassault Systems did not have an expertise in marketing, they had revenue sharing tie-up with IBM which proved extremely fruitful to both the companies to market CATIA. In the early stages, CATIA was extensively used in the design of the Mirage aircrafts; however the potential of the software soon made it a popular choice in the automotive sector as well. As CATIA was accepted by more and more manufacturing companies, Dassault changed the product classification from CAD / CAM software to Project Lifecycle Management. The company also expanded the scope of the software.
CATIA can be used at different stages of the design - ideate, draw, test and iterate. The software comes with different workbenches (“modules”) that allow CATIA to be used across varied industries – from parts design, surface design and assembly to sheet metal design. CATIA can also be used for CNC.

CATIA offers many workbenches that can be loosely termed as modules. A few of the important workbenches and their brief functionality description is given below:

Part Design: The most essential workbench needed for solid modelling. This CATIA module makes it possible to design precise 3D mechanical parts with an intuitive and flexible user interface, from sketching in an assembly context to iterative detailed design.

Generative Shape Design: allows you to quickly model both simple and complex shapes using wireframe and surface features. It provides a large set of tools for creating and editing shape designs. Though not essential, knowledge of Part Design will be very handy in better utilization of this module.

Assembly: The basics of product structure, constraints, and moving assemblies and parts can be learned quickly. This is the workbench that allows connecting all the parts to form a machine or a component.

Kinematic Simulation: Kinematics involves an assembly of parts that are connected together by a series of joints, referred to as a mechanism. These joints define how an assembly can perform motion. It addresses the design review environment of digital mock-ups. This workbench shows how a machine will move in the real world**.**

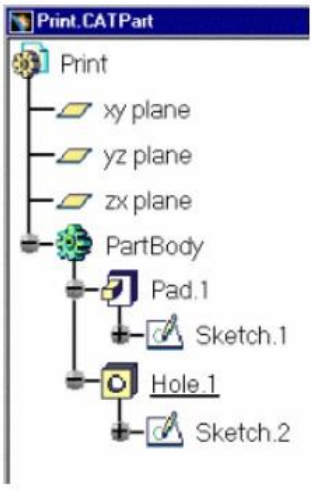
Surface Modelling:

Generative Surface Design workbench the Generative Surface Design workbench enables users to create wireframe construction elements and enrich existing mechanical part design with wireframe and surface features.

COMMANDS

SPECIFICATION TREE

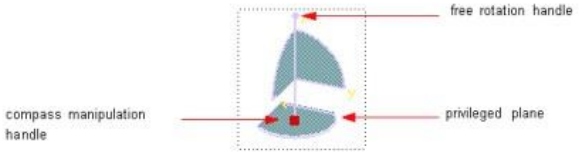
- The Specification Tree is displayed on the left side of the screen while you are working
- Provides access to the history of how a part was constructed, and shows the product structure
- Product entities can be selected from the spec. tree or in the geometry area
- Parts can be modified by selecting them from the spec. tree.
- Click on + to open a tree branch
- Solid Parts are stored in the PartBody branch of the Part tree



MOUSE CONTROLS

- Pan
  - Press and hold the middle mouse button and move the mouse to pan
- Rotate
  - Press and hold the middle mouse button then the left mouse button and move the mouse to rotate
- Zoom
  - Press and hold the middle mouse button and click the left mouse button then move the mouse to zoom in and out

- Using the compass



- Drag the axes or planes of the compass to dynamically rotate the display
- Multi-select entities by holding down the ***Shift*** key

VIEW TOOLBAR

- Display Commands
  - Fly Through
  - Fit View
  - Layer control
  - Pan
  - Rotate
  - Zoom
  - Normal View
  - Standard Views
  - View Types: Shaded/ Hidden Line/ Wireframe/ User Defined
- Hide/ Show
  - Hide
  - Swap Visible Space
- Properties
  - Display Characteristics for an object are set by selecting the entity, then pressing the right mouse button and selecting **Properties** from the menu

OTHER COMMONLY USED TOOLS:

- Copy/ Paste

- Geometry entities can be copied and pasted from one part to another.

- Paste Special allows you to:
  - Paste a complete copy with history
  - Paste a linked copy
  - Paste the result without linking

- Undo/ Redo

- Allows you to undo previous actions

- Redo repeats an action that has been undone

- Hide/ Show

- Allows you to temporarily hide entities from the display

- Hidden entities can be recovered by clicking on the "Swap visible space" icon, and then selecting the entity to make visible

- Update

- Used to update the part after modification

Step 2: Sketcher Module

The Sketcher workbench is a set of tools that helps you create and constrain 2D geometries. Features (pads, pockets, shafts, etc...) may then be created solids or modifications to solids using these 2D profiles. You can access the Sketcher workbench in many ways. Two simple ways are by using the top pull down menu (Start – Mechanical Design – Sketcher), or by selecting the Sketcher icon. When you enter the sketcher, CATIA requires that you choose a plane to sketch on. You can choose this plane either before or after you select the Sketcher icon. To exit the sketcher, select the Exit Workbench icon.

The Sketcher workbench contains the following standard workbench specific toolbars.

- Entering the sketcher

- Click on the Sketcher icon or select Start -> Mechanical Design -> Sketcher

- Exiting from the Sketcher

- Click on the Exit icon to leave the sketcher and return to the 3D workspace

- Geometry Creation

- Geometry Operations

- Constraint Creation

- Tools Toolbar

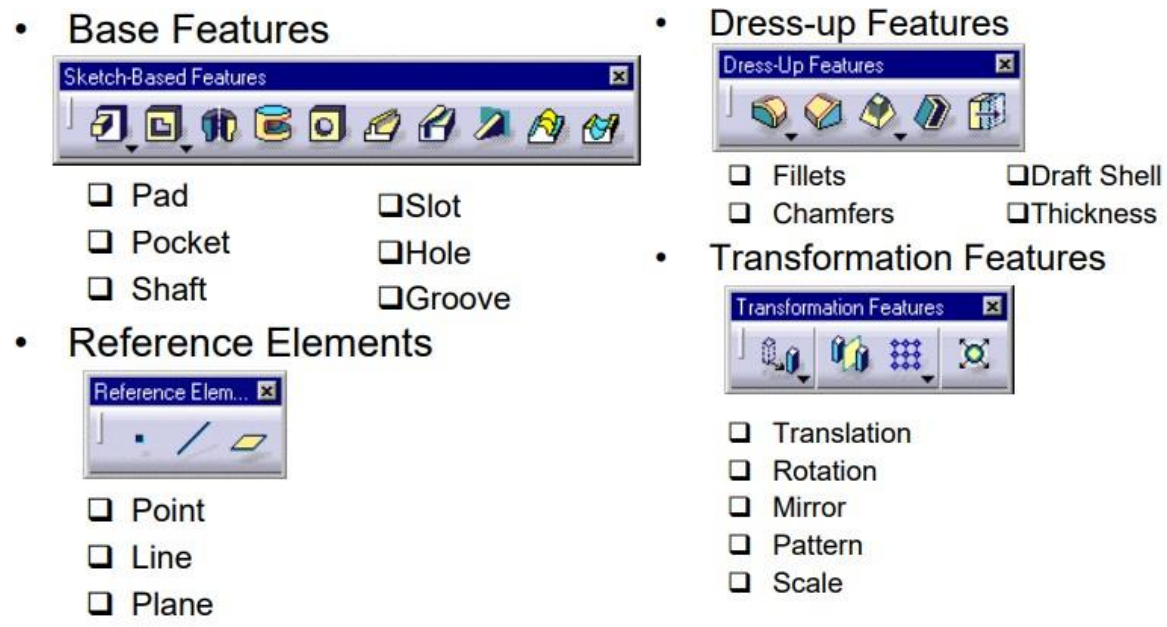
- Snap to point
- Construction Geometry
- Constraint

• **Profile toolba**r: The commands located in this toolbar allow you to create simple geometries (rectangle, circle, line, etc...) and more complex geometries (profile, spline, etc...).

• **Operation toolbar:** Once a profile has been created, it can be modified using commands such as trim, mirror, chamfer, and other commands located in the Operation toolbar.
• **Constraint toolbar:** Profiles may be constrained with dimensional (distances, angles, etc...) or geometrical (tangent, parallel, etc...) constraints using the commands located in the Constraint toolbar.
• **Sketch tools toolbar:** The commands in this toolbar allow you to work in different modes which make sketching easier.

Step 3: Part Design Module

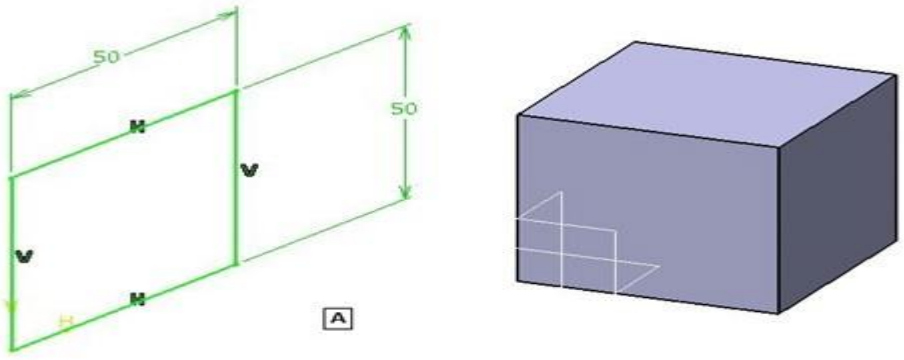
Part design environment is used to create 3D models from the basic 2D sketches created in sketcher environment.



Some of the commands in workbench explained below

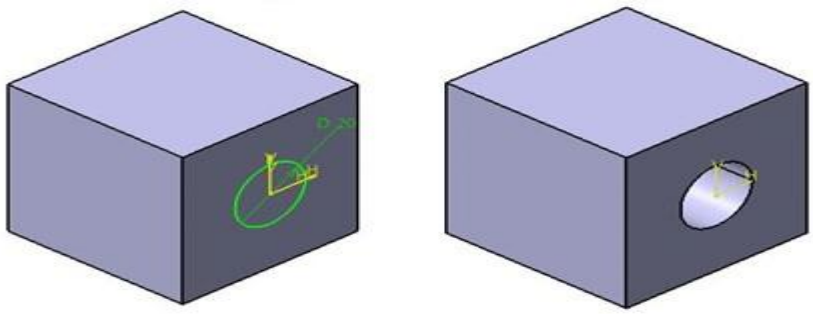
PAD command

In most CAD software, the equivalent of this is called EXTRUDE, but in CATIA we call it PAD. This command adds material in the third direction, a direction other than the sketch.



