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COMPUTER AIDED MACHINE DRAWING
LAB MANUAL
(20A03404)

PREPARED
BY
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DEPARTMENT OF MECHANICAL ENGINEERING

PREFACE

CATIA IS THE MOST EMERGING TECHNOLOGY IN THE FIELD OF MECHANICAL ENGINEERING.

The Version 5 Part Design application makes it possible to design precise 2D & 3D mechanical parts with an intuitive and flexible user interface, from sketching in an assembly context to iterative detailed design. Version 5 Part Design application will enable you to accommodate design requirements for parts of various complexities, from simple to advanced.

This application, which combines the power of feature-based design with the flexibility of a Boolean approach, offers a highly productive and intuitive design environment with multiple design methodologies, such as post-design and local 3D parameterization.

As a scalable product, Part Design can be used in cooperation with other current or future companion products such as Assembly Design and Generative Drafting. The widest application portfolio in the industry is also accessible through interoperability with CATIA Solutions Version 4 to enable support of the full product development process from initial concept to product in operation.

The Part Design User's Guide has been designed to show you how to create a part. There are several ways of creating a part and this book aims at illustrating the several stages of creation you may encounter.

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

COURSEOUT COMES

CO1	Demonstrate the conventional representations of materials and machine components
CO2	Model riveted, welded and key joints using CAD system
CO3	Create solid models and sectional views of machine components
CO4	Generate solid models of machine parts and assemble them.
CO5	Translate 3D assemblies into 2D drawings. • Create manufacturing drawing with dimensional and geometric tolerances.

JAWAHARLAL NEHRU TECHNOLOGICAL UNIVERSITY ANANTAPUR

B. Tech II-II Sem. (ME)

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COMPUTER AIDED MACHINE DRAWING LABORATORY(20A03404)

The following contents are to be done by any 2D software package

Conventional representation of materials and components:

Detachable joints: Drawing of thread profiles, hexagonal and square-headed bolts and nuts, bolted joint with washer and locknut, stud joint, screw joint and foundation bolts.

Riveted joints: Drawing of rivet, lap joint, butt joint with single strap, single riveted, double riveted double strap joints.

Welded joints: Lap joint and T joint with fillet, butt joint with conventions.

Keys: Taper key, sunk taper key, round key, saddle key, feather key, woodruff key.

Couplings: rigid – Muff, flange; flexible – bushed pin-type flange coupling, universal coupling, Old hams' coupling

The following contents to be done by any 3D software package
Sectional views
Creating solid models of complex machine parts and create sectional views.

Assembly 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,

Manufacturing drawing: Representation of limits, fits and tolerances for mating parts.

Use any four parts of above assembly
drawings and prepare manufacturing drawing with dimensional and geometric tolerances.

LIST OF EXPERIMENTS

COMPUTER AIDED MACHINE DRAWING LABORATORY(20A03404)

The following contents are to be done by any 2D software package

1. Modelling on Conventional representation of materials and components
2. Modelling on Detachable joints in Drawing Of Thread Profiles
3. Modelling Detachable joints in Hexagonal and Square-Headed Bolts
4. Modelling on Detachable joints in Hexagonal Headed Bolt With A Nut &Washer
5. . Modelling on Detachable joints in Stud Joint
6. Modelling Of Component In Feather Keys & Saddle Keys
7. Modelling on Riveted joints in Single Riveted Lap Joint
8. Modelling on Muff coupling Components

The following contents to be done by any 3D software package

Sectional views

9.Modelling of components in Sectional Views of Machine Part

10.Modelling on Assembly Drawing in Axle Bearing Engine Component and prepare manufacturing drawing with dimensional and geometric tolerances

11 Modelling on Assembly Drawing in piston Component and prepare manufacturing drawing with dimensional and geometric tolerances

12 Modelling on Assembly Drawing in Plummer Block Component and prepare manufacturing drawing with dimensional and geometric tolerances

INSTRUCTIONS TO STUDENTS

1. Students are required to remove their footwear outside the center 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 center.
11. Edibles are strictly prohibited in the center. No internet browsing allowed during the lab hours
12. can be standardized and be used whenever required in any future drawings.

INTRODUCTION TO CATIA

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 Systems, 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 Modeling 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 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.

Part Design Workbench

The Part Design workbench is a parametric and feature-based environment, in which you we can create solid models.

Wireframe and Surface Design Workbench

1. The Wireframe and Surface Design workbench is also a parametric and feature - based environment, in which you can create wireframe or surface models.
2. The tools in this workbench are similar to those in the Part Design workbench, with the only difference that the tools in this environment are used to create basic and advanced surfaces.

Assembly Design Workbench

1. The **Assembly Design** workbench is used to assemble the components using the assembly constraints available in this workbench.
2. There are two types of assembly design approaches:
   1. Bottom-up
   2. Top-down

Drafting Workbench

1. The **Drafting** workbench is used for the documentation of the parts or the assemblies created earlier in the form of drawing views and their detailing.
2. There are two types of drafting techniques:
   1. Generative drafting
   2. Interactive drafting

SYSTEM REQUIREMENTS:

The following are the system requirements to ensure smooth running of CATIA V5R16 on our system:

• **System unit:** An Intel Pentium III or Pentium 4 based workstation running Microsoft 2000 Professional Edition or Windows XP Professional Edition.

• **Memory:** 256 MB of RAM is the minimum recommended for all applications. 512 MB of RAM is recommended for DMU applications.

• **Disk drive:** 4 GB Disk Drive space (Minimum recommended size)

• **Internal/External drives:** A CD-ROM drive is required for program installation.

• **Display:** A graphic color display compatible with the selected platform-specific graphic adapter. The minimum recommended monitor size is 17 inches.

• **Graphics adapter:** A graphics adapter with a 3D OpenGL accelerator is required with minimum resolution of 1024x768 for Microsoft Windows workstations and 1280 x 1024 for UNIX workstations.

IMPORTANT TERMS AND DEFINITIONS:

Feature-based Modeling

- A feature is defined as the smallest building block that can be modified individually.
- A model created in CATIA V5 is a combination of a number of individual features and each feature is related to the other directly or indirectly.

Parametric Modeling

The parametric nature of a software package is defined as its ability to use the Standard properties or parameters in defining the shape and size of a geometry.

CAT Part

CAT Part is a file extension associated with all the files that are created in Sketcher, Part Design, and Wireframe and Surface Design workbenches of CATIA V5.

CAT Product

CAT Product is a file extension associated with all the files that are created in Assembly Design workbench of CATIA V5.

CAT Drawing

CAT Drawing is a file extension associated with all the files that are created in Drafting workbench of CATIA V5.

Specification Tree

1. The specification tree keeps a track of all the operations that are carried on the part, as shown in the figure.
2. The specification tree that appears when you start a new file under the Part Design workbench, is as shown in the figure.

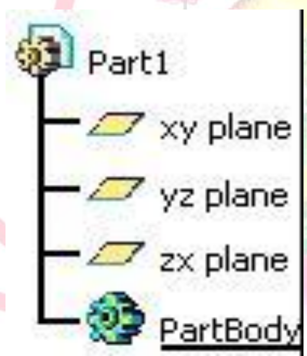
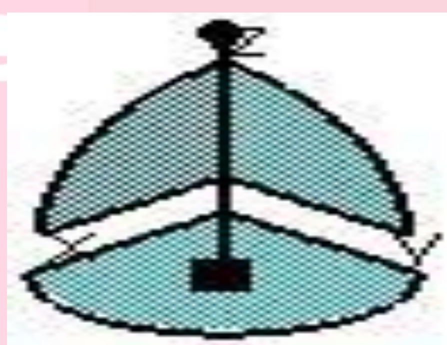


FIG.: The specification tree that appears on starting a new CAT Part file

Compass

1. It is a tool that is used to manipulate the orientation of parts, assemblies, or sketches.
2. You can also orient the view of the parts and assemblies.
3. By default, it appears on the top right corner of the geometry area.



Compass

Constraints:

1. Constraints are logical operations that are performed on the selected element to define its size and location with respect to other elements or reference geometries.

2. The constraints in **Sketcher** workbench are called geometric constraints and the constraints available in the **Assembly Design** workbench are called assembly constraints.

Geometric Constraints:

These are the logical operations performed on sketched elements to define their size and position with respect to other elements.

BASIC COMMANDS IN CATIA:

1.Profile: Create line from one point to another point

2.Line: Create line from one point to another point

3.Circle: Create circle from centre point

4.Spline: Create spline curve

5.Rectangle: Create Rectangle

6.Elipse: Create ellipse curve

7.Parabola: Create Parabola curve

8.Axis: Create axis line

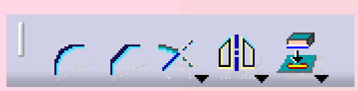
9.Chamfer: Create chamfer inclined line two lines

10.Fillet: Create chamfer curve two lines

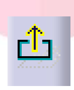
11.Mirror: Create same object from the axis another side

12.Trim or quick trim: Remove Extra lines or object

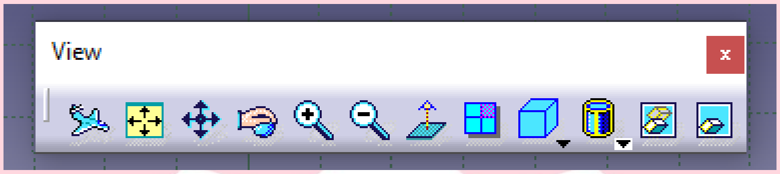
13.offset: Create line Parallel & perpendicular directions of the object



14.Work bench: it uses to convert 2d to 3d objects



15.View: fit,zoom,rotate,fly mode,box commands



16.Rotae: Rotate object in any directions
17.Zoom: The object is Enlarge
18.Fit: The object is Fit on the Screen

19.SKETCHED BASE FEATURES::

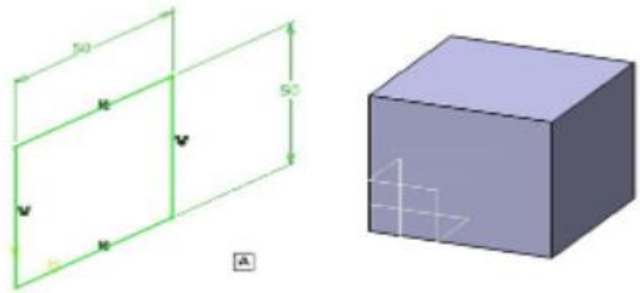
When it is required to make a simple solid component, we usually go to Part Design workbench. Important commands in this workbench include

PAD,
POCKET,
SHAFT,
GROOVE,
RIB and SLOT.

All these commands come under a toolbar, named Sketch-Based Features. That means, we are required to have a Sketch to make use of a particular command. Let's see how each of these command are used.

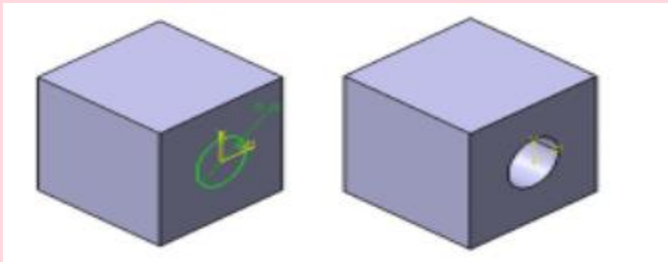
PAD command

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



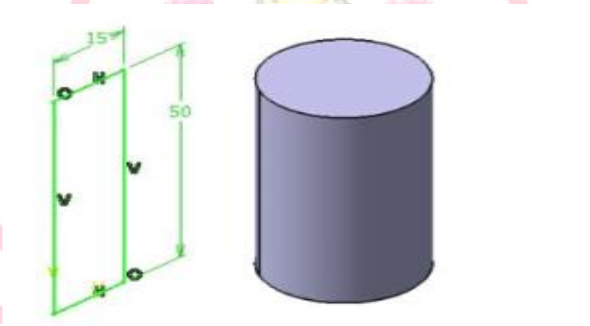
POCKET command

In CATIA, The POCKET command is 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 us to create the cylindrical hole in the middle of the cube.



SHAFT command

Similar to 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.

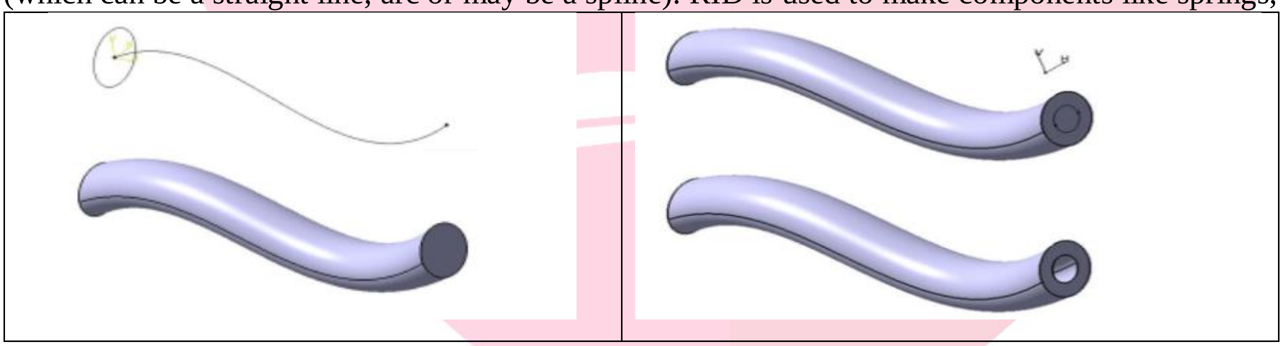


GROOVE command

As said earlier, there is another command in CATIA to subtract geometry from shaft like components, called GROOVE. This command allows you to remove material by revolving a sketch.

