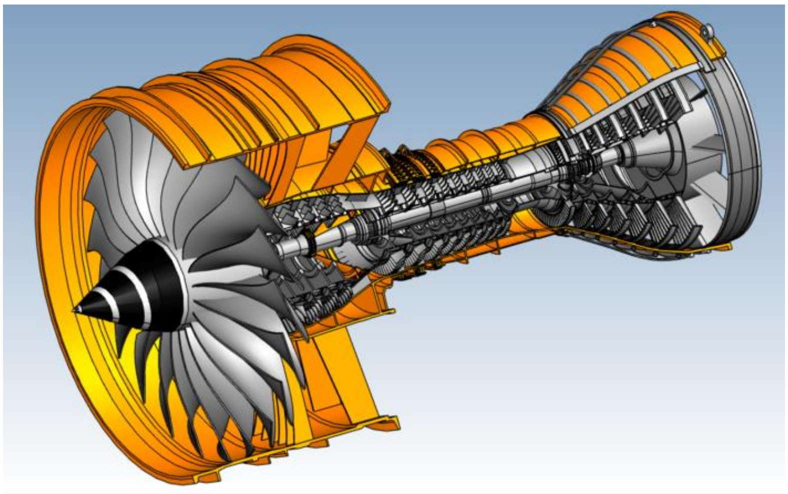
VINAYAKA MISSION'S RESEARCH FOUNDATION
(Deemed to be University)
AARUPADAI VEEDU INSTITUTE OF TECHNOLOGY

DEPARTMENT OF MECHANICAL ENGINEERING

17MECC82
MACHINE DRAWING LABORATORY MANUAL



NAME OF THE STUDENT :…………………………………….

REG.NO :…………………………………….

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HOD/MECH

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.

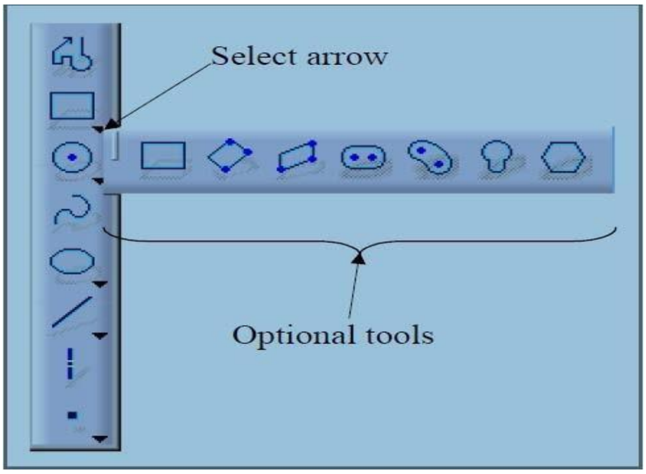
Tool Bars

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.

