# LABORATORY MANUAL

## III Year B. Tech II- Semester

## MECHANICAL ENGINEERING

## AY: 2025-26

# COMPUTER AIDED DESIGN
# AND
# COMPUTER AIDED MANUFACTURING LAB

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Secunderabad-500100,TelanganaState,India.
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# MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

**(AUTONOMOUS INSTITUTION - UGC, GOVT. OF INDIA)**

**Affiliated to JNTUH; Approved by AICTE, NBA-Tier 1 & NAAC with A-GRADE | ISO 9001:2015**

Maisammaguda, Dhulapally, Komaplly, Secunderabad - 500100, Telangana State, India


# LABORATORY MANUAL & RECORD

Name:.............................................................................

Roll No:.......... Branch:.......................................................

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# Certificate

Certified that this is the Bonafide Record of the Work Done by

Mr./Ms.……………………………………………........................Roll.No……………..of

B.Tech year …..........................… Semester for Academic year **2025-2026**

in……………………………………………………………………Laboratory.

Date: Faculty Incharge HOD

Internal Examiner External Examiner

# INDEX

| S.No | Date | Name of the Activity/Experiment | Grade/ Marks | Faculty Signature |
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# INDEX

| S.No | Date | Name of the Activity/Experiment | Grade/ Marks | Faculty Signature |
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# CONTENTS

# MALLAREDDYCOLLEGEOFENGINEERING&TECHNOLOGY

(Autonomous Institution–UGC,Govt.of India)

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## VISION

- To establish a pedestal for the integral innovation, team spirit, originality and competence in the students, expose them to face the global challenges and become technology leaders of Indian vision of modern society.

## MISSION

- To become a model institution in the fields of Engineering, Technology and Management.
- To impart holistic education to the students to render them as industry ready engineers.
- To ensure synchronization of MRCET ideologies with challenging demands of International Pioneering Organizations.

## QUALITYPOLICY

- To implement best practices in Teaching and Learning process for both UG and PG courses meticulously.
- To provide state of art infrastructure and expertise to impart quality education.
- To groom the students to become intellectually creative and professionally competitive.
- To channelize the activities and tune them in heights of commitment and sincerity, the requisites to claim the never – ending ladder of **SUCCESS** year after year.

For more in formation: www.mrcet.ac.in

# VISION

To become an innovative knowledge center in mechanical engineering through state-of-the-art teaching-learning and research practices, promoting creative thinking professionals.

# MISSION

The Department of Mechanical Engineering is dedicated for transforming the students into highly competent Mechanical engineers to meet the needs of the industry, in a changing and challenging technical environment, by strongly focusing in the fundamentals of engineering sciences for achieving excellent results in their professional pursuits.

# Quality Policy

- ✓ To pursuit global Standards of excellence in all our endeavors namely teaching, research and continuing education and to remain accountable in our core and Support functions, through processes of self-evaluation and continuous improvement.
- ✓ To create a midst of excellence for imparting state of art education, industry-oriented training research in the field of technical education.

## PROGRAMOUTCOMES

Engineering Graduates will be able to:

1. **Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
2. **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.
3. **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.
4. **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.
5. **Modern tool usage**: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
6. **The engineer and society** :Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
7. **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.
8. **Ethics**: Apply ethical principle sand commit professional ethics and responsibilities and norms of the engineering practice.
9. **Individual and teamwork**: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
10. **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.
11. **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.

12. **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.

## PROGRAMSPECIFICOUTCOMES(PSOs)

**PSO1** Ability to analyze, design and develop Machine learning systems to solve the Engineering problems by integrating design and manufacturing Domains.

**PSO2** Ability to succeed in competitive examinations or to pursue higher studies or research.

**PSO3** Ability to apply the learned Mechanical Engineering knowledge for the Development of society and self.

## Program Educational Objectives (PEOs)

The Program Educational Objectives of the program offered by the department are broadly listed below:

**PEO1: PREPARATION**

To provide sound foundation in mathematical, scientific and engineering fundamentals necessary to analyze, formulate and solve engineering problems.

**PEO2: CORE COMPETANCE**

To provide thorough knowledge in Mechanical Engineering subjects including theoretical knowledge and practical training for preparing Artificial models pertaining to Automobile Engineering, Element Analysis, Production Technology, Mechatronics etc.,

**PEO3: INVENTION,INNOVATION AND CREATIVITY**

To make the students to design, experiment, analyze, interpret in the core field with the help of other inter disciplinary concepts wherever applicable.

**PEO4: CAREER DEVELOPMENT**

To inculcate the habit of lifelong learning for career development through successful completion of advanced degrees, professional development courses, industrial training etc.

**PEO5: PROFESSIONALISM**

To impart technical knowledge, ethical values for professional development of the student to solve complex problems and to work in multi-disciplinary ambience, whose solutions lead to significant societal benefits.

# GENERAL LABORATORY INSTRUCTIONS

1. Students are advised to come to the laboratory at-least 5 minutes before (to the starting time), those who come after 5minutes will not be allowed into the lab.
2. Plan your task properly much before to the commencement, come prepared to the lab with the synopsis / program / experiment details.
3. Student should enter in to the laboratory with:

   a) Laboratory observation notes with all the details (Problem statement, Aim, Algorithm, Procedure, Program, Expected Output, etc.,) filled in for the lab session.

   b) LaboratoryRecordupdateduptothelastsessionexperimentsandotherutensils (if any) needed in the lab.

   c) Proper Dress code and Identity card.
4. Sign in the laboratory login register, write the TIME-IN, and occupy the computer system allotted to you by the faculty.
5. Execute your task in the laboratory, and record the results/output in the lab observation notebook, and get certified by the concerned faculty.
6. All the students should be polite and cooperative with the laboratory staff, must maintain the discipline and decency in the laboratory.
7. Computer labs are established with sophisticated and high end branded systems, which should be utilized properly.
8. Students / Faculty must keep their mobile phones in SWITCHED OFF mode during the lab sessions. Misuse of the equipment, misbehaviors with the staff and systems etc., will attract severe punishment.
9. Students must take the permission of the faculty incase of any urgency to go out; if anybody found loitering outside the lab / class without permission during working hours will be treated seriously and punished appropriately.
10. Students should LOG OFF/ SHUT DOWN the computer system before he/she leaves the lab after completing the task (experiment) in all aspects. He/she must ensure the system / seat is kept properly.

## MALLAREDDY COLLEGE OF ENGINEERING AND TECHNOLOGY

**III Year B.Tech.ME-II Sem** **L/T/P/C**

**-/-/-2/1**

# (R22A0388)CAD/CAM LAB

**COURSE OBJECTIVES:**

1. To design 3D part models
2. To enhance the students with computer aided modeling skills.
3. To impart the students with knowledge of assemblies.
4. To understand about CNC programming.
5. To develop CNC programming for the given simple turning and milling operations.

**LIST OF EXPERIMENTS**

1. ISOMETRIC VIEW – 1
2. ISOMETRIC VIEW – 2
3. ISOMETRIC VIEW – 3
4. ISOMETRIC VIEW – 4
5. 3D Modeling & Assembly of Gib and cotter joint.
6. 3D Modeling & Assembly of knuckle joint.
7. 3D Modeling of flanged coupling.
8. 3D Modeling and Assembly of screw jack.
9. 3D Modeling and Assembly of plumber block.
10. 3D Modeling and Assembly of of Bushed Bearing.
11. Manual part program for simple turning and facing operation for component
12. Manual Part Programming For Step Turning Operation
13. Manual part program for Taper turning.

Note: At least 10 experiments are to be conducted.

Any Two Software Packages from the following:
Use of AutoCAD, CATIA, Creo, Solid works

**Course Outcomes**

1. Designing 3D part models

2. Enhancing computer aided modeling skills.

3. Gaining knowledge on assemblies.

4. Under standing the basics of CNC programming.

5. Developing CNC programming for the given simple turning and milling operations.

# INTRODUCTION

In cad laboratory we are going to study about how to create a model of engineering objects and also how to create an assembly of modeled objects. The modeling software's like CATIA, Pro e, Unigraphics are generally used in mechanical engineering field for the modeling. In this lab catia-v5r18 software is used to do the exercises.

CATIA (Computer Aided Three-dimensional Interactive Application) is a multi-platform CAD/CAM/CAE. It is written in the C++ programming language. Commonly referred to as a 3D Product Lifecycle Management software suite, CATIA supports multiple stages of product development (CAx), from conceptualization, design (CAD), manufacturing (CAM), and engineering (CAE).CATIA can be customized via application programming interfaces (API). V4 can be adapted in the Fortran and C Programming languages under an API called CAA (Component Application Architecture). V5 can be adapted via the Visual Basic and C++ programming languages, an API called CAA2 or CAA V5 that is a component object model (COM)-like interface. Although later versions of CATIA V4 implemented NURBS, V4 principally used piecewise polynomial surfaces.

#### AN OVERVIEW OF CATIA DESIGN SOFTWARE

### Optimal Sharing

Catia V6 users will get access to a unique, collaborative 3 dimensional environments that can be access by an unlimited number of people online. This allows people from across the globe to collaborate in a virtual environment. It has been designed to not only allow for online cooperation, but also makes offline sharing and designing easy to integrate as well.

### Simplified Product Development

Creating a new product can be a long and complex process. It encompasses multiple design phases including the initial design, overall development, and manufacturing. Catia V6 decreases the complexity and length of the entire project because it integrates various stages of the development process so that they can be controlled and modified on a single platform. It does this by using an approach to systems engineering known as RFLP. This allows you to create several versions of the same product using different sets of requirements. This gives you a comprehensive look at what the final product could be Seamless Transitioning.

Every version of the CATIA design software is designed to allow for seamless integration with previous versions. This makes upgrading a simple process and can be completed without losing any of the information that has already been stored. CATIA design software has found its way into more and more industries with each passing year. Traditionally, it gained notoriety through 3 main industries; however, every industry that is involved in engineering has found ituseful.

CATIA has become a leader in product development software and may be exactly what is needed to overcome the shortcomings of CAD software.

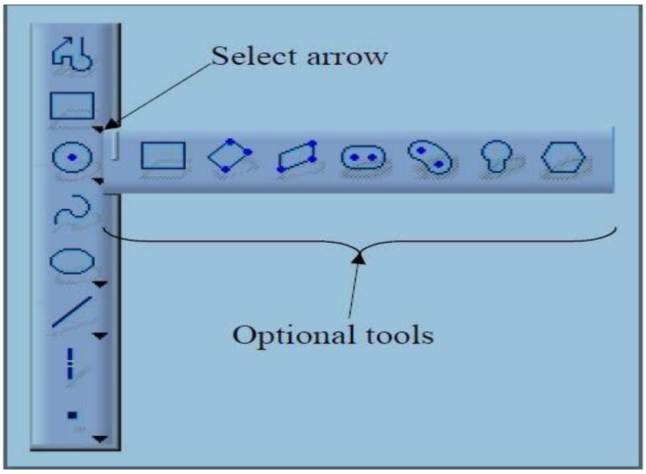
**Tool Bars**

Many standard toolbars are used in the different modes like sketcher mode operational mode etc, in CATIA software. Here we discuss about the tool bars used in sketcher with some examples.

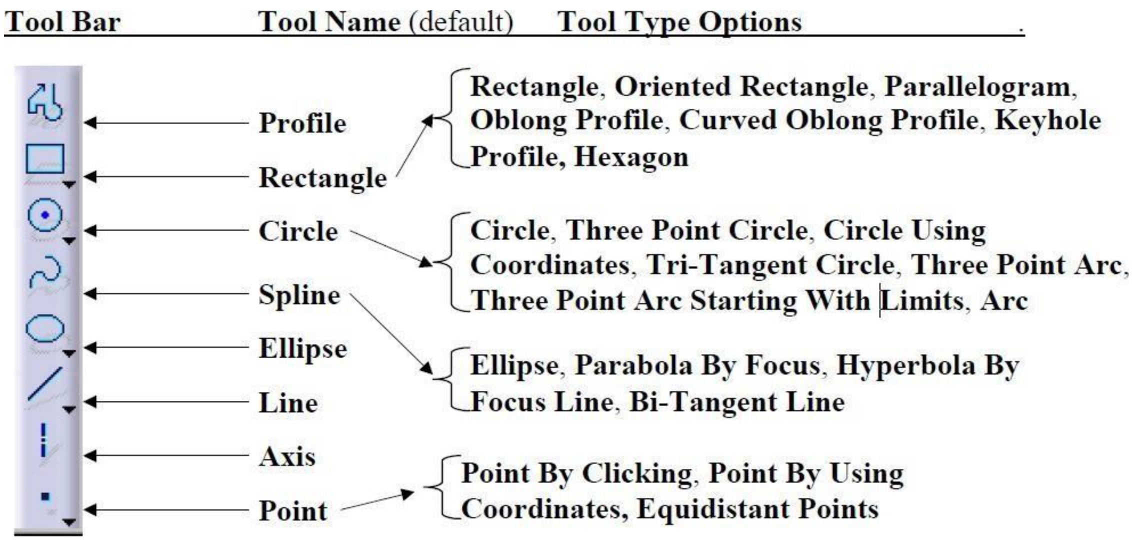
**Sketcher Work Bench Tool Bars**

There are three standard tool bars found in the Sketcher Work Bench. The three toolbars are shown below. The individual tools found in each of the three tools are labeled to the right of the tool icon. Some tools have an arrow located at the bottom right of the tool icon. The arrow is an indication that there is more than one variation of that particular type of tool. The tools that have more than one option that are listed to the right of the default tool. To display the other tool options you must select and hold the left mouse button on the arrow as shown in Figure1.1. This will bring up the optional tools Select Arrow Optional tools.

THE OPERATION TOOL BAR



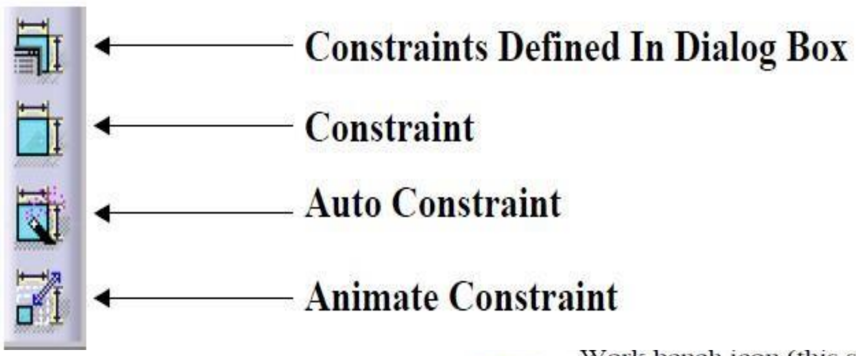

THE PROFILE TOOL BAR

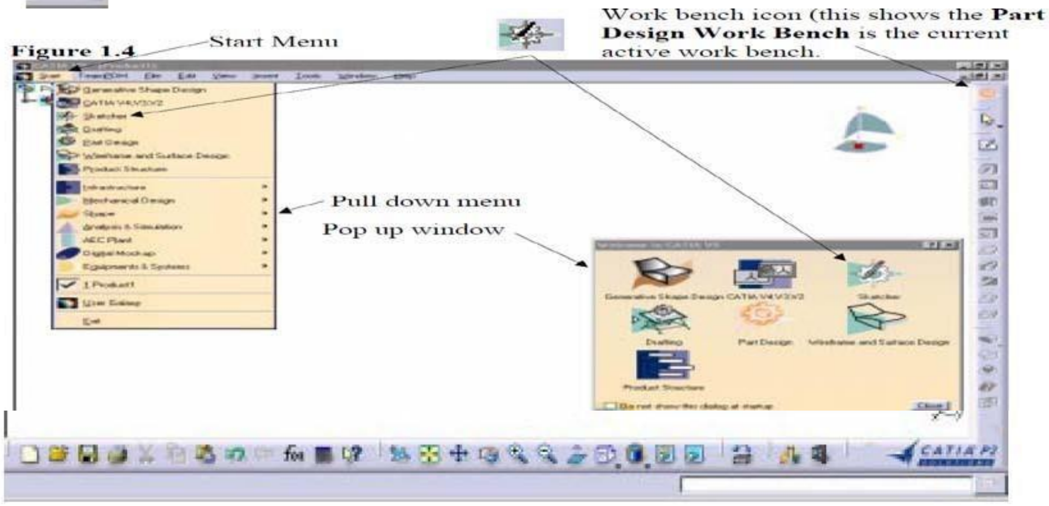



Tools covered in this lesson: **Profile, Rectangle**, **Circle**, **Line** and **Point**.

THE CONSTRAINTS TOOL BAR

**Tool Bar** **Tool Name** (default) **Tool Type Options**

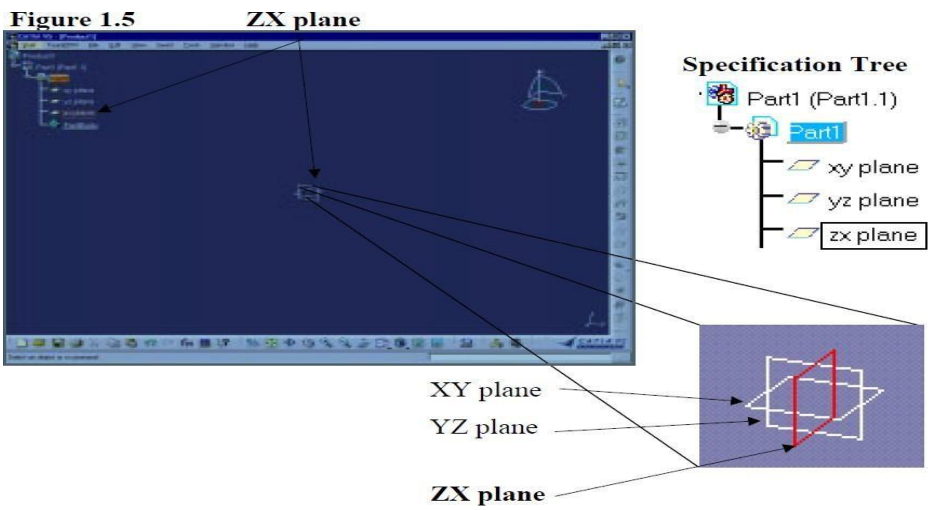



**Specify a Working Plane**

The next step is to create a 2 dimensional profile of the part. The Sketcher Work Bench is a two dimensional (planar) work area. To use the Sketcher Work Bench, you must specify which plane the profile is to be created on. Specifying, the plane can be done several different ways.

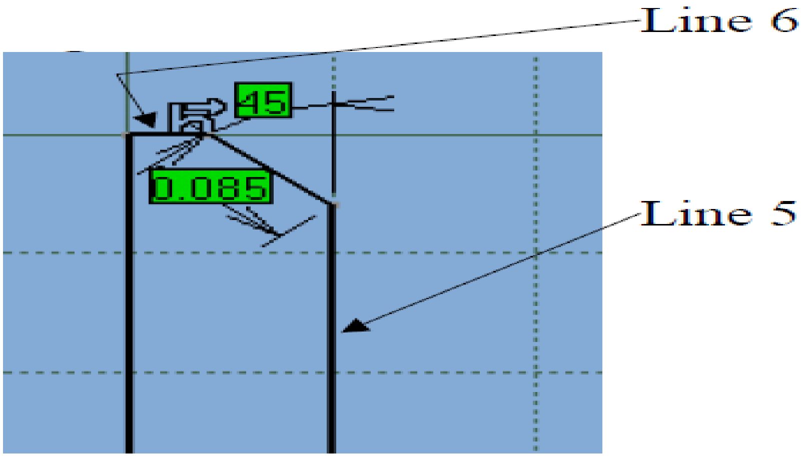
**ZX plane**



**MODIFYING THE PROFILE USING CHAMFER**

The Chamfer icon is also located in the Operations tool bar. This procedure assumes you know what a chamfer is. The steps required to create a chamfer are almost identical to creating a corner. Select the Chamfer icon.

The command prompt at the bottom left hand of the screen, will prompt you with the following: “Select the first curve, or a common point”. For this exercise select line 5. The next command prompt will ask you to “Select the second curve”.




**Constraint**

This tool allows you to create individual constraints, one at time. You have already applied a constraint and may not even know it. The Anchor icon is a constraint. The values attached to the Chamfer and Corner is constraints. To apply Dimensional Constraints, complete the following steps:

Select the Constraint icon.

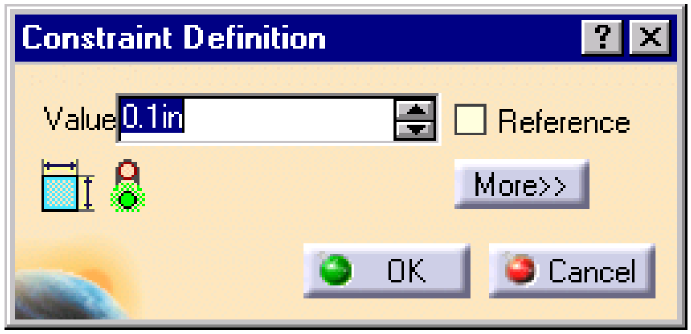
Select the line and/or Sketcher element to be constrained.

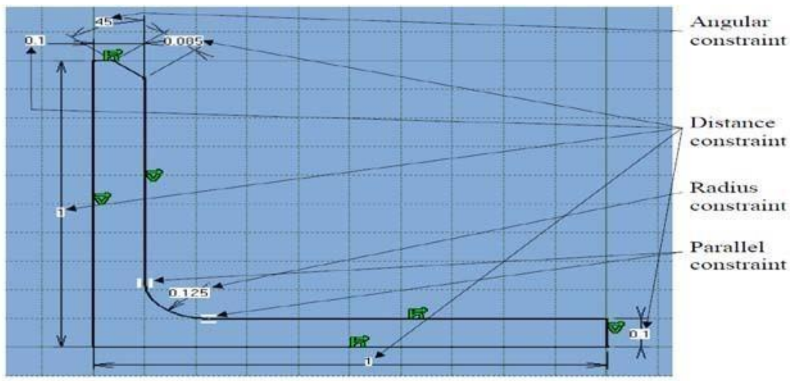
The Sketcher element will turn green (constraint symbol) along with the appropriate dimension and box with the value in it. To re-locate the constraint value, select the value box and drag the mouse to the desired location.