RIB command

The command which is usually known as SWEEP is called RIB in CATIA. It adds material along a guide curve (which can be a straight line, arc or may be a spline). RID 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.

EXP NO:1

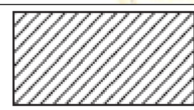
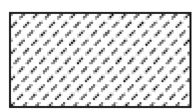
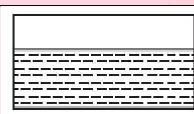
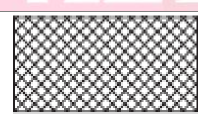
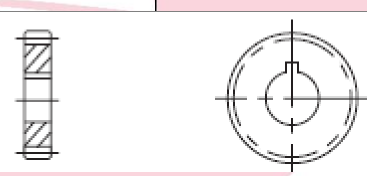
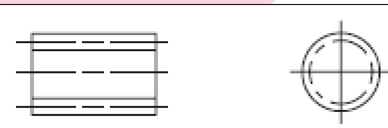
MODELLING ON CONVENTIONAL REPRESENTATION OF MATERIALS AND COMPONENTS

Aim: To Draw the **Conventional Representation Of Materials And Components as** Shown In Drawing By Using 2d Drafting Commands

Software Required:

PC with CATIA software Package

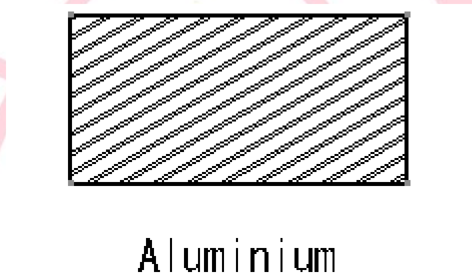
Drawing:

S.NO	Type	Convention	Materials & Component
1	**Metals**		**Steel, Cast Iron, Copper and its Alloys, Aluminium and its Alloys, etc.**
2	**Glass**		**Glass**
3	**Packing and Insulating material**		**Porcelain, Stoneware, Marble, Slate, etc.**
4	**Liquids**		**Water, Oil, Petrol, Kerosene, etc.**
5	**Metals**		**Lead, Zinc, Tin, White-metal, etc.**
6	**Spur gear**		
7	**Worm**		

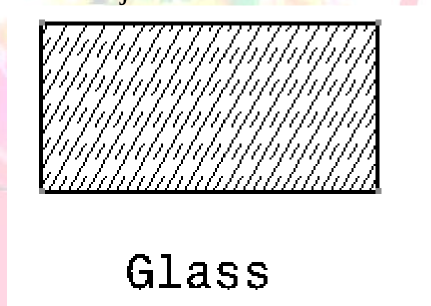
SEQUENCE OF COMMANDS REQUIRED TO DRAW THE MODEL

PROCEDURE:

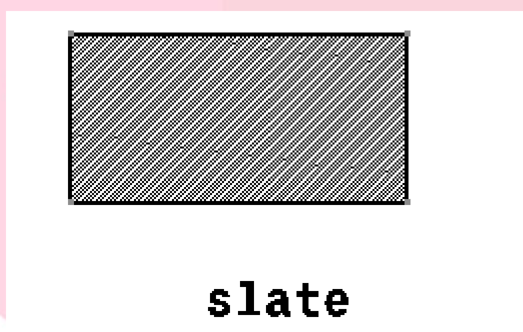
- Click on the **CATIA** Icon, present on the desk top.
- Go to file menu and click on new ,a dialogue box is opened, select Mechanical design & Drafting feature From the list in the box---select new sheet----ok
- Select the Rectangle command create Rectangle with dimensions
- Select area fill creation command—select object & click on it **Steel, Cast Iron, Copper and its Alloys, Aluminum and its Alloys, etc.** material.



- Select the Rectangle command create Rectangle with dimensions
- Select area fill creation command—select object & click on it **Glass** material.

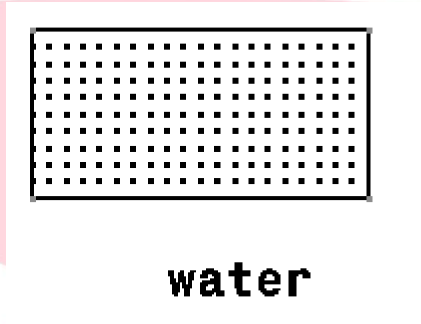


- Select the Rectangle command create Rectangle with dimensions
- Select area fill creation command—select object & click on it **Porcelain, Stoneware, Marble, Slate,** material.

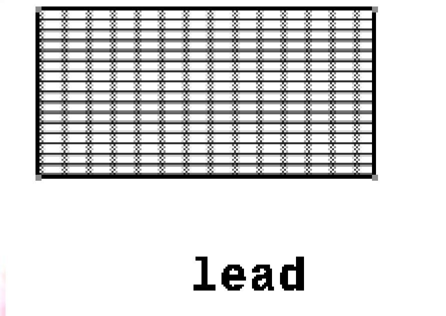


- Select the Rectangle command create Rectangle with dimensions

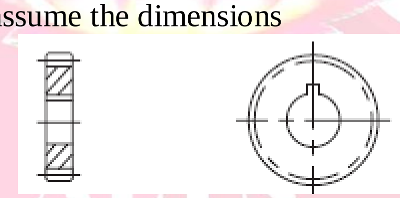
- Select area fill creation command—select object & click on it **Water, Oil, Petrol, Kerosene, etc.** material.



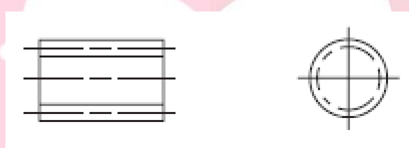
- Select the Rectangle command create Rectangle with dimensions
- Select area fill creation command—select object & click on it **Lead, Zinc, Tin, White-metal,** material.



- Select **dimension** command ae per assume the dimensions
- Select **Profile, circle** commands to create **spur gear** machine component
- Select **dimension** command ae per assume the dimensions



- Select **Profile, circle** commands to create **spur gear** machine component
- Select **dimension** command ae per assume the dimensions



- Select Tool command---select image-camera select---save the file in desk top folder

Result: Thus the 2D model of conventional representation of Materials & components is Created by CATIA Software.

EXP NO:2

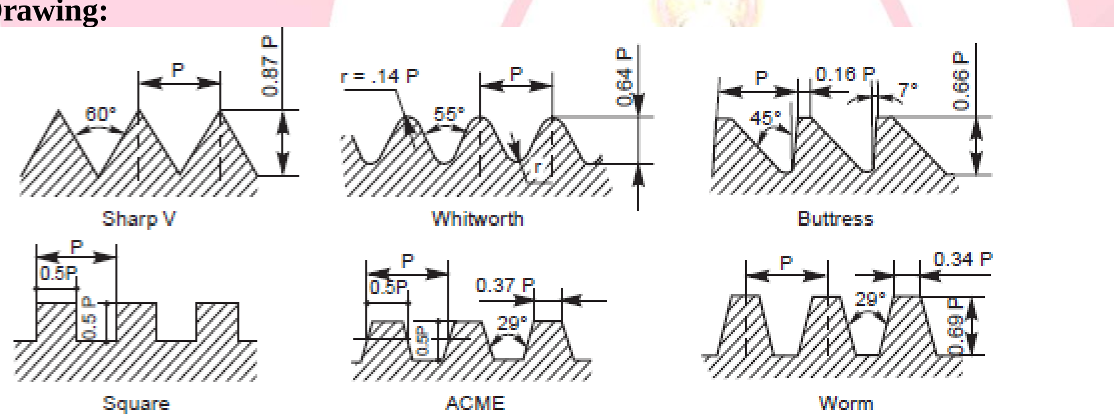
MODELLING ON DETACHABLE JOINTS IN DRAWING OF THREAD PROFILES

Aim: To Draw The **DRAWING OF THREAD PROFILES as** Shown In Drawing By Using 2d Drafting Commands

Software Required:

PC with CATIA software Package

Drawing:



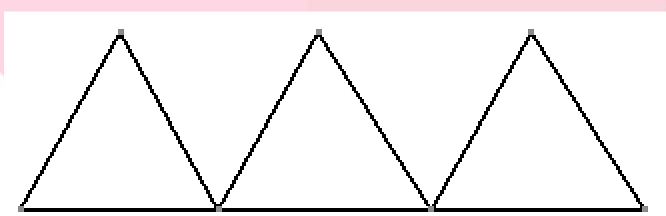
Sequence of Commands Required to draw the Model:

NOTE: Pitch P=30mm

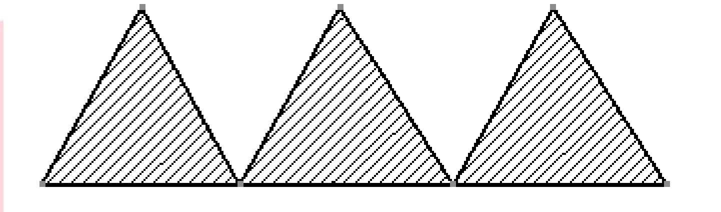
SEQUENCE OF COMMANDS REQUIRED TO DRAW THE MODEL

PROCEDURE:

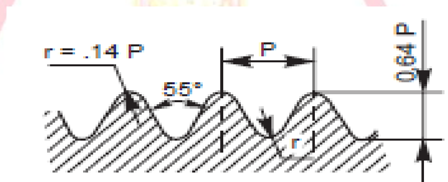
- Click on the **CATIA** Icon, present on the desk top.
- Go to file menu and click on new ,a dialogue box is opened, select Mechanical design & Drafting feature From the list in the box---select new sheet----ok
- Select the **Profile** command create sharp-v thread ,as per pitch is 30mm,angle is 60^0,height is 26.1mm with dimensions.



- Select area fill creation command—select object & click on its steel material

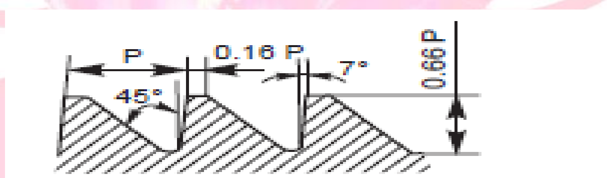


- Select the **Profile** command ,circle command create with worth thread ,as per pitch is 30mm,angle is 55^0,height is 19.2,radius is 4.2mm mm with dimensions.
- Select area fill creation command—select object & click on it material



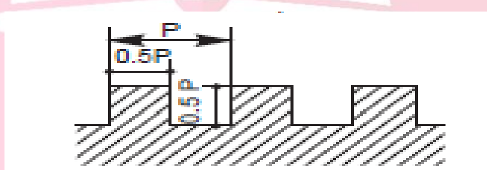
Whitworth

- Select the **Profile** command create Buttress thread ,as per pitch is 30mm,angle is 45^0,height is 19.8,width is 4.8mm mm with dimensions.
- Select area fill creation command—select object & click on it material



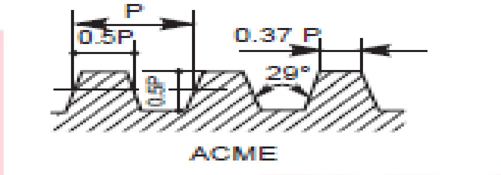
Buttress

- Select the **Profile** command create Square thread ,as per pitch is 30mm,angle is 90^0,height is 15mm,width is 15mm mm with dimensions.
- Select area fill creation command—select object & click on it material

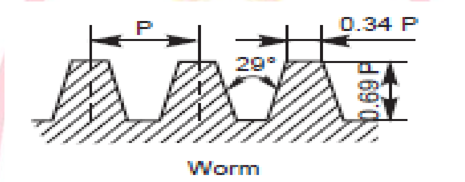


Square

- Select the **Profile** command create ACME Thread ,as per pitch is 30mm,angle is 29^0,height is 15mm,width is 15mm mm with dimensions.
- Select area fill creation command—select object & click on it material



- Select the **Profile** command create WORM Thread ,as per pitch is 30mm,angle is 29^0,height is 20.7mm,width is 10.2mm mm with dimensions.
- Select area fill creation command—select object & click on it material



- Select Tool command---select image-camera select---save the file in desk top folder

Result: Thus The 2d Modelling On Detachable Joints In Drawing Of Thread Profiles done by using Catia software.