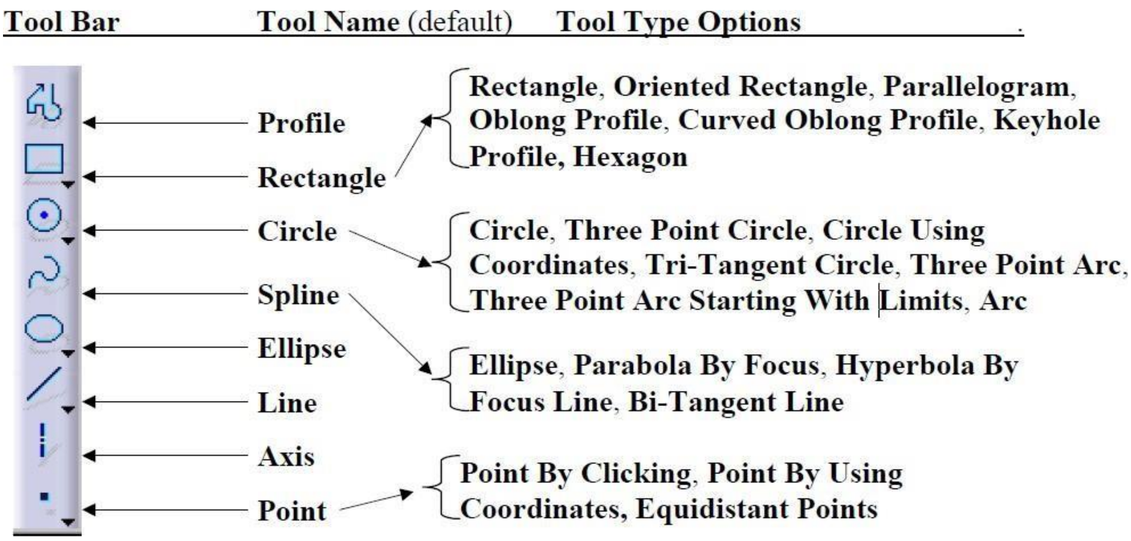
Sketcher Work Bench Tool Bars

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

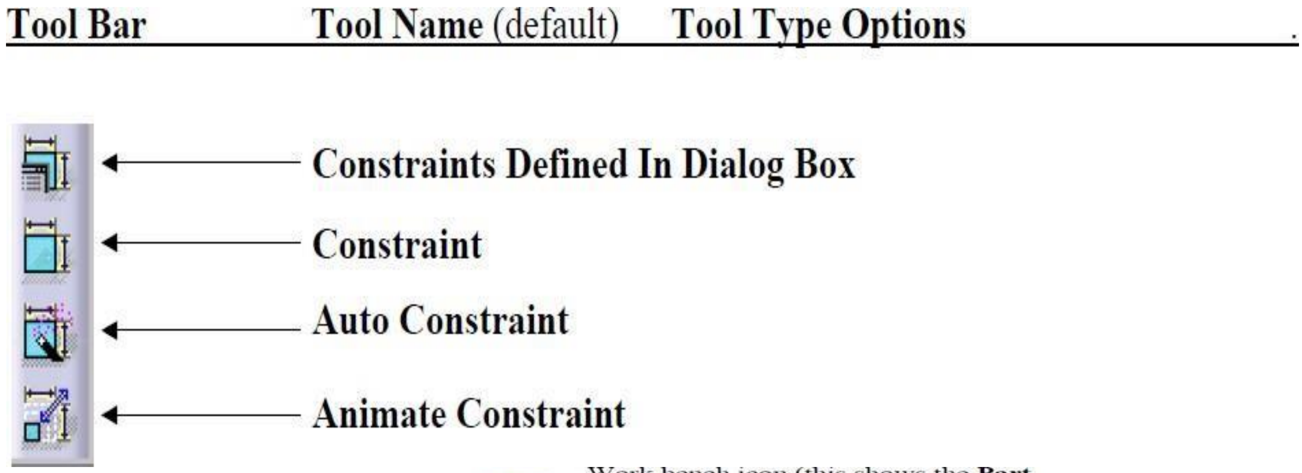


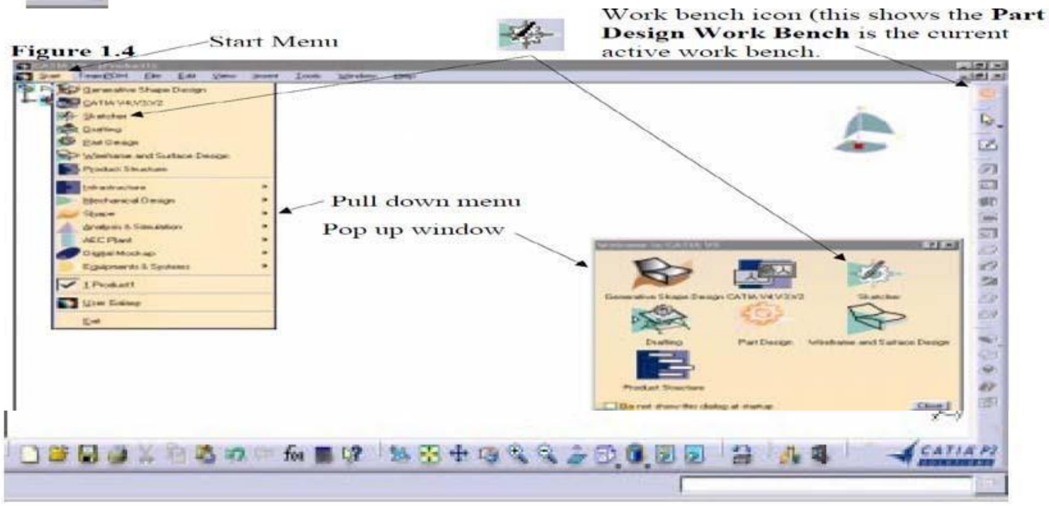
THE PROFILE TOOL BAR



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

THE CONSTRAINTS TOOL BAR