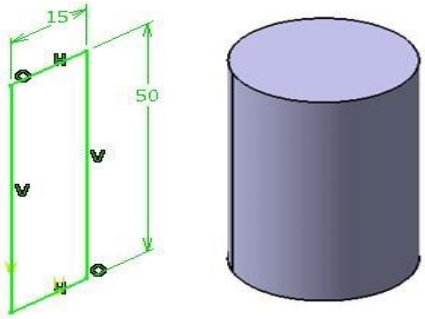
POCKET command

The POCKET commands somehow the opposite of PAD command. It simply helps remove geometry belonging to an already create part. On the figure below the POCKET command is helping to create the cylinder hole in the middle of the cube.



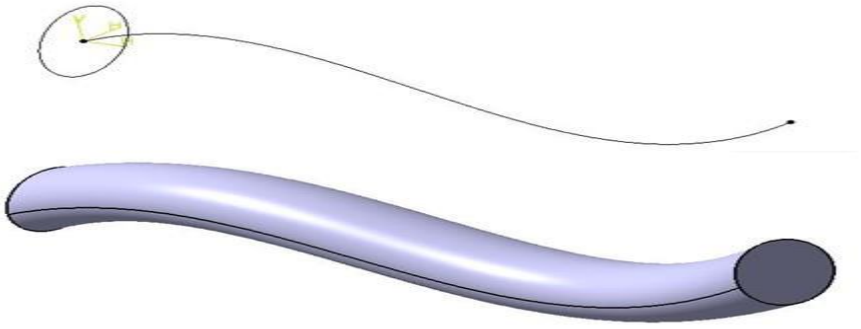
SHAFT command

It is Like revolve command in other CAD software, the SHAFT command is mostly used to make shaft like parts. It requires an axis, around which the sketch will be revolved.



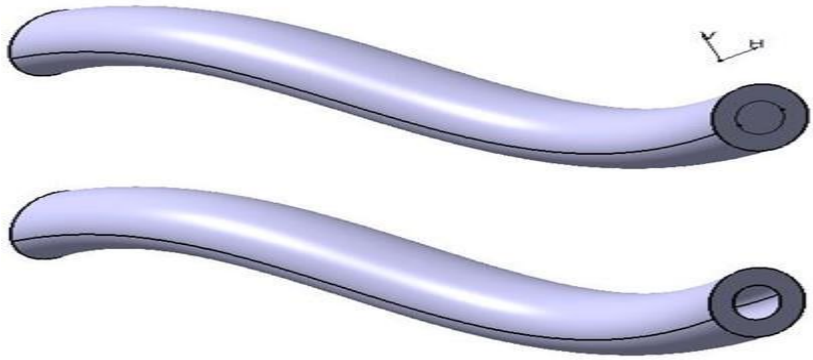
RIB command

This command which is usually known as SWEEP is called RIB IN CATIA. It adds material along a guide curve. RIB is used to make components like springs, pipes etc.



SLOT command

SLOT removes the material along a guide curve. Here is an example of slot. While using SLOT, I have used the same guide curve that was used for RIB. This ensures that the cross section will be uniform throughout.



Step 4: Assembly Module

Assembly environment is used to provide mating to two or more part models to from complete assembly

- Product Structure Tools

- ❑ Insert New Component
- ❑ Insert New Product
- ❑ Insert New Part
- ❑ Insert Existing Component
- ❑ Replace Component
- ❑ Reorder Tree
- ❑ Generate Numbers
- ❑ Load Components
- ❑ Unload Components
- ❑ Manage Representations
- ❑ Multi-Instantiation

- Move Toolbar

- ❑ Manipulate
- ❑ Snap
- ❑ Explode and Assembly

- Constraints Toolbar

- ❑ Coincidence
- ❑ Contact
- ❑ Offset
- ❑ Angular
- ❑ Anchor
- ❑ Fix Together

.**Step 5: Drafting Module**: Drafting is a process of generating 2D machine drawing for the 3D part models to send it to the manufacturer's.

Catia drafting is of two types

1.Interactive Drafting 2. Generative Drafting

- Views Toolbar

- ❑ Create a Front View (other views available underneath icon)
- ❑ Create a section view
- ❑ Create a detail view
- ❑ Create a Clipping View
- ❑ Create Views Via Wizard

- Automatic Dimension Creation

- ❑ Auto-dimension
- ❑ Semi-Automatic Dimensions

- Geometry Creation

  - ❑ Point
  - ❑ Line
  - ❑ Circle
  - ❑ Arc
  - ❑ Profile
  - ❑ Curve
  - ❑ Pre-Define Profiles

- Transformations Toolbar

  - ❑ Translate, Rotate, Scale, Mirror

- Relimitations Toolbar

  - ❑ Corner
  - ❑ Chamfer
  - ❑ Trim
  - ❑ Break

- Annotation

  - ❑ Text
  - ❑ Symbols

- Dimensions Toolbar

  - ❑ Create Dimension
  - ❑ Create Tolerance

- Dress up Toolbar

  - ❑ Centreline
  - ❑ Thread
  - ❑ Axis
  - ❑ Fill
  - ❑ Arrow

EXPERIMENT NO:1

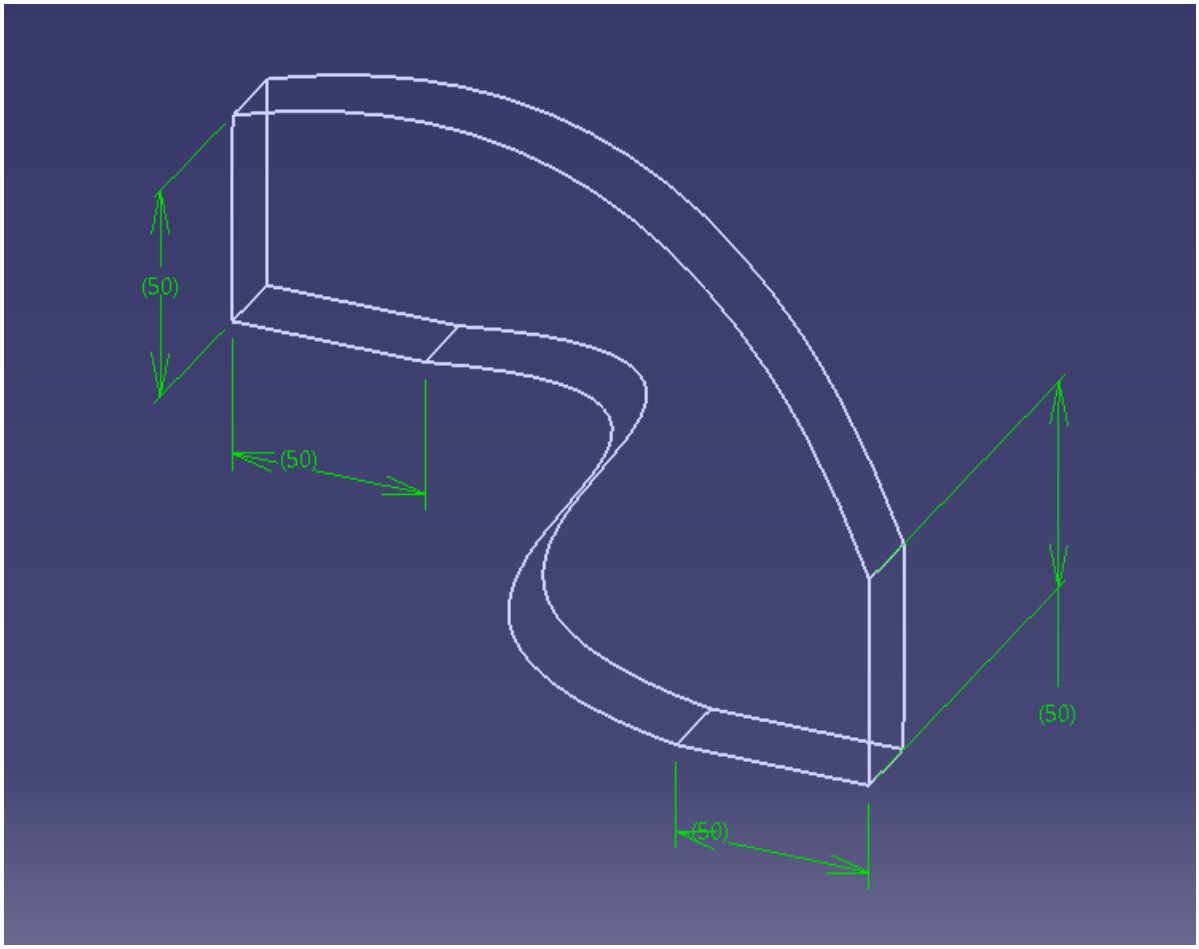
EXERCISE ON 3D DRAWING IN B-SPLINE CURVE

Aim: To Create the 3D Model by the showing figure with specified dimensions.

Software required

Personal Computer with CATIA software Package

Drawing:



Description:

Concept of B-spline curve came to resolve the disadvantages having by Bezier curve, as we all know that both curves are parametric in nature. In Bezier curve we face a problem, when we change any of the control point respective location the whole curve shape gets change. But here in B-spline curve, the only a specific segment of the curve-shape gets changes or affected by the changing of the corresponding location of the control points.

In the B-spline curve, the control points impart local control over the curve-shape rather than the global control like Bezier-curve.