EXP NO:3

MODELLING ON COMPONENT OF HEXAGONAL HEADED BOLT

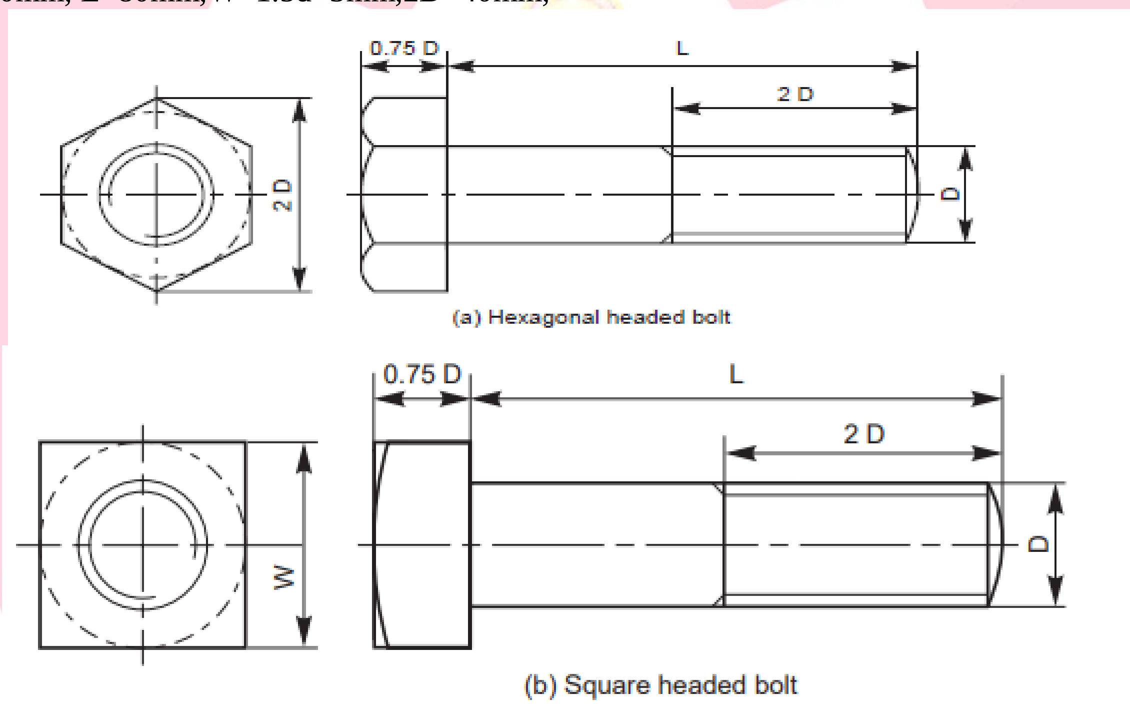
Aim: To Draw The hexagonal and square-headed bolts **as** Shown In Drawing By Using 2d Drafting Commands

Software Required:

PC with CATIA software Package

Drawing:

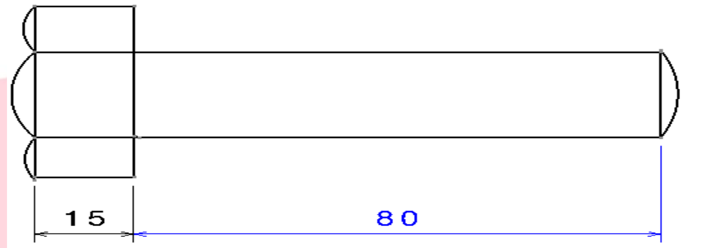
D=20mm, L=80mm,W=1.5d+3mm,2D=40mm,



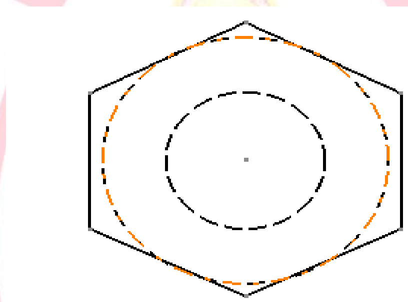
SEQUENCE OF COMMANDS REQUIRED TO DRAW THE MODEL

PROCEDURE:

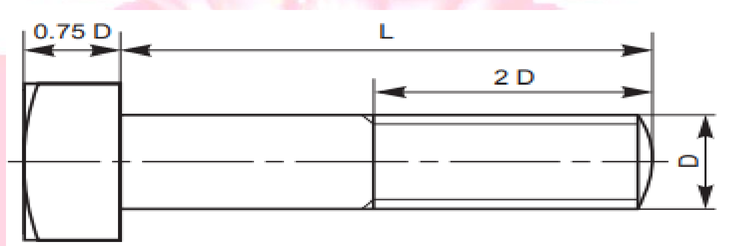
- Click on the **CATIA** Icon, present on the desk top.
- Go to file menu and click on new ,a dialogue box is opened, select Mechanical design & Drafting feature From the list in the box---select new sheet----ok
- Select the **Profile** command create line length is 100mm ,height is 20mm.
- Select the **Profile** command create line length is 15mm ,height is 40mm, create one Rectangle .
- Select the **ARC** command create the arcs as per given the dimensions
- Select **dimension** command ae per the dimensions



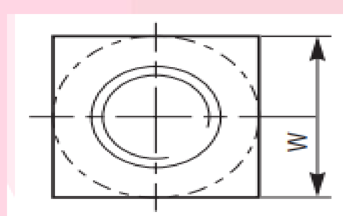
- Select the **Profile** command create hexagon length is 40mm
- Select circle command create two circles of side view with axis circle line thickness is 2 mm
- Select **dimension** command ae per the dimensions



- Select the **Profile** command create line length is 15mm ,height is 33mm, create one Rectangle .
- Select the **ARC** command create the arcs as per given the dimensions
- Select **dimension** command ae per the dimensions



- Select the **Profile** command create length and height is 33mm
- Select circle command create two circles of side view with axis circle line thickness is 2 mm
- Select **dimension** command ae per the dimensions



- Select Tool command---select image-camera select---save the file in desk top folder

Result: Thus The 2d Modelling On Detachable Joints In Drawing Of hexagonal & square headed bolts done by using Catia software.

EXP NO:4

MODELLING OF COMPONENT IN HEXAGONAL HEADED BOLT WITH A NUT &WASHER

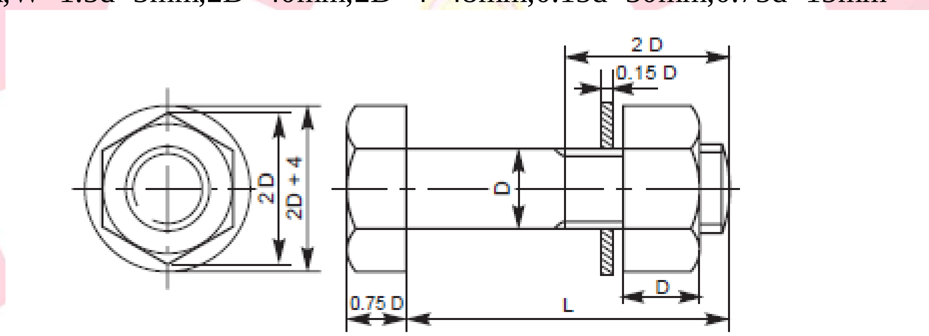
Aim: To Draw The hexagonal headed bolt with a nut and a washer in position **as** Shown In Drawing By Using 2d Drafting Commands

Software Required:

PC with CATIA software Package

Drawing:

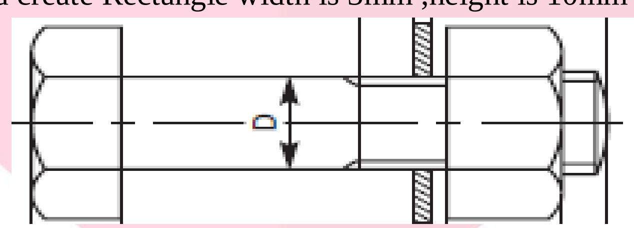
D=20mm, L=80mm,W=1.5d+3mm,2D=40mm,2D+4=48mm,0.15d=30mm,0.75d=15mm



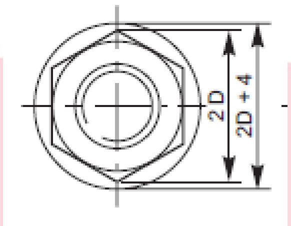
SEQUENCE OF COMMANDS REQUIRED TO DRAW THE MODEL

PROCEDURE:

- Click on the **CATIA** Icon, present on the desk top.
- Go to file menu and click on new ,a dialogue box is opened, select Mechanical design & Drafting feature From the list in the box---select new sheet----ok
- Select the **Profile** command create line length is 100mm ,height is 20mm.
- Select the **Profile** command create line length is 15mm ,height is 40mm, create one Rectangle .
- Select the **ARC** command create the arcs as per given the dimensions on both sides of the bolts
- Select **dimension** command ae per the dimensions
- Select the **Profile** command create Rectangle width is 3mm ,height is 10mm



- Select the **Profile** command create hexagon length ,height is 40mm
- Select circle command create three circles of side view with diameters is 40mm,20mm,18mm
- Select circle command create circle of side view with diameters is 44mm



- Select **dimension** command ae per the dimensions
- Select Tool command---select image-camera select---save the file in desk top folder

Result: Thus The 2d Modelling On Detachable Joints In Drawing Of hexagonal headed bolt with nut and washer done by using Catia software

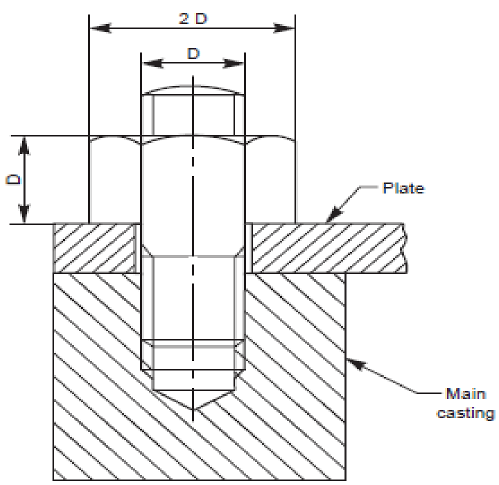
EXP NO: 5
MODELLING OF COMPONENT IN STUD JOINT

Aim: To Draw Modelling Of Component In Stud Joint **as** Shown In Drawing By Using 2d Drafting Commands

Software Required:

PC with CATIA software Package

Drawing:
D=20mm, ,2D=40mm



SEQUENCE OF COMMANDS REQUIRED TO DRAW THE MODEL
PROCEDURE:

- Click on the **CATIA** Icon, present on the desk top.
- Go to file menu and click on new ,a dialogue box is opened, select Mechanical design & Drafting feature From the list in the box---select new sheet----ok
- Select the **Profile** command create rectangular block with out dimension(main casting part)
- Select the **Profile** command create rectangular block with out dimension(plate part)
- Select the **Profile** command create rectangular block line width is is 40mm,height is 20mm length is assume to create stud part.
- Select the **ARC** command create the arcs as per given the dimensions
- Select **dimension** command ae per the dimensions
- Select the **Profile** command create rectangular block line width is 20mm length is assume to create box
- Select the **ARC** command create the arcs as per given the dimensions
- Select **dimension** command ae per the dimensions
- Select Tool command---select image-camera select---save the file in desk top folder
- **Result:** Thus The 2d Modelling On Detachable Joints In Drawing Of stud joint is done by using Catia software

EXP NO: 6

MODELLING OF COMPONENT IN FEATHER KEYS & SADDLE KEYS

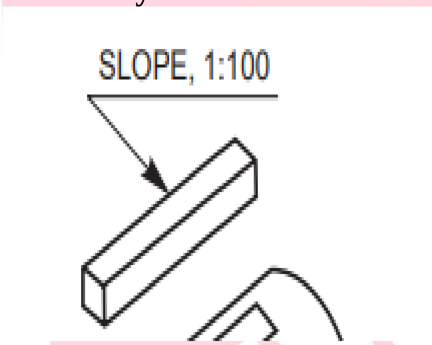
Aim: To Draw The Feather keys, saddle key, **as** Shown In Drawing By Using Catia software

Equipment:

PC with CATIA software Package

Drawing:

Saddle key



FEATHER KEYS

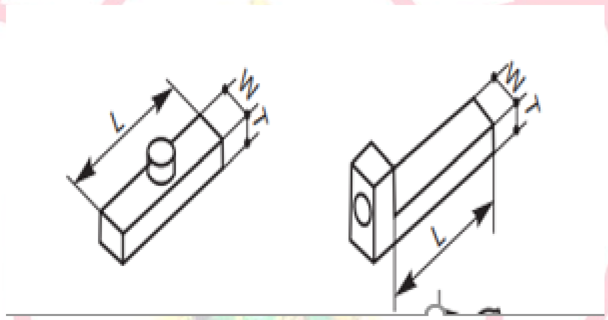


FIG NO.7.1

SADDLE KEY PRAPORTIONS:

D=20mm,L=60mm,
W=0.25*D+2=7mm,
t=0.67w=0.67*7=4.7mm

FEATHER KEYS PRAPORTIONS

D=20mm, L=80mm,
W=0.25*20+2=7mm,
t=0.67w=0.67*7=4.74mm

DESCRIPTION::

Keys are machine elements used to prevent relative rotational movement between a shaft and the parts mounted on it, such as pulleys, gears, wheels, couplings, etc. shows the parts of a keyed joint and its assembly. For making the joint, grooves or keyways are cut on the surface of the shaft and in the hub of the part to be mounted. After positioning the part on the shaft such that, both the keyways are properly aligned, the key is driven from the end, resulting in a firm joint. For mounting a part at any intermediate location on the shaft, first the key is firmly placed in the keyway of the shaft and then the part to be mounted is slid from one end of the shaft, till it is fully engaged with the key. Keys are classified into three types, viz., saddle keys, sunk keys and round keys

Sequence of Commands Required to draw the Model:

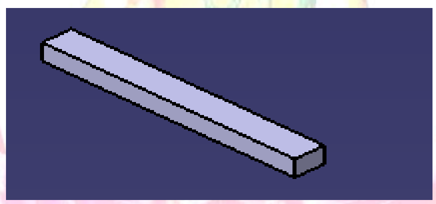
PROCEDURE:

STEP:1: Click on the CATIA icon, present on the desktop

- Go to file 'MENU' and click on 'NEW' ,a dialogue box is opened .select 'part' option from the list in the box.
- Select the Required plane YZ
- Select 'sketcher' option from 'sketcher tool bar'