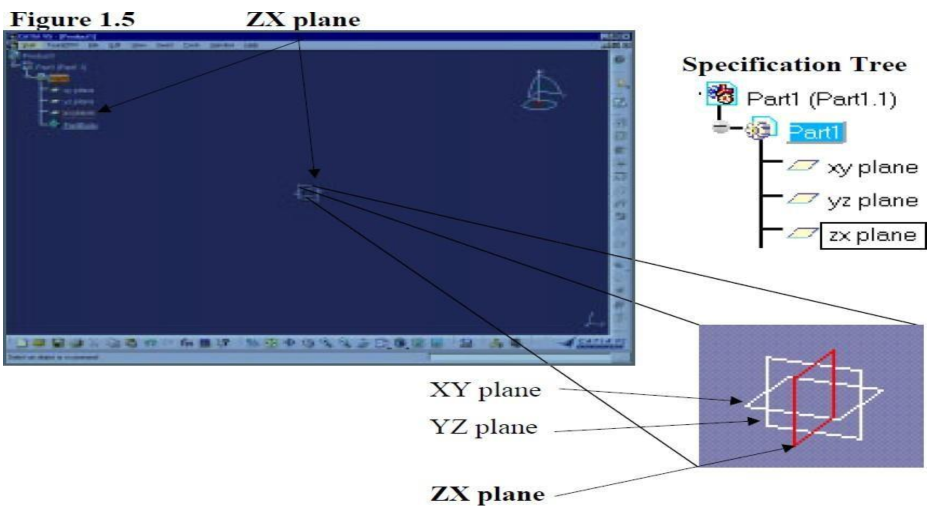




Specify a Working Plane

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.

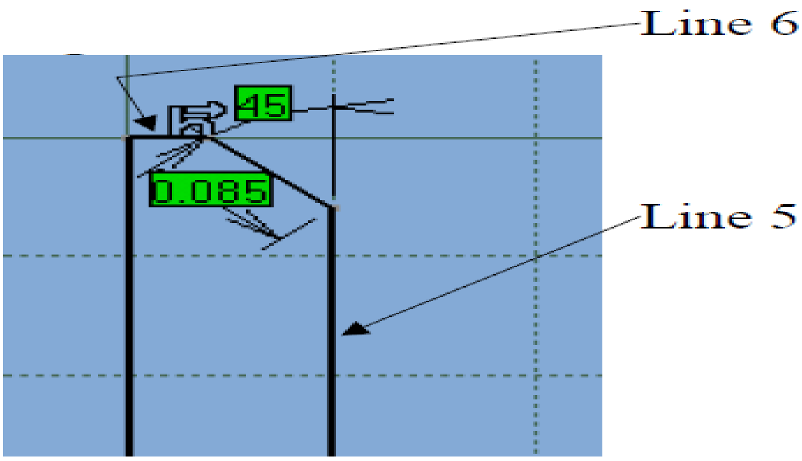
ZX plane



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.

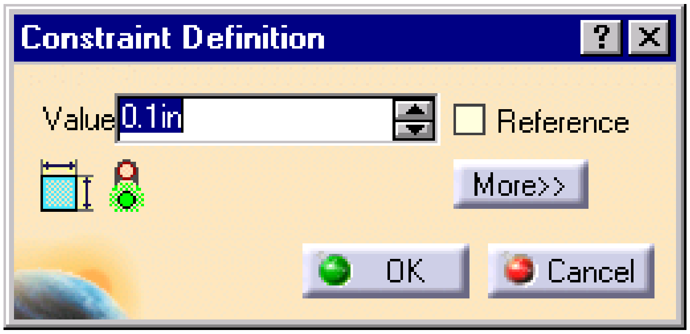
Select the line and/or Sketcher element to be constrained.