PROCEDURE:

- Click on the CATIA icon, present on the desktop

- Go to file ‘MENU’ and click on ‘NEW’ a dialogue box is opened select mechanical design and select Part design.
- Select the Required YZ or XY plane go to select sketcher command
- Select **line command** draw a line length is 50mm and use the **constrain define command** fix the line.
- Select **line command** draw a line length is 50mm and use the **constrain define command** fix the line.
- Select **constrain command** and given length is 80mm of both lines
- Select **spline curve command** to create the curve between two lines by using two points
- Select **line command** draw a line length is 50mm and use the **constrain define command** fix the line
- Select **line command** draw a line length is 50mm and use **the constrain define command** fix the line
- Select **spline curve command** to create the curve between two lines by using two points
- Select **Exit work bench** command to convert 2d to 3d
- Select **PAD** Command and given the length is 30mm to crate 3d model
- Select **constrain command** to given the dimension of 3d model

RESULT: Hence the Exercise On 3d Drawing In B-Spline Curve is created by using Catia software

EXPERIMENT NO:2

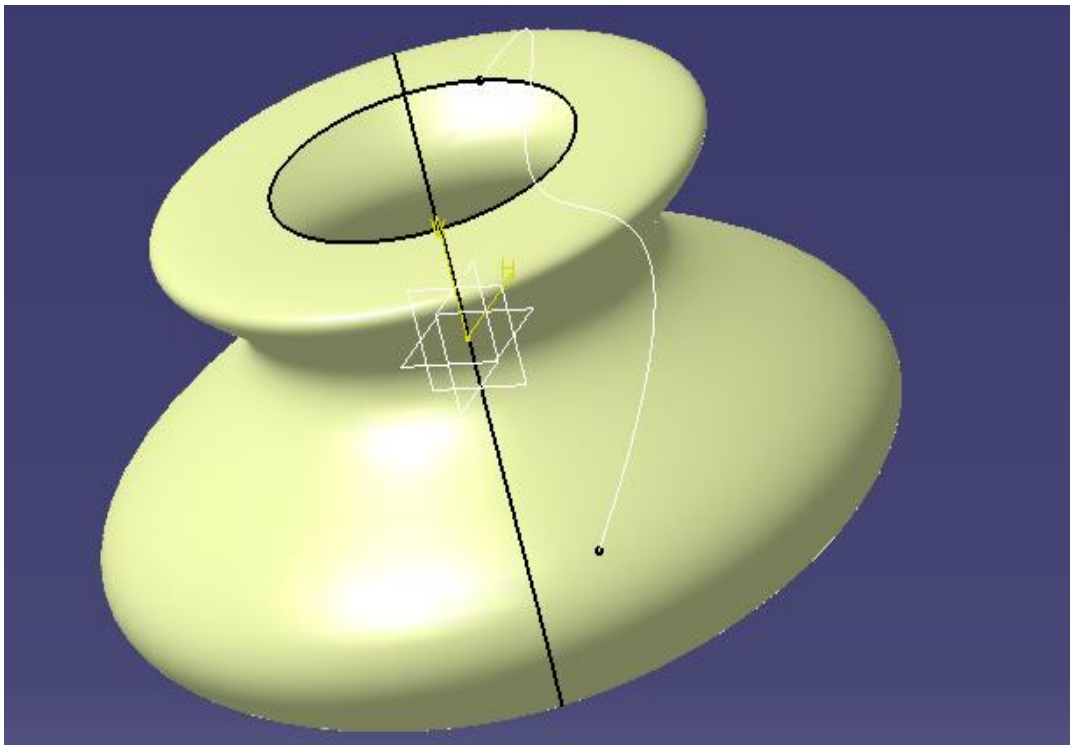
EXERCISE ON 3D DRAWING IN B-SPLINE SURFACES

Aim: To Create the 3D Model **POT** in b-spline command by the showing figure with specified dimensions.

Software required

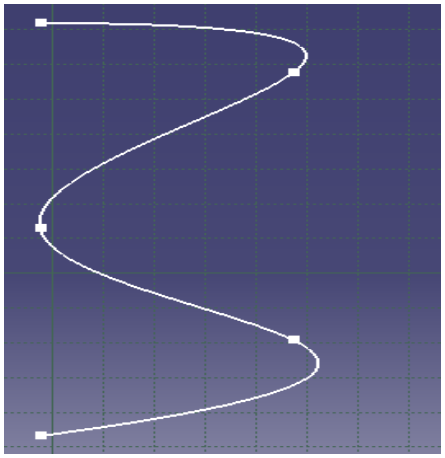
Personal Computer with CATIA software Package

Drawing:

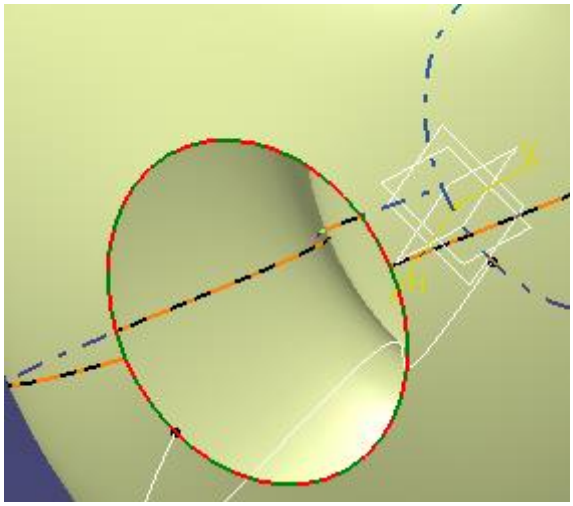


PROCEDURE:

- Click on the CATIA icon, present on the desktop
- Go to file 'MENU' and click on 'NEW' a dialogue box is opened select mechanical design and select wire frame and surface design
- Select the Required YZ or XY plane go to select sketcher command
- Select spline curve command to create one spline curve without dimension



- Select exit work bench command and select Revolve operation one window box is open select revolution axis and angle one is given 360 degrees and angle two is given 0 degrees.. then select ok.
- Select rotate command Rotate object & select bottom side of the model and select fill command one window box is open select deviation value is 0.001mm ok



- The surface area is full filled of the model
- The complete 3D model POT is created

RESULT: Hence the Exercise On 3d Drawing POT is Created by using B-Spline surface command is done by Catia software.

EXPERIMENT NO:3

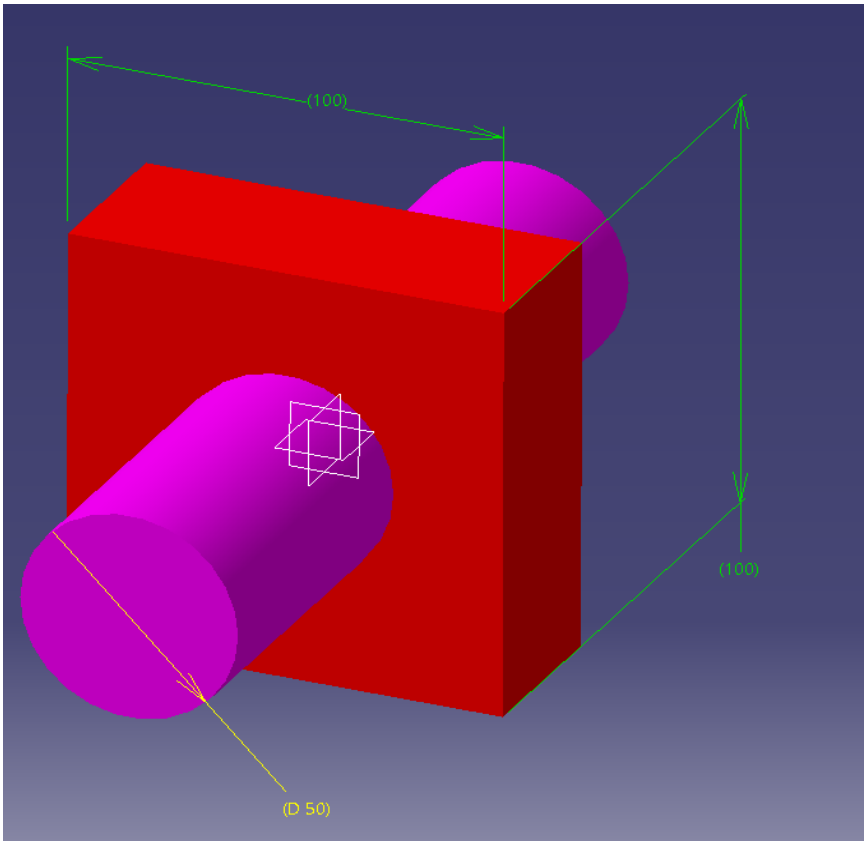
EXERCISE ON 3D SOLID MODEL

Aim: To Create the 3Dsolid Model by using Boolean Operation command by the showing figure with specified dimensions.

Software required

Personal Computer with CATIA software Package

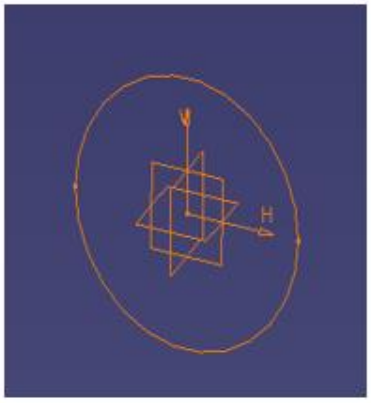
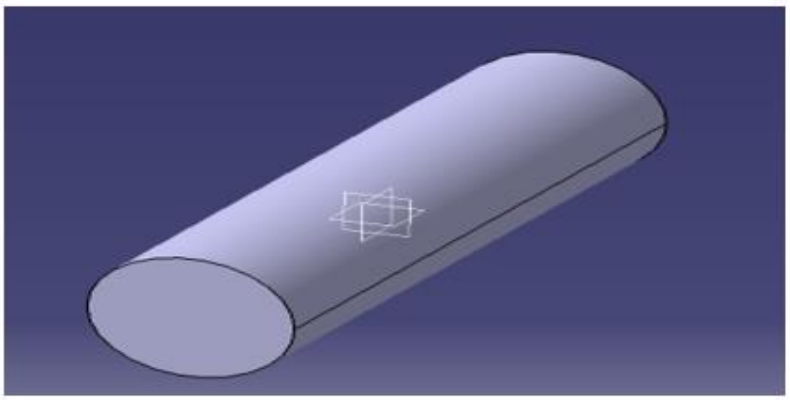
Drawing:



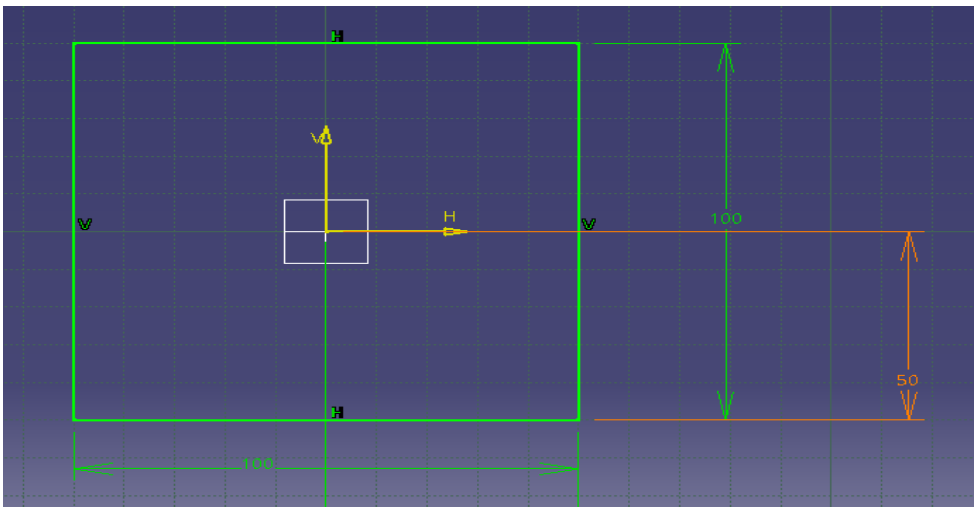
PROCEDURE:

- Click on the CATIA icon, present on the desktop
- Go to file 'MENU' and click on 'NEW' a dialogue box is opened select mechanical design and select Part design.
- Select the Required YZ or XY plane go to select sketcher command

- Select **Circle** command create two circles with dimensions of 50mm by using constrain command
- Select exit work bench command -ok.
- Select pad command by giving extrusion length is 100mm both mirror side-ok
- Select hide option-remove body1-ok

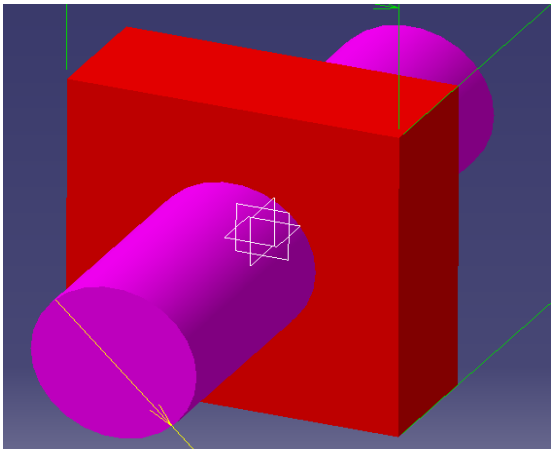



- Select insert tool-body command-Right click define in work object-select yz plane-sketcher-ok
- Select Rectangle command create rectangle length and height is 100mm x 100mm by using constrain command ok

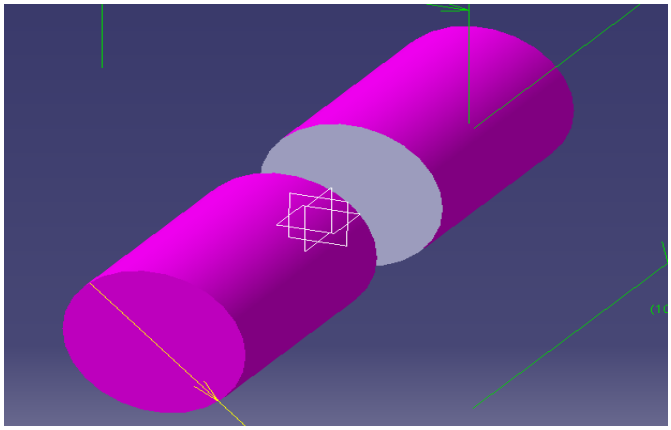


- Select exit work bench command -ok.
- Select pad command by giving extrusion length is 20mm both mirror side-ok

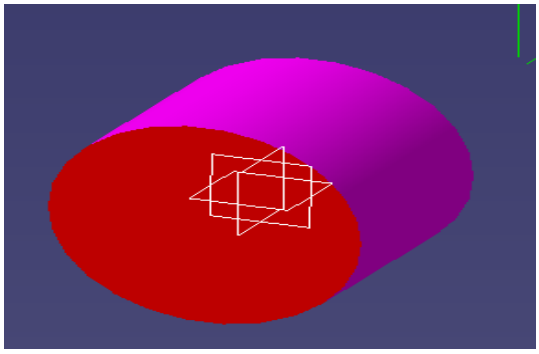
- Select show option-body1-ok
- Select Boolean operation—**ADD** Command select part body & body 2-ok-success



- Select Boolean operation—**Remove** Command select part body & body 2-ok-success



- Select Boolean operation—**intersect Command** select part body & body 2-ok-success



RESULT: Hence the Exercise On 3d Drawing exercise is Created by using Boolean operation command is done by Catia software

EXPERIMENT NO:4

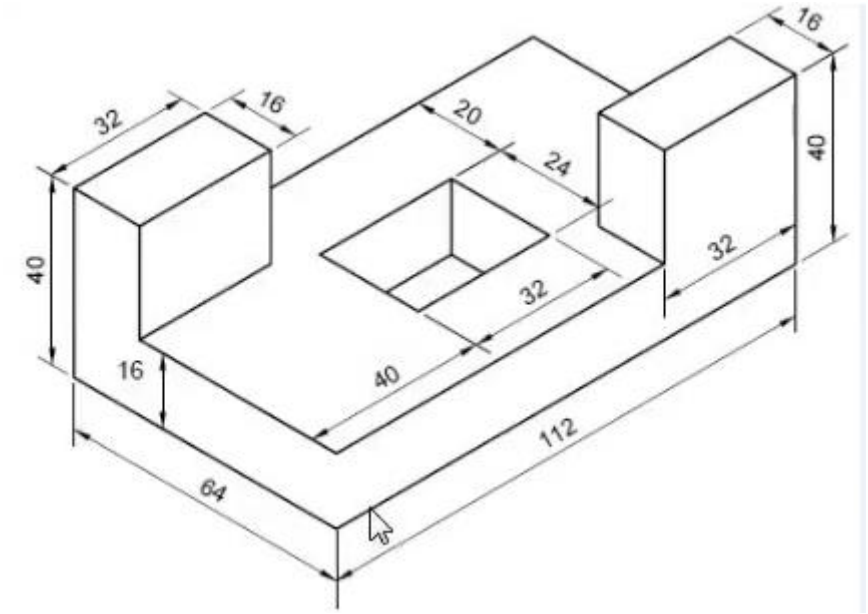
EXERCISE ON WIREFRAME &SURFACE MODELLING IN BRACKET

Aim: To Create Wireframe &Surface Modelling in Bracket in the showing figure with specified dimensions.

Software required

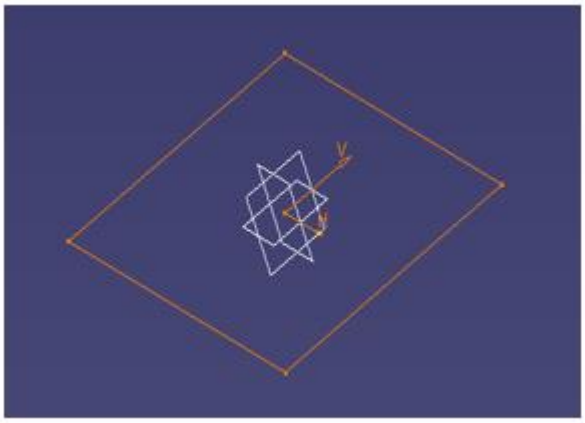
Personal Computer with CATIA software Package

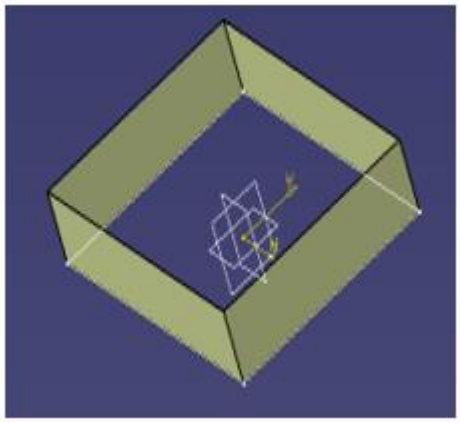
Drawing:



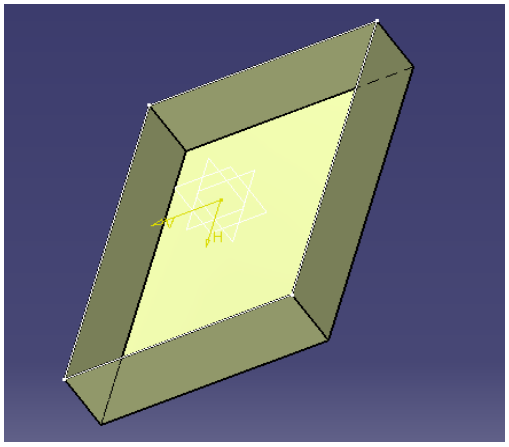
PROCEDURE:

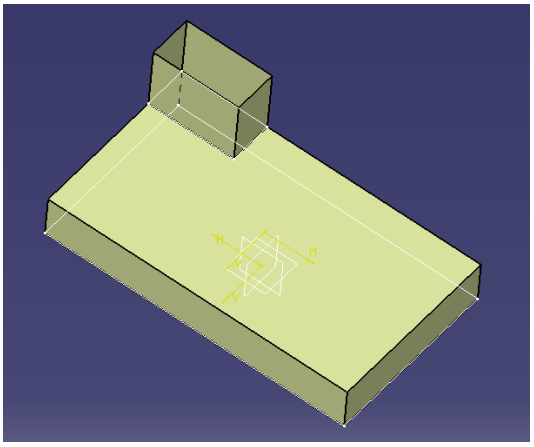
- Click on the CATIA icon, present on the desktop
- Go to file 'MENU' and click on 'NEW' a dialogue box is opened select mechanical design and select wire frame Surface and design
- Select the Required YZ or XY plane go to select sketcher command.
- Select rectangle command to create rectangle with length is 112mm and height is 64mm
- Select exit work bench command—ok
- Select Extrude command ---just click on the profile given the extrusion length is 16mm ok



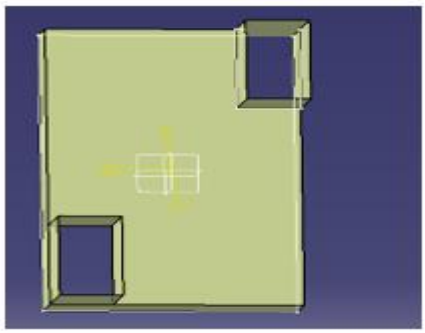


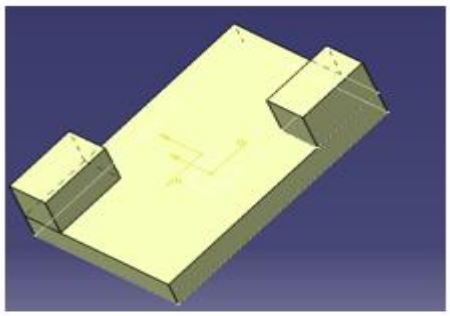
- Select fill command –select four sides of Rectangle ok—it formed surface area of the rectangle.