If the initial location of the constraint is not satisfactory re-select the dimension and drag and drop it at the new location.

To edit the value of the constraint double click on the value box. This will bring up the Constraint Definition pop up window shown in. This window shows the existing value for the Sketcher element. This value can be edited by typing the new value over the existing value. Then select OK or hit the Enter key. The entities linked to the constraint will automatically be updated to the new value.

If the constraint is between two different entities, such as lines, select the first line and then the second line. CATIA V5 will constrain the distance between the two entities. The constraint value will appear near the constraint to move the constraint value. For this lesson constrain your “L Shaped Extrusion” similar to the one shown in Figure.




**Suggested Steps for practice exercises:**

1. Select the XY plane (the plane the profile will be sketched on). Enter the **SketcherWork Bench**.
2. Sketch the profile of the part. Hint: use the **Profile** tool.
3. Anchor the lower left hand corner of the sketch. For anchoring a profile.
4. Constrain the profile to match the dimensions given in the profile.
5. Exit the **Sketcher Work Bench**, return to the **Part Design Work Bench** (the 3D environment). **Sketcher Work Bench** and entering the **Part Design Work Bench**.
6. Once in the **Part Design Work Bench** extrude the profile to the dimension. It'sExtrude or cut the profile.
7. Finally save a part drawing.

********************************

# TUTORIAL - 1

**Creating the "Swivel. CATPart" Using Multiple Sketches**

1.1 Start CATIA V5.
1.2 Verify that you are in the **Part Design Work Bench** and the default **Properties** are set the way you want them such as **Units**. For this step, set the Units to mm.
1.3 In the **Specification Tree** rename **Part.1** to "**Swivel**".
1.4 Enter the **Sketcher Work Bench** using the **ZX Plane** as shown in figure 1.1

**ZX Plane**

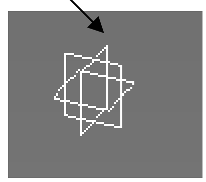


**Figure 1.1**

1.5 Create a circle at the coordinates (0,0) with a diameter of 25 mm as shown in figure 5.2

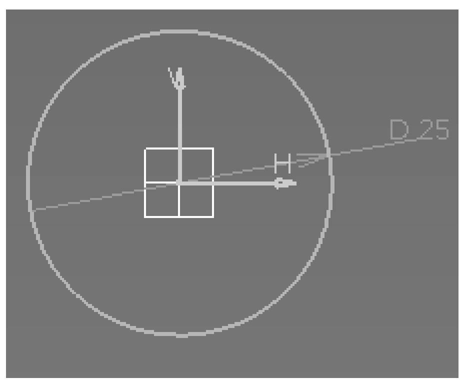



**Figure 1.2**

1.6 Exit the Sketcher Work Bench. Remember, this will put you back into the **Part Design Work bench**.
1.7 Select the **Pad** tool.
**1.8** When the **Pad Definition** window appears, , select the **More** button. This will expand the **Pad Definition** window to show the **Second Limit** box. Figure 1.3 shows the **Pad Definition** window expanded to include the **Second Limit.**

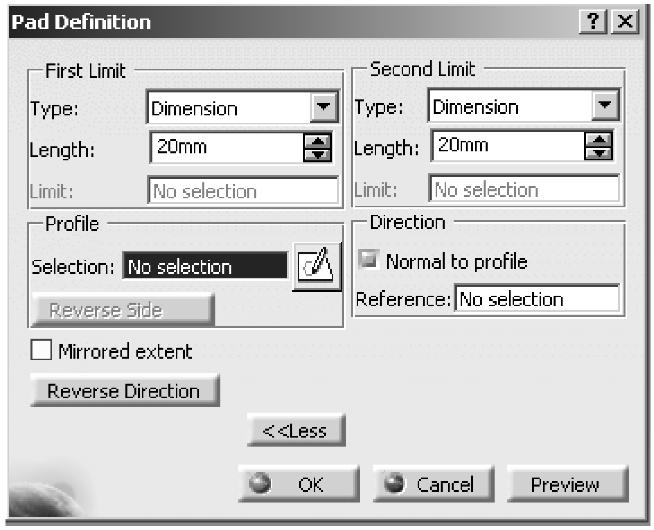



**Figure 1.3**

1.9 In the **First Limit** area enter "**20mm**" for the **Length** Box. Leave the **Type** box set at **"Dimension",** as shown in figure 1.5.

1.10 In the **Second Limit** area, enter "**20mm"** for the **Length** box, as shown in figure 1.5.

1.11 Select the **OK** button. Notice that the circle has been extruded 1 inch in both directions, reference figure 1.4

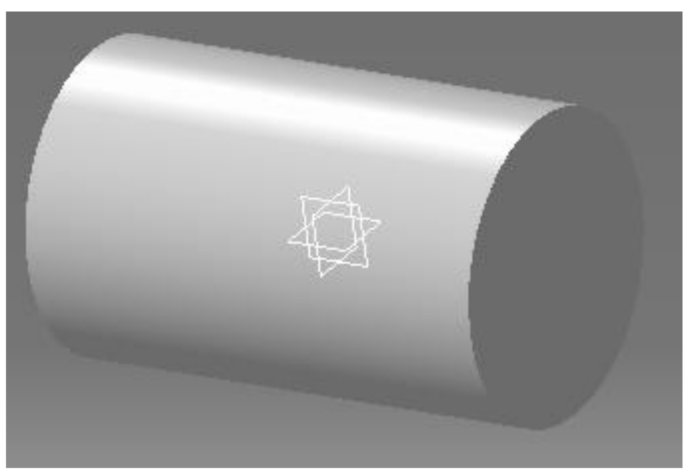


**Figure 1.4**

1.12 Select the ZX Plane (figure 5.1) and enter the **Sketcher Work Bench**.

1.13 Create another circle at the Coordinates (0,0) with a diameter of 18 (Figure 1.5).

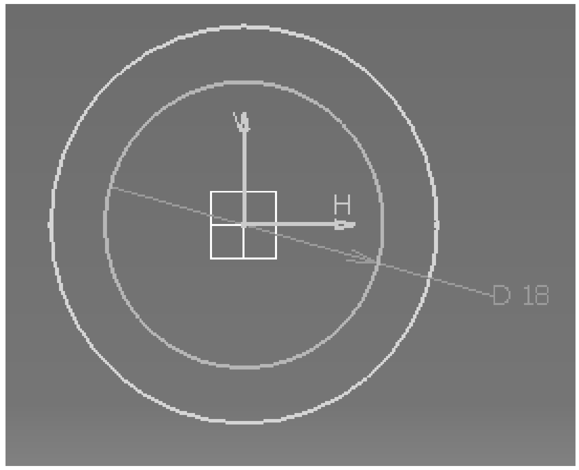



**Figure 1.5**

1.14 Exit the **Sketcher Workbench**. Select the **Pad** tool and select the **More** button to expand the Pad Definition window.

1.15 Enter “40” as the First Length and “40” as the Second length.

1.16 Select the OK button. The part should look similar the part shown in figure 1.6.

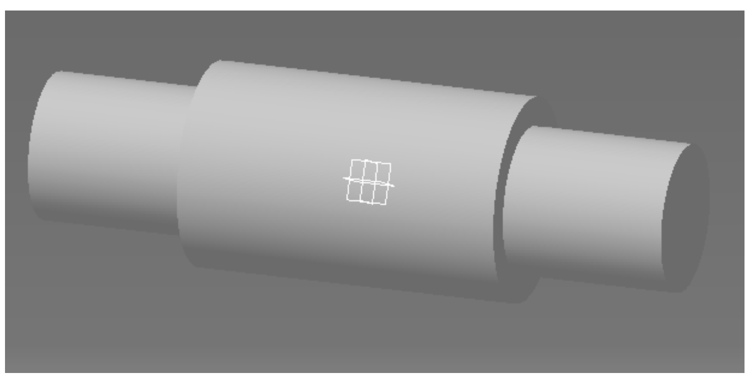


**Figure 1.6**

1.17 Select the YZ Plane (Figure 1.7) and enter the Sketcher Work Bench.

**YZ Plane**

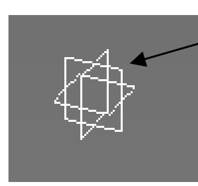


**Figure 1.7**

1.18 Create a circle at the coordinates(0,0) with a diameter of **25mm**(Figure1.2)
1.19 Exit the **Sketcher Work Bench** and select the **Pad** tool.
1.20 Select the More button from the Pad Definition window.
1.21 Enter “20” for both the First and Second lengths.
1.22 Select the OK button. The part should look similar the one shown in figure 1.8

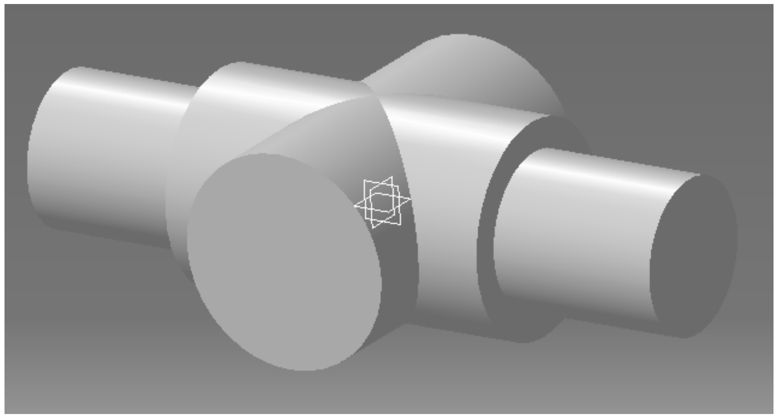


**Figure 1.8**

1.23 Select the **YZ Plane** reference figure 1.7
**1.24** Enter the **Sketcher Work Bench.**
1.25 Create a circle at the coordinates (0,0) with a diameter of **18mm**, as shown in figure 1.5
1.26 Exit the **Sketcher Work Bench**.
1.27 Extrude the part using the **Pad** tool and **More** button. Enter **“40”** mm for both the First and Second Lengths.
1.28 Select the **OK** button. At this point your part should look similar to the one shown in Figure 1.9.

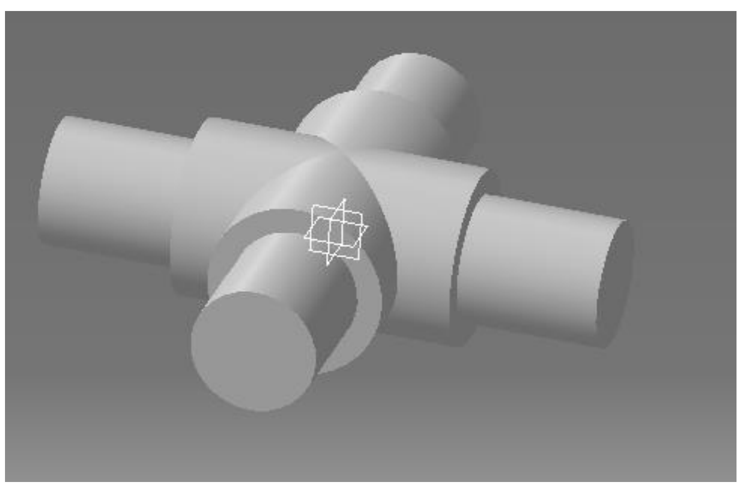


**Figure 1.9**

Save the part as “Swivel.CATPart”.

# TUTORIAL - 2

## Creating The "Top U-Joint" Using Multiple Sketches

2.1 The **"Top U-Joint"** part is a new and completely separate part from the **"Swivel"** part you just created. Since the **"Top U-Joint**" part is a new part, you will need to go to the file option in the top left pull down menu. Select New.
2.2 Selecting the **Part** option automatically puts you in the **Part Design Work Bench**.
2.3 Select the **YZ Plane**, shown in Figure 2.1

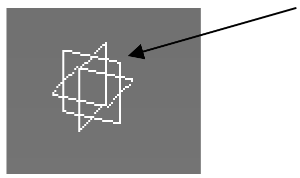



**Figure 2.1**

**2.4** Enter the **Sketcher Work Bench.**
2.5 Sketch the Profile shown in Figure 2.2
2.6 **Constrain** the profile as shown in Figure 2.2

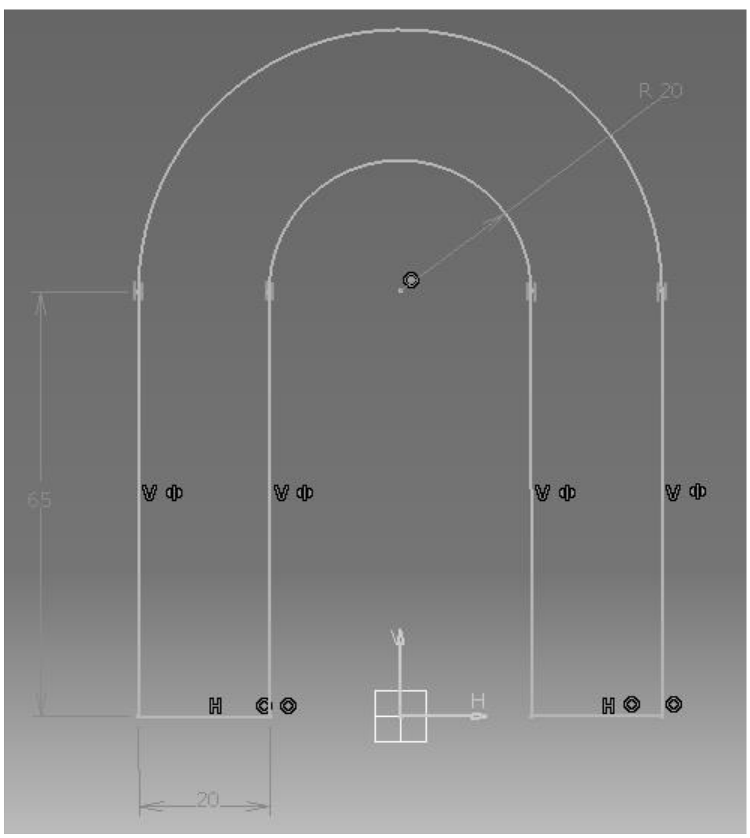



**Figure 2.2**

**2.7** Exit the **Sketcher Work Bench**

2.8 Use the **Pad** tool to extrude the profile 40mm. Your part should look similar to the one shown in figure 2.3

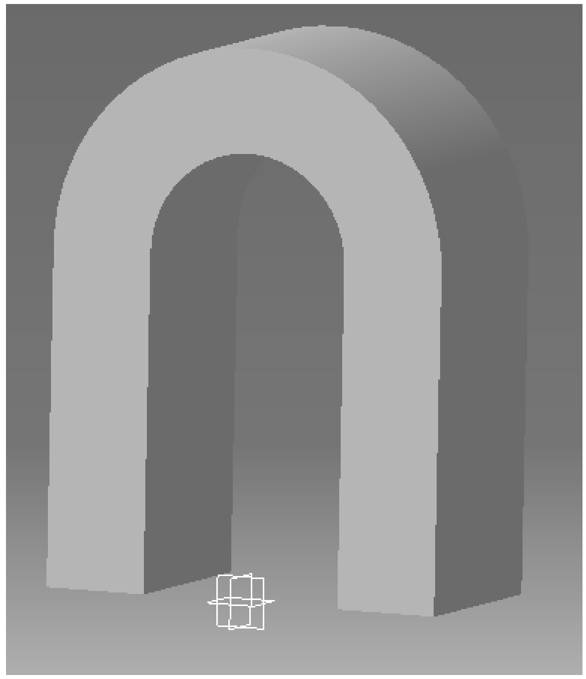


**Figure 2.3**

2.9 The next step is to create a 40mm diameter rounded edge on the bottom of both legs. This process can simplified by using the **Tritangent Fillet tool**. By default the Tritangent Fillet tool will be a sub option under the Edge Fillet tool .

Select the **Tritangent Fillet tool**. 

2.10 Selecting the **Tritangent Fillet** tool will bring up the **Tritangent Fillet Definition** window (Figure 2.4). The first box is the **Faces To Fillet** box. This box allows you to select two faces to be joined with a fillet. The second box is **Face To Remove** box. This box allows you to select the face that will be removed and replaced with the fillet.

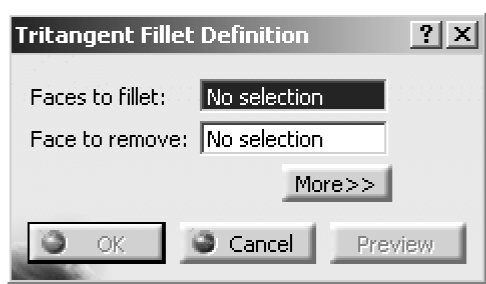



**Figure 2.4**

2.11 Select the front and back surfaces as the **Faces to Fillet** on the **"Top U-Joint",** as shown in Figure 2.5

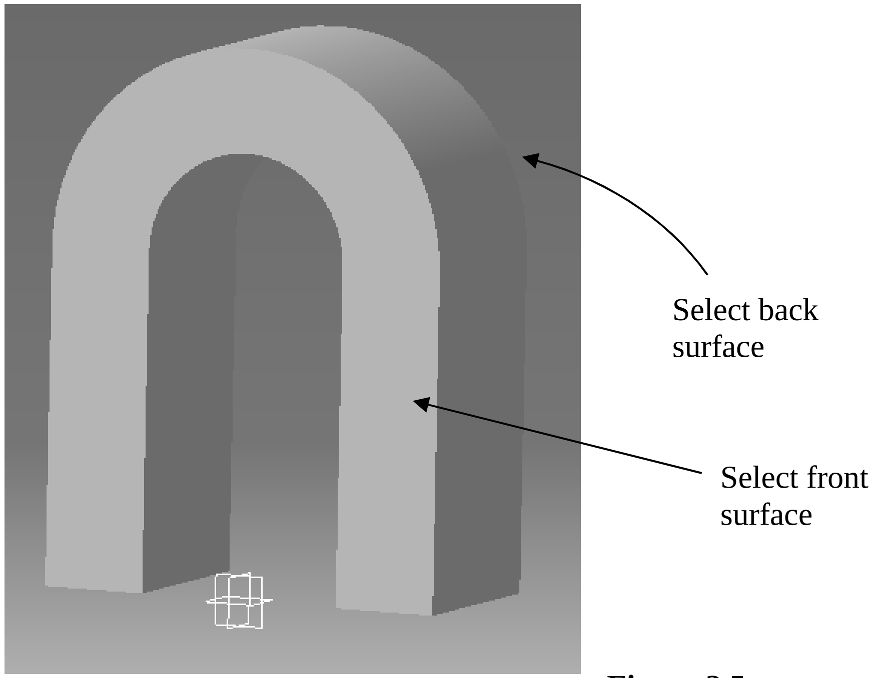
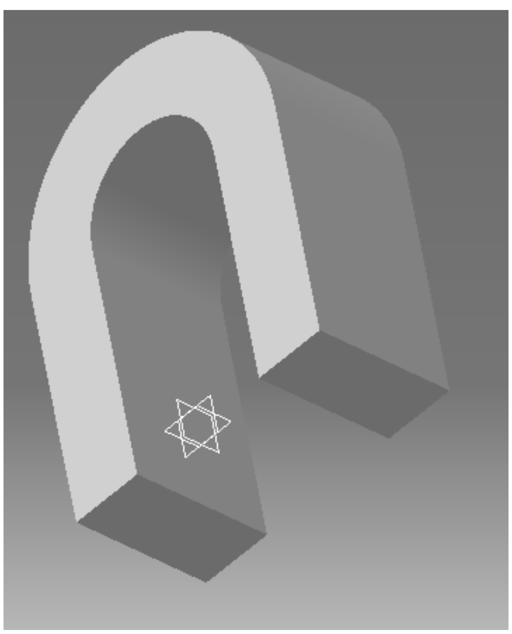



**Figure 2.5**

2.12 Select the bottom surface of the right leg. This surface joins the front and back surfaces. It is the **Face to Remove**, as shown in figure 2.6



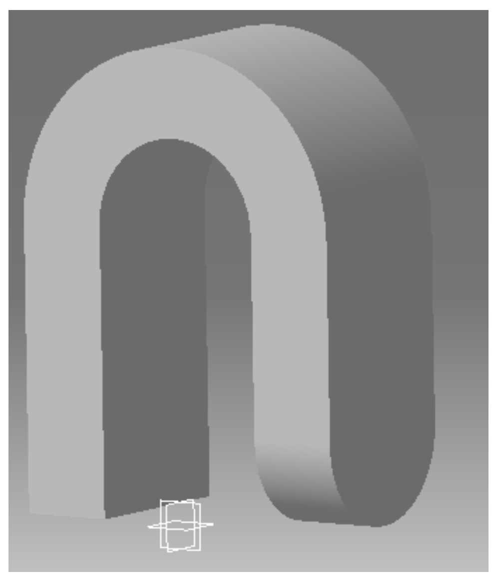
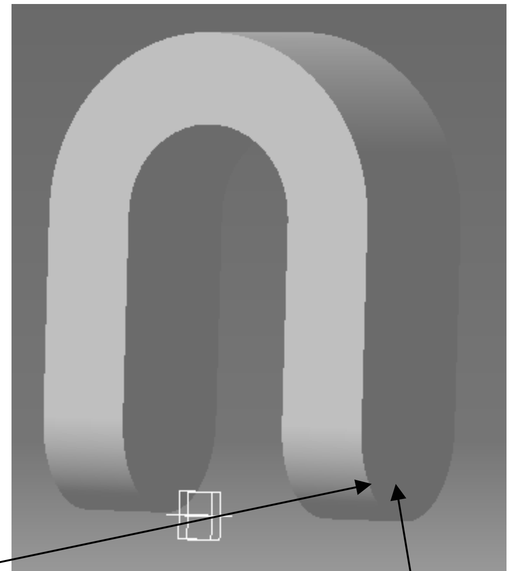


**Figure 2.6** **Figure 2.7**

2.13 Select the **OK** button. The bottom surface selected will be removed and replaced with a radius. The radius will be the same size as the length of the surface it replaced, reference Figure 2.7.