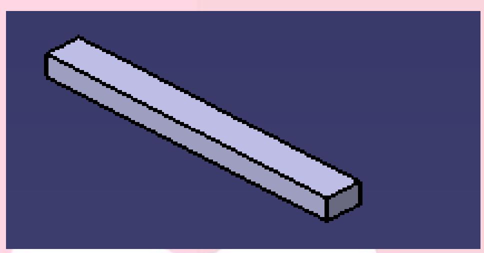
STEP:2: Create the following rough sketch using 'profile' icon 'profile tool bar'

- Give the dimensions to the rough sketch as shown below, using 'constraint' Icon of 'constraint tool bar'

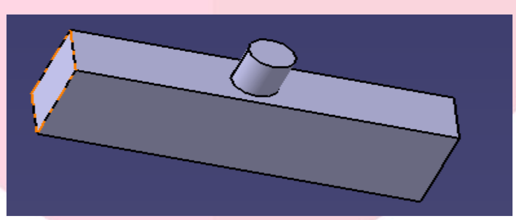


SADDLE KEY

STEP:3: Create the following rough sketch using 'profile' icon 'profile tool bar'



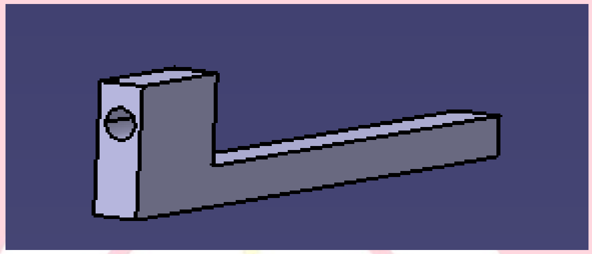
- Select one side of the key go to sketcher command
- Create the following rough sketch using 'circle' icon 'profile tool bar'



- Give the dimensions to the rough sketch as shown below, using 'constraint' Icon of 'constraint tool bar'

STEP:4: Create the following rough sketch using ‘profile’ icon ’profile tool bar’

- Create the following rough sketch using ‘circle’ icon ’profile tool bar’



- Select Tool command---select image-camera select---save the file in desk top folder

Result: Thus The Modelling Of Component In Feather Keys & Saddle Key are created by using Catia Software.

EXP NO: 7

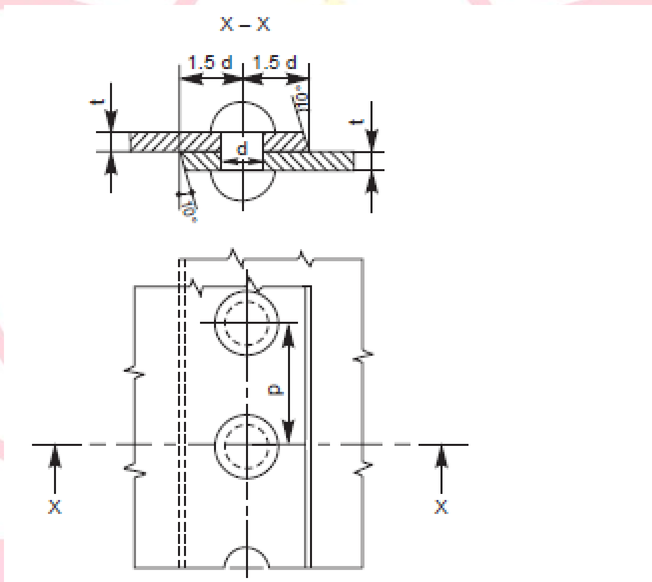
MODELLING OF COMPONENT IN SINGLE RIVETED LAP JOINT

Aim: To Draw The Single riveted lap joint **as** Shown In Drawing By Using 2d Drafting Commands

Equipment: PC with AUTO CAD software Package

Drawing:

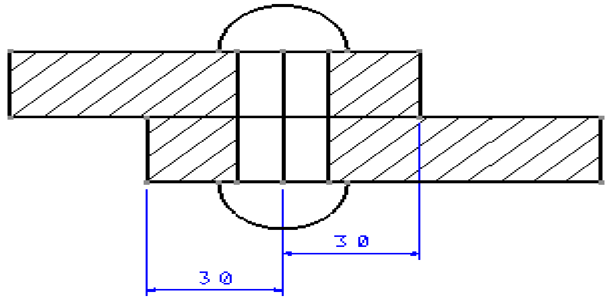
T =20mm, pitch =50mm
d=4.47mm,
1.5d=30mm



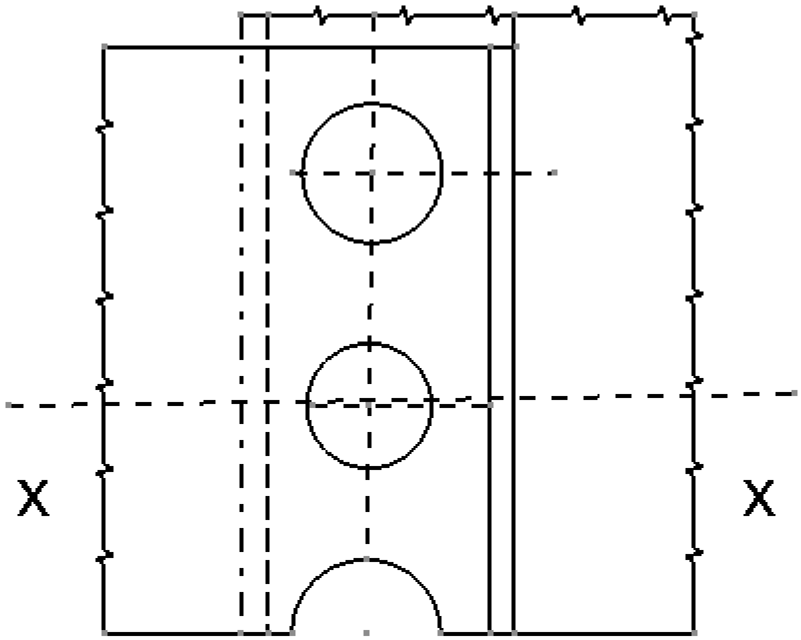
Description: Riveted joints are permanent fastenings and riveting is one of the commonly used method of producing rigid and permanent joints. Manufacture of boilers, storage tanks, etc., involve joining of steel sheets, by means of riveted joints. These joints are also used to fasten rolled steel sections in structural works, such as bridge and roof trusses.

PROCEDURE:

- Click on the **CATIA** Icon, present on the desk top.
- Go to file menu and click on new ,a dialogue box is opened, select Mechanical design & Drafting feature From the list in the box---select new sheet----ok
- Select the **Profile** command create rectangular block length is 90mm,thickness is20mm
- Select the **Profile** command create rectangular block length is 90mm,thickness is20mm
- Select **line** command draw a line length is 10mm left side and right side of the plates create center line of rivets
- Select **circl**e command draw a circle radius is 15mm and cut the parts is using **trim** command.



- Select the **Profile** command create projections lines with out dimension(plate part)
- Select the **line** command create projections lines with out dimension(plate part) by using break line
- Select the **line** command create axis projections lines with out dimension(plate part)
- Select **circle command** radius 15mm to create two circles
- Select **circle command** radius 15mm to create circle and remove half circle by using trim command



- Select **dimension** command ae per the dimensions
- Select Tool command---select image-camera select---save the file in desk top folder

Result: Thus The 2d Modelling On The Single riveted lap joint is done by using Catia software

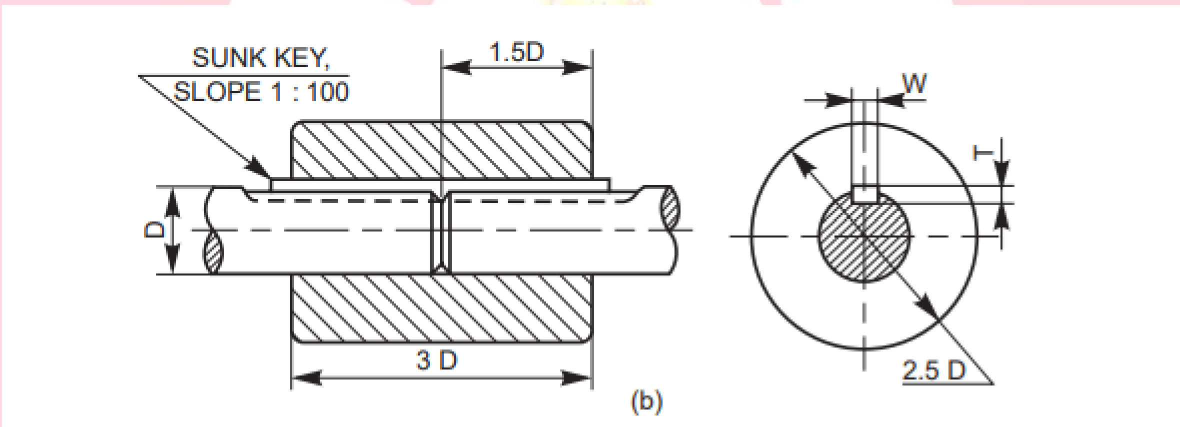
EXP NO: 8
MODELLING OF COMPONENT IN BUTT MUFF COUPLING

Aim: To Draw Modelling Of Component In Butt Muff Coupling **as** Shown In Drawing By Using Catia Software

Equipment:

PC with CATIA software Package

Drawing::



Proportions of Butt Muff Coupling:

D=20mm, L=80mm, 3D=60mm, 1.5D=30mm, 2.5D=50mm,

W=0.25*D+2=7mm,

t=0.67w=0.67*7=4.7mm

DESCRIPTION::

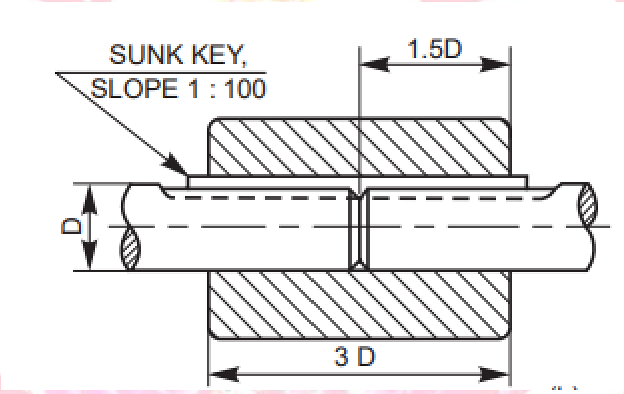
Shaft couplings are used to join or connect two shafts in such a way that when both the shafts rotate, they act as one unit and transmit power from one shaft to the other. Shafts to be connected or coupled may have collinear axes, intersecting axes or parallel axes at a small distance.

Sequence of Commands Required to draw the Model:

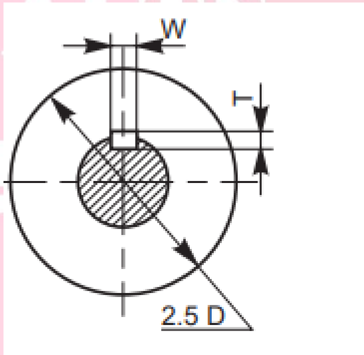
PROCEDURE:

- 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 & Drafting feature From the list in the box---select new sheet----ok

-
- Select **Profile** command create a line length is 100mm, height is 20mm.Create center line by using line command.
- Select **Profile** command create a line length is 60mm, height is 15mm.
- Select **Profile** command create a line length is 60mm, height is 15mm.
- Select line command and create key with values thickness is4.7mm and length is 80mm
- Select **dimension** command ae per the dimensions



- Select **Profile** command and draw the projection lines to create side view of the coupling
- Select **circle command** radius 20 mm to create circle
- Select **Profile** command and create width is 7mm and 4.7mm thickness to create key side view.
- Select **circle command** radius 50 mm to create circle



- Select **dimension** command ae per the dimensions
- Select Tool command---select image-camera select---save the file in desk top folder

Result: Thus The 2d- Modelling Of Component In Butt Muff Coupling is done by using Catia software

THE FOLLOWING CONTENTS TO BE DONE BY ANY 3D SOFTWARE PACKAGE

EXP NO: 9

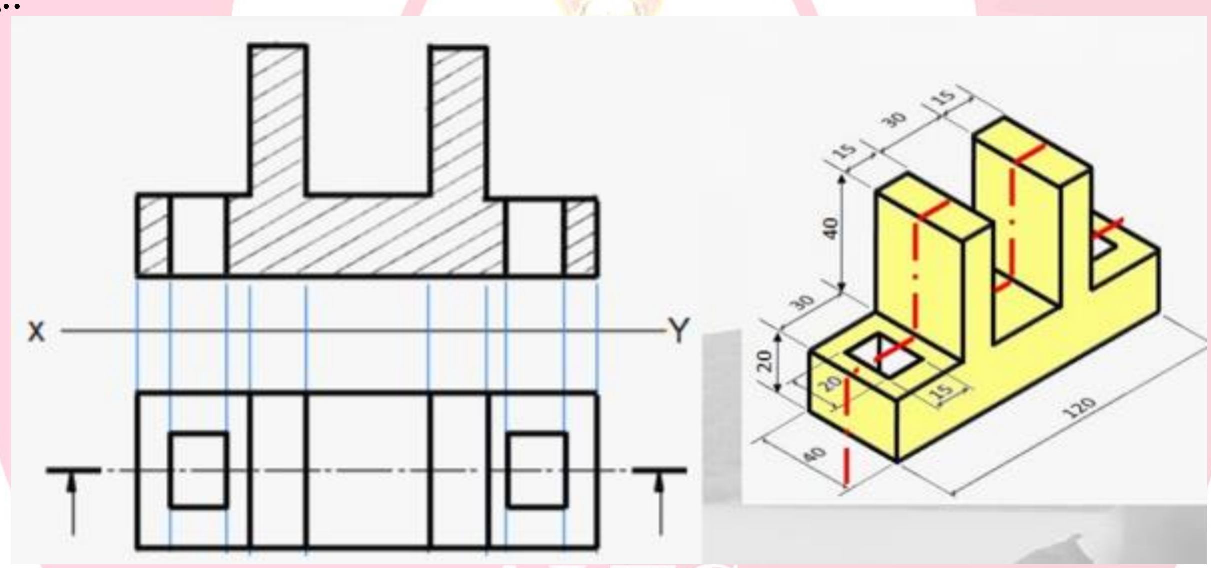
MODELLING OF COMPONENT IN SECTIONAL VIEWS OF MACHINE PART