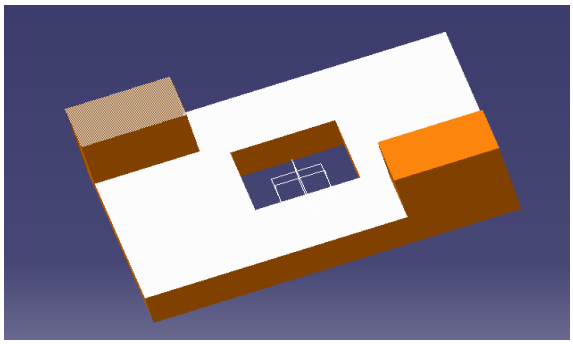


- Select surface area of the bracket-go to sketcher command crate rectangle by using Rectangle command with the length 32mm and height is 16mm-ok
- Select work bench command –ok
- Select Extrude command to given the extrusion length is 24mm to create one surface area automatically--ok
- Select translate command –go to point point to option—select surface area of the hole-select the position of points—crate automatically another rectangle portions.
- Select trim option to remove the Edges of both rectangle holes.





- Select fill command –select four sides of two Rectangular holes ok—it formed surface area of the rectangle.
- Select surface area of the bracket go to sketcher command ok
- Select centre of rectangle to create rectangle at centre of the bracket with dimension of 32mm length and 24 mm height
- Select Extrude command to given the extrusion length is 16mm to create one surface area automatically--ok
- Select trim option to remove the Edges of centre rectangle hole.
- Select join command to join all entities-ok
- Go to file ‘MENU’ and click on ‘NEW’ a dialogue box is opened select mechanical design and select part design---go to closed surface area option--ok



RESULT: Hence the Exercise On Wireframe &Surface Modelling in Bracket by the showing figure with specified dimensions is Created by using Catia software

EXPERIMENT NO:5

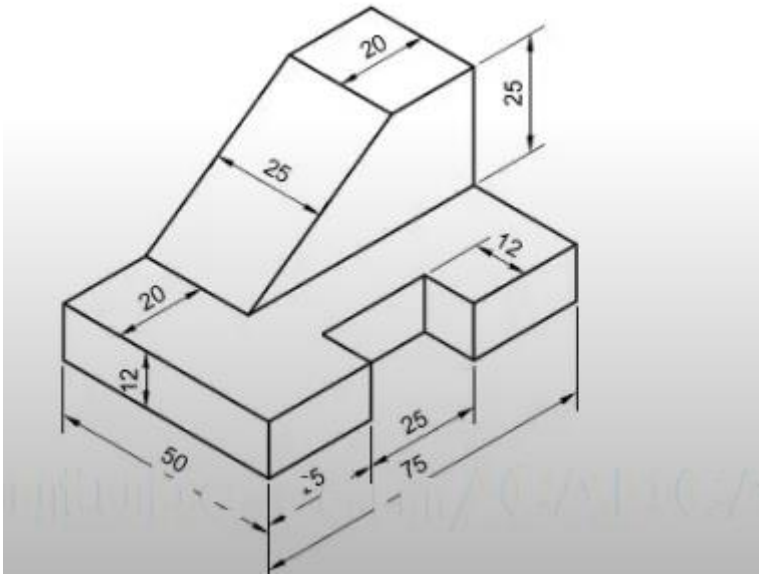
EXERCISE ON WIREFRAME &SURFACE MODEL IN ANGULAR BRACKET

Aim: To Create the wire frame and surface model in Angular Bracket in the showing figure with specified dimensions.

Software required

Personal Computer with CATIA software Package

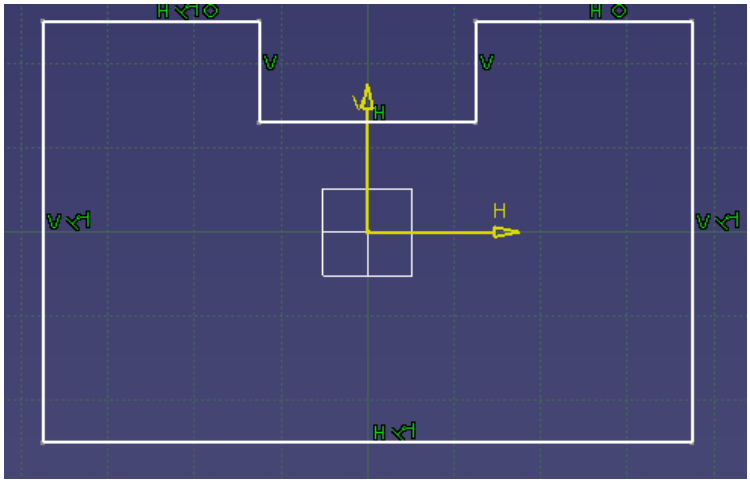
Drawing:



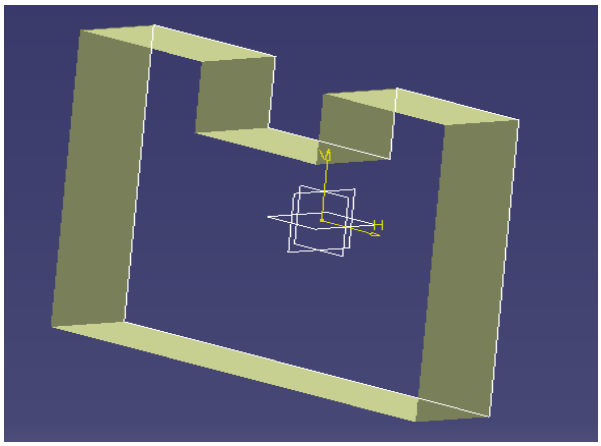
PROCEDURE:

- Click on the CATIA icon, present on the desktop
- Go to file 'MENU' and click on 'NEW' a dialogue box is opened select mechanical design and select wire frame Surface and design
- Select the Required YZ or XY plane go to select sketcher command
- Select Centred Rectangle command and create Rectangle by the length 75mm and width is 50mmm
- Select Rectangle command and create Rectangle by the length 25mm and width is 12mmm

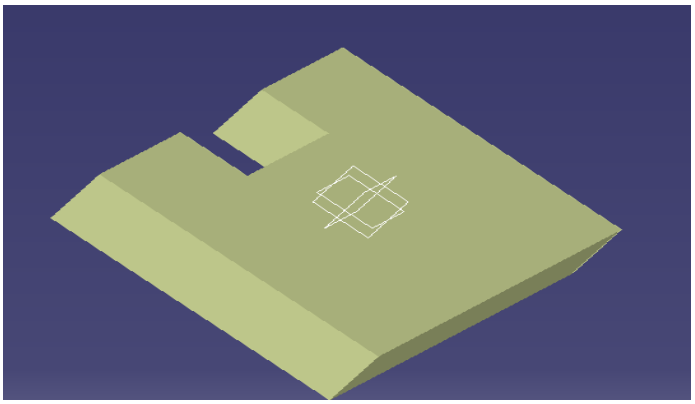
- Select trim command to trim Excess length of rectangle block



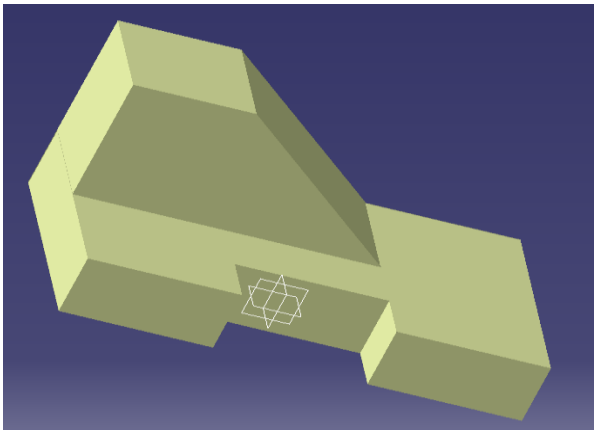
- Select exit work bench command to exit
- Select Extrude command and extrude the length is 12mm



- Select Surface fill command to select All edges of the Object.. Just click on enter.. ok



- Select Line or Profile command To create Line length is 50mm and again create line length is 25mm
- Select line command to create line length is 20mm and join the line of corner point.
- Select Profile command to create a line length is 50mm ,height is 25mm,length is 20mm Create one mounted model.
- Select exit work bench command
- Select Extrude command to extrude the length is 25mm
- Select Surface fill command and select the all lines of surface area select it.. Fill the surface area of the model.



RESULT: Hence the Exercise On Wireframe &Surface Modelling by the showing figure with specified dimensions is Created by using Catia software

EXPERIMENT NO:6

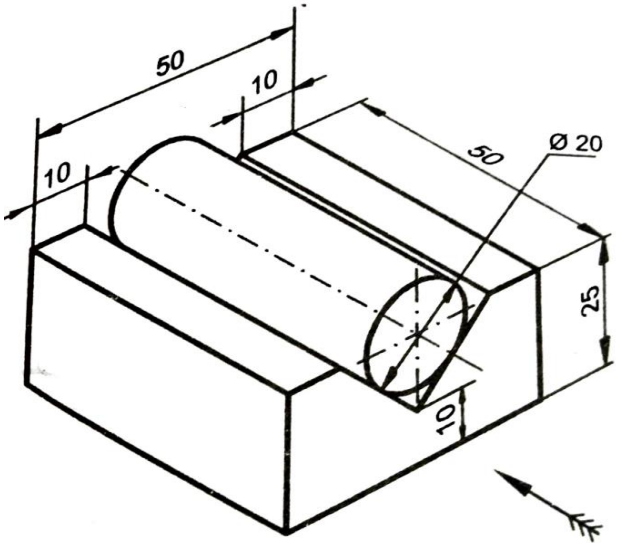
WIREFRAME &SURFACE MODELLING IN VBLOCK WITH CIRCULER ROD

Aim: To Create the wire frame and surface model in v-block with circular rod in the showing figure with specified dimensions..