2.14 Repeat steps 2.9 thru 2.13 to create the radius on the other leg. With the radii created on both legs, the part should look similar to the one shown in figure 2.8

2.15 To put the 18mm diameter hole in the two leg of the part, multi select the edge and surface shown in figure 2.8



Edge

Surface

**Figure 2.8**

2.16 Select the **Hole** tool. When the **Hole Definition** Window appears, enter **"18 mm"** for the **Diamete**r box and select "Up To Last" as the Hole Type. This will create the Hole in both legs at the same time. Select OK to create the hole. Your part should look similar to the one in Figure 2.9.

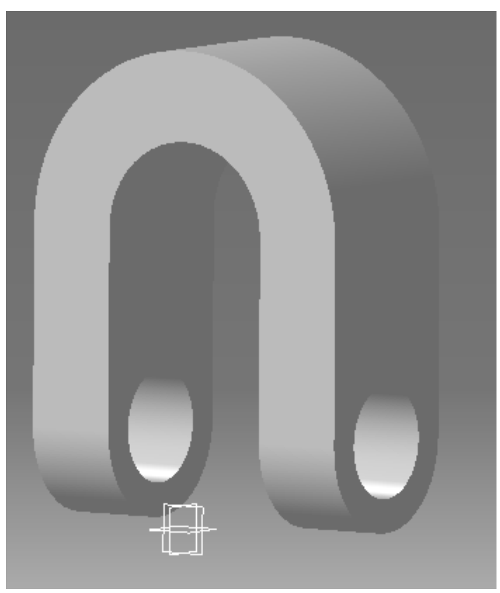
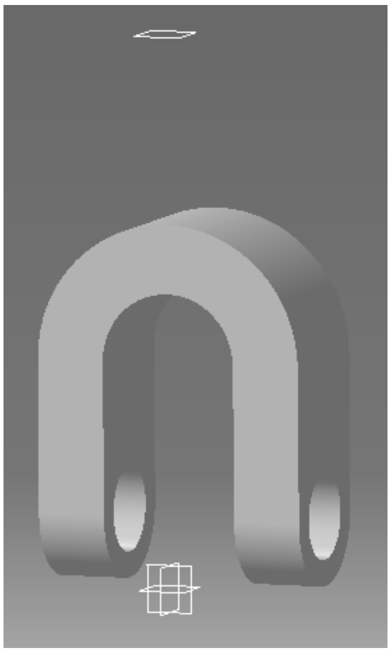


**Figure 2.9**

2.17 The next step is to create the shaft on the top of the **“Top U-Joint**”. To accomplish this you will need to create a **Plane** that will represent the top of the shaft. This **Plane** is where you will create the sketch for the shaft.

2.18 Select the **Plane** tool from the **Reference element tool bar** and with the **Plane Definition** window set **to “Offset from Plane”,** create a plane **150mm** from the **XY Plane** as shown in Figure 2.10



. **Figure 2.10**

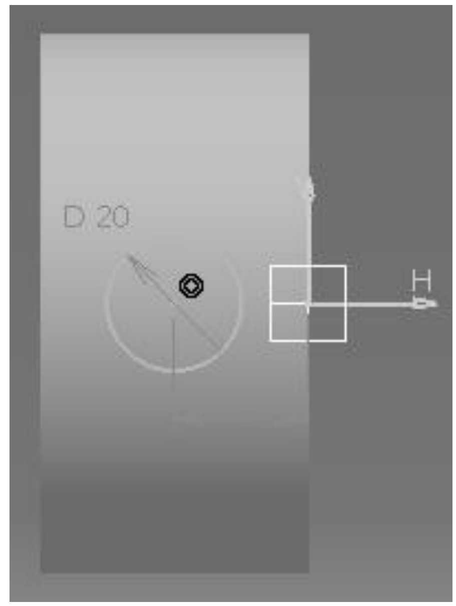



**Figure 2.11**

**2.19** Select the new **Plane** and enter the **Sketcher Work Bench.**
2.20 Create a **Circle,** with a radius of 20mm and **Constrain** it as shown in figure 2.11
2.21 Exit the **Sketcher Work Bench** and extrude the circle down to the top surface of the **“Top U-Joint”** as shown in figure 2.12

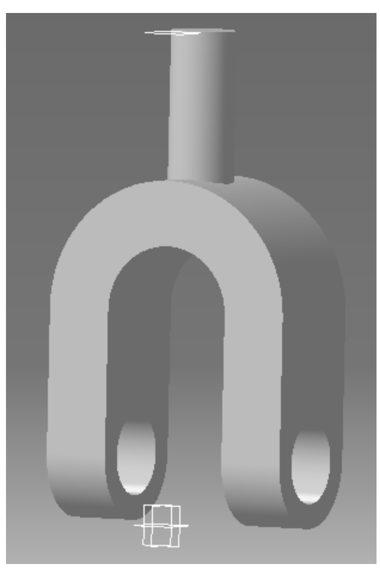


**Figure 2.12**

2.22 Add a **2mm Chamfer** to the top edge of the shaft as shown in figure 2.13

2.23 Add a **5mm** radius **Fillet** to all of the exterior edges of the solid as shown in Figure 2.13

**2.24** The **"Top U-Joint"** has now been created. Rename **Part.1** in the **Specification Tree** to **"Top U-Joint"**

**2.25** Save the part as **'Top U-Joint.CATPart."**

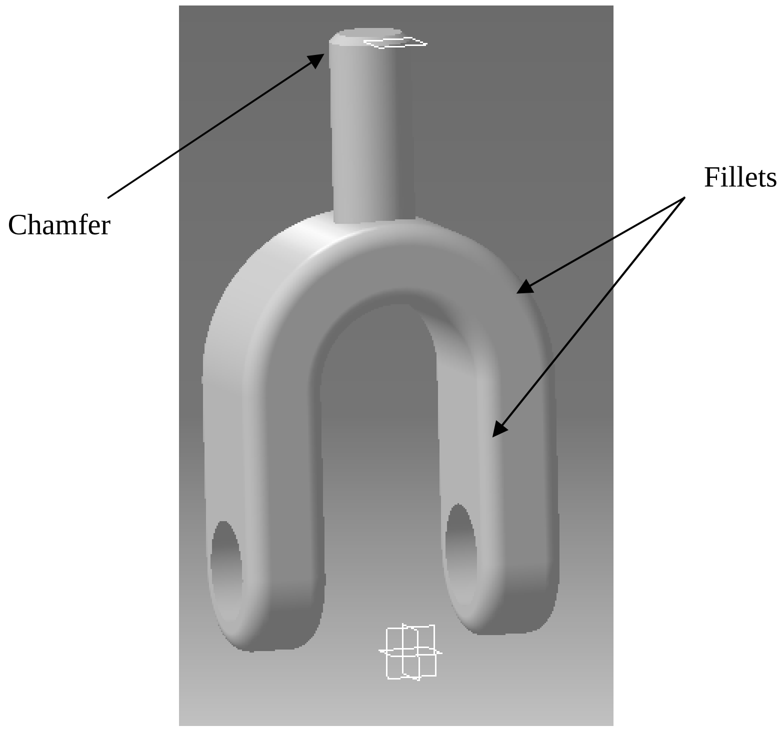



**Figure 2.13**

# Tutorial – 3

**Creating the "Bottom U-Joint" using Boolean Geometry.**

The third and last part you will create in this lesion is the Bottom U-Joint; The Bottom U-Joint is identical to the Top U-Joint. In industry you would create the Bottom U-Joint as efficiently as possible, which would be by duplicating the Top U-Joint. Since it is this books objective to show you how to step will use another method of creating the Bottom U-Joint. At the moment this may not be the most efficient method, but it will help you be a more efficient and knowledgeable CATIA V5 user. The method still has solids using Boolean geometry. The following instructions step you through the process of creating the Bottom U-Joint using Boolean Operations.

3.1 Start a new part. rename it to Bottom U-Joint.

3.2 Select the ZX Plane and enter the Sketcher Work bench.

3.3 Create a rectangle of 80mm x 85mm in the Sketcher Work Bench .

3.4 Exit the Sketcher Work Bench and extrude the box 40mm as shown in figure 3.1

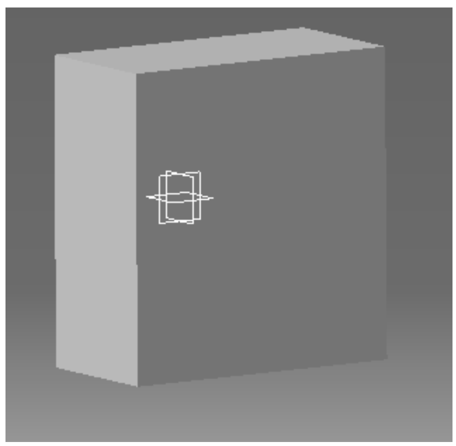


**Figure 3.1**

3.5 Select the Insert tab from the top pull down menu as shown in figure 3.2

3.6 From the Insert window, select the Body option. This will insert a new body into the Specification Tree (Figure 3.3). The purpose of this step will be explained later.

3.7 Select the front surface of the box as shown in figure 3.4 and enter the Sketcher Work Bench. Selecting the surface is the same as selecting a plane; the selection is where the sketch will be created.

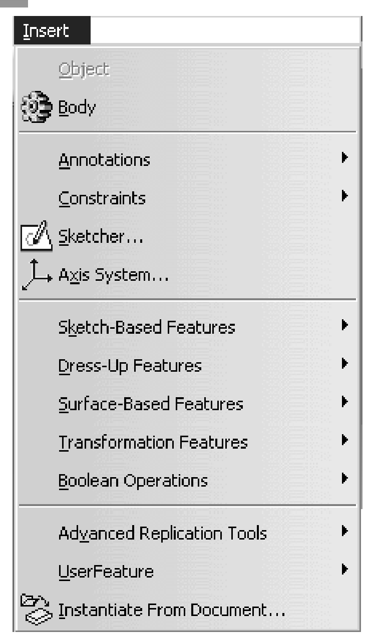
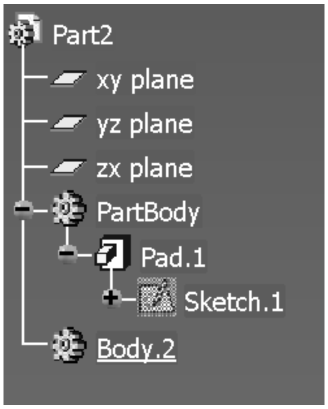
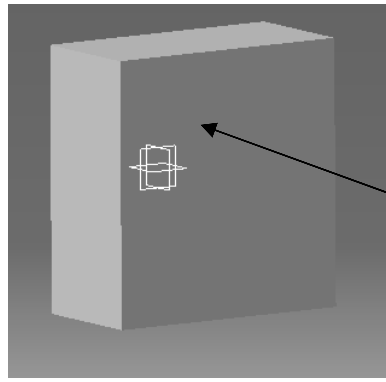



**Figure 3.2**

3.8 Sketch the profile shown in figure 3.5 CATIA V5 will allow you to Constrain the new sketch to the edges of the existing box (Sketch.1) as shown in figure 3.5



Select this face

**Figure 3.3** **Figure 3.4**

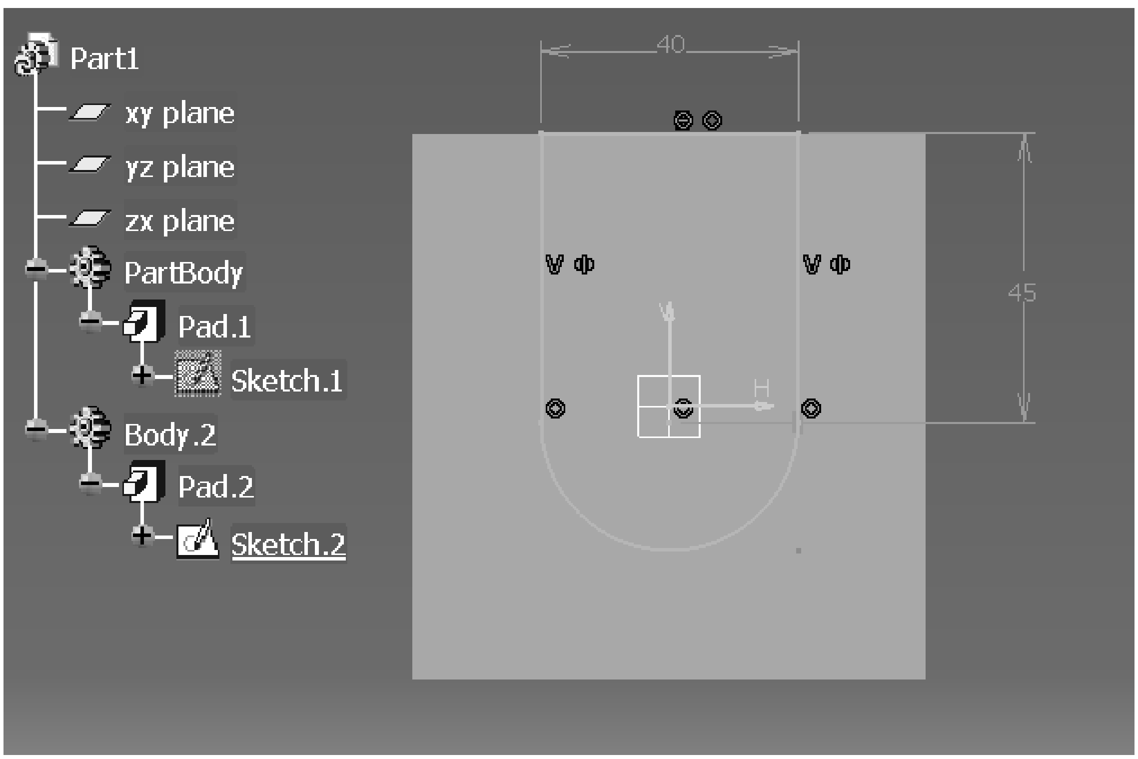



**Figure 3.5**

3.9 Exit the Sketcher Work Bench and select the Pad tool.

3.10 In the Pad Definition window, select the More button. Figure 3.6 shows the Pad Definition window already expanded.

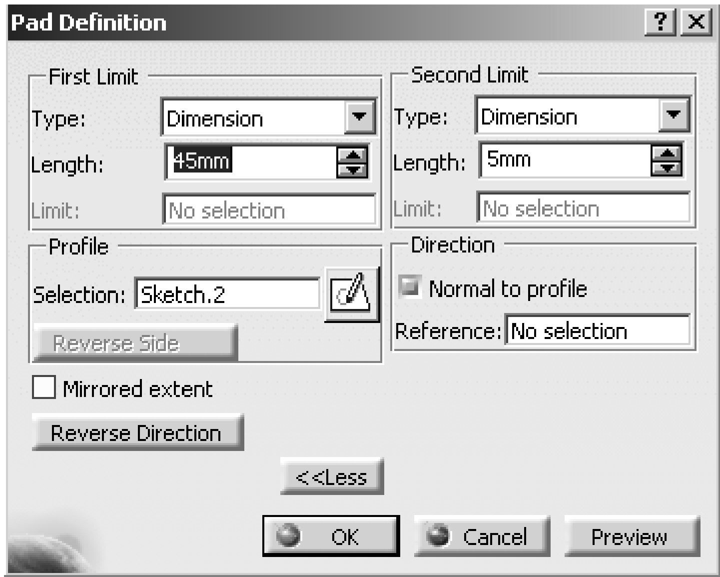



**Figure 3.6**

3.11 In the First Limit box, for the Length, enter “45mm” as shown in figure 3.6
3.12 In the Second Limit box, for the Length enter “.5mm” as shown in figure 3.6. Select the OK button. The extruded solid should look similar to the one shown in figure 3.7. if the solid was extruded in the wrong direction you may need to hit the Reverse Direction button to reverse the direction of the extrusion.

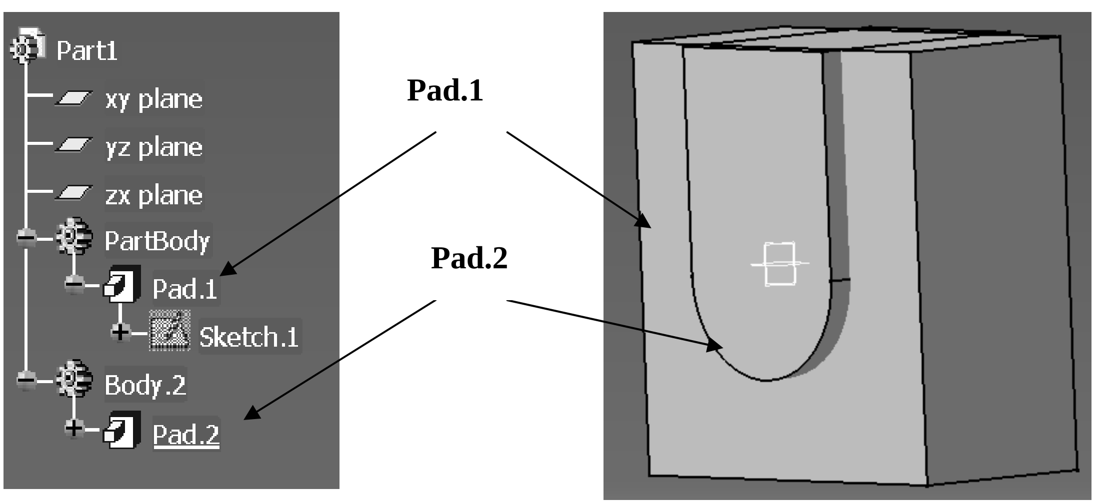



**Figure 3.7**

3.13 Select the branch, Body.2 from the Specification Tree. This should highlight indicating it has been selected. The corresponding solid on the screen Body.2 will also highlight. If Body.2 is not selected, the following steps will not work.
3.14 Select the Edit tab from the top pull down menu as shown in figure 3.8
3.15 Select the body.2 Object from the bottom of the edit window Figure 3.8.

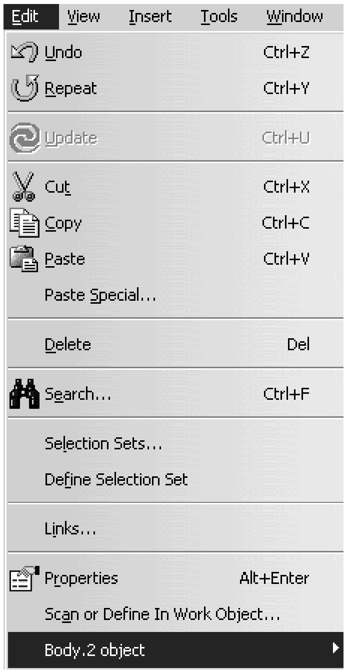




**Figure 3.8**

3.16 Select Remove from the list of Boolean Operations as shown in figure 3.8. this will remove the second profile from the first profile. Reference figure 3.9. selecting the Insert, Boolean Operation, Remove option will give you the same result.

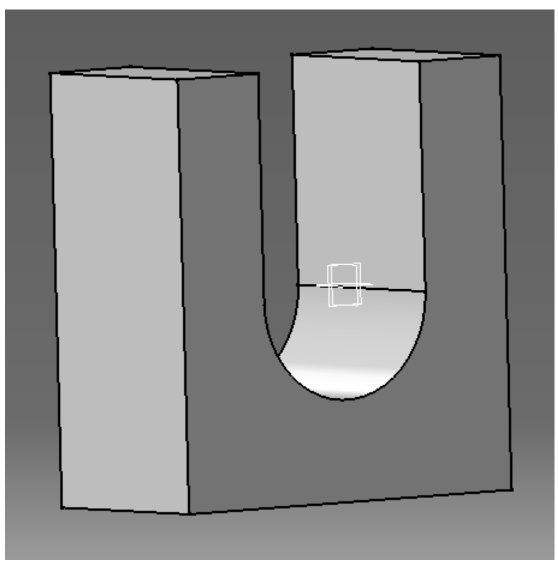


**Figure 3.9**

3.17 Create a 40mm radius Fillet on the bottom two edges of the par. Reference figure 3.10

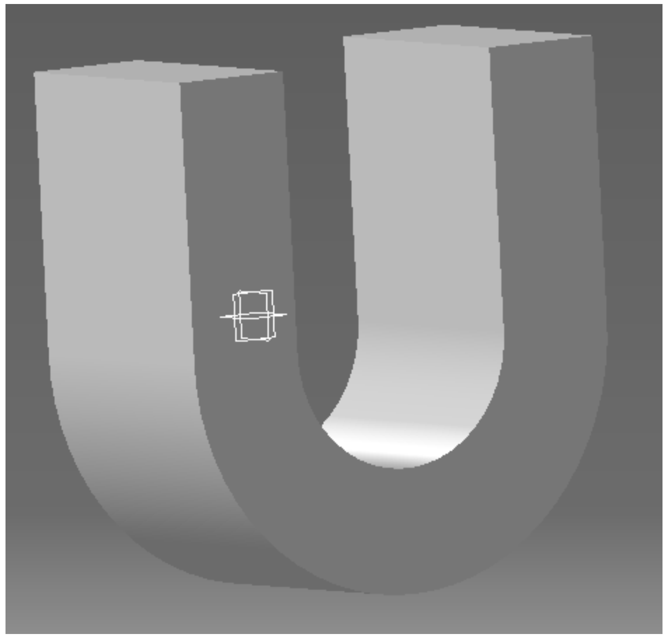


**Figure 3.10**

3.18 By using step 2.9 thru 2.23 then your can construct part, which will be like fig 3.11.

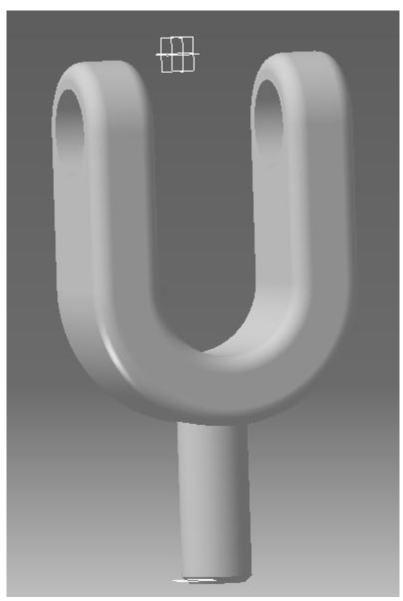


**Figure 3.11**

3.19If your part looks similar the one shown in figure 3.11, you are ready to save your newly created CATPart. Save the part as Bottom U-Joint.CATOPart.