Aim: To Draw Modelling Of Component In Sectional Views Of Machine Part **as** Shown In Drawing By Using Catia Software

Equipment:

PC with CATIA software Package

Drawing::

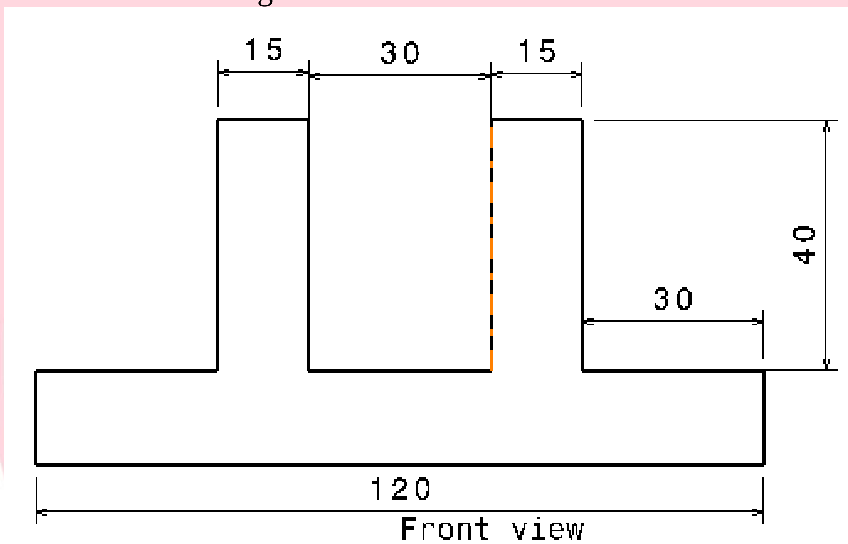


Sequence of Commands Required to draw the Model:

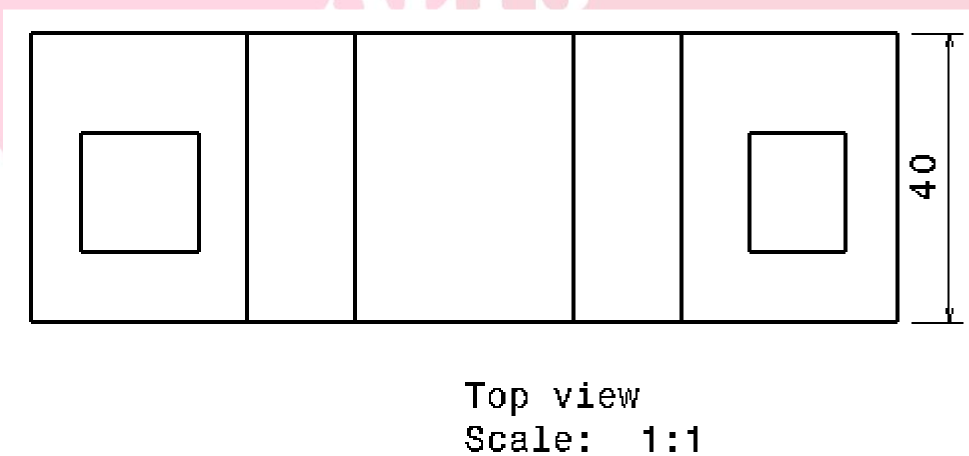
PROCEDURE:

- Click on the CATIA icon, present on the desktop
- Go to file 'MENU' and click on 'NEW' ,a dialogue box is opened .select 'part' option from the list in the box.
- Select the Required plane YZ
- Select 'sketcher' option from 'sketcher tool bar'
- Select profile command create line length is 120mm
- Select profile command create line height is 20mm
- Select profile command create line length is 30mm
- Select profile command create line length is 40mm
- Select profile command create line length is 15mm
- Select profile command create line length is 40mm
- Select profile command create line length is 30mm

- Select **profile** command create line length is 40mm
- Select p**rofile** command create line length is 15mm
- Select **profil**e command create line length is 40mm
- Select **profil**e command create line length is 30mm
- Select p**rofil**e command create line length is 20mm



- Select **exit work bench** command to create 3d view
- Select **pad** command create pad command length is 40mm
- Select top view of object go to sketcher command
- Select **Rectangle** command create rectangle length and width is 15mm and 20 mm
- Select **exit work bench** command to create 3d view
- Select **pocket** command create pocket
- Select **dimension** command ae per the dimensions
- Select Tool command---select image-camera select---save the file in desk top folder



Result: Thus, the 3d- Modelling of Component In sectional view of a machine part is done by using Catia software

EXP NO: 10

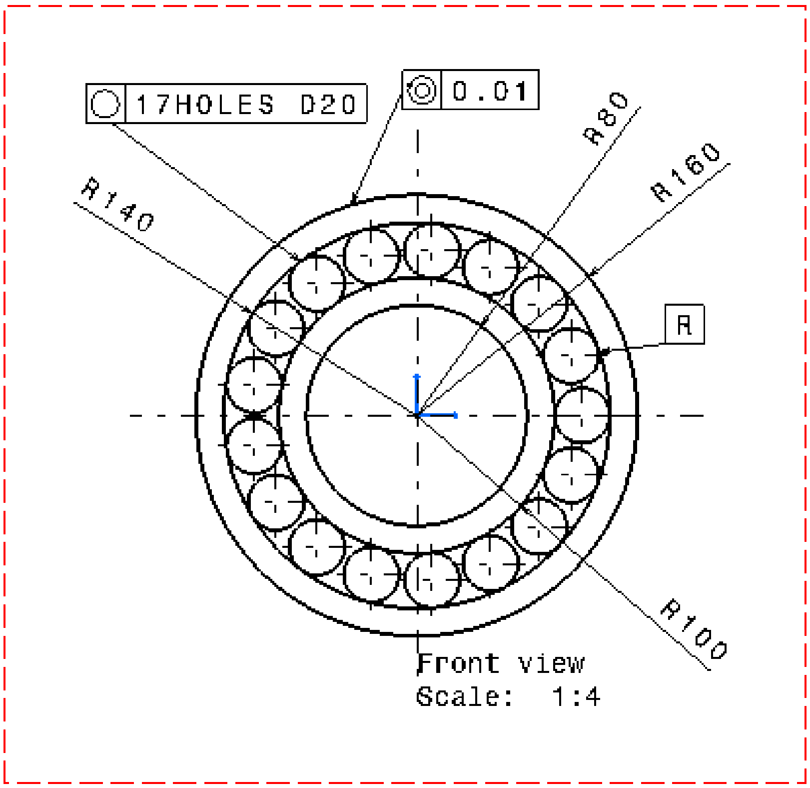
MODELLING ON ASSEMBLY DRAWING ON AXLE BEARING COMPONENT & PREPARE WITH DIMENSIONAL AND GEOMETRIC TOLERANCES

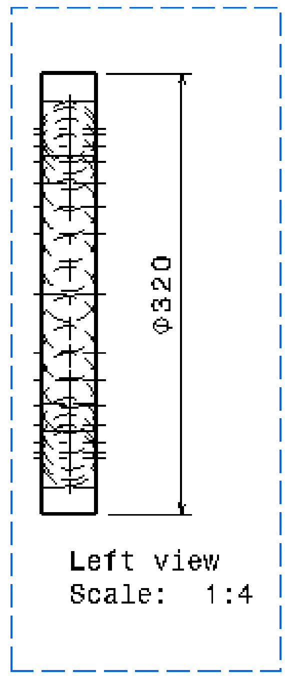
Aim: To Draw Modelling On assembly drawing on axle bearing **as** Shown In Drawing By Using Catia Software

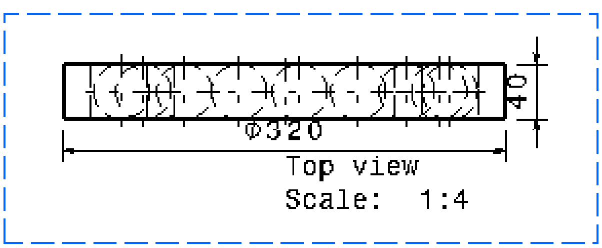
Equipment:

PC with CATIA software Package

Drawing::



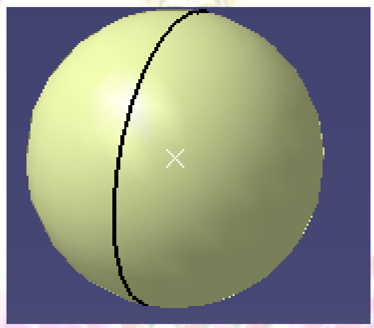




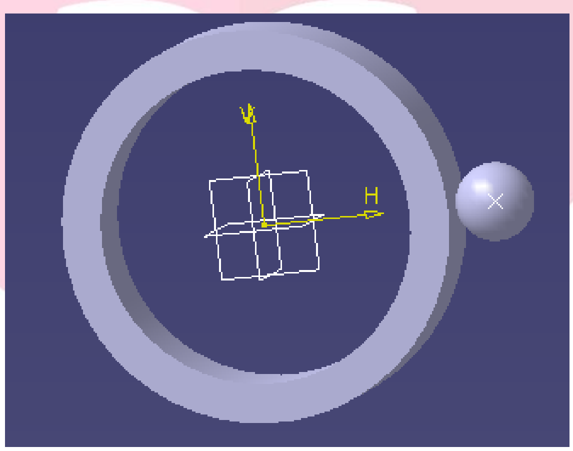
SEQUENCE OF COMMANDS REQUIRED TO DRAW THE MODEL:

PROCEDURE:

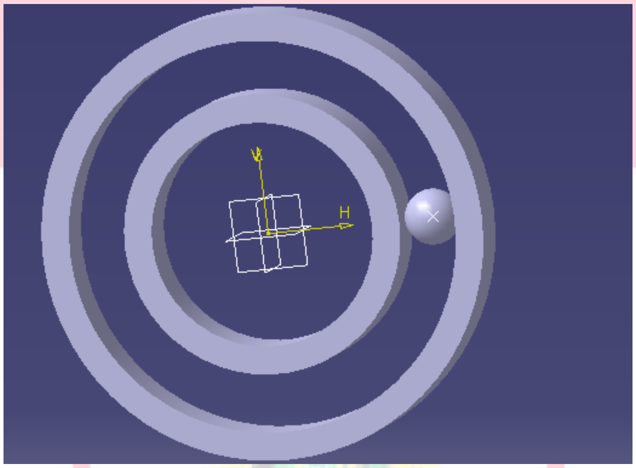
- Click on the CATIA icon, present on the desktop
- Go to file ‘MENU’ and click on ‘NEW’ ,a dialogue box is opened select mechanical design and select wire frame and composite design.
- Select the Required YZ plane go to select sketcher
- Select **point** command and create one point and give the dimensions 120mm by using constraint command
- Select **exit work bench** command
- Select **sphere** command and give the radius as 20mm and select create the **whole sphere option** to form a sphere**.**



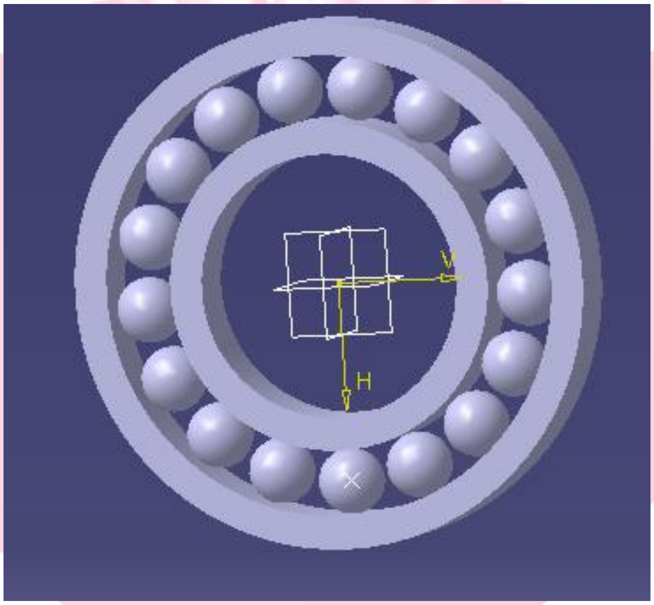
- Select start --- mechanical design and select part design and select **close surface select sphere surface ok**
- Select **Part body---c**lick **sketcher-1-**right click on **sphere option** and select **hide and show option**
- **Select YZ** plane go to **sketcher—**select **circle** command and give the diameter of **200mm** by **constrain command.**
- Select **pad** command and give the thickness1 value is 20mm and mirror extent commands, length is 20mm.