Software required

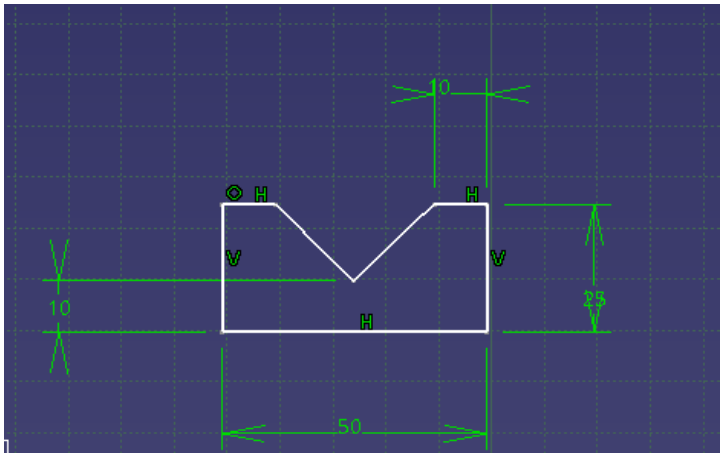
Personal Computer with CATIA software Package

Drawing:

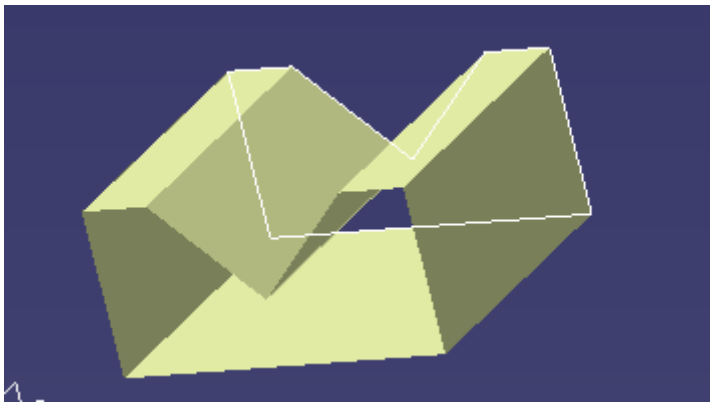


PROCEDURE:

- Click on the CATIA icon, present on the desktop
- Go to file 'MENU' and click on 'NEW' a dialogue box is opened select mechanical design and select wire frame Surface and design
- Select the Required YZ or XY plane go to select sketcher command.
- Select Profile command to create Line Length is 50mm, height is 25mm, length is 10mm with the help of constrain command
- Select Profile command to create Line height is 25mm length is 10mm with the help of constrain command
- Select Profile command create centre point of both sections with th height is 10mm.

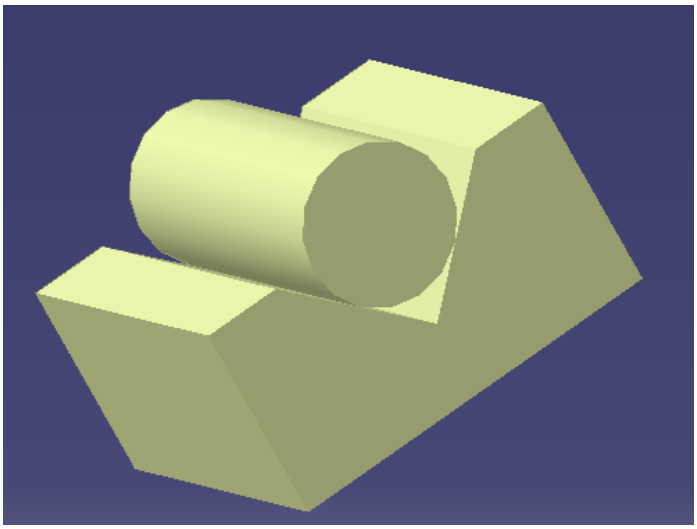


- Select Exit work bench command
- Select Extrusion command dimension length is 50mm then Extrude the v-block



- Select fill command Select one side of all 7 edges of the v-block then fill the surface area.
- Select one side of v-block go to sketcher command draw a circle by using circle command to given the circular diameter is 20mm by using constraint command
- Select circle & V-Block and crate tangency of the both v-block and circular rod
- Select Exit work bench command
- Select Extrusion command dimension length is 50mm then Extrude the circular rod by using constraint command
- Select fill command Select one side of all edges of the circle then fill the surface area.

- Select fill command Select another side of all edges of the circle then fill the surface area.
- Select fill command Select another side of all 7 edges of the v-block then fill the surface area.



RESULT: Hence the Exercise the wire frame and surface model in v-block with circular rod is Created by using Catia software

EXPERIMENT NO:7

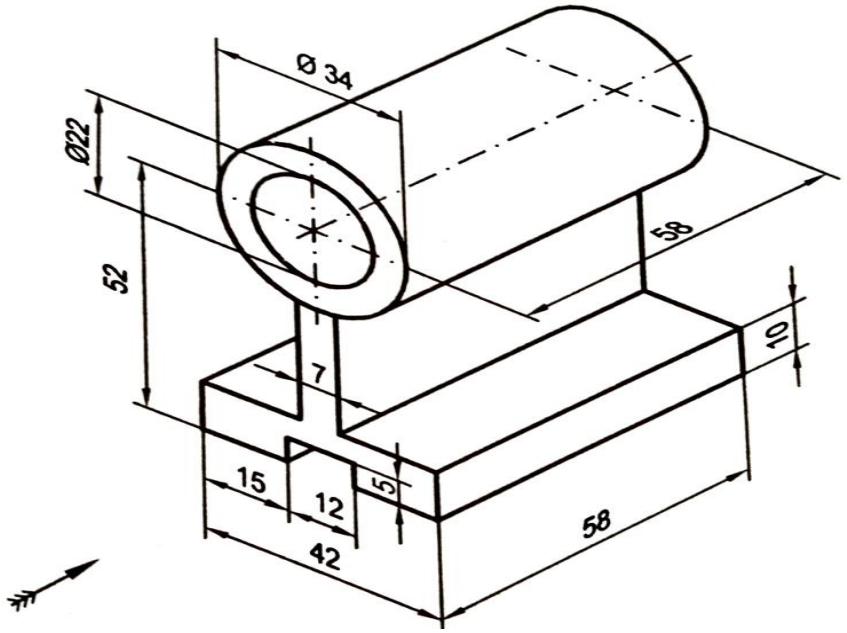
3D DRAFTING IN DRUM WITH MOUNTING

Aim: To Create 3d Drafting In Drum With Mounting in the showing figure with specified dimensions.

Software required

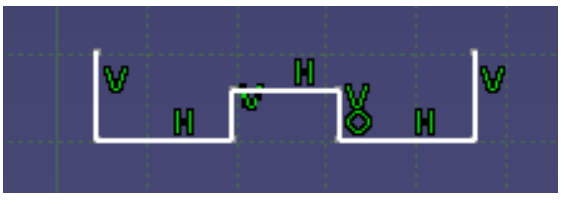
Personal Computer with CATIA software Package

Drawing:

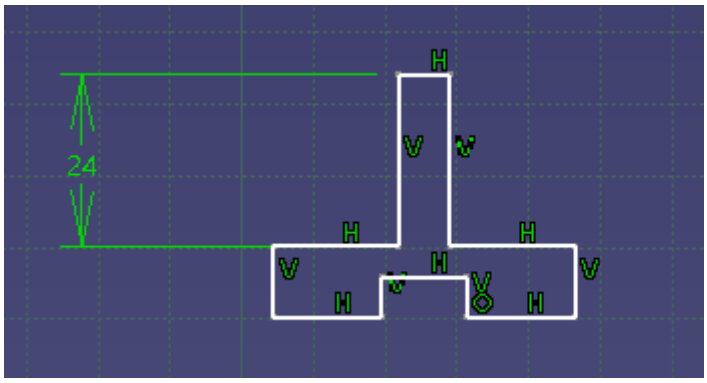


PROCEDURE:

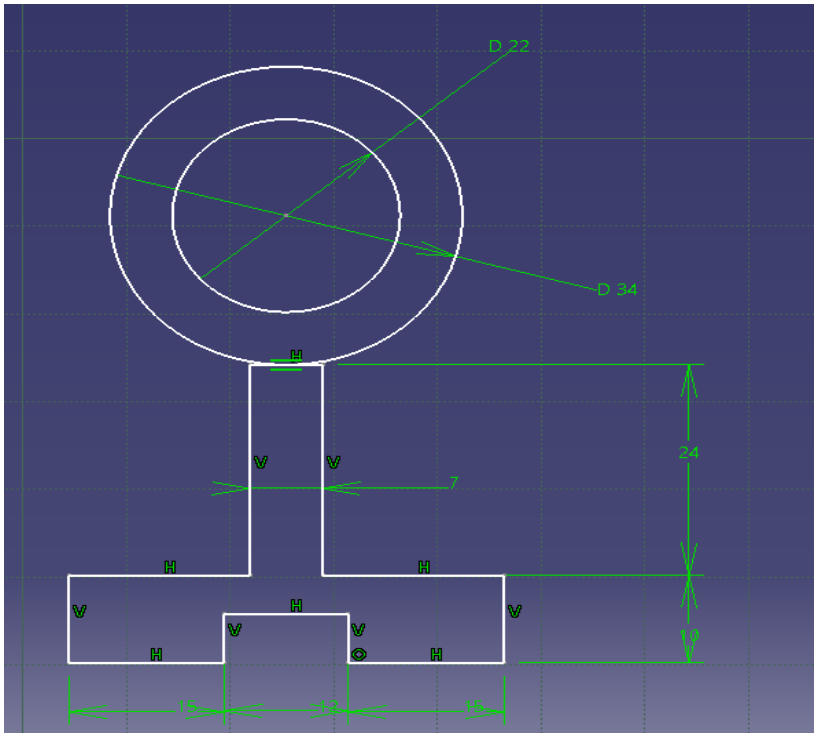
- Click on the CATIA icon, present on the desktop
- Go to file ‘MENU’ and click on ‘NEW’ a dialogue box is opened select mechanical design and select Part Design
- Select the Required YZ or XY plane go to select sketcher command.
- Select Profile command create line length is 10mm,15mm,12mm,15mm,10mm by using constrain command ---ok