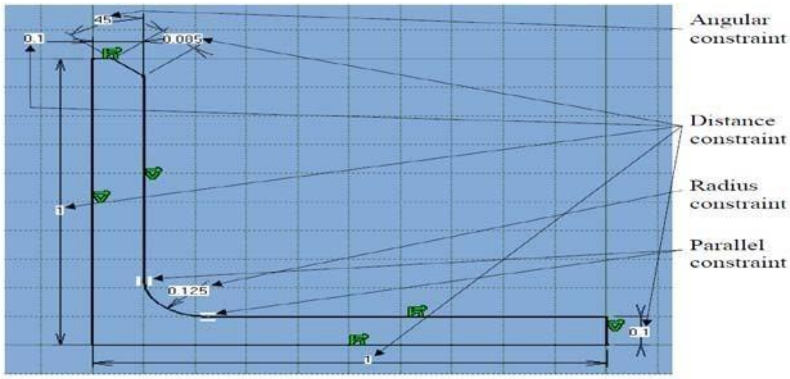
The Sketcher element will turn green (constraint symbol) along with the appropriate dimension and box with the value in it. To re-locate the constraint value, select the value box and drag the mouse to the desired location.

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

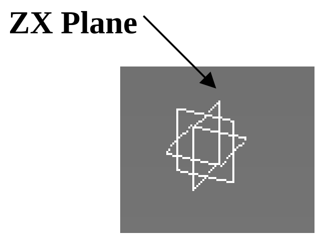


Figure 1.1

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

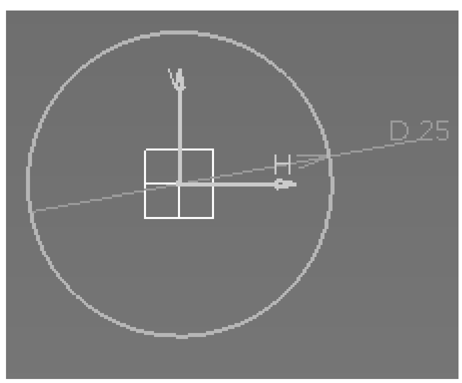


Figure 1.2

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.**

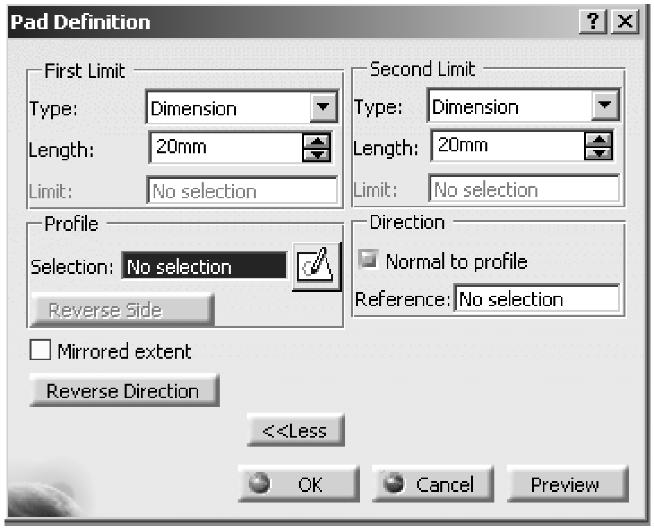


Figure 1.3

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

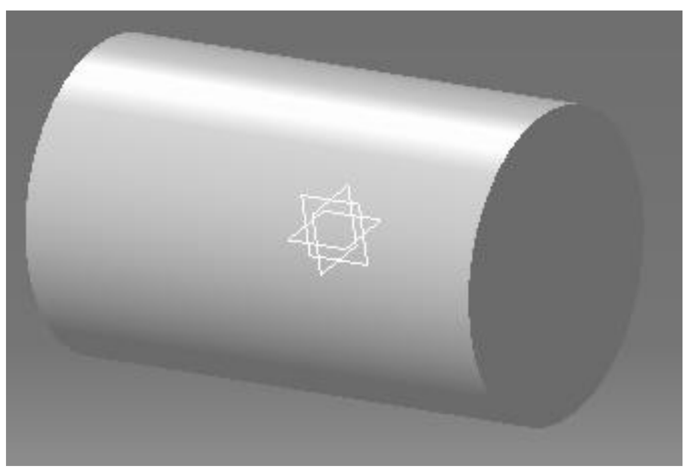


Figure 1.4

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).