# Tutorial – 4

**Assembling Detail Parts**
**Inserting Components into the Assembly Design Work Bench**

The detail parts that make up the assembly were created in assignment 1,2and3. before the detail parts can be assembled, you will need to insert them into the Assembly Work Bench. The follow steps explain how this is done.

1.1 The top of the Specification Tree must be selected before components can be inserted into the Assembly Design Work Bench. In this case, the top of the Specification Tree is labeled "product1". If your specification Tree has a different number following the "Product", it is OK; the process is the same. That number will change depending on the number of times you have entered the Assembly Design Work Bench.

1.2 With 'Product1" highlighted, select the Existing Component tool. The Insert an Existing Component widow will appear on the screen as shown in figure 4.1

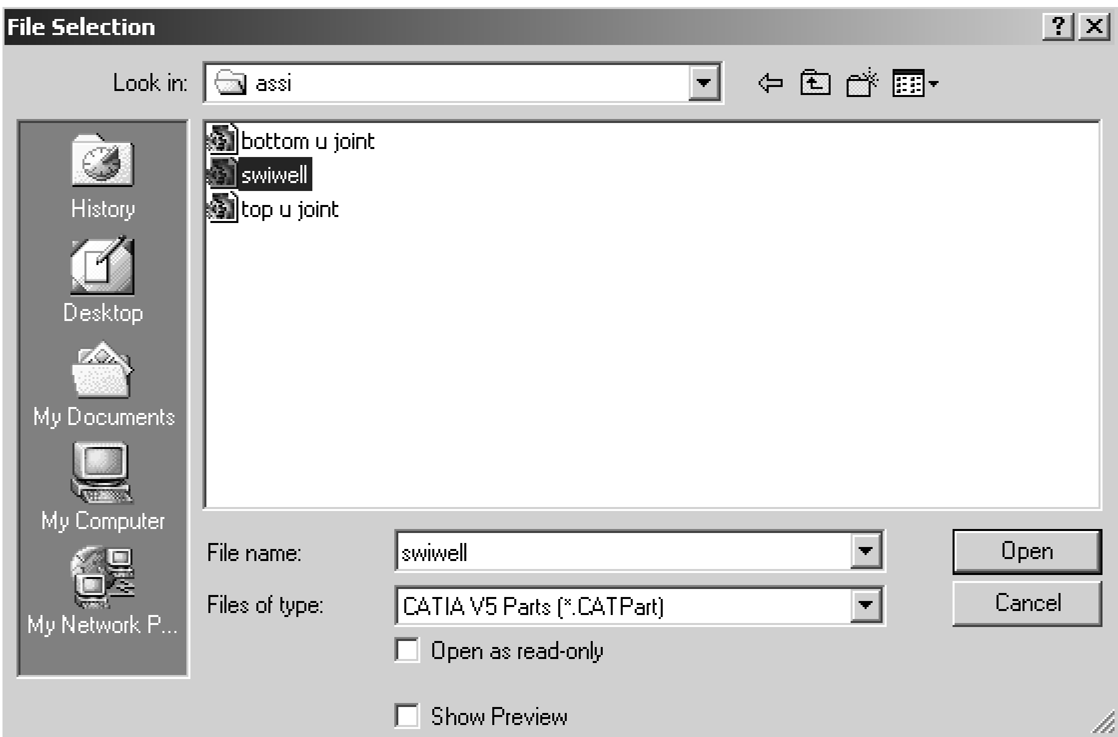



**Figure 4.1**

1.3 In the Insert An Existing Component window, find the file that was saved earlier assignment 1 named "Swivel". Select the open button. Reselect the top of the Specification Tree and select the Existing Component tool as you did in Step 2.2

1.4 In the Insert An Existing Component window, select the file labeled "Top U-Joint". Select the OK button.

1.5 Repeat steps 1.4, but instead of opening the file "Top U-Joint" open the file "Bottom U-Joint". Your screen should look similar to figure 4.2.

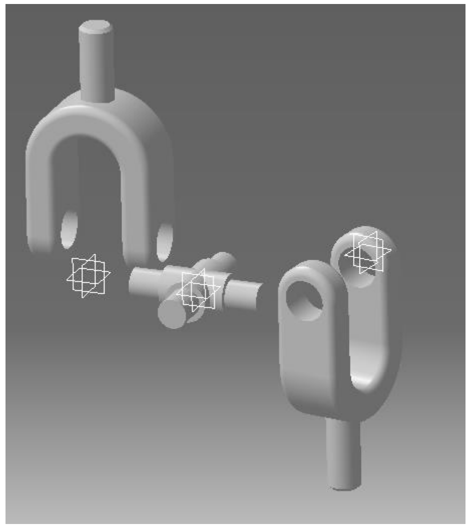


**Figure 4.2**

## 2 Assembling Existing Components

To assemble the components, constraints will need to be created that define the relationship each object has with respect to the others. The steps below will show you how to create these constraints.

2.1 Select the Fix Component tool  from the right side of the screen and then select the "Bottom U-Joint part. It can be selected from the Specification Tree or by selecting the object on the screen. An Anchor symbol will appear on the part to show that it has been fixed as shown in figure 4.3

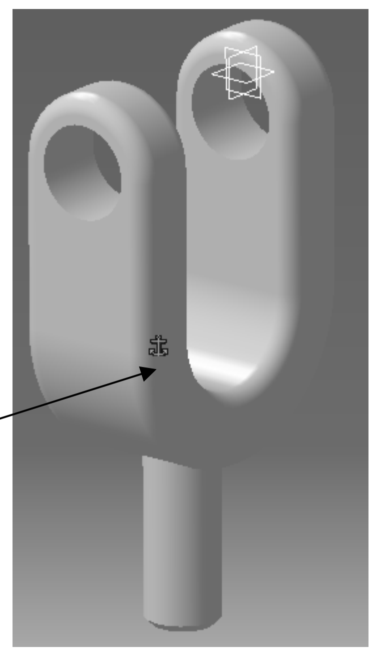
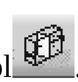
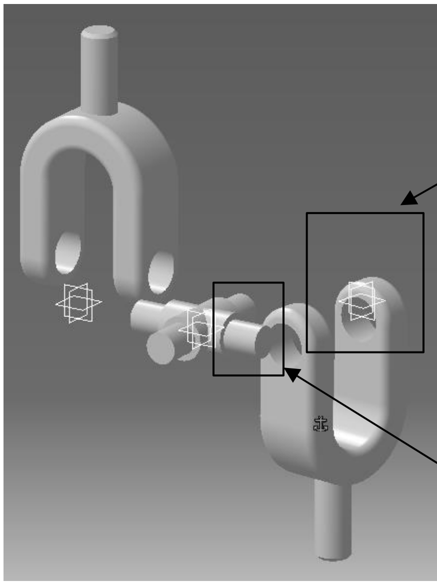
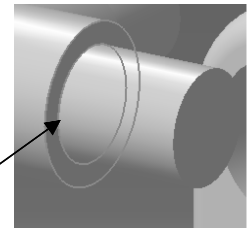



**Figure 4.3**

2.2 Select the contact constraint tool . This tool will place the selected surfaces of one part onto the selected surface of another part. Select the surfaces shown in figure 4.4 views A & B. The Contact constraint symbol will appear on each surface selected. A line connecting the two surfaces will show that they share a common plane as shown in figure 4.5



**View A**

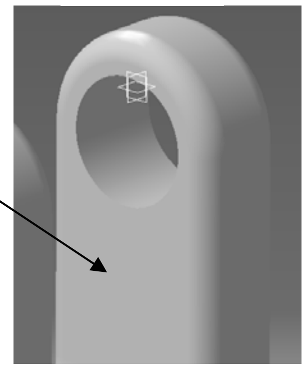
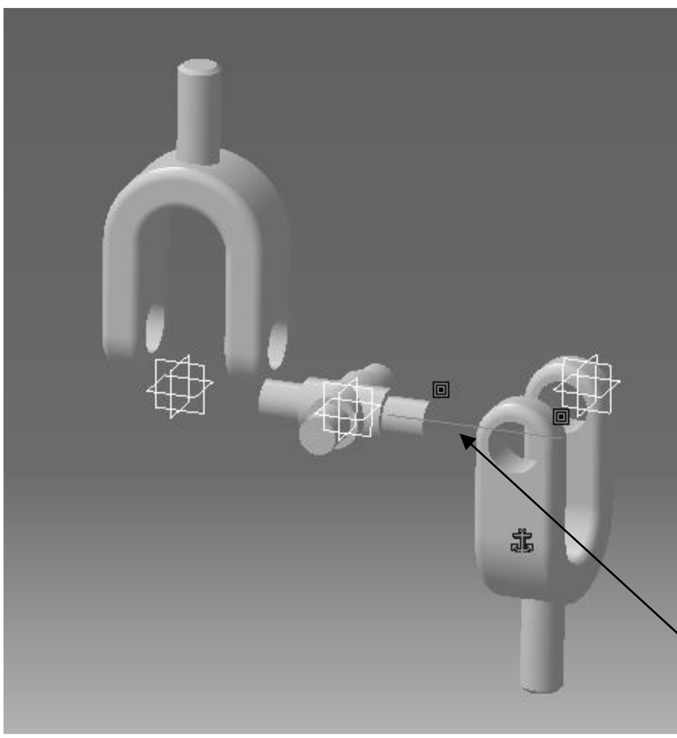


**Figure 4.4**

**View B**



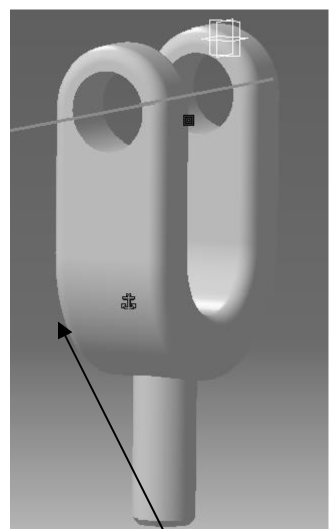




**Figure 4.5**

**Figure 4.6**

2.3 Select the coincidence constraint tool . This command will align the centers of two holes or cylinders. This command will be used to place the shafts of the swivel part though the hiles in the Bottom U-Joint.

2.4 Place the curser on the hole of the Bottom U-Joint part and move it around until the axis that runs through the center of the holes appears as shown in figure 4.6. This can sometimes be tricky; the axis is hard to activate to make it visible. The pointer needsto be in just the right place in order for the Axis Line to appear. Moving the mouse around in the area of the axis should allow you to pick it up. When it appears, press down the left mouse button to select it. Once the Axis line is selected it will show upas highlighted.

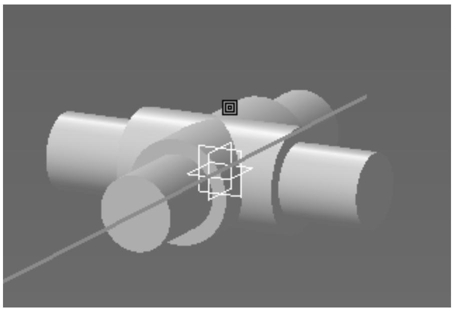
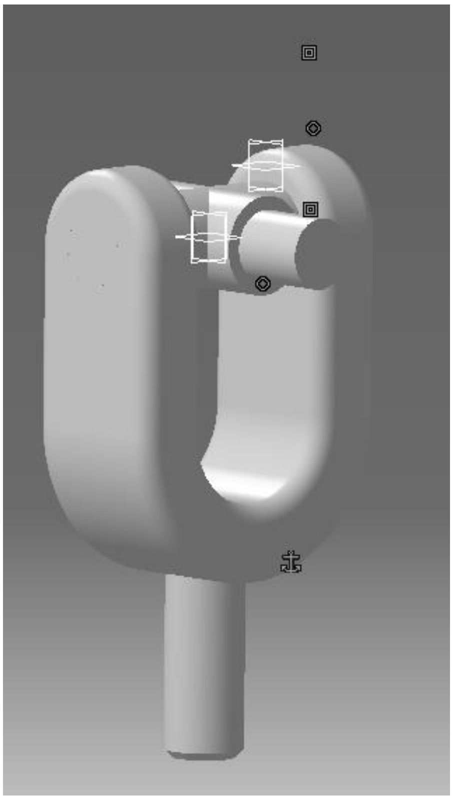


**Figure 4.7**



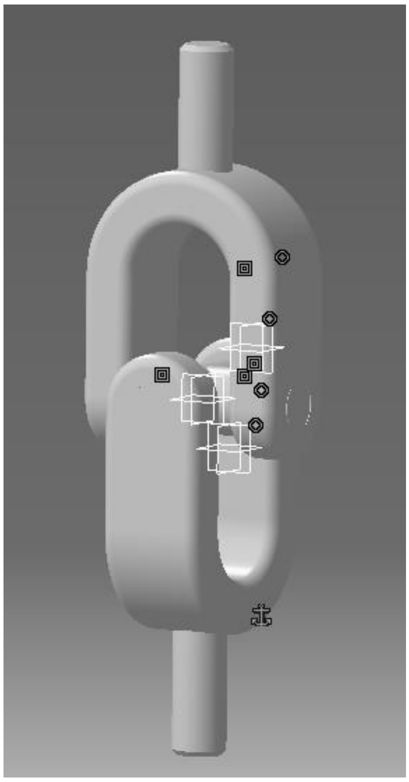


**Figure 4.7** **Figure 4.8**

2.5 Repeat the same process to locate the Axis Line that passes through the center of the cylinder of the swivel part as shown in figure 4.7.

2.6 The swivel should now be lined up with the Bottom U-Joint so that the shafts of the Swivel are inside the holes in the Bottom U-Joint as shown in figure 4.8.

**EX. NO. 1** **ISOMETRIC VIEW - 1**

**DATE:**

## AIM:

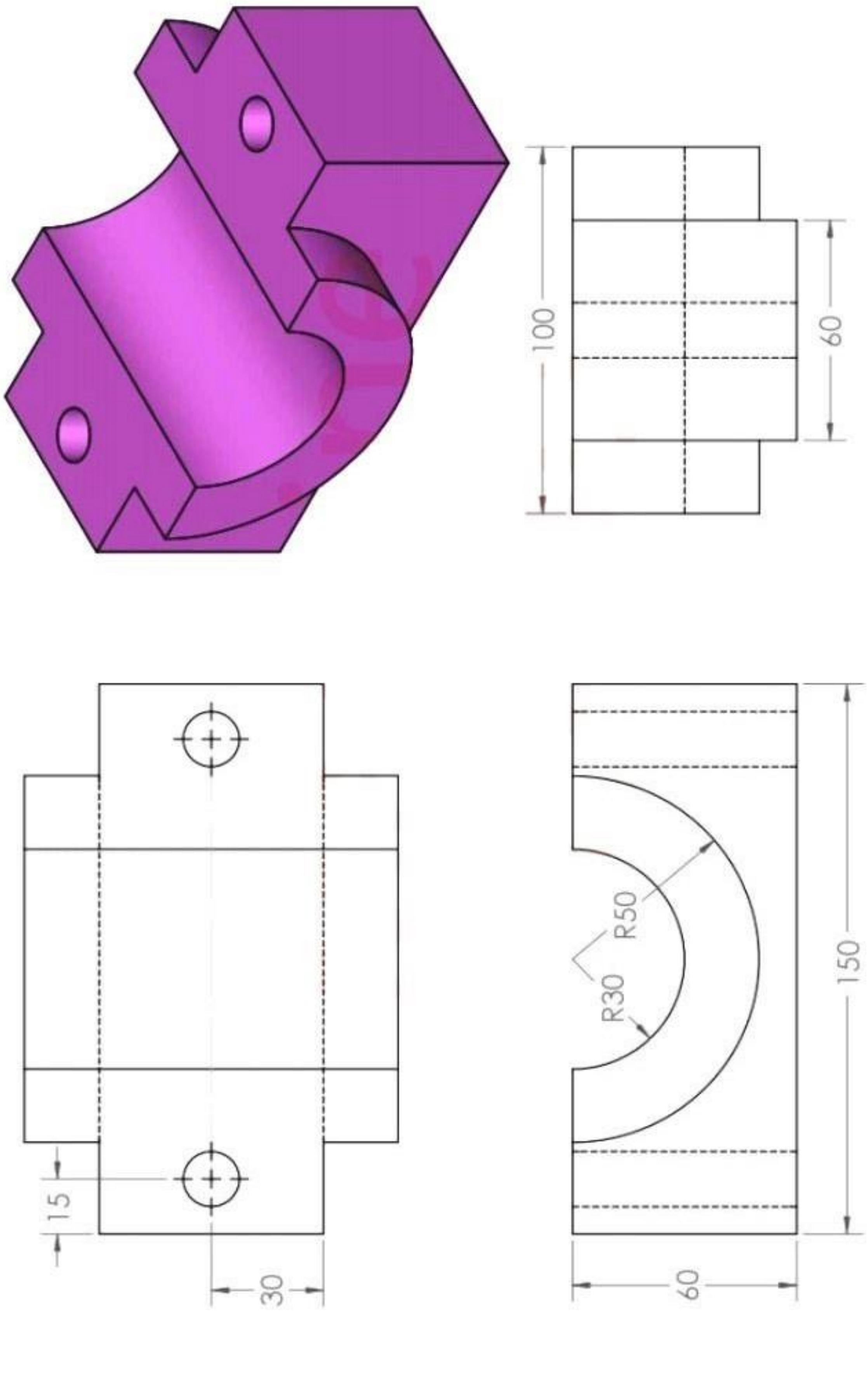
Preparation of 3D model using CATIA V5.18 software

## TOOLS USED:

Pad, Pocketing etc.,

## PROCEDURE:

1. Click the Sketcher icon to start the *Sketcher workbench*.
2. Select XY, YZ, ZX plane to define the sketch plane, now the Sketcher workbench isdisplayed, it contains the tools needed for sketching any profile.
3. Select the profile and draw the part which is given in the model.
4. Using *Constraint* command the dimensions are modified as per the given model.
5. Exit the Sketcher workbench, click *Pad* and give the thickness for the part.
6. Select the face to define the work plane and draw the second element.
7. Using *Pocket* command material is removed and the final model is created.


100
60
R50
R30
150
15
30
60

**RESULT:**

Thus the given 3D model as per the drawing is modelled using CATIA V5.18 software.

**EX. NO. 2** **ISOMETRIC VIEW - 2**

**DATE:**

## AIM:

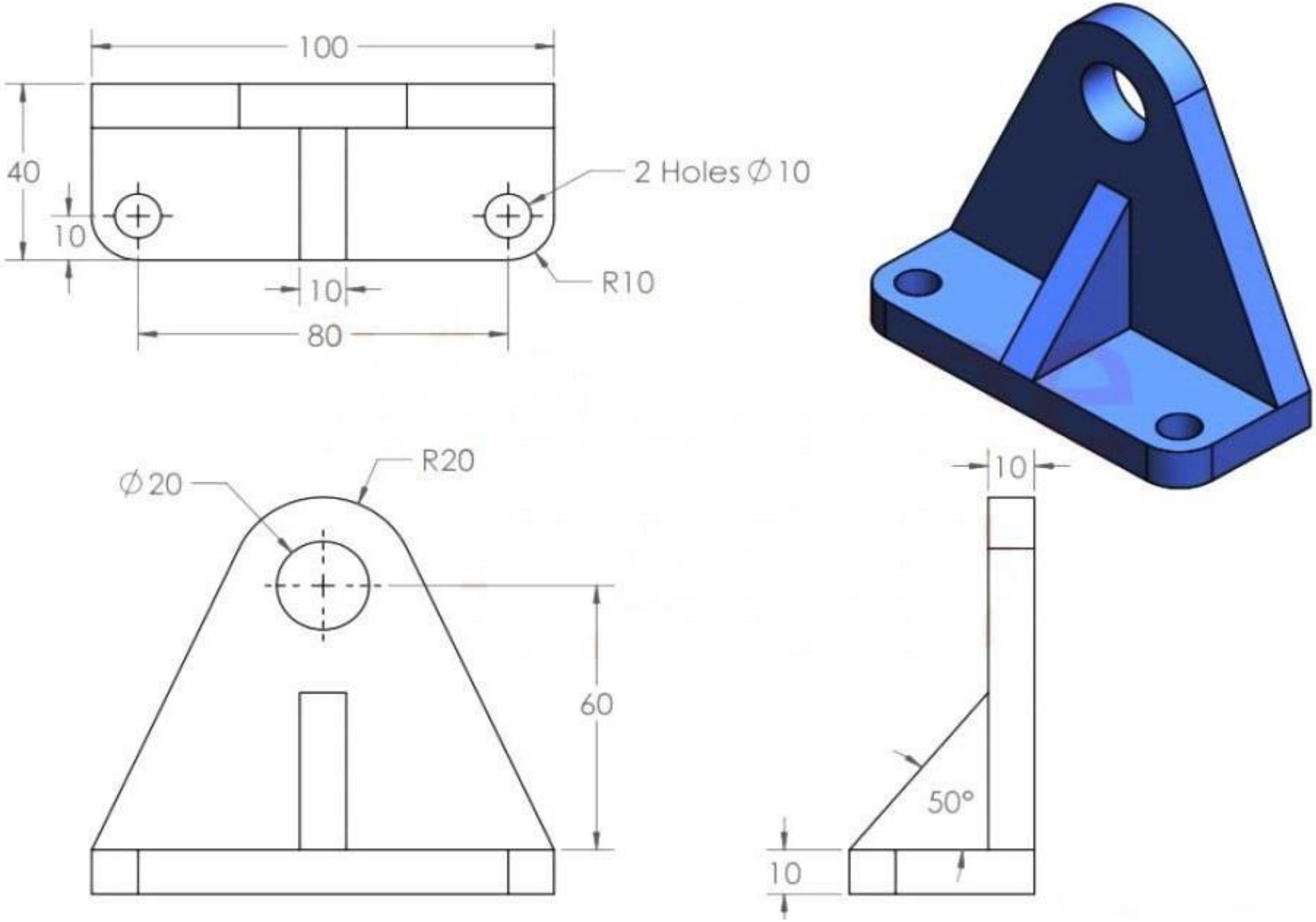
Preparation of 3D model using CATIA V5.18 software

## TOOLS USED:

Pad, Pocketing etc.,

## PROCEDURE:

1. Click the Sketcher icon to start the *Sketcher workbench*.
2. Select XY, YZ, ZX plane to define the sketch plane, now the Sketcher workbench isdisplayed, it contains the tools needed for sketching any profile.
3. Select the profile and draw the part which is given in the model.
4. Using *Constraint* command the dimensions are modified as per the given model.
5. Exit the Sketcher workbench, click *Pad* and give the thickness for the part.
6. Select the face to define the work plane and draw the second element.
7. Using *Pocket* command material is removed and the final model is created.