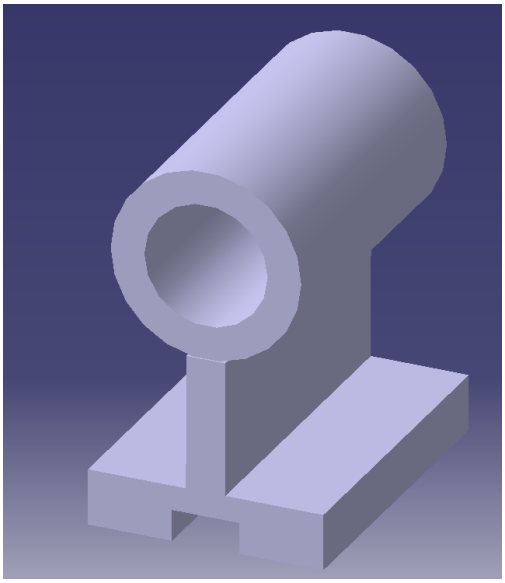
- Select Profile command create line length is 17.5mm and right side length is 17.5mm by using constrain command ---ok
- Select Profile command create line length is 24mm and right side length is 24mm by using constrain command ---ok



- Select axis line crate one horizontal axis line and vertical axis line to create a circles.
- Select constrain command to crate a distances is 52mm is horizontal axis line and vertical axis line is 3.5mm to create centre point.
- Select circle command to create a circle diameter is 22mm
- Select circle command to create a circle same centre point diameter is 34mm.



- Select Exit work bench command.
- Select pad command to given extrusion length is 58mm the extrude total object—ok



RESULT: Hence the Exercise on the 3d Drafting In Drum With Mounting is Created by using Catia software

EXPERIMENT NO:8

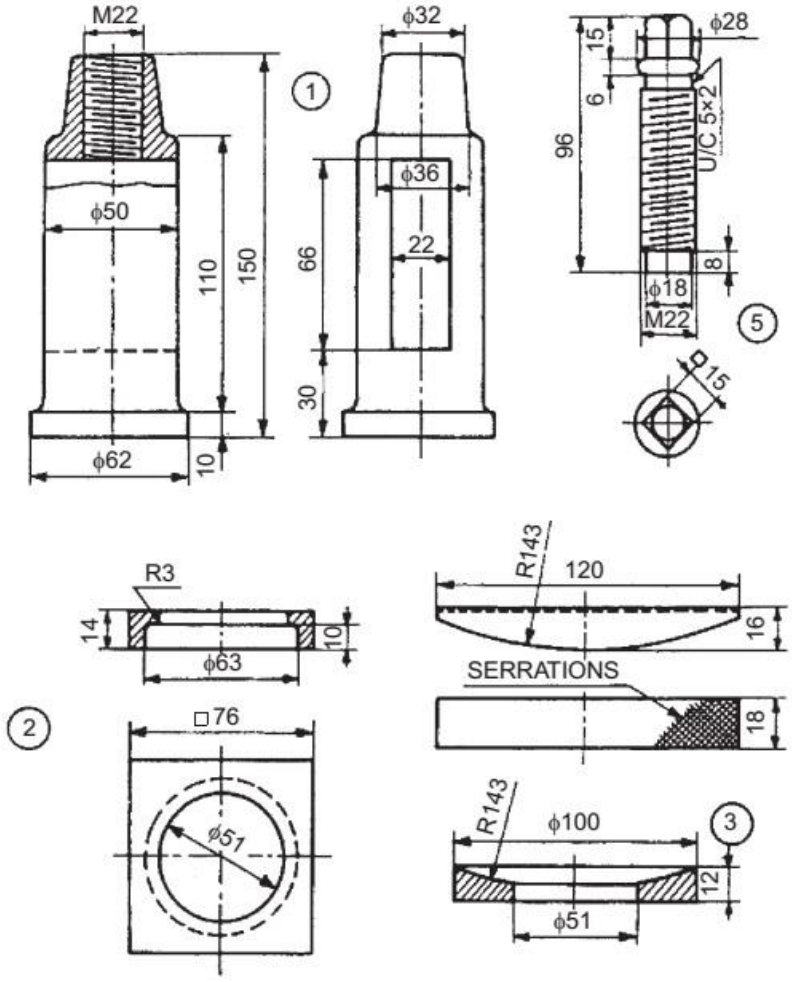
PRODUCTION DRAWING WITH GEOMETRIC DIMENSIONING AND TOLERANCES ON LATHE TOOL POST

Aim: To Create Production Drawing with Geometric Dimensioning and Tolerances in the Lathe Tool Post as showing figure with specified dimensions.

Software required

Personal Computer with CATIA software Package

Drawing:



Parts list

No.	Name	Matl	Qty
1	Piller	MCS	1
2	Block	MCS	1
3	Ring	MS	1
4	Wedge	MCS	1
5	Screw	TS	1

PROCEDURE:

- Click on the CATIA icon, present on the desktop
- Go to file 'MENU' and click on 'NEW' a dialogue box is opened select **mechanical design and select Drafting**
- Select the Required YZ or XY plane go to select sketcher command.

PART-1: PILLER PROCEDURE

- Select **Line** command to create line length 62mm and height is 10mm and create Rectangle.
- Select **Line** command to create line length 50mm and height is 110mm and create Rectangle.
- Select **Line** command to create line length 36mm and height is 30mm and another line length is 32mm and create Tapered shape.
- Select **Line** command to create line length 62mm and height is 10mm and create Rectangle.
- Select **Line** command to create line length 50mm and height is 86mm and create Rectangle.
- Select **Line** command to create line length 36mm and height is 54mm and another line length is 32mm and create Tapered shape.
- Select **Line** command to create line length 22mm and height is 66mm to create Rectangle shape.
- Select dimension command to give the dimensions of whole part

PART-2: BLOCK PROCEDURE

- Select **Line** command to create line length 76mm and height is 14mm to create Rectangle shape.

- Select **Line** command to create line length 66mm and height is 10mm to create Rectangle shape
- Select **arc** command to create arc radius 3mm and height is 4mm to create arcs both sides.
- Select dimension command to give the dimensions of whole part

PART-3: RING PROCEDURE

- Select **Line** command to create line length 100mm and height is 12mm to create Rectangle shape.
- Select **Line** command to create line length 51mm and height is 5mm to create Rectangle shape
- Select **arc** command to create arc radius 143mm and height is 7mm to create arcs both sides.
- Select dimension command to give the dimensions of whole part

PART-4: WEDGE PROCEDURE

- Select **Line** command to create line length 120mm create line
- Select **Arc** command to create arc 143mm and height is 16mm to crate a shape.
- Select **Rectangle** command to create line length 120mm and height is 18mm to crate rectangle.
- Select **area fill** command to create knurling shape.
- Select dimension command to give the dimensions of whole part

PART-5: SCREW PROCEDURE

- Select **Line** command to create line length 18mm and height is 8mm to create Rectangle.

- Select **Line** command to create line length 22mm and height is 62mm to create Rectangle.
- Select **Line** command to create line length 18mm and height is 5mm to create Rectangle
- Select **Line** command to create line length 18mm and height is 6mm to create Rectangle
- Select **Line** command to create line length 18mm and height is 15mm to create Rectangle
- Select dimension command to give the dimensions of whole part.

RESULT: Hence the Production Drawing with Geometric Dimensioning and Tolerances in the Lathe Tool Post created by using CATIA Software

EXPERIMENT NO:9

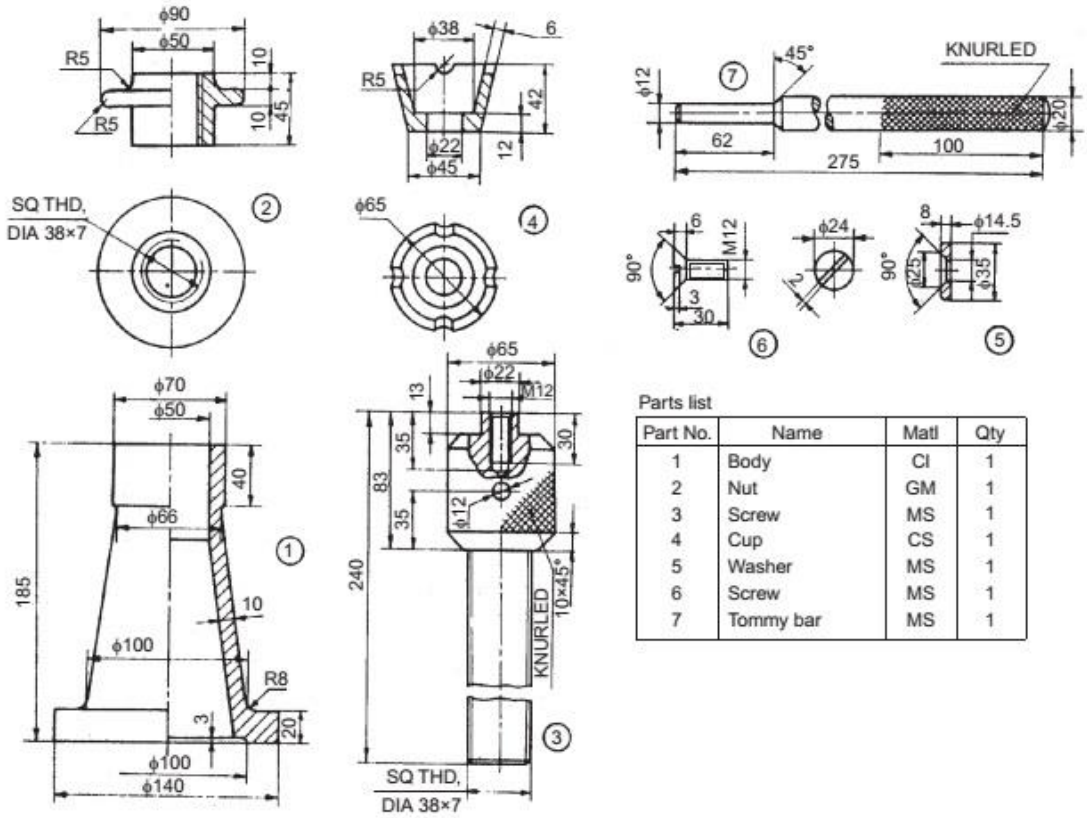
PRODUCTION DRAWING WITH GEOMETRIC DIMENSIONING AND TOLERANCES ON SCREW JACK

Aim: To Create Production Drawing with Geometric Dimensioning and Tolerances in the Screw jack as showing figure with specified dimensions.