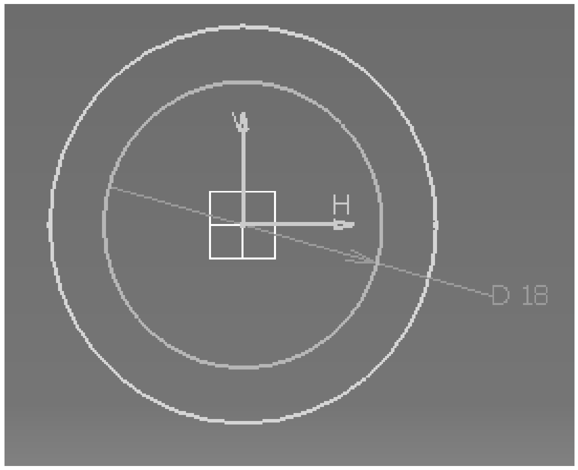


Figure 1.5

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.

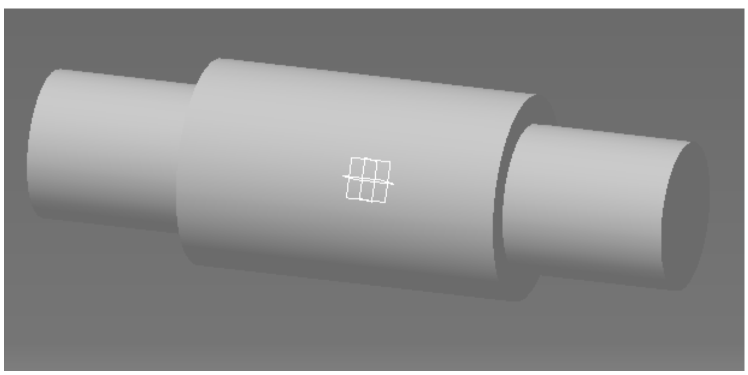


Figure 1.6

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

YZ Plane

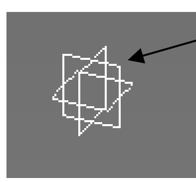


Figure 1.7

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

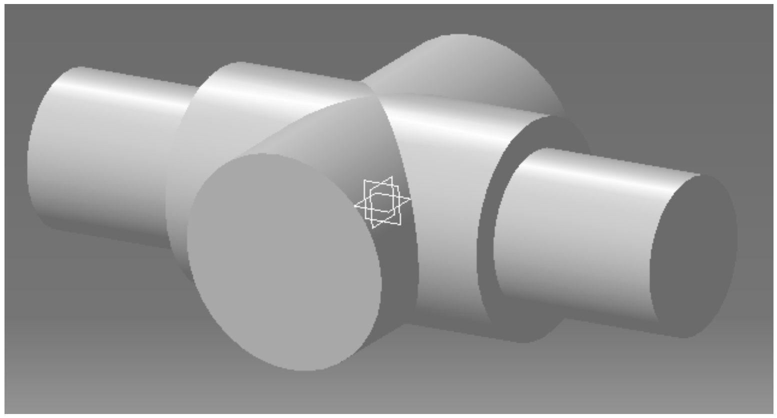


Figure 1.8

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.