100
40
10
2 Holes Ø10
R10
10
80
Ø20
R20
60
10
50°
10

**RESULT:**

Thus the given 3D model as per the drawing is modelled using CATIA V5.18 software.

**EX. NO. 3** **ISOMETRIC VIEW - 3**

**DATE:**

## AIM:

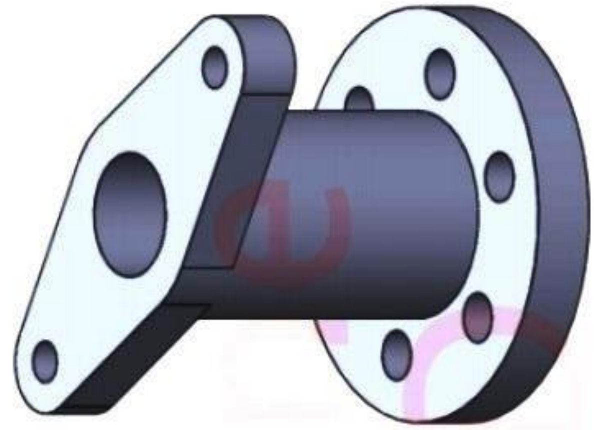
Preparation of 3D model using CATIA V5.18 software

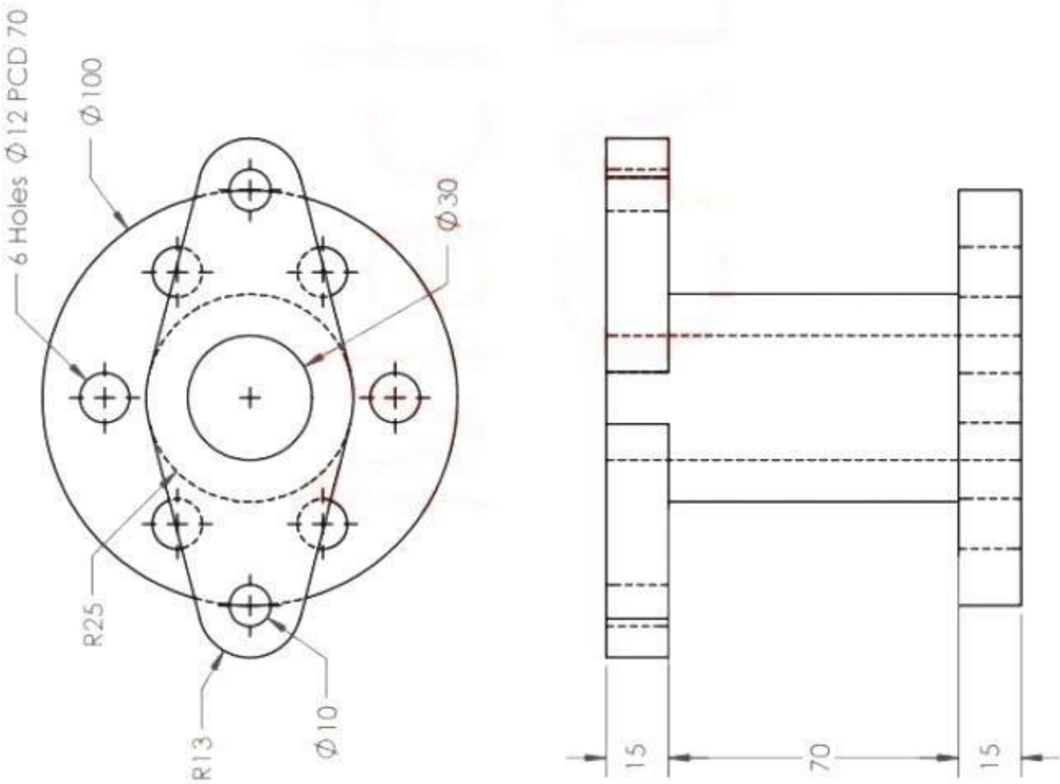
## TOOLS USED:

Pad, Pocketing etc.,

## PROCEDURE:

1. Click the Sketcher icon to start the *Sketcher workbench*.
2. Select XY, YZ, ZX plane to define the sketch plane, now the Sketcher workbench isdisplayed, it contains the tools needed for sketching any profile.
3. Select the profile and draw the part which is given in the model.
4. Using *Constraint* command the dimensions are modified as per the given model.
5. Exit the Sketcher workbench, click *Pad* and give the thickness for the part.
6. Select the face to define the work plane and draw the second element.
7. Using *Pocket* command material is removed and the final model is created.




6 Holes Ø12 PCD 70
Ø100
Ø30
R25
R13
Ø10
15
70
15

**RESULT:**

Thus the given 3D model as per the drawing is modelled using CATIA V5.18 software.

**EX. NO. 4** **ISOMETRIC VIEW - 4**

**DATE:**

**AIM:**

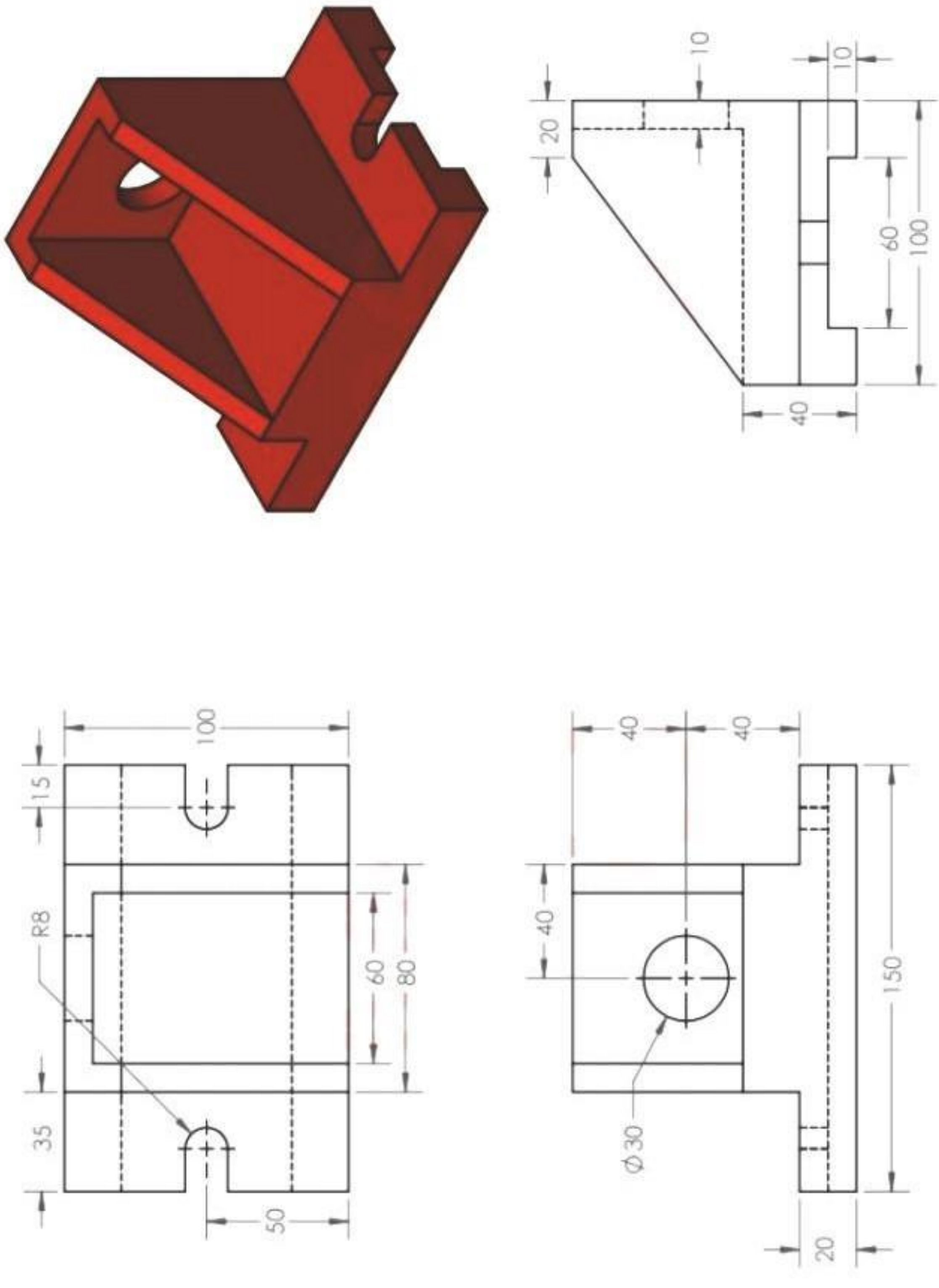
Preparation of 3D model using CATIA V5.18 software

**TOOLS USED:**

Pad, Pocketing etc.,

**PROCEDURE:**

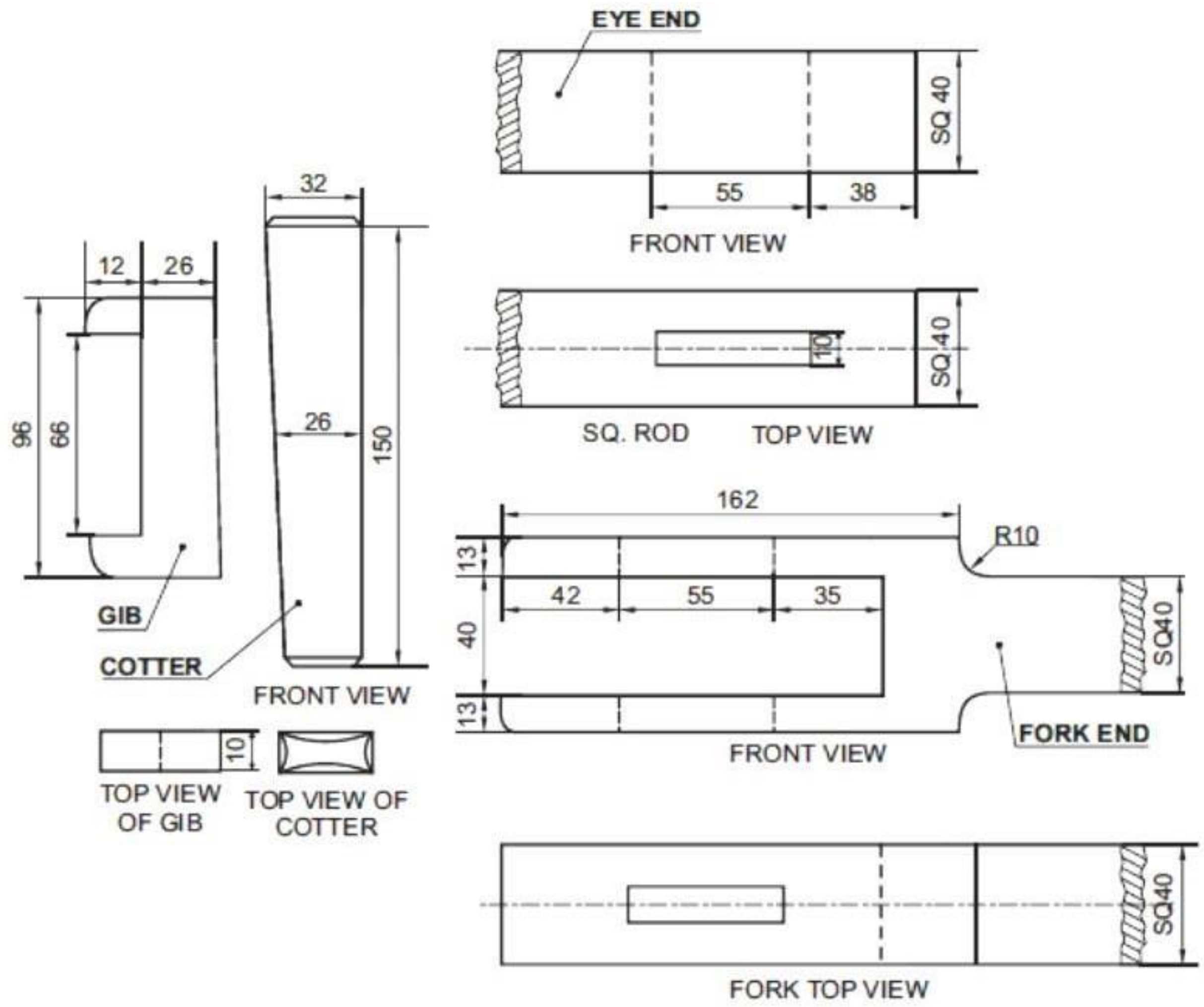
1. Click the Sketcher icon to start the *Sketcher workbench*.
2. Select XY, YZ, ZX plane to define the sketch plane, now the Sketcher workbench isdisplayed, it contains the tools needed for sketching any profile.
3. Select the profile and draw the part which is given in the model.
4. Using *Constraint* command the dimensions are modified as per the given model.
5. Exit the Sketcher workbench, click *Pad* and give the thickness for the part.
6. Select the face to define the work plane and draw the second element.
7. Using *Pocket* command material is removed and the final model is created.



10
10
20
60
100
40
100
15
R8
35
50
60
80
40
40
40
150
Ø30
20

**RESULT:**

Thus the given 3D model as per the drawing is modelled using CATIA V5.18 software.




DETAILS OF A GIB AND COTTER JOINT

**EX. NO. 5** **ASSEMBLY OF GIB AND COTTER JOINT**

**DATE:**

## AIM:

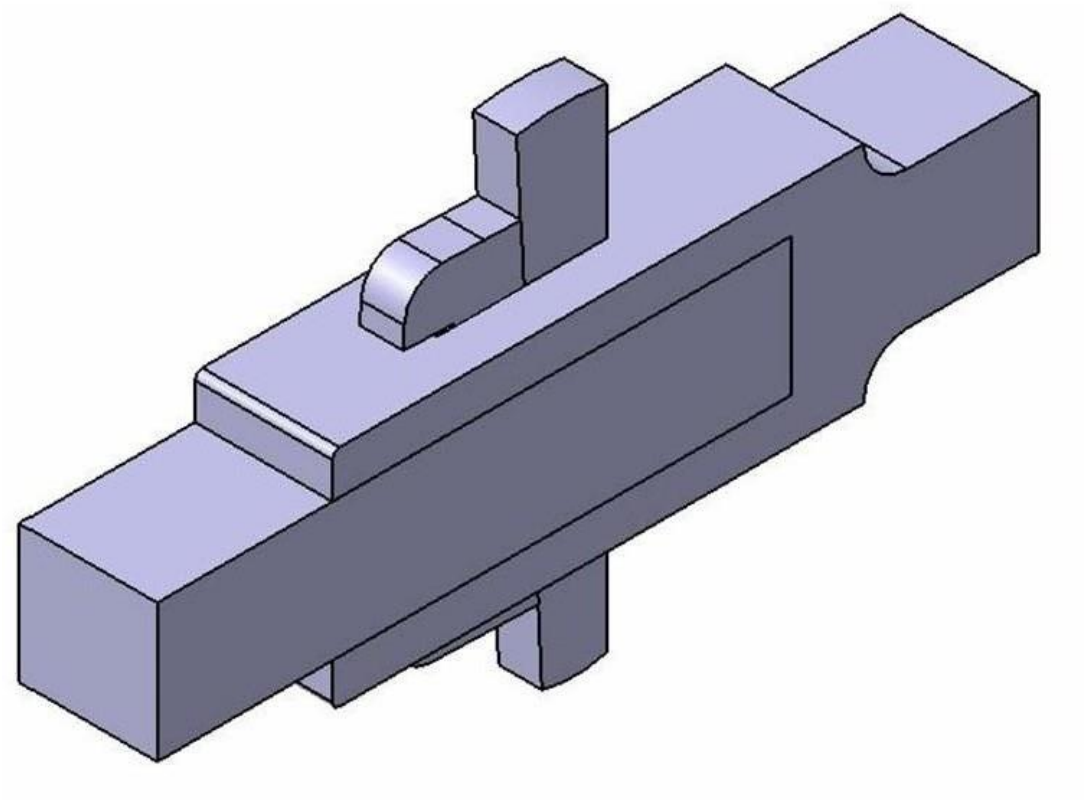
Preparation of 3D Assembly model using CATIA V5.18 software

## TOOLS USED:

Existing Component icon, Smart move, Constraining etc.,

## PROCEDURE:

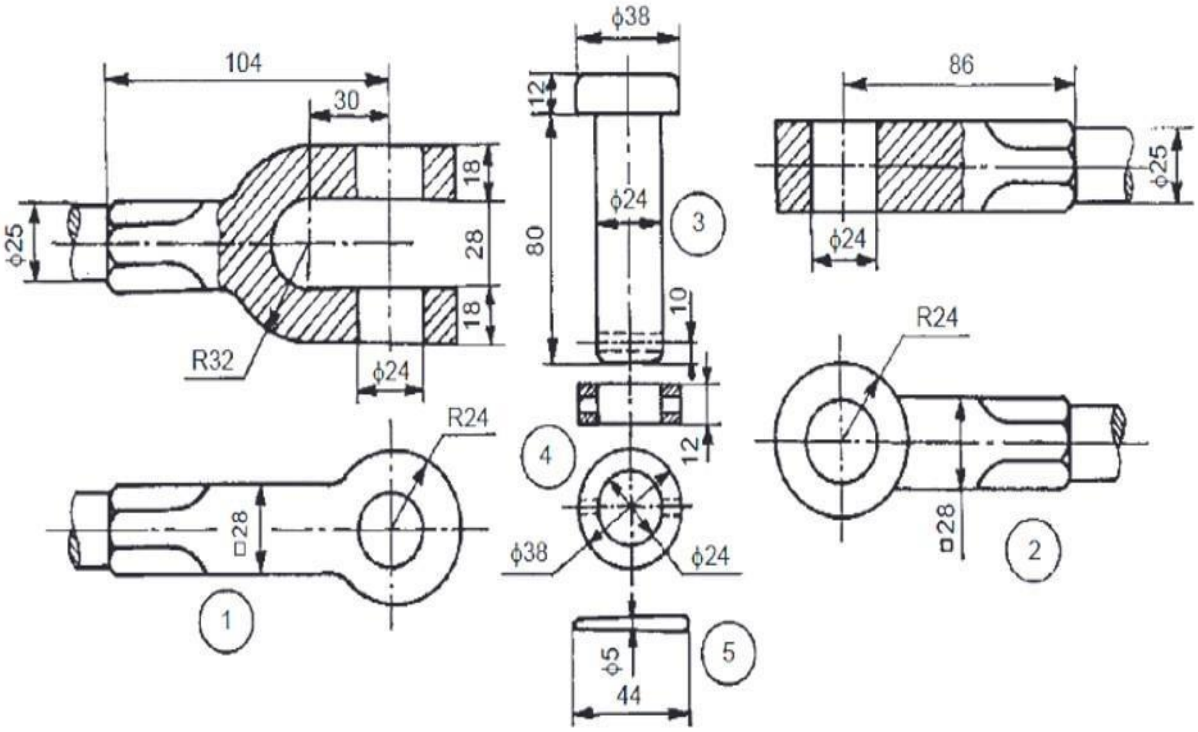
1. *Assembly Design* command is activated to launch the required workbench.
2. The commands for assembling parts are available in the toolbar on the right side of the application window.
3. Click the *Existing Component icon* in the Product Structure toolbar, then the File Selection dialog box is displayed were required the part model is selected.
4. To constrain a model for positioning parts correctly is carried using the various constraints tools like *Constraining and Manipulating* and Select the fix component icon,to *Fix* a part in the space.
5. Then the parts are assembled in the following order ***Fork, Block, Gib, Cotter*** by adopting above methods
6. Select the update icon for assembly to be updated, all assembly design models are ensured that it is in constraints condition before the models are finished.



Assembly of Gibb and Cotter joint

**RESULT:**

Thus the given 3D assembly model as per the drawing is modelled usingCATIA V5.18 software.