- **Select YZ** plane go to **sketcher—**select **circle** command and give the diameter of **280mm** by **constrain command.**
- Select **pad** command and give the thickness2 value is 20mm and mirror extent commands, length is 20mm.

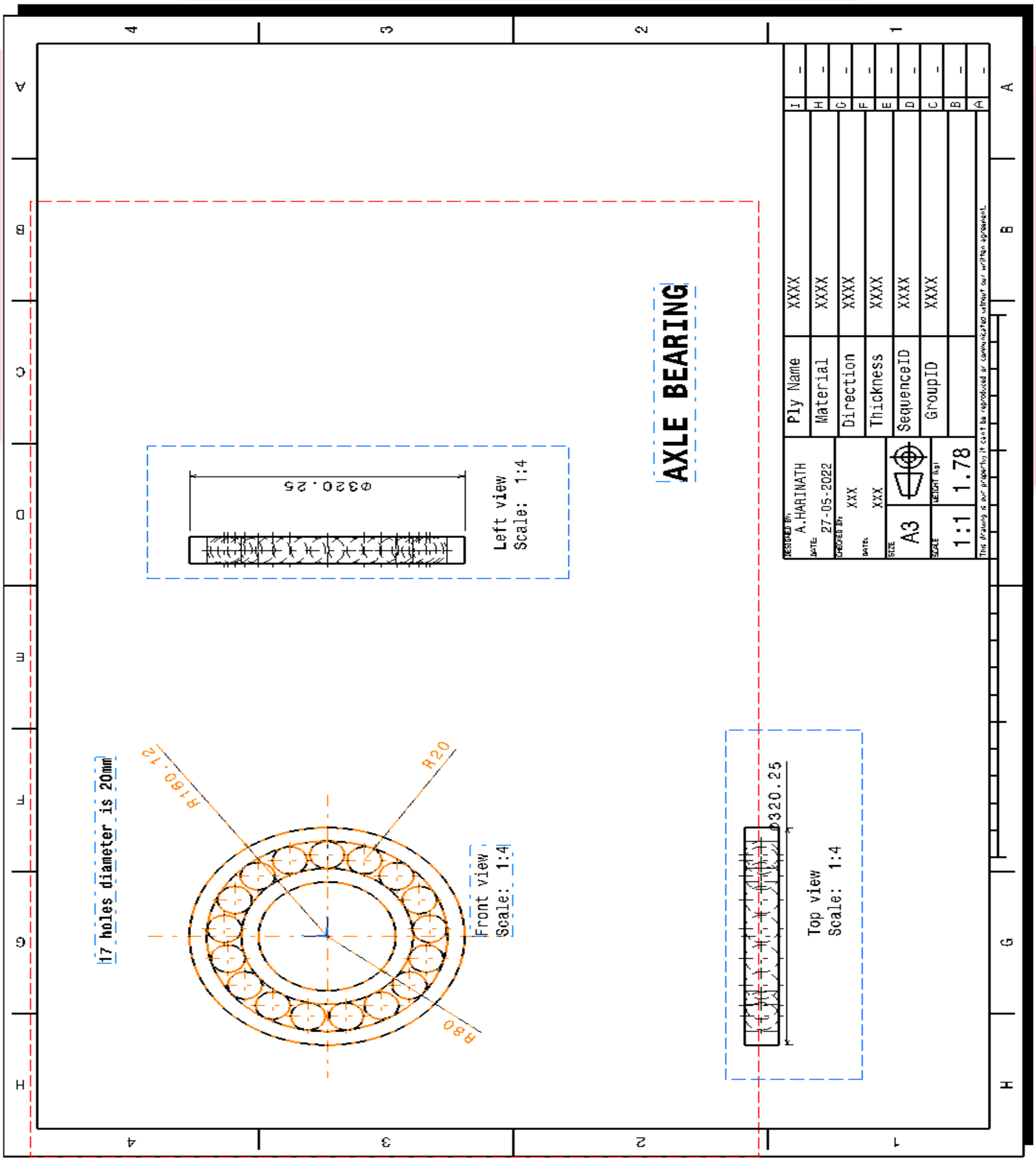


- Select circular pattern command—select parameters: **complete crown option**—give the **instances:17**—reference element **:v-axis— ok**



ASSEMBLY DRAWING

Result: Thus, The 3D-Modelling On Assembly Drawing In Axial Bearing Component & Prepare With Dimensional And Geometric Tolerances is done by using Catia software.



EXP NO: 11

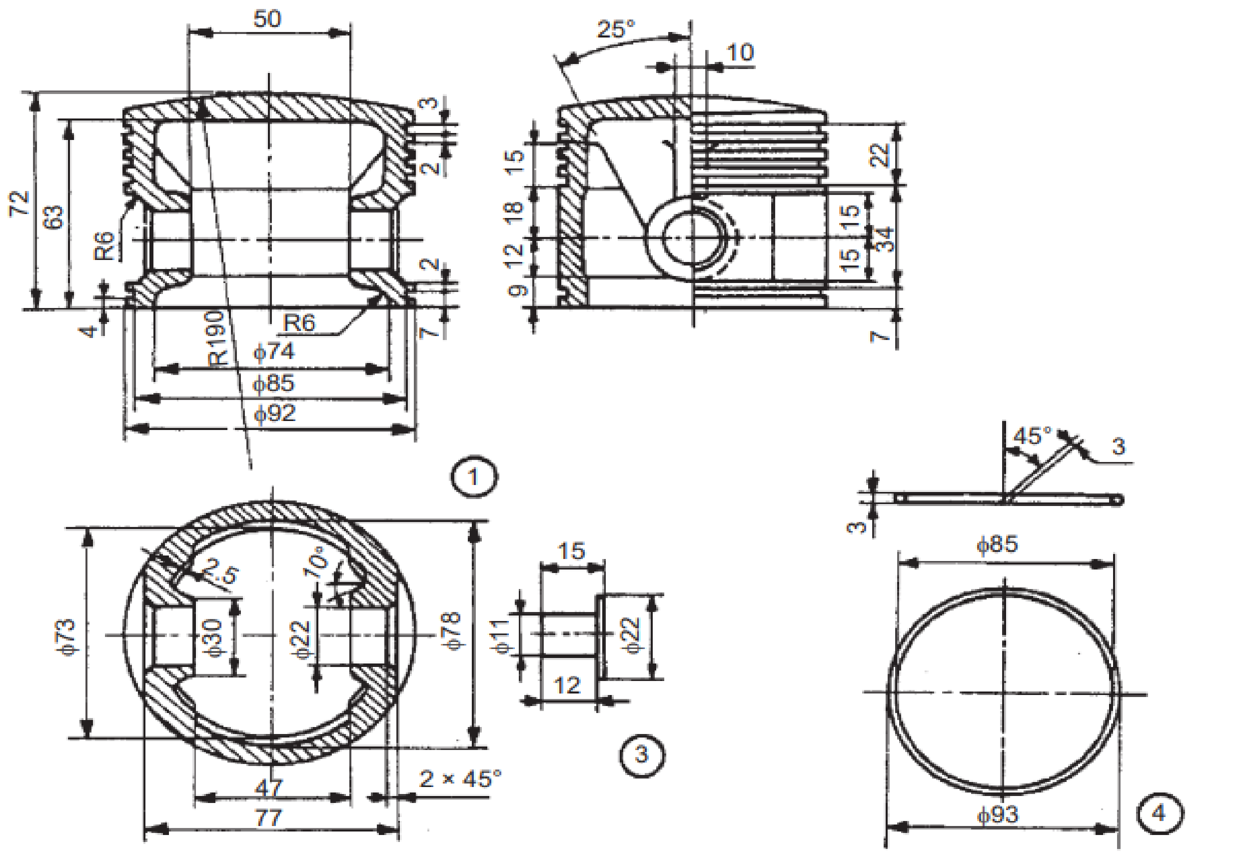
MODELLING ON ASSEMBLY DRAWING IN PISTON ENGINE COMPONENT & PREPARE WITH DIMENSIONAL AND GEOMETRIC TOLERANCES

Aim: To Draw Modelling On assembly drawing in piston component Part **as** Shown In Drawing By Using Catia Software

Equipment:

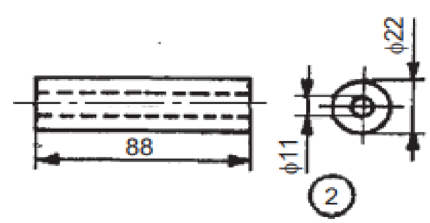
PC with CATIA software Package

Drawing::



Parts list

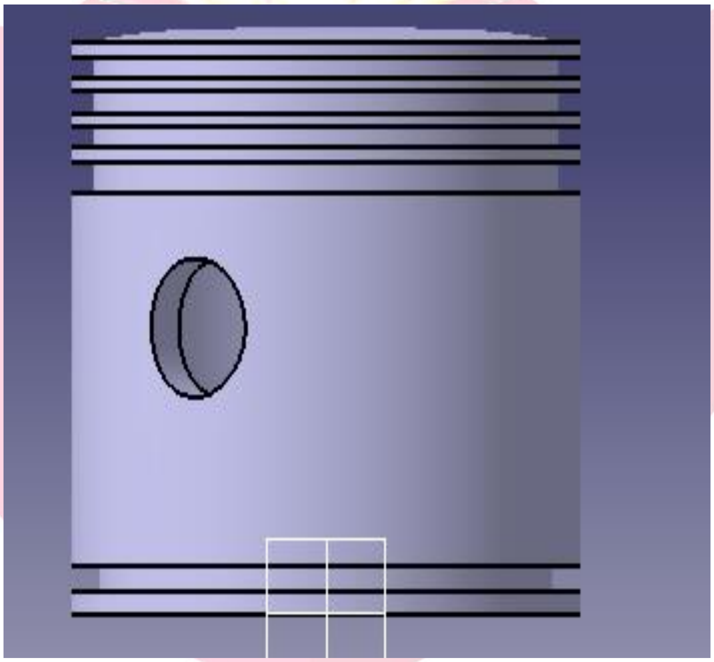
No.	Name	Matl	Qty
1	Piston	Al-alloy	1
2	Piston pin	HCS	1
3	Piston pin plug	HCS	2
4	Piston ring	CI	5



SEQUENCE OF COMMANDS REQUIRED TO DRAW THE MODEL:

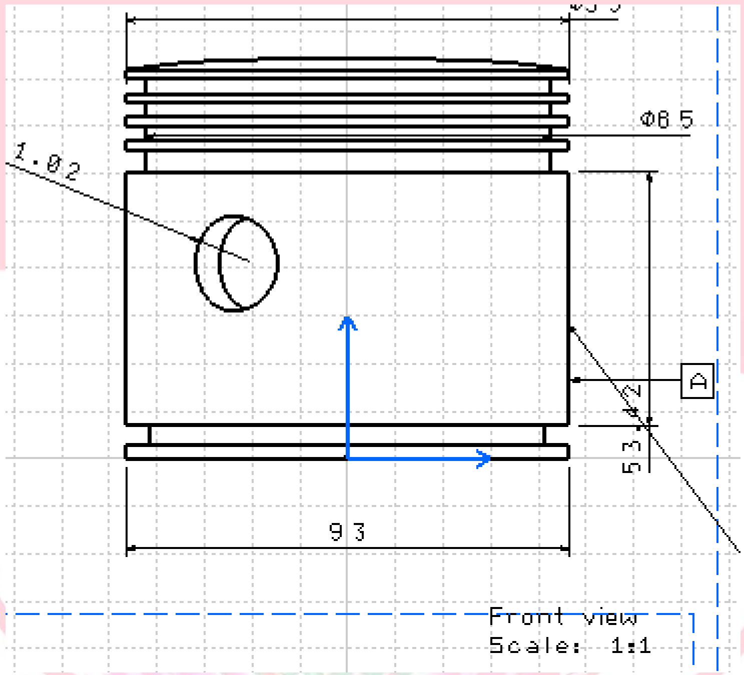
PROCEDURE:

- Click on the CATIA icon, present on the desktop
- Go to file 'MENU' and click on 'NEW' ,a dialogue box is opened .select 'part' option from the list in the box.
- Select the Required plane YZ
- Select 'sketcher' option from 'sketcher tool bar'
- Select profile command create line height is 2mm ,width is 4 mm, height is 3mm
- Select profile command create line height is 7mm
- Select profile command create line height is 34mm
- Select profile command create line height is 2mm ,width is 4 mm, height is 3mm
- Select profile command create line height is 2mm ,width is 4 mm, height is 3mm
- Select profile command create line height is 2mm ,width is 4 mm, height is 3mm
- Select profile command create line height is 2mm ,width is 4 mm, height is 3mm(22mm height create 3 rings)
- Select **arc** command create **arc** to center point as radius is 190mm and coincide of the center of axis
- **Select profil**e command create line height is 10mm
- **Select exit work bench command create one 2d too 3d at particular object**
- Select **shaft** command rotate 360 degrees of object and select yz command .it create 3d object automatically.