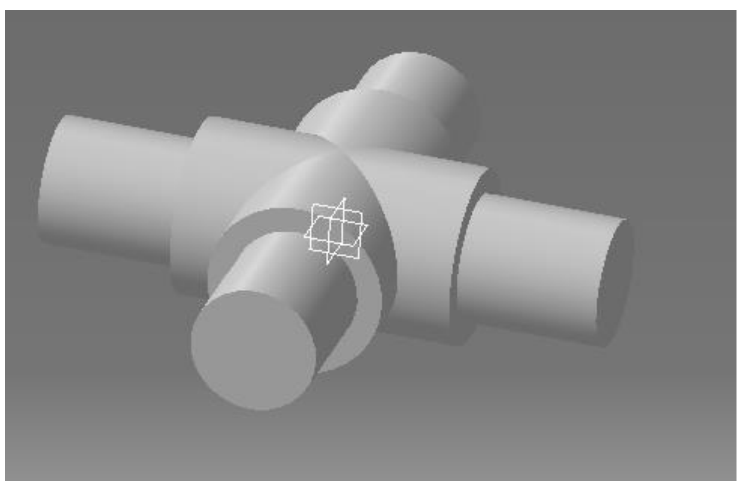


Figure 1.9

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

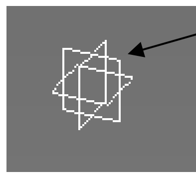


Figure 2.1

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

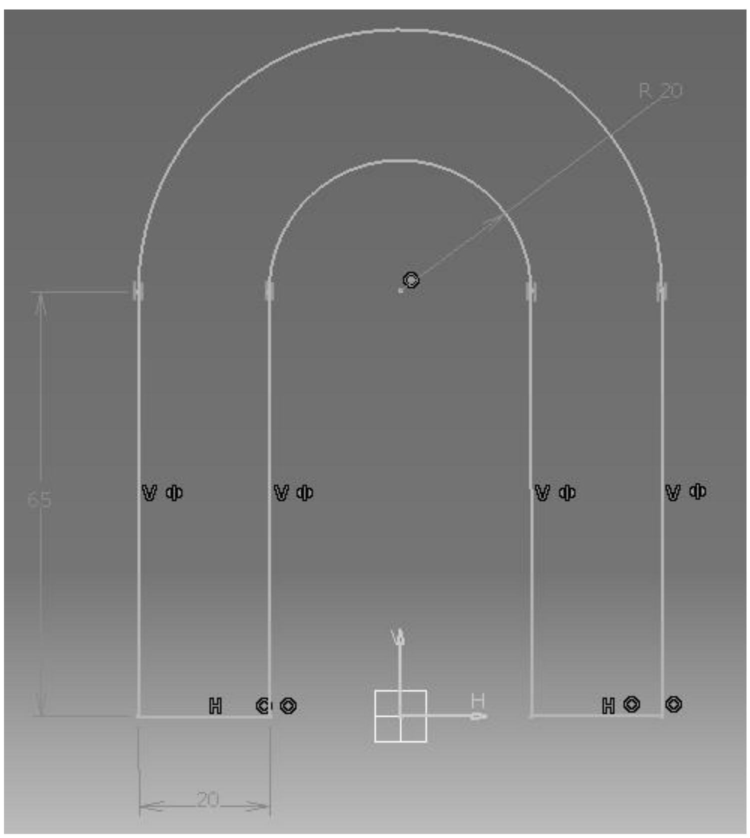


Figure 2.2

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

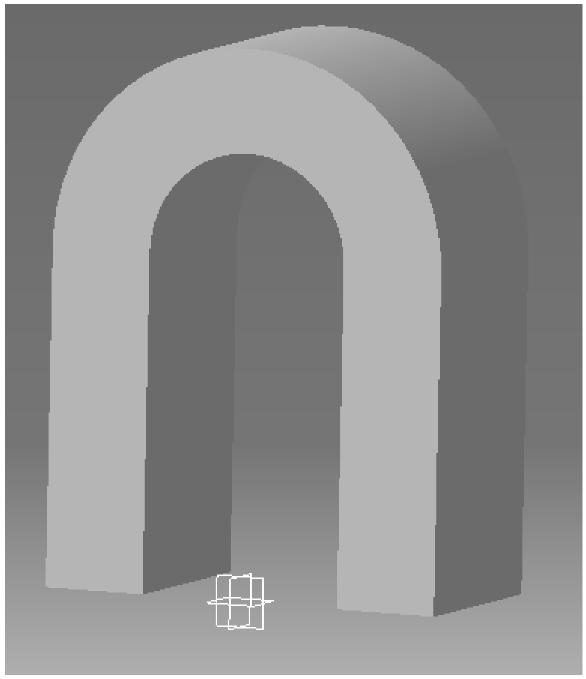


Figure 2.3

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.