Parts list

| Sl. No. | Name | Matl. | Qty. |
|---|---|---|---|
| 1 | Fork end | Forged steel | 1 |
| 2 | Eye end | Forged steel | 1 |
| 3 | Pin | Mild steel | 1 |
| 4 | Collar | Mild steel | 1 |
| 5 | Taper pin | Mild steel | 1 |

Details of Knuckle joint

**EX. NO. 6** **ASSEMBLY OF KNUCKLE JOINT**

**DATE:**

## AIM:

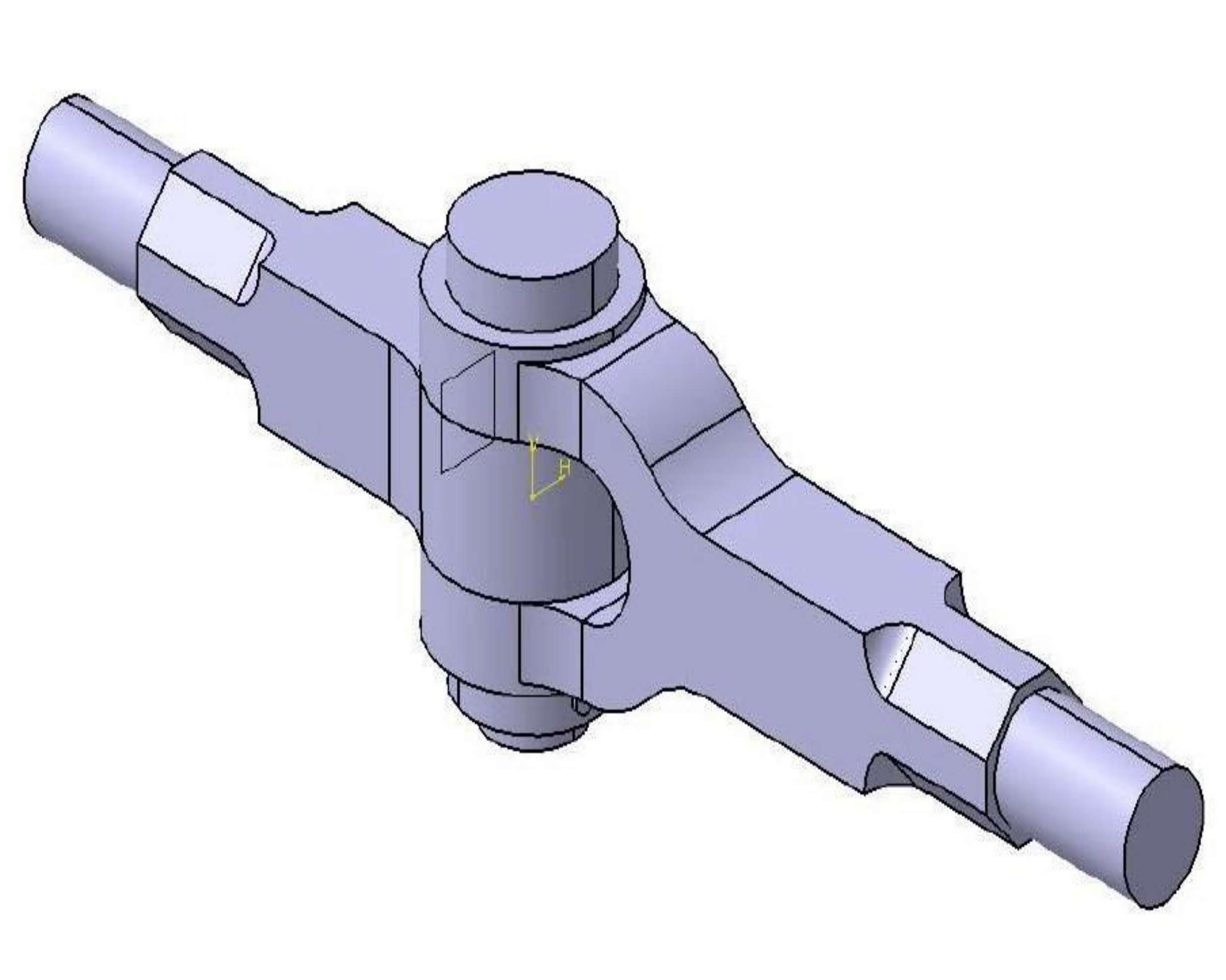
Preparation of 3D Assembly model using CATIA V5.18 software

## TOOLS USED:

Existing Component icon, Smart move, Constraining etc.,

## PROCEDURE:

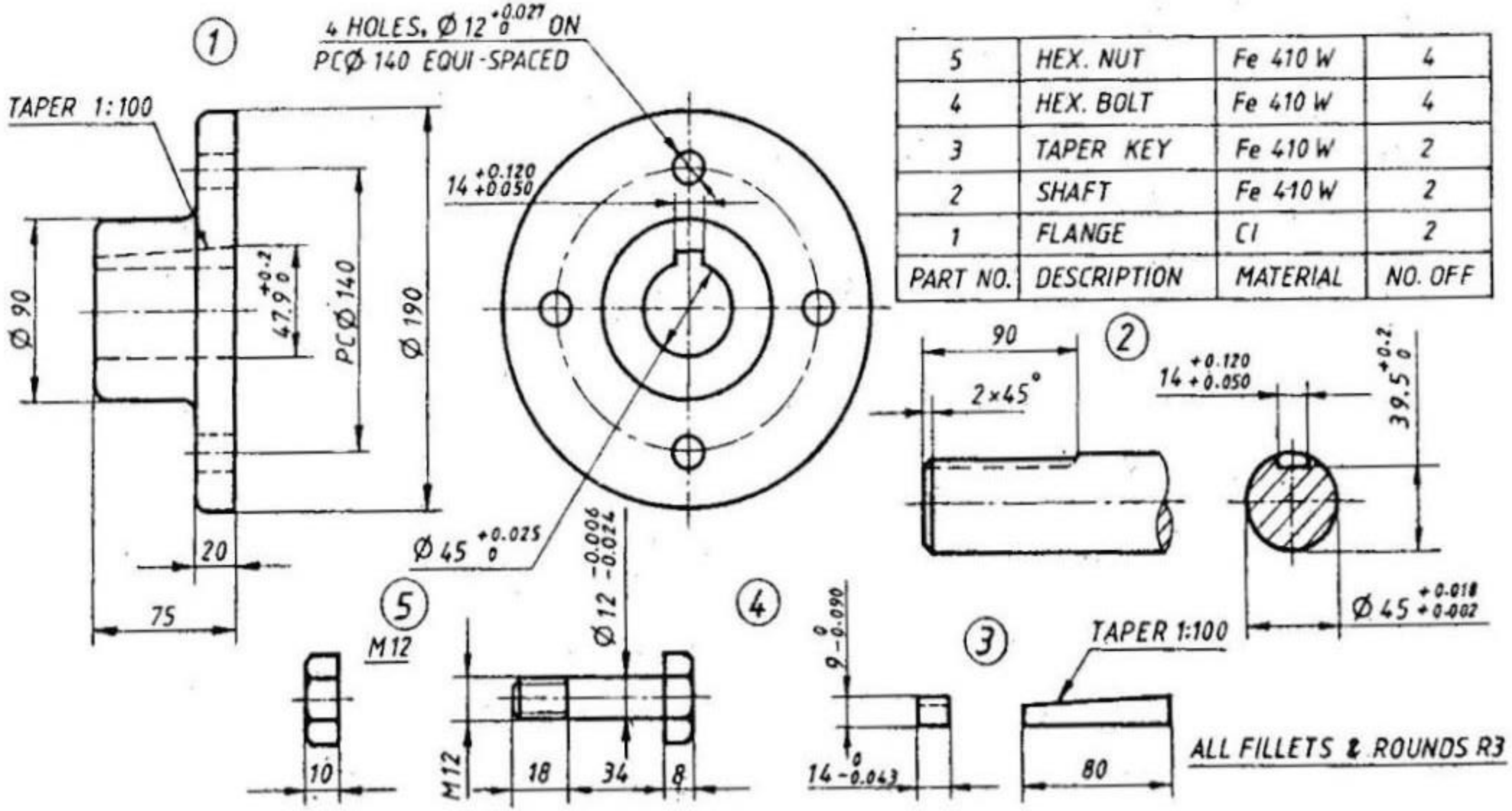
1. *Assembly Design* command is activated to launch the required workbench.
2. The commands for assembling parts are available in the toolbar on the right side of the application window.
3. Click the *Existing Component icon* in the Product Structure toolbar, then the File Selection dialog box is displayed were required the part model is selected.
4. To constrain a model for positioning parts correctly is carried using the various constraints tools like *Constraining and Manipulating* and Select the fix component icon,to *Fix* a part in the space.
5. Then the parts are a assembled in the following order ***Fork end, Eye end, Pin, Collar, Taper pin*** by adopting above methods
6. Select the update icon for assembly to be updated, all assembly de sign models are ensured that it is in constraints condition before the models are finished.



Assembly of Knuckle Joint

**RESULT:**

Thus the given 3D assembly model as per the drawing is modelled usingCATIA V5.18 software.




| PART NO. | DESCRIPTION | MATERIAL | NO. OFF |
|---|---|---|---|
| 5 | HEX. NUT | Fe 410 W | 4 |
| 4 | HEX. BOLT | Fe 410 W | 4 |
| 3 | TAPER KEY | Fe 410 W | 2 |
| 2 | SHAFT | Fe 410 W | 2 |
| 1 | FLANGE | CI | 2 |

*All Dimensions in mm*

**Details of Flanged Coupling — Unprotected Type**

**EX. NO. 7** **ASSEMBLY OF FLANGED COUPLING**

**DATE:**

## AIM:

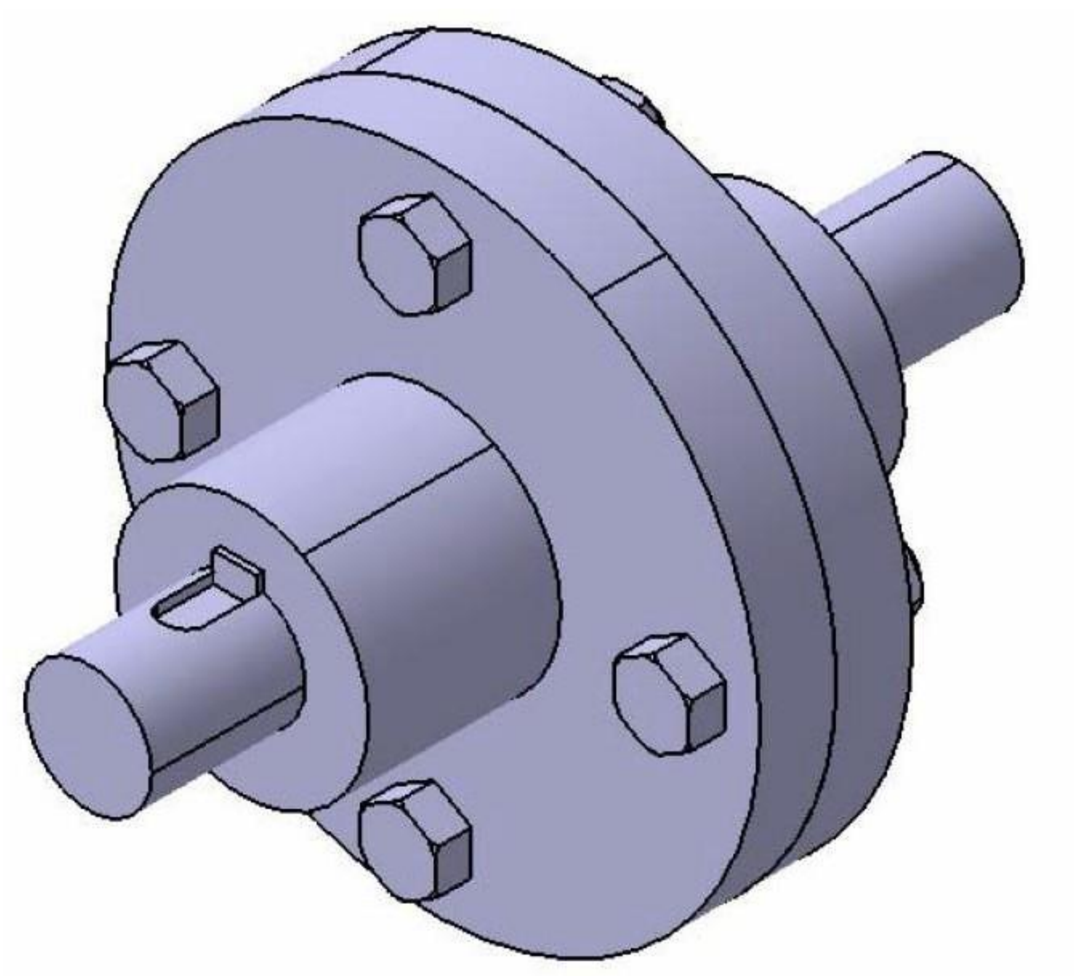
Preparation of 3D Assembly model using CATIA V5.18 software.

## TOOLS USED:

Existing Component icon, Smart move, Constraining etc.,

## PROCEDURE:

1. *Assembly Design* command is activated to launch the required workbench.
2. The commands for assembling parts are available in the toolbar on the right side of theapplication window.
3. Click the *Existing Component icon* in the Product Structure toolbar, then the File Selection dialog box is displayed were required the part model is selected.
4. To constrain a model for positioning parts correctly is carried using the various constraints tools like *Constraining and Manipulating* and Select the fix component icon,to *Fix* a part in the space.
5. Then the p arts are assembled in the following order ***Flange (Male), Flange (Female), Shaft, Key*** by adopting above methods.
6. Select the update icon for assembly t o b e updated, all assembly de sign models are ensured that it is in constraints condition before the models are finished.

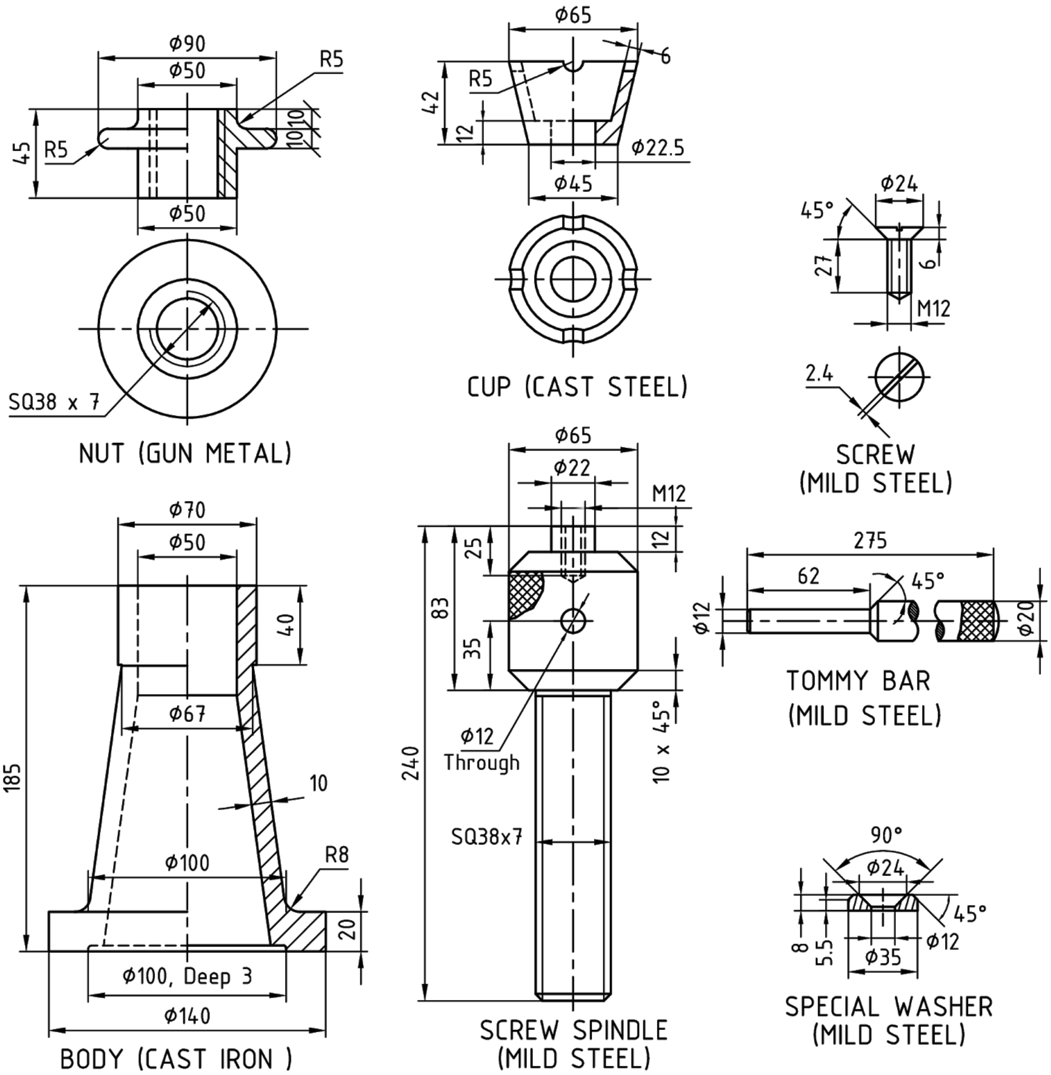


Assembly of Unprotected type Flange Coupling

**RESULT:**

Thus the given 3D assembly model as per the drawing is modelled usingCATIA V5.18 software.

# DETAILS OF SCREW JACK



**EX. NO. 10** **ASSEMBLY OF SCREW JACK**

**DATE:**

## AIM:

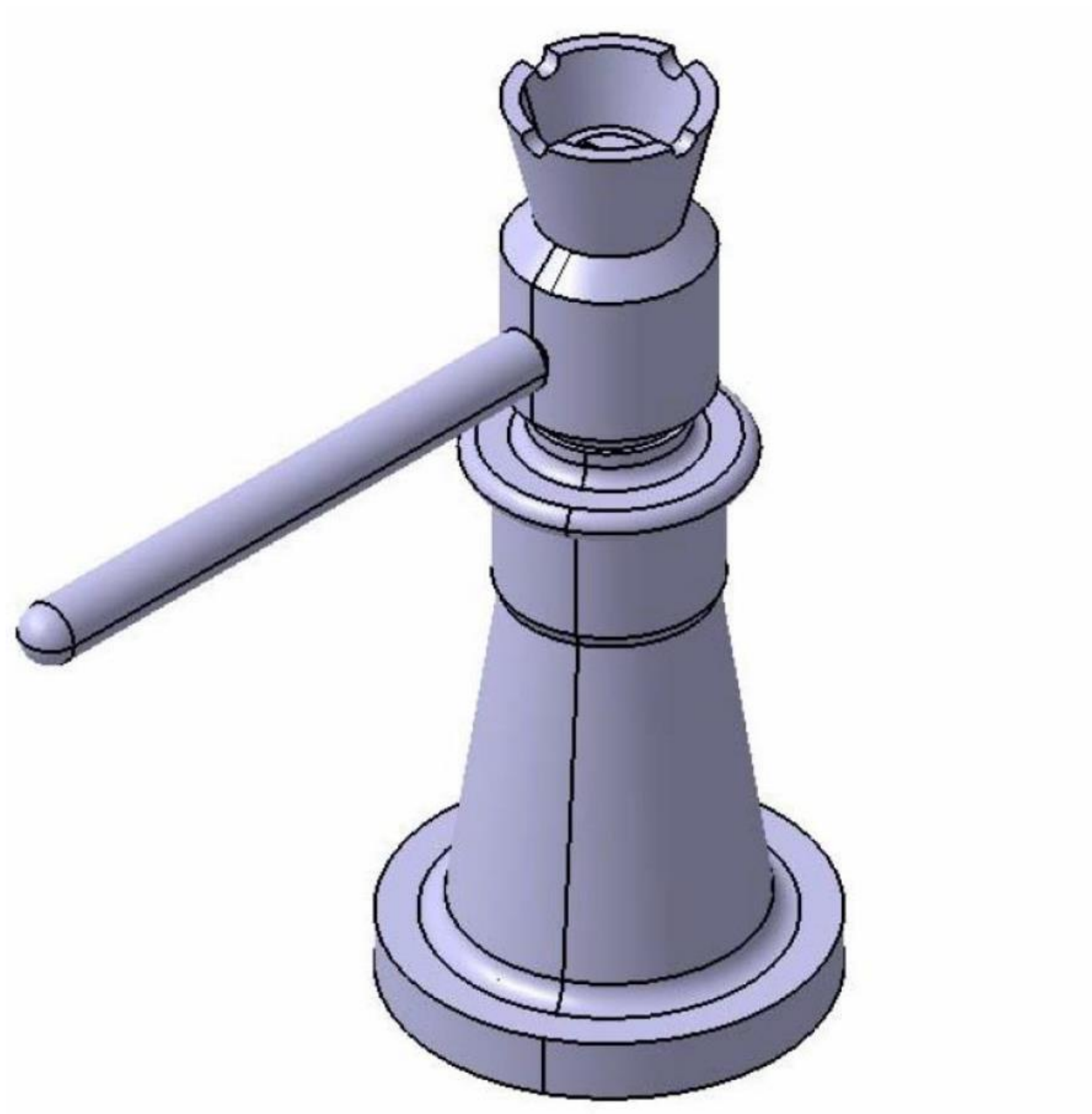
Preparation of 3D Assembly model using CATIA V5.18 software

## TOOLS USED:

Existing Component icon, Smart move, Constraining etc.,

## PROCEDURE:

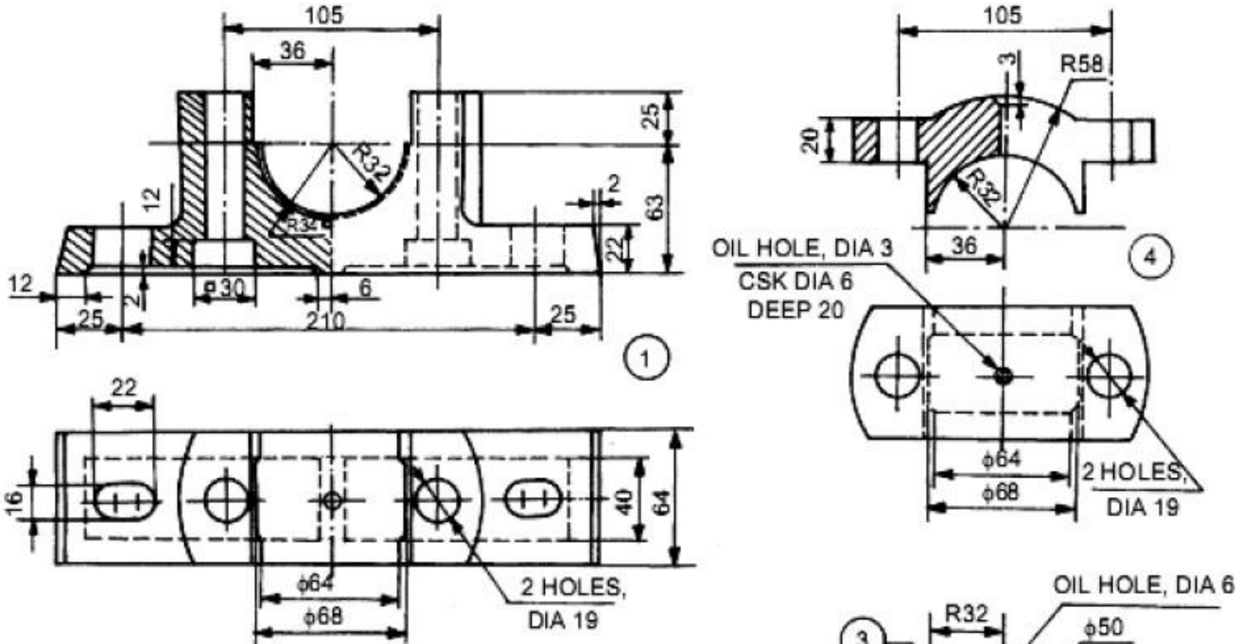
1. *Assembly Design* command is activated to launch the required workbench.
2. The commands for assembling parts are available in the toolbar on the right side of theapplication window.
3. Click the *Existing Component icon* in the Product Structure toolbar, then the File Selection dialog box is displayed were required the part model is selected.
4. To constrain a model for positioning parts correctly is carried using the various constraints tools like *Constraining and Manipulating* and Select the fix component icon,to *Fix* a part in the space.
5. Then the parts are assembled in the following or der ***Body, Screw spindle, Nuts, Cup,Tommy bar, Sunk screw, washer*** by adopting above methods
6. Select the update icon for assembly to be updated, all assembly de sign models are ensured that it is in constraints condition before the models are finished.