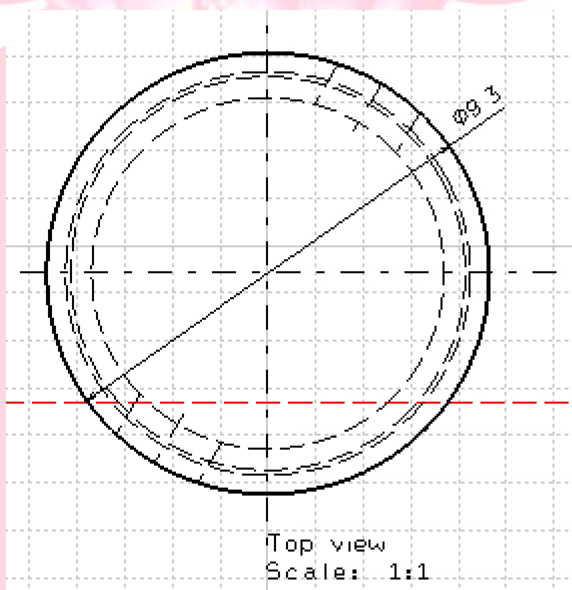


- Select **hole** command create the hole diameter is 22mm and depth is 100mm create hole at both sides of the piston.
- Select **constraint** command to give the dimensions of all elements.

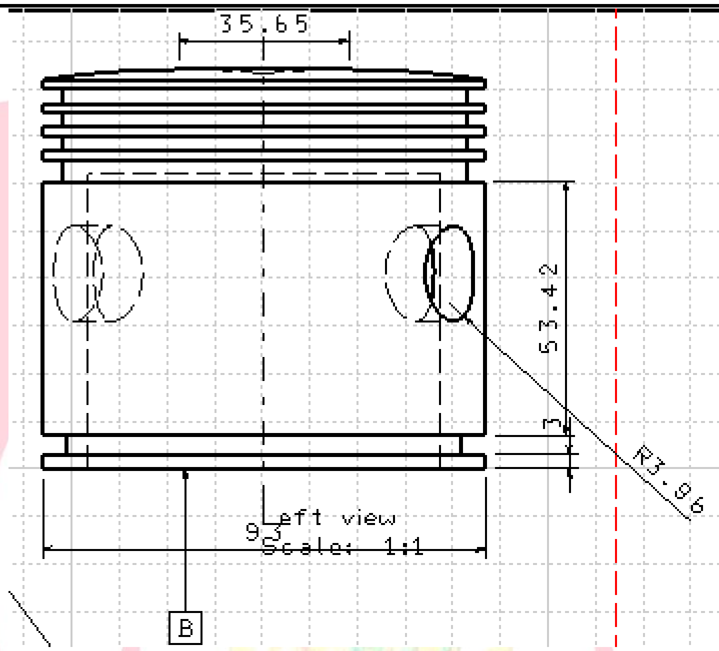
- Select mechanical design -----drafting part---open it
- Select **view tools command** select front view to create **front view** object of the piston



- Select **view tools command** select projection view to **create top view object** of the piston automatically.



- Select **view tools command** select projection view to create **side view** object of the piston automatically



- Select **datum featu**re command to create letters of the tolerance's symbols of piston assembly drawings
- Select **geometrical tolerances** command to create letters of the tolerance's symbols of piston assembly drawings
- Select **edit** option and go to sheet background option and select drawing sheet format to create new sheet apply ok
- Select **dimension** command ae per the dimensions
- Select Tool command---select image-camera select---save the file in desk top folder.

Result: Thus, The 3D-Modelling On Assembly Drawing In Piston Engine Component & Prepare With Dimensional And Geometric Tolerances is done by using Catia software.

EXP NO: 12

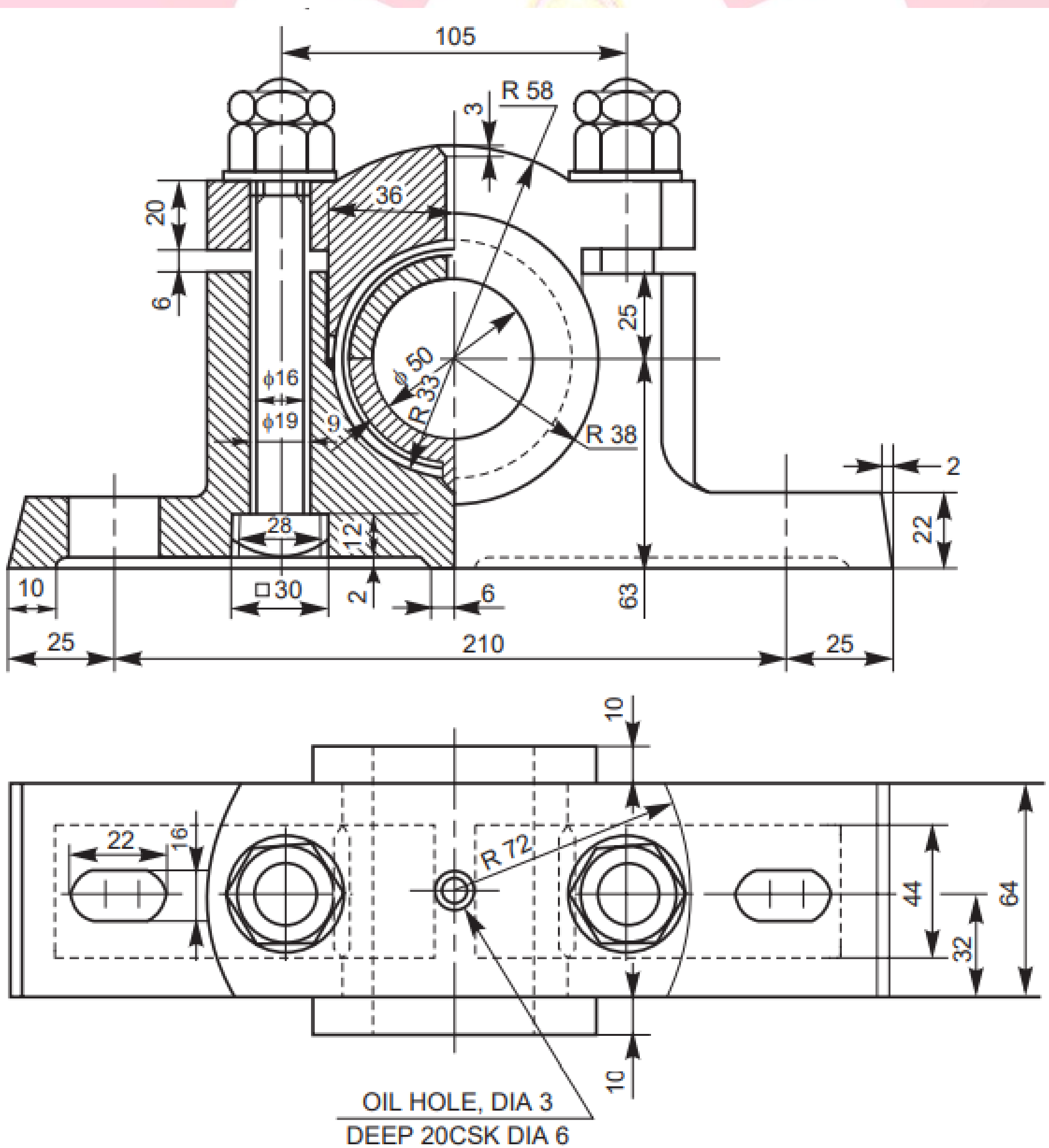
MODELLING ON ASSEMBLY DRAWING IN PLUMMER BLOCK & PREPARE WITH DIMENSIONAL AND GEOMETRIC TOLERANCES

Aim: To Draw Modelling On assembly drawing in Plummer Block Part **as** Shown In Drawing By Using Catia Software

Equipment:

PC with CATIA software Package

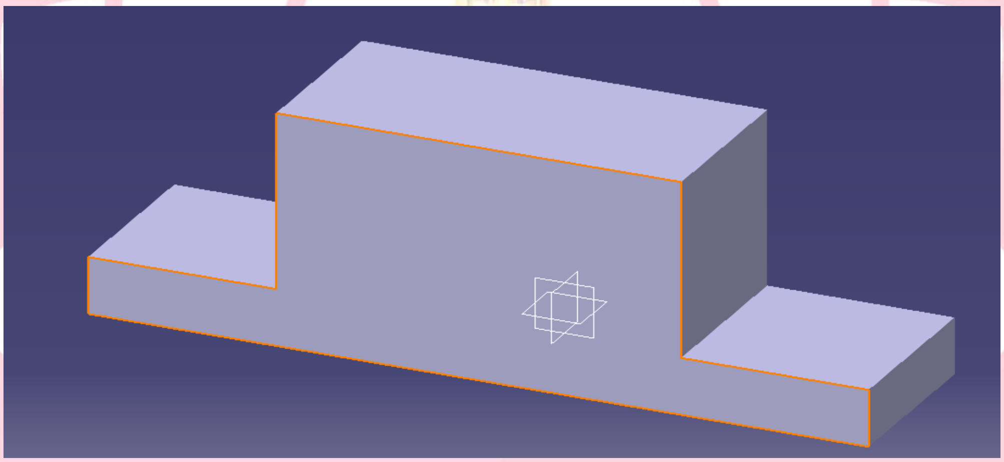
Drawing::



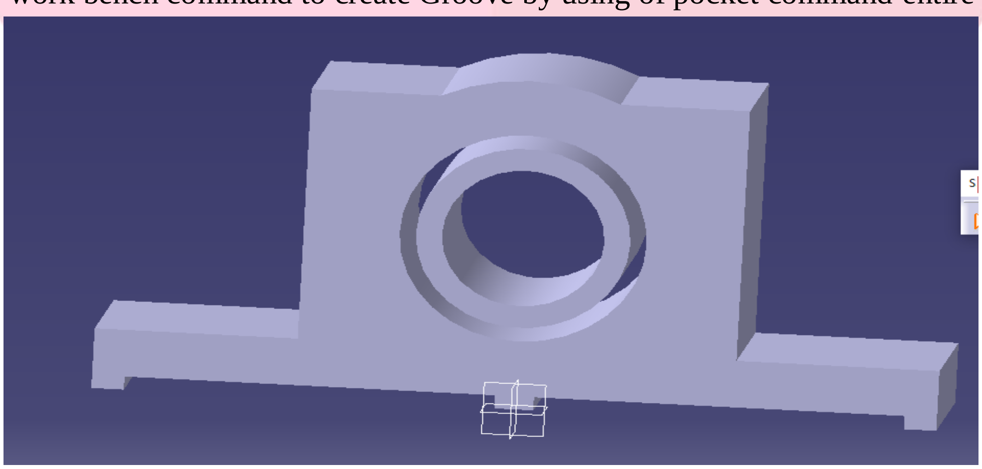
SEQUENCE OF COMMANDS REQUIRED TO DRAW THE MODEL:

PROCEDURE:

- Click on the CATIA icon, present on the desktop
- Go to file 'MENU' and click on 'NEW' ,a dialogue box is opened .select 'part' option from the list in the box.
- Select the Required plane YZ
- Select 'sketcher' option from 'sketcher tool bar'
- Select Profile command create a line. line length is 260mm ,again create a line height is 22mm
- Select Profile command create a line. line height is 114mm and length is 62.5mm
- Select exit work bench command ,Using pad command extrude object the length is 64mm

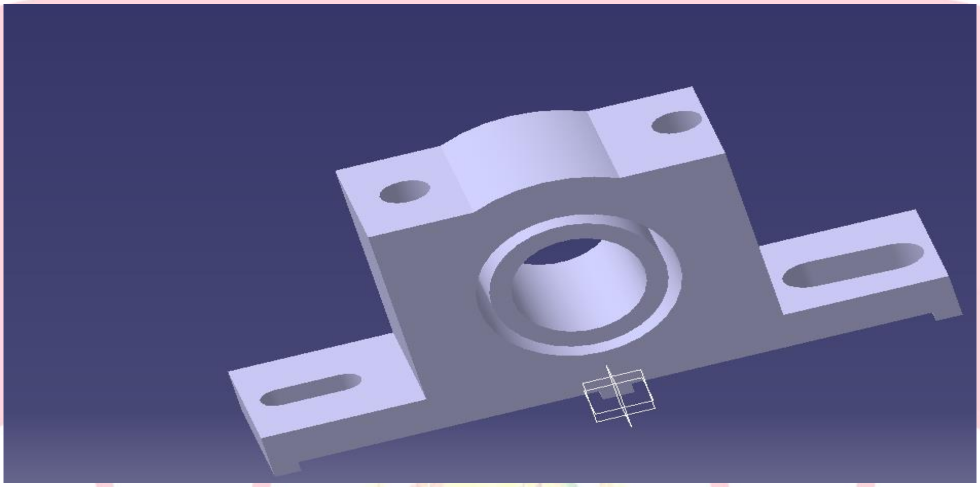


- Select circle command create a circle the axis Hight of 63mm and create 4 circles with different diameters,116mm,76mm,66mm,50mm
- Select Rectangle command given length is 10mm at both sides,6mm to the main axis line as both left and right side of the center point.
- Select Exit work bench command to create Groove by using of pocket command entire object.

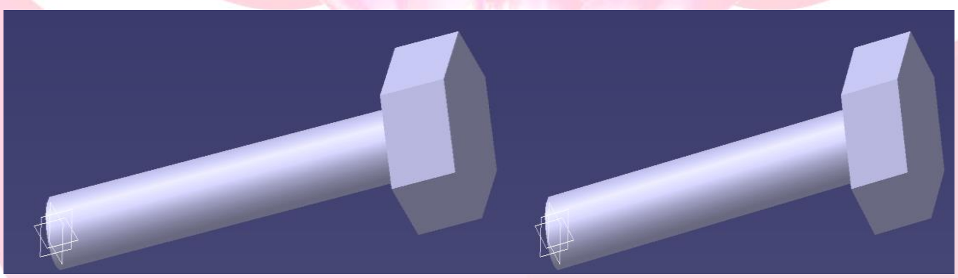


- Select Pocket command diameter is 19mm and depth length is 100mm to create hole at both sides.