Software required

Personal Computer with CATIA software Package

Drawing:



Parts list

Part No.	Name	Matl	Qty
1	Body	CI	1
2	Nut	GM	1
3	Screw	MS	1
4	Cup	CS	1
5	Washer	MS	1
6	Screw	MS	1
7	Tommy bar	MS	1

PROCEDURE:

- Click on the CATIA icon, present on the desktop
- Go to file ‘MENU’ and click on ‘NEW’ a dialogue box is opened select **mechanical design and select Drafting**
- Select the Required YZ or XY plane go to select sketcher command.

PART-1: BODY PROCEDURE

- Select **Line** command to create line length 140mm and height is 20mm to create Rectangle
- Select **Line** command to create line length 100mm and height is 125mm and length is 66mm and height is 125mm to create Tapered shape of the body
- Select **Line** command to create line length 70mm and height is 40mm to create Rectangle
- Select **Line** command to create line length 10mm height is 40mm ,145mm to create lines total body.
- Select **area fill** command to create any shape of sectioning to the body
- Select **arc** command to create arc radius is 8mm.
- Select dimension command to give the dimensions of whole part.

PART-2: NUT PROCEDURE

- Select **Line** command to create line length 50mm and height is 25mm to create Rectangle.
- Select **Line** command to create line length 90mm and height is 10mm to create Rectangle
- Select **Line** command to create line length 50mm and height is 10mm to create Rectangle.
- Select **Line** command to create line height is 10mm and Select **area fill command** to create any shape of sectioning to the body.
- Select **arc** command to arc radius is 5mm to crate arc both sides of the rectangle portion.
- Select dimension command to give the dimensions of whole part.

PART-3: SCREW PROCEDURE

- Select **Line** command to create line length 38mm and height is 202mm to create Rectangle.
- Select **Line** command to create line length 65mm and height is 10mm with 45 degrees angle to join the Rectangle portion
- Select **Line** command to create line length 65mm and height is 50mm to create Rectangle portion
- Select **Line** command to create line length 65mm and height is 10mm with 45 degrees angle to join the Rectangle portion
- Select **Rectangle** command to create line length 20mm and height is 30mm to create Rectangle portion
- Select **circle** command to create circle at radius 12mm centre of the small rectangle portion
- Select **area fill** command to create any shape of sectioning to the body
- Select **area fill** command to create knurling shape of sectioning to the body
- Select dimension command to give the dimensions of whole part.

PART-4: CUP PROCEDURE

- Select **Line** command to create line length 45mm and height is 42mm create rectangle with 45 degrees angle.
- Select **Line** command to create line length 26 mm and height is 33mm create rectangle with 45 degrees angle.
- Select **area fill command** to create any shape of sectioning to the body
- Select dimension command to give the dimensions of whole part.

PART-5: WASHER PROCEDURE

- Select **Rectangle** command to create line length 8mm and height is 25mm to create Rectangle .
- Select **area fill** comm band to create any shape of sectioning to the body
- Select dimension command to give the dimensions of whole part.

PART-6: SCREW PROCEDURE

- Select **Rectangle** command to create line length 24mm and height is 22mm to create Rectangle.
- Select **Line** command to create line length 24mm and height is 6mm create rectangle with 45 degrees angle.
- Select **area fill** command to create any shape of sectioning to the body
- Select dimension command to give the dimensions of whole part.

PART-7: TOMMYBAR PROCEDURE

- Select **Line** command to create line length 213mm and height is 20mm create rectangle with 45 degrees angle.
- Select **Line** command to create line length 62mm and height is 12mm create rectangle with 45 degrees angle.
- Select **area fill** command to create knurling shape of sectioning to the body.
- Select dimension command to give the dimensions of whole part.

RESULT: Hence the Production Drawing with Geometric Dimensioning and Tolerances in the **SCREW JACK** created by using CATIA Software

EXPERIMENT NO:10

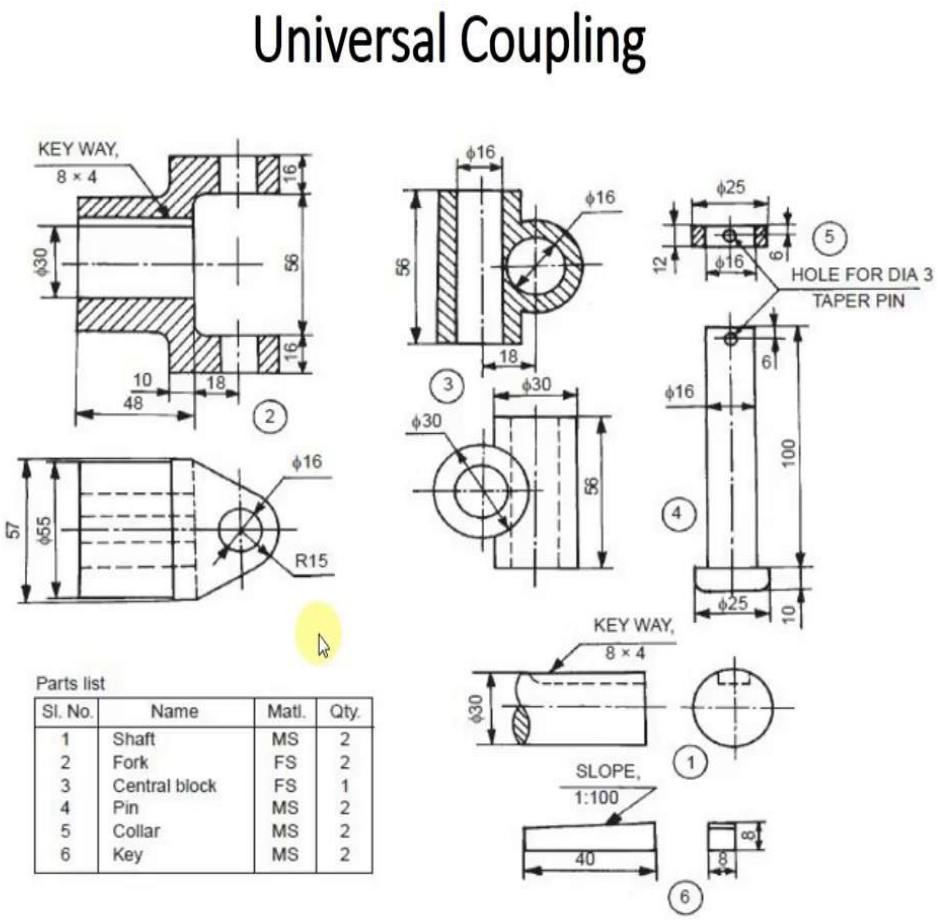
PRODUCTION DRAWING WITH GEOMETRIC DIMENSIONING AND TOLERANCES ON UNIVERSAL COUPLING

Aim: To Create Production Drawing with Geometric Dimensioning and Tolerances in the Universal Coupling as showing figure with specified dimensions.

Software required

Personal Computer with CATIA software Package

Drawing:



Parts list

Sl. No.	Name	Matl.	Qty.
1	Shaft	MS	2
2	Fork	FS	2
3	Central block	FS	1
4	Pin	MS	2
5	Collar	MS	2
6	Key	MS	2

PROCEDURE:

- Click on the CATIA icon, present on the desktop
- Go to file ‘MENU’ and click on ‘NEW’ a dialogue box is opened select **mechanical design and select Drafting**
- Select the Required YZ or XY plane go to select sketcher command.

PART-1: SHAFT PROCEDURE

- Select **Line** command to create line length 48mm and height is 30mm to create Rectangle
- Select **Circle** command to create circle diameter is 30mm.
- Select dimension command to give the dimensions of whole part.

PART-2: FORK PROCEDURE

- Select **Line** command to create line height 56mm and length is 38mm ,height 16mm,length is 46mm,height is 16mm,line length is 36mm,height is 56mm,length is 36mmand 10mm,height 18mm,length 38mm..Enter .
- Select **area fill** command to create any shape of sectioning to the body
- Select **Line** command to create line height 57mm and length is 48mm,create line height is55mm,length is 38mm…Enter.
- Select **Circle** command to create circle diameter is 30mm and 16mm.
- Select dimension command to give the dimensions of whole part.

PART-3: CENTRAL BLOCK PROCEDURE

- Select **Line** command to create line height 56mm and length is 30mm,create Rectangle
- Select **Line** command to create line height 57mm and length is 16mm..enter
- . Select **Line** command to create line length is 18mm,From Centre of the axis line…Enter.
- Select **Circle** command to create circle diameter is 30mm and 16mm
- Select **area fill** command to create any shape of sectioning to the body.
- Select dimension command to give the dimensions of whole part.

PART-4: PIN PROCEDURE

- Select **Line** command to create line length 25mm and height is 10mm to create Rectangle.
- Select **Line** command to create line length 16mm and height is 100mm to create Rectangle
- Select **Circle** command to create circle diameter is 3mm.
- Select dimension command to give the dimensions of whole part.

PART-5: COLLAR PROCEDURE

- Select **Line** command to create line length 25mm and height is 12mm to create Rectangle.
- Select **Line** command to create line length 16mm height is 10mm to create Rectangle.
- Select **Circle** command to create circle diameter is 3mm.
- Select dimension command to give the dimensions of whole part.

PART-6: KEY PROCEDURE

- Select **Line** command to create line length 40mm and height is 8mm to create Rectangle.
- Select dimension command to give the dimensions of whole part.

RESULT: Hence the Production Drawing with Geometric Dimensioning and Tolerances in the **UNIVESAL COUPLING** created by using CATIA Software