Assembly of Screw Jack

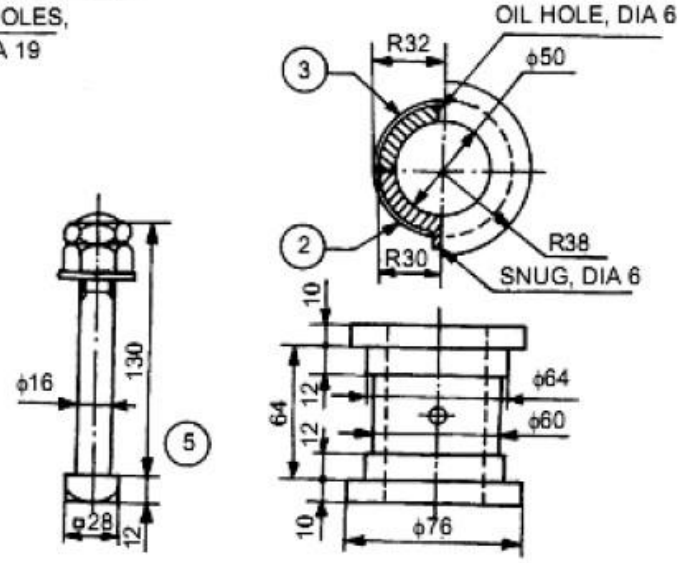
**RESULT:**

Thus the given 3D assembly model as per the drawing is modelled usingCATIA V5.18 software.




Parts list

| Sl. No. | Name | Matl. | Qty. |
|---|---|---|---|
| 1 | Base | CI | 1 |
| 2 | Bearing brass | Bronze | 1 |
| 3 | Bearing brass | Bronze | 1 |
| 4 | Cap | CI | 1 |
| 5 | Bolt with nuts | MS | 2 |




Details of Plummer Block

**Ex. No. 11** **ASSEMBLY OF PLUMMER BLOCK**

**DATE:**

**AIM:**

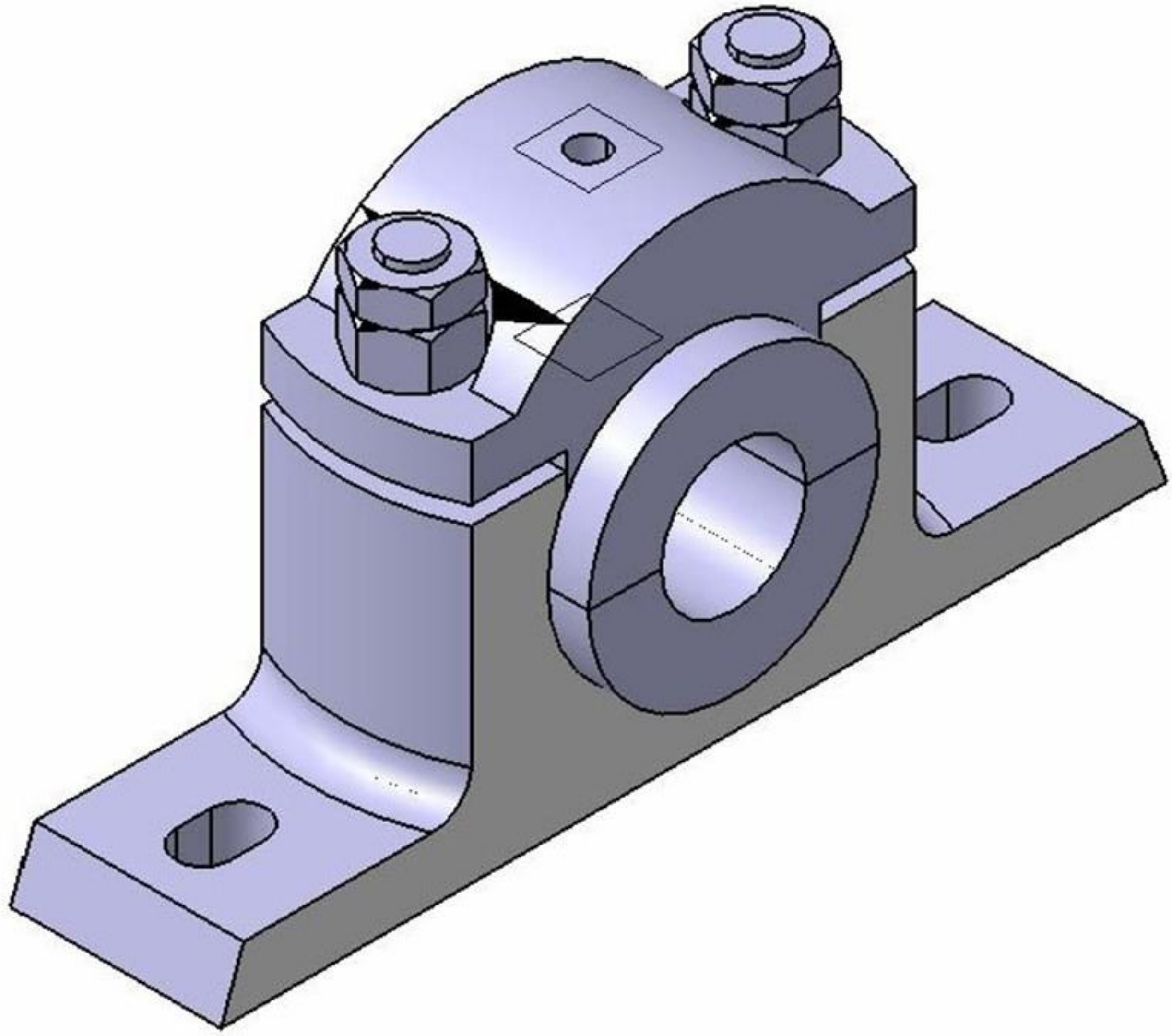
Preparation of 3D Assembly model using CATIA V5.18 software

**TOOLS USED:**

Existing Component icon, Smart move, Constraining etc.,

**PROCEDURE:**

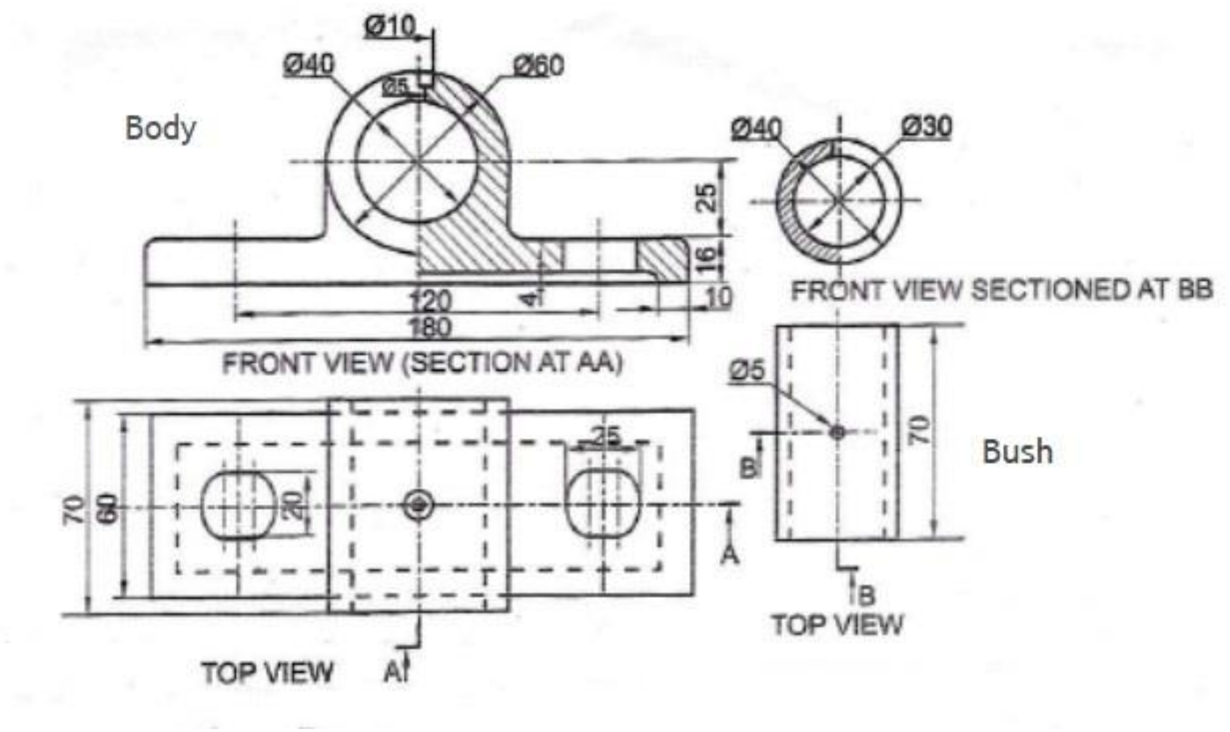
1. *Assembly Design* command is activated to launch the required workbench.
2. The commands for assembling parts are available in the toolbar on the right side of the application window.
3. Click the *Existing Component icon* in the Product Structure toolbar, then the File Selection dialog box is displayed were required the part model is selected.
4. To constrain a model for positioning parts correctly is carried using the various constraints tools like *Constraining and Manipulating* and Select the fix component icon,to *Fix* a part in the space.
5. Then the parts are assembled in the following or der ***Body, Brass, Cap, Bolt & Nuts, Lock Nuts*** by adopting above methods.
6. Select the update icon for assembly to be updated, all assembly de sign models are ensured that it is in constraints condition before the models are finished.



Assembly of Plummer Block

**RESULT:**

Thus the given 3D assembly model as per the drawing is modelled usingCATIA V5.18 software.




Details of Bushed Bearing

**EX. NO. 12**

# ASSEMBLY OF BUSHED BEARING

**DATE:**

## AIM:

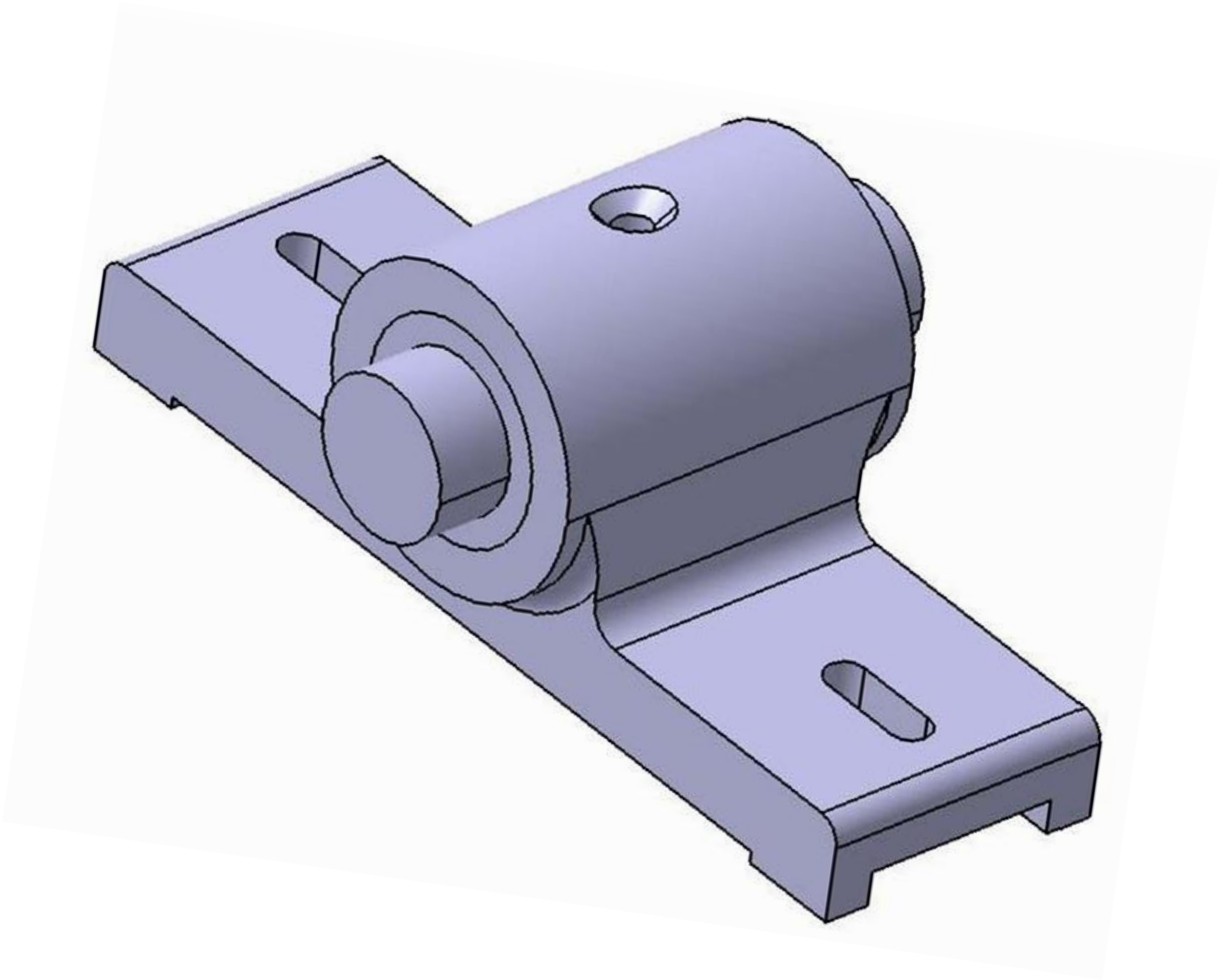
Preparation of 3D Assembly model using CATIA V5.18 software

## TOOLS USED:

Existing Component icon, Smart move, Constraining etc.,

## PROCEDURE:

1. *Assembly Design* command is activated to launch the required workbench.
2. The commands for assembling parts are available in the toolbar on the right side of the application window.
3. Click the *Existing Component icon* in the Product Structure toolbar, then the FileSelection dialog box is displayed were required the part model is selected.
4. To constrain a model for positioning parts correctly is carried using the various constraints tools like *Constraining and Manipulating* and Select the fix component icon, to Fix a part in the space.
5. Then the parts are assembled in the following order ***Body, Bush, Shaft*** by adopting above methods.
6. Select the update icon for assembly to be updated, all assembly de sign models are ensured that it is in constraints condition before the models are finished.



Assembly of Bushed Bearing

**RESULT:**

Thus the given 3D assembly model as per the drawing is modelled usingCATIA V5.18 software.

*****************************************************************

## INTRODUCTION TO CNC

### DEFINITION OF CNC

**“**A system in which the actions are controlled by direct insertion of numerical data at some point .The system must automatically interpret at least some portion of this data”

### WHY IT IS CALLED AS CNC?

Since the information required to actuate and control slides of the machine are coded numerically, this technology came to be known as Numerical Control.

### WHAT IS CNC?

**CNC is acronym for <u>Computer Numerical Control.</u>**

A dedicated computer is used to perform all the basic NC functions. The complete part programme to produce a component is input and stored in the computer memory and the information for each operation is fed to the machine tools. The program can be stored and used in future

### AXIS IN CNC MACHINES

**THE BASIS OF AXIS IDENTIFICATION IS THE 3-DIMENSIONAL CARTESIAN CO-ORDINATE SYSTEM AND THREE AXIS OF MOVEMENT ARE IDENTIFIED AS X,Y AND Z AXIS**

### Z AXIS.

Z- Axis The Z Axis of motion is always the axis of the main spindle of the machine.It doses not matters whether the spindle carries the work piece or the cutting tool . On vertical machining centers Z axis is vertical and on horizontal machining center and turning centers Z axis is horizontal

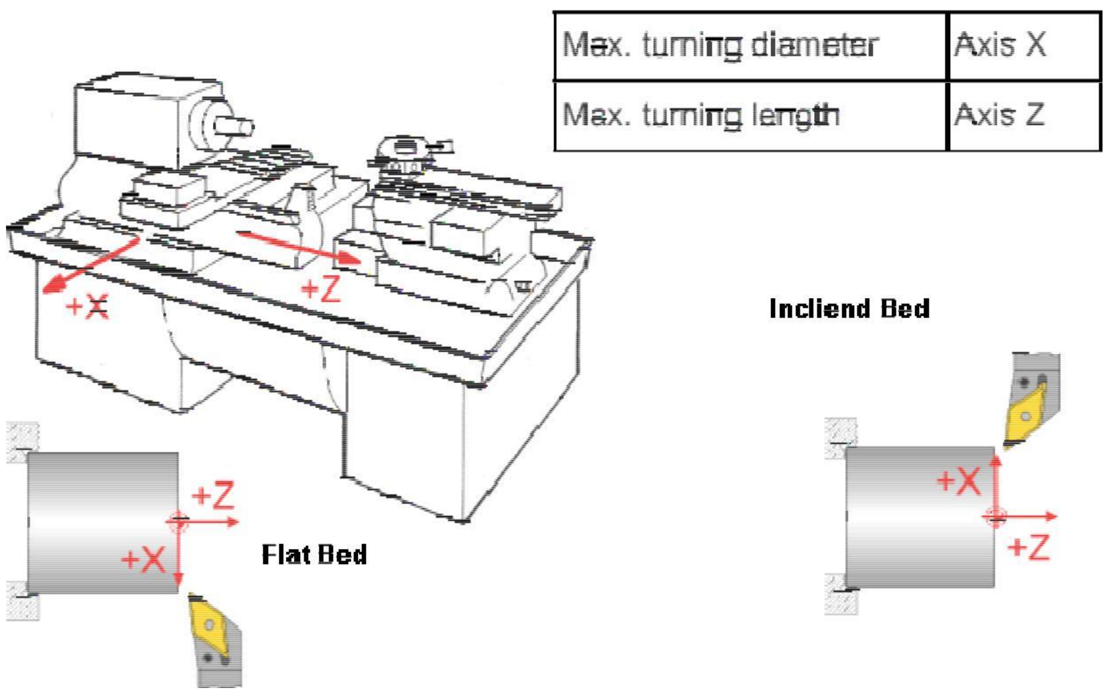
Positive Z Movement is away from the spindle

**X-Axis** The axis is always horizontal and is always parallel to the work holding surface. Positive X Axis movement is identified as being to the right, when looking from the spindle towards its supporting column.

**Y- Axis** The axis is always at right angle to both X-Axis and Z-Axis

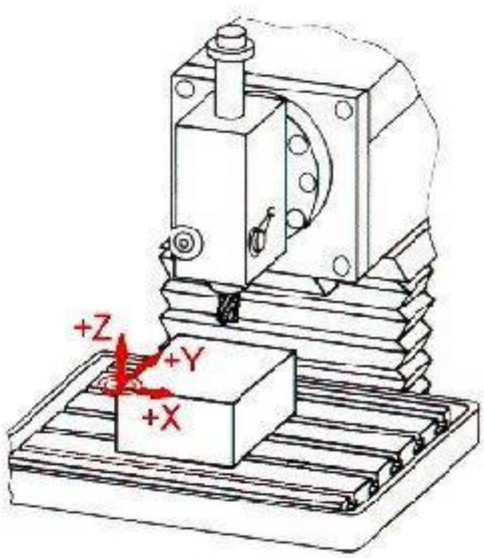
**Rotary axis** .The rotary motion about the X,Y and Z-Axis are identified by A,B,C respectively .Clockwise is designated as +VE. .Positive rotation is identified looking in x ,y and z direction respectively

**AXIS IN CNCLATHE**



## AXIS IN MILLING MACHINE

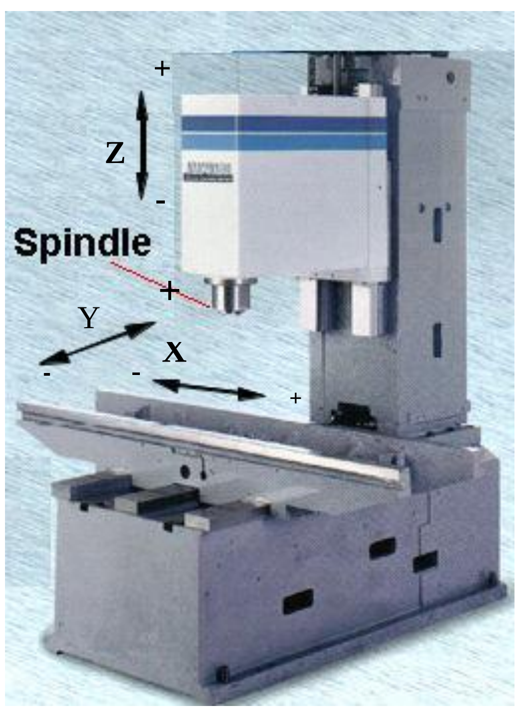
A milling machine has 3 axes of movement identified by X, Y & Z axes

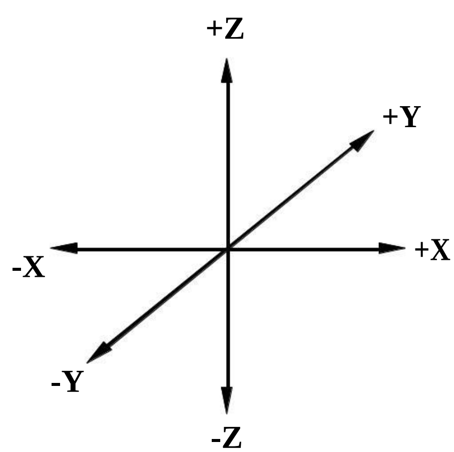



| | |
|---|---|
| Max. workpiece length | Axis X |
| Max. workpiece width | Axis Y |
| Max. workpiece height | Axise Z |

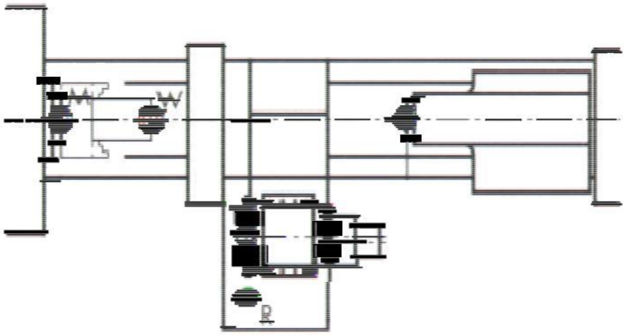
The maximum work piece dimensions correspond to the possible traversing path of the tool in the particular axis.