- Select enlarged hole to create holes line length is 22mm and width is 16mm create hole at both sides of the body.



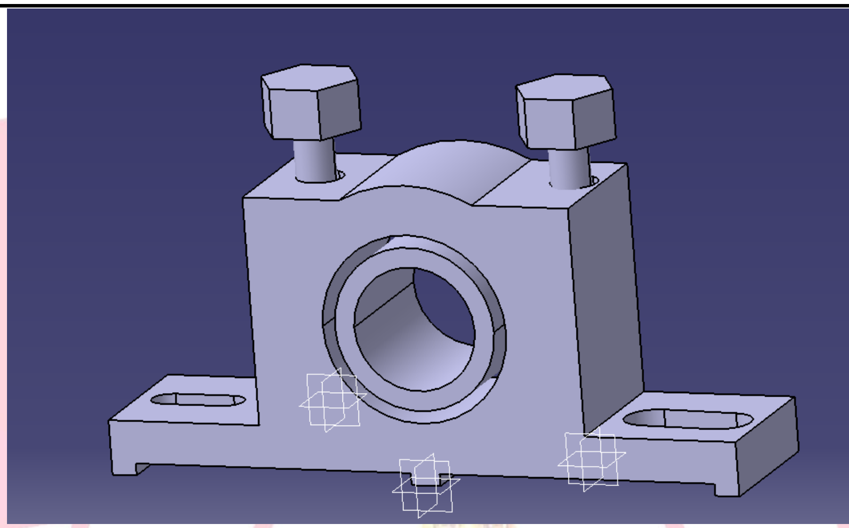
- Select **Circle** command create 2 circles of diameter is 16mm and select **work bench** command to To extrude the circle length is 130mm by using **pad** command.
- Select one side of plane circle. .go to sketcher command and create one Hexagonal shape by the length of 20mm.To create up side of the circle.
- Select exit work bench command to extrude length of the head of bolt is 20mm



- Select Mechanical design ---Assembly design ---and select Exist component Command ..to select the Product option …All parts will dump on the main Catia software.
- Select **Explode** command to separate all assemble parts of the Plummer block

- Select **snap** command to coincide the lines of the Assemble parts of Plummer block

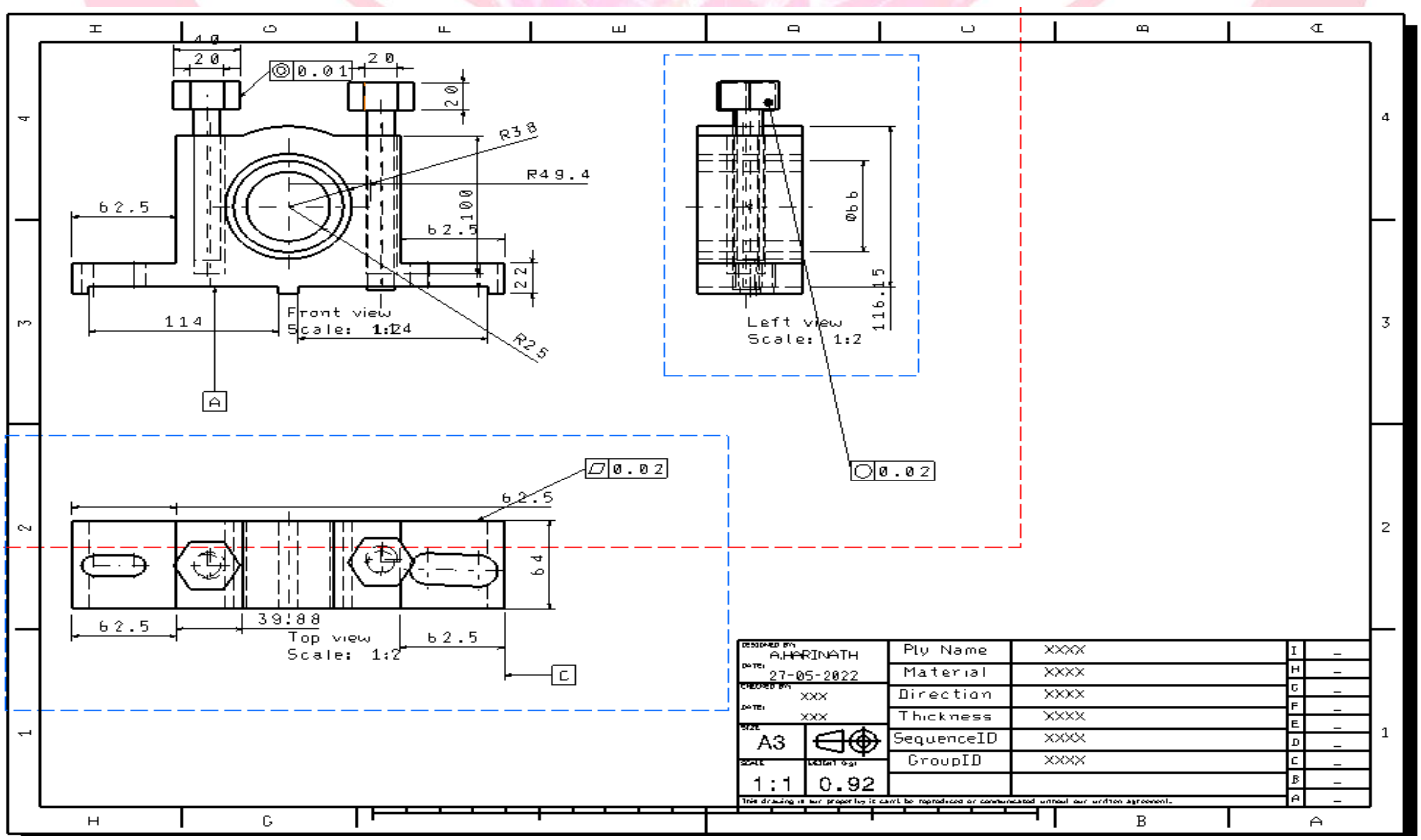
- Select Manipulation Command to use assemble all parts of the Plummer block Assembly drawing



ASSEMBLY DRAWING

- Select **dimension** command ae per the dimensions
- Select Tool command---select image-camera select---save the file in desk top folder.

Result: Thus, The 3D-Modelling On Assembly Drawing on Plummer Block & Prepare With Dimensional And Geometric Tolerances is done by using Catia software.



VIVA QUESTIONS

1.What is Convention Representation?

Certain droughting conventions are used to represent materials in section and machine elements in engineering drawings.

2.Which commands are used to draw Convention Representation of materials?

- Rectangle
- Trim
- Area fill cover
- Dimension commands

3. What is the full form of Catia?

Computer aided integrative Application

4.Profile: Create line from one point to another point

5.Line: Create line from one point to another point

6.Circle: Create circle from centre point

7.Spline: Create spline curve

8.Rectangle: Create Rectangle

9.Elipse: Create ellipse curve

10.Parabola: Create Parabola curve

11.Axis: Create axis line

12. **Chamfer:** Create chamfer inclined line two lines

13.Fillet: Create chamfer curve two lines

14.Mirror: Create same object from the axis another side

15.Trim or quick trim: Remove Extra lines or object

16.offset: Create line Parallel & perpendicular directions of the object

17. **Work bench:** it uses to convert 2d to 3d objects

18.View: fit,zoom,rotate,fly mode,box commands

19.Rotae: Rotate object in any directions

20.Zoom: The object is Enlarge

21.Fit: The object is Fit on the Screen

22. Sketched Base Features?

When it is required to make a simple solid component, we usually go to Part Design workbench. Important commands in this workbench include

PAD, POCKET, SHAFT, GROOVE, RIB and SLOT.

23.What is sectional view?

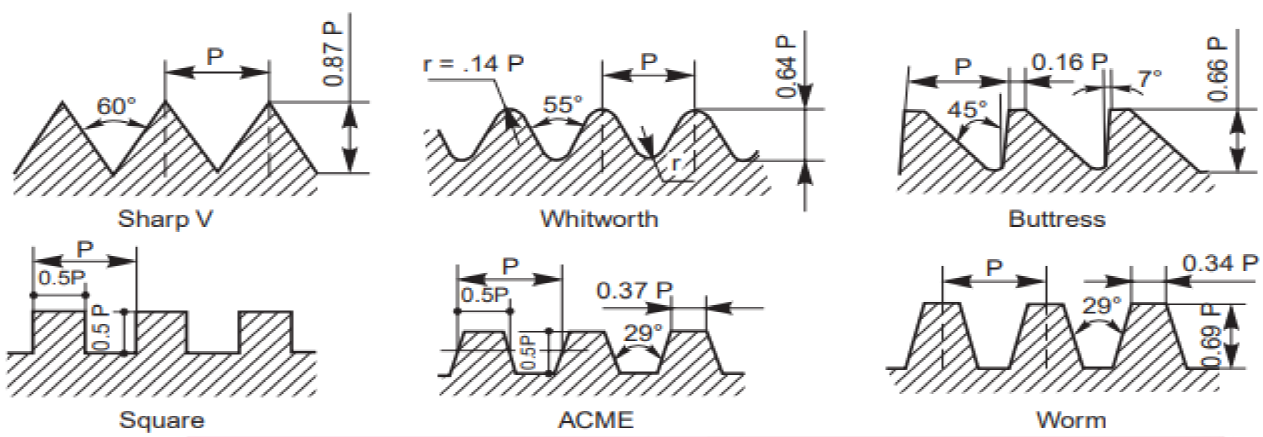
A sectional view obtained by assuming that the object is completely cut by a plane is called a full section or sectional view

24.Define fastener?

A machine element used for holding or joining two or more parts of a machine or structure is known as a fastener.

25.Types of threads?

Sharp-v thread, worm threads, with worth thread, square thread, acme thread, Buttress threads



26.What is bolt and nut?

A bolt and nut in combination is a fastening device used to hold two parts together. The body of the bolt, called shank is cylindrical in form, the head; square or hexagonal in shape, is formed by forging

27.What is main purpose of washer?

It is used to give a perfect seating for the nut and to distribute the tightening force uniformly to the parts under the joint. It also prevents the nut from damaging the metal surface under the joint

28.What is main purpose of key?

Keys are machine elements used to prevent relative rotational movement between a shaft and the parts mounted on it, such as pulleys, gears, wheels, couplings, etc.

29.Types of keys?

Keys are classified into three types, viz., saddle keys, sunk keys and round keys.

30.What is main purpose of coupling?

Shaft couplings are used to join or connect two shafts in such a way that when both the shafts rotate, they act as one unit and transmit power from one shaft to the other.

31. What is main purpose of Riveted joints? uses of Riveted joints?

Riveted joints are permanent fastenings and riveting is one of the commonly used method of producing rigid and permanent joints. Manufacture of boilers, storage tanks, etc., involve joining of steel sheets, by means of riveted joints. These joints are also used to fasten rolled steel sections in structural works, such as bridge and roof trusses

32. What is main purpose of Bearings? uses of Bearings?

Bearings are supports for shafts, providing stability, and free and smooth rotation. The importance of bearings may be understood from the supporting requirement of machine tool spindles, engine crankshafts, transmission or line shafts in workshops, etc.

33.Types of bearings?

Bearings are broadly classified into two categories:

- sliding contact bearings
- Rolling contact bearings or antifriction bearings

34.What is limit? Define Upper and lower limits?

The two extreme permissible sizes between which the actual size is contained are called **limits.** The maximum size is called the **upper limit** and the minimum size is called the **lower limit.**

35.What is Tolerances?

Great care and judgement must be exercised in deciding the tolerances which may be applied on various dimensions of a component. If tolerances are to be minimum, that is, if the accuracy requirements are severe, the cost of production increases. In fact, the actual specified tolerances dictate the method of manufacture. Hence, maximum possible tolerances must be recommended wherever possible.

36.What is Grade?

Tolerance is denoted by two symbols, a letter symbol and a number symbol, called the grade.

37.What is assembly drawing?

A machine is an assembly of various links or parts. It is necessary to understand the relation between the various parts of the unit for the purpose of design and production. An assembly drawing is one which represents various parts of a machine in their working position. These drawings are classified as design assembly drawings, working assembly drawings, sub-assembly drawings, installation assembly drawings, etc.

38.Explain piston?

A piston is cylindrical in form and reciprocates in a cylinder. The petrol engine piston is generally die cast in aluminum alloy. It is connected to the small end of the connecting rod by means of a gudgeon pin.Five piston rings 4 are positioned in the piston 1; four at the top and one at the bottom. The top piston rings, known as compression rings, prevent leakage of gases from combustion chamber into the crank case. The bottom one; oil or scraper ring, prevents the lubricating oil from entering the combustion chamber. The piston is connected to the small end of the connecting rod, by means of the gaugeon or piston pin 2; the axial movement of which is prevented by piston plugs 3

39.Explain Plummer block?

This bearing is used for long shafts, requiring intermediate support, especially when the shaft cannot be introduced in the bearing end-wise. It consists of a pedestal or base, a cap and a bush, split into two halves, called ‘bearing brasses’. The split parts used in the assembly, facilitate easy assembly and periodical replacement of the worn-out brasses.

40. What is the main Purpose of Explode command?

Select **Explode** command to separate all assemble parts of the Plummer block

41. **What is the main Purpose of snap command?**

Select **snap** command to coincide the lines of the Assemble parts of Plummer block

42. What is the main Purpose of Manipulation command?

Select Manipulation Command to use assemble all parts of the Plummer block Assembly drawing