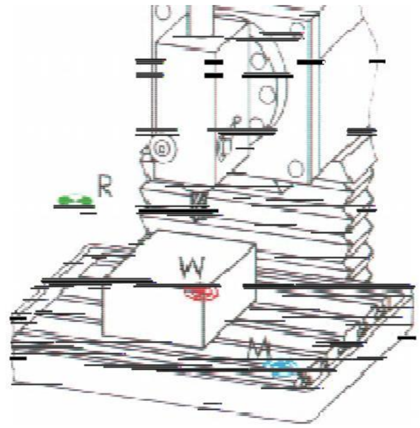
## AXIS IN MILLING MACHINE



## ZERO POINTS REFERENCE POINT




**Machine zero M**

The manufacturer defines the machine zero M and this cannot be changed.
It is located at the origin of the machine coordinate system.

**Workpiece zero W**

The workpiece zero W (also known as program zero) is at the origin of the workpiece coordinate system. It can be freely selected, and should be located at the point where most of the dimensions originate in the drawing.

**Reference point R**

The reference point R is approached to set the measuring system to zero, as the machine zero point can generally not be approached. The control starts to count in its incremental position measuring system.

The reference point R serves for calibrating and for controlling of measuring systems of the slides and tool traverses. The position of the reference point is accurately predetermined in every traverse axis by the trip dogs and limit switches. Therefore, the reference point coordinates always have the same, precisely known numerical value in relation to the machine zero point.

After initiating the control system, the reference point must always be approached from all axes to calibrate the traverse measuring system.

**DIMENSION SYSTEM**

Dimensional information in a work piece drawing can be stated in two ways Absolute Dimension System and Incremental Dimension System.

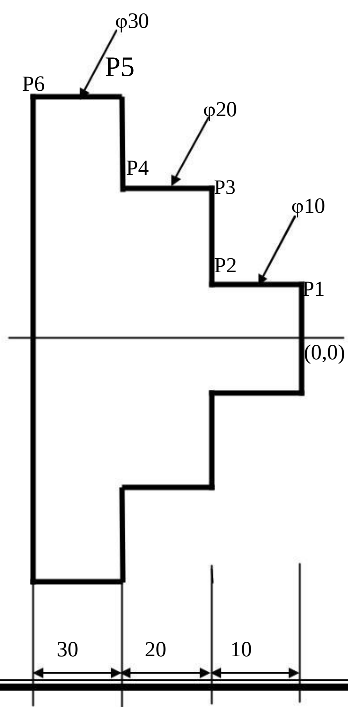
**Absolute Dimension System**

Data in absolute dimension system always refer to a fixed reference point. This point has the function of a coordinate zero point. The dimension lines run parallel to the coordinate axes and always start at the reference point. Absolute dimensions are also called as 'Reference dimensions'

**Incremental Dimension System**

When using Incremental Dimension system, every measurement refers to a previously dimensioned position Incremental dimensions are distance between adjacent points. These distances are converted into incremental coordinates by accepting the last dimension point as the coordinate origin for the new point. This may be compared to a small coordinate system, i.e., shifted consequently form point to point (P1..P2..through P9). Incremental dimensions are also frequently called 'Relative dimensions' or 'Chain dimensions'.

**DIMENSION SYSTEM**




| ABSOLUTE DIMENSIONING | | | INCREMENIAL DIMENSIONING | | |
|---|---|---|---|---|---|
| **POINTS** | **X** | **Z** | **POINTS** | **U** | **W** |
| P1 | 10 | 0 | P1 | 10 | 0 |
| P2 | 10 | -10 | P2 | 0 | -10 |
| P3 | 20 | -10 | P3 | 10 | 0 |
| P4 | 20 | -25 | P4 | 0 | -15 |
| P5 | 30 | -25 | P5 | 10 | 0 |
| P6 | 30 | -35 | P6 | 0 | -10 |

# STUDY OF ISO CODES

# CNC TURNING

**AIM**: To study various CNC Turning codes and addresses

**G Codes**

G00: Point to point positioning (Rapid traverse)

G01: Linear interpolation

G02: Circular interpolation clockwise

G03: Circular interpolation counter clockwise

G04: Dwell, Exact stop

G17: X Y Plane selection

G18: Z X plane selection

G19: Y Z Plane selection

G20: Input in inch

G21: Input in metric (mm)

G28: Return to reference point

G32: Thread Cutting

G40: Cutter compensation cancel

G41: Cutter Compensation left

G42: Cutter Compensation right

G49: Tool length compensation cancel

G50: Work co-ordinate Change / Max Spindle speed setting

G70: Finishing cycle

G71: Stock removal in turning

G72: Stock removal in facing

G73: Pattern repeating

G74: Peck drilling in z axis

G75: Grooving in x axis

G76: Thread Cutting Cycle

G80: Canned cycle cancel

G90: Cutting cycle A

G92: Thread cutting cycle

G94: Cutting cycle B

G96: Constant surface speed control

G97: Constant surface speed control cancel

G98: Feed per minute

G99: Feed per revolution

**M Codes**

M00: Program stop

M01: Optional (planned) stop

M02: End of program

M03: Spindle forward clockwise

M04: Spindle forward counter clockwise

M05: Spindle stop

M06: Tool change

M08: Coolant ON

M09: Coolant OFF

M10: Chuck open

M11: Chuck close

M62: Output 1 ON

M63: Output 2 ON

M64: Output 1 OFF

M65: Output 2 OFF

M66: Wait input 1 ON

M67: Wait input 2 ON

M76: Wait input 1 OFF

M77: Wait input 2 OFF

M98: Sub program Call

M99: Sub program Exit

## Common Lathe G-Codes

*G00 **Rapid linear move**. [X, Z, U, W] Moves the machine at the fastest rate possible to the X, Z location specified or incrementally U (X) W(Z) distance.

G01 **Linear feed move** [X, Z, U, W, F] Moves the machine at the specified feed rate (F) to the X, Z location specified or incrementally U (X) W(Z) distance.

G02 **Circular interpolation; CW.** [X, Z, U, W, F, R (or I, K)] Moves the machine, in a clockwise circular path to the X, Z, location (or incrementally by U,W) with radius R, or with a center point defined relative to the start point in the X,& Z axis by I, & K respectively.

G03 **Circular interpolation; CCW.** [X, Z, U, W, F, R (or I, K)] Same as G02, but opposite direction of movement.

G28 **Machine home**. Causes the machine to return to it's X0, Z0 position at a rapid rate.

*G40 **Tool nose compensation, *CANCEL.*** [X, Z, U, W, I, K, F] Cancels the G41 or G42 Cutter compensation listed below. Causes a feed move to X and/or Z at feed rate F (or at modal feed F, if not specified). The distance of the move must be greater than the radius of the tool.

G41 **Tool nose compensation, *LEFT***. [X, Z, U, W, D, F] Looking from the spindle toward the part, G41 offsets the position of the tool *left* of the programmed tool path by the value stored in the offsets register position called by the D word. Causes a feed move from the current position to the *compensated* position specified by X, Y, at feed F (or at modal feed F if not specified). The distance of this move must be greater than the radius of the tool.

G42 **Tool nose compensation, RIGHT**. Same as G41 above except that the tool is compensated to the *right* of the programmed tool path.

**NOTE:** Cutter compensation may be accomplished at the machine, through the tool path generated by the CAM program, or both.

G50 **Spindle speed clamp.** Specifies the maximum RPM the spindle can run during constant surface speed operation.

G70 **Finishing Cycle**

G71 **Multiple Turning Cycle.** It is used when the major direction of cut is along the Z axis. It causes the profile to be roughed out by turning.

G72 **Multiple Facing Cycle,** used when the major direction of cut is along the X axis. This cycle causes the profile to be roughed out by facing.

G73 **Pattern Repeating Cycle,** provides for roughing out of a form by repeating the desired tool path a set number of times, the tool path being incremented into the work piece until the full form is completed.

G75 **Grooving Cycle**

G92 **Single Thread cycle.** This is a Box type cycle producing a single pass of the threading tool.

G76 **Multiple Threading Cycle.** This is a box type cycle that is repeated a given number of times. After the first pass subsequent passes cut with one edge of the threading tool only to reduce the load at the tool tip.

G74 **End Face Peck Drilling.** This cycle is designed for deep hole drilling, the drill entering the work piece by a predetermined amount then by backing off by another set amount to provide breaking and allowing swarf to clear the drill flutes.

G90 **Single Turning Cycle**, used to produce either a parallel or tapered tool path. It performs four distinct moves with one line of information and it is equivalent to rapid to X position, feed to Z position, feed to start X position, rapid to start Z position

G94 **Facing Cycle** .This cycle is used for stock removal either parallel or at an angle to work piece face. It is equivalent of rapid to Z position, feed to X Position, and feed to start Z position and rapid to start X position. If R value is specified tapering will be performed.

**Common Lathe M-Codes**

M00 **Program stop**. Stops the machine, requiring the operator to restart the program to continue.

M01 **Optional stop**. Stops the machine as above only when the optional stop button has been pressed prior to this command in the program.

M03 **Spindle start**, **CW.** [S] Starts the spindle in a clockwise direction, at the RPM specified by the S word accompanying the code.

M04 **Spindle start, CCW**. [S] Similar to above, but rotation is reversed**.**

M05 **Spindle stop.**

M06 **Tool Change.** The M06 in conjunction with “T” word, is used to call up the required tool on an automatic indexing turret machine, and to activate its tool offsets.

M08 **Coolant on**.

M09 **Coolant off.**

M10 **Chuck Open**

M11 **Chuck Close**

M13 **Spindle Forward, Coolant ON**

M14 **Spindle Reverse, Coolant ON**

M19 **Spindle orientation.**

M25 **Quill (**Tailstock**) Extend**

M26 **Quill (**Tailstock**) Retract**

M30 **Program stops and rewind.** To program start

M38 **Door Open**

M97 **Local subroutine call.** [N] Causes the program to skip to a subprogram contained inside the current program at line number N.

M98 **Subprogram call** [P, L] the program will call another program number, specified by the P word, and execute it L times.

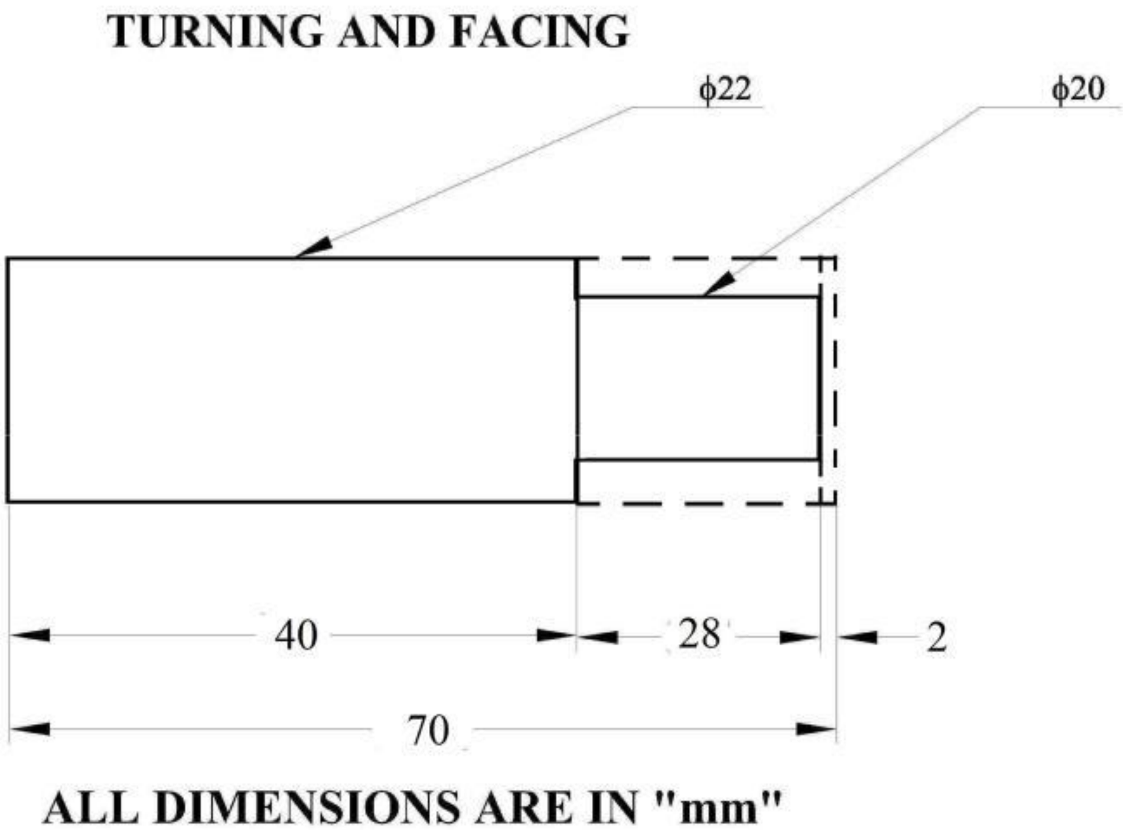
M99 **Subprogram return**. Contained at the end of the subprogram (or subroutine) will return the control to the main program.

Note: only one M Code can be used per line and will be executed after all other operations specified in the line.

**RESULT:**

Thus various CNC Turning codes and addresses were studied.

Write a manual part program for simple **Turning and Facing** operation for component shown in **DWG.NO.T01**




**DWG.NO.T01**

| PLANNING AND OPERATIONS SHEET | | | | | | | |
|---|---|---|---|---|---|---|---|
| **BILLET SIZE:** 22x60 | | | | **MATERIAL:** Aluminium | | | |
| **PROGRAM NO:**1001 | | | | **DWG.NO:1** | | | |
| **Sl.No** | **Operation** | **Tool type** | **Tool dia., mm** | **Tool Station No.** | **Tool offset No.** | **Spindle speed, rpm** | **Feed, mm/min** |
| 1 | Simple Facing | SDJCR121H11 | DCMT11T304 | 1 | 1 | 1200 | 45 |

## PLAIN TURNING AND FACING

**EX.NO: L01**

**DATE:**

**AIM:**

To write a manual part program for simple turning and facing operation for component shown in DWG.NO.T01

**MATERIAL REQUIRED:**

Material : Aluminium
Size : Length 70mm, Diameter 22mm

**PROGRAM:**

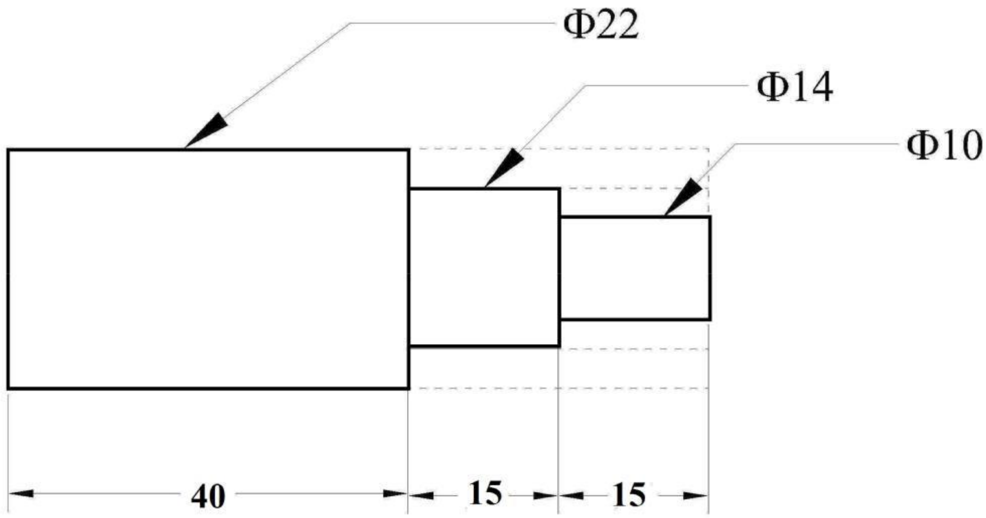
```
O0001
G21 G98
G28 U0 W0
M06 T0101
M03 S1200
G00 X23 Z3
G00 Z-0.5
G01 X-1 F45
G00 Z3
G00 X23
G00 Z-1
G01 X-1 F45
G00 Z3
G00 X23
G00 Z-1.5
G01 X-1 F45
G00 Z3
G00 X23
G00 Z-2
G01 X-1 F45
G00 Z3
G00 X21.5
G01 Z-30 F45
G00 X23
G00 Z3
G00 X21
G01 Z-30 F45
G00 X23
G00 Z3
G00 X20.5
G01 Z-30 F45
G00 X23
G00 Z3
G00 X20
G01 Z-30 F45
G00 X23
G28 U0 W0
M05
M30
```

**RESULT:**

Thus the manual part program was written to the given dimensions and executed in CNC Lathe.

Write a manual part program for **Step Turning** operation with G90 cycle for component shown in **DWG.NO.T02**

**STEP TURNING**




**ALL DIMENSIONS ARE IN "mm"**

**DWG.NO.T02**

| **PLANNING AND OPERATIONS SHEET** | | | | | | | |
|---|---|---|---|---|---|---|---|
| **BILLET SIZE:** 22x60 | | | | **MATERIAL:** Aluminium | | | |
| **PROGRAM NO:**1002 | | | | **DWG.NO:2** | | | |
| **Sl.No** | **Operation** | **Tool type** | **Tool dia., mm** | **Tool Station No.** | **Tool offset No.** | **Spindle speed, rpm** | **Feed, mm/min** |
| 1 | Step Turning | SDJCR121H11 | DCMT11T304 | 1 | 1 | 1200 | 30 |

**Cutting Cycle**

G90 X(U) Z(W) F(*f)

X Diameter to which the movement is being made.

U The incremental distance from the current tool position to the required final Diameter Z The Z Axis Co-ordinate to which the movement is being made.

W The incremental distance from the current tool position to the required Z axis Position

fFeed rate

## STEP TURNING

**EX.NO: L02**

**DATE:**

**AIM:**

To write a manual part program for step turning operation with G90 cycle for component shown in DWG.NO.T02

**MATERIAL REQUIRED:**

Material : Aluminium
Size : Length 70mm, Diameter 22mm

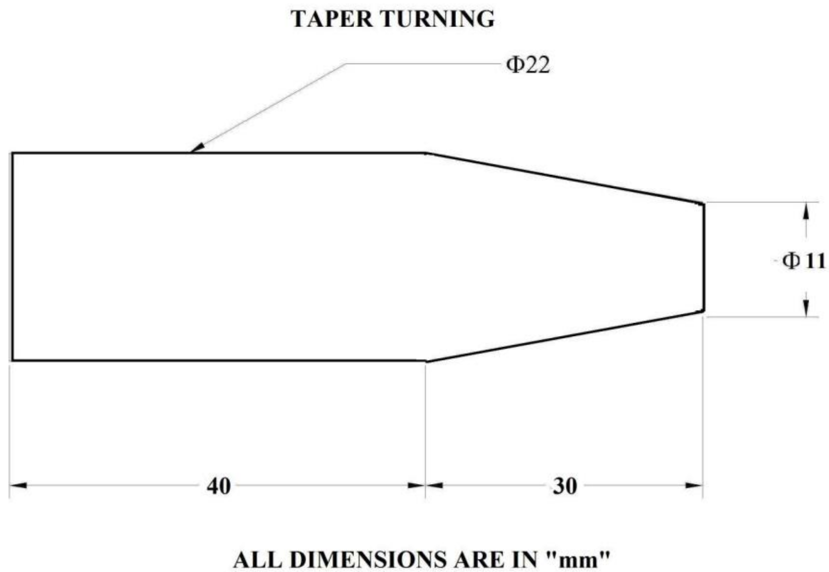
**PROGRAM:**

```
O0001
G21 G98
G28 U0 W0
M06 T0101
M03 S1200
G00 X22 Z1
G90 X22 Z-30 F30
X21
X20
X19
X18
X17
X16
X15
X14
G00 X14 Z1
G90 X14 Z-15 F30
X13
X12
X11
X10
G28 U0 W0
M05
M30
```

**RESULT:**

Thus the manual part program was written to the given dimensions and executed in CNC Lathe.

Write a manual part program for **Taper Turning** operation for component shown in **DWG.NO.T03**




**DWG.NO.T03**

| PLANNING AND OPERATIONS SHEET | | | | | | | |
|---|---|---|---|---|---|---|---|
| **BILLET SIZE:** 22x60 | | | | **MATERIAL:** Aluminium | | | |
| **PROGRAM NO:**1003 | | | | **DWG.NO:3** | | | |
| **Sl.No** | **Operation** | **Tool type** | **Tool dia., mm** | **Tool Station No.** | **Tool offset No.** | **Spindle speed, rpm** | **Feed, mm/min** |
| 1 | Taper Turning | SDJCR121H11 | DCMT11T302 | 3 | 3 | 1000 | 30 |

**Cutting Cycle**
G90 X(U) Z(W) R F
X Diameter to which the movement is being made.
U The incremental distance from the current tool position to the required final Diameter Z The Z Axis Co-ordinate to which the movement is being made.
W The incremental distance from the current tool position to the required Z axis Position R The difference in incremental of the cut start radius value and the cut finish radius value.
fFeed rate

# TAPER TURNING

**EX.NO:L03**

**DATE:**

**AIM:**

To write a manual part program for taper turning operation for component shown in DWG.NO.T03

**MATERIAL REQUIRED:**

Material : Aluminium
Size : Length 70mm, Diameter 22mm

**PROGRAM:**

```
O0002
G21 G98
G28 U0 W0
M06 T0101
M03 S1200
G00 X22 Z1
G90 X22 Z-30 R0 F30
X22 R-0.5
X22 R-1
X22 R-1.5
X22 R-2
X22 R-2.5
X22 R-3
X22 R-3.5
X22 R-4
X22 R-4.5
X22 R-5
X22 R-5.5
G28 U0 W0
M05
M30
```

**RESULT:**

Thus the manual part program was written to the given dimensions and executed in CNC lathe.