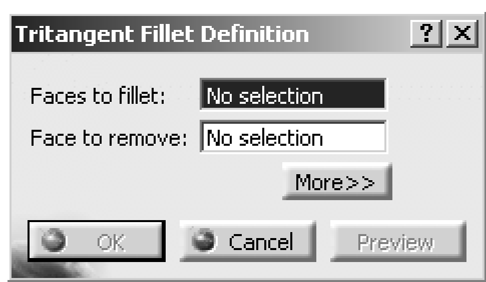


Figure 2.4

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

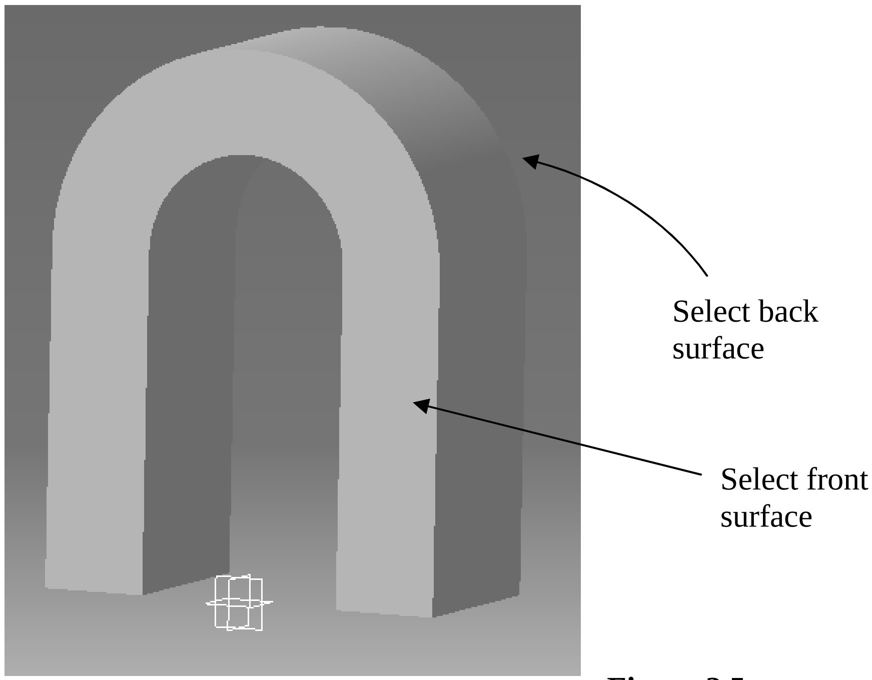
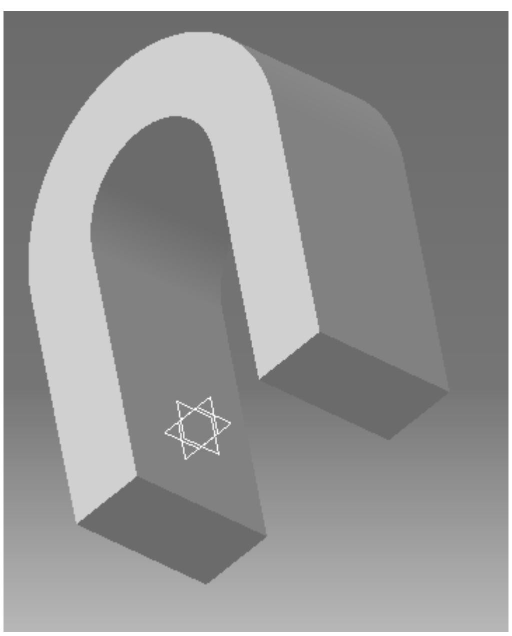


Figure 2.5

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



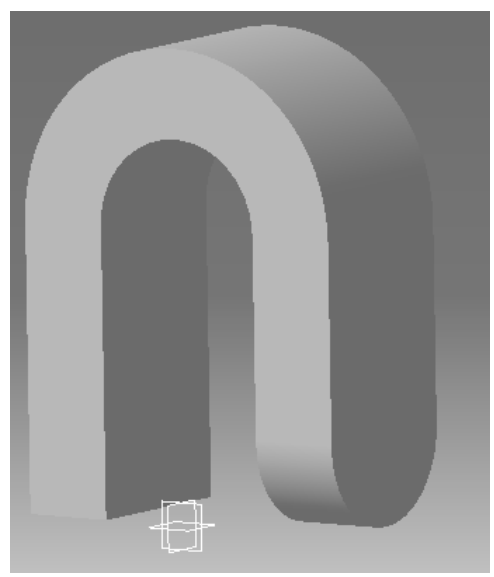


Figure 2.6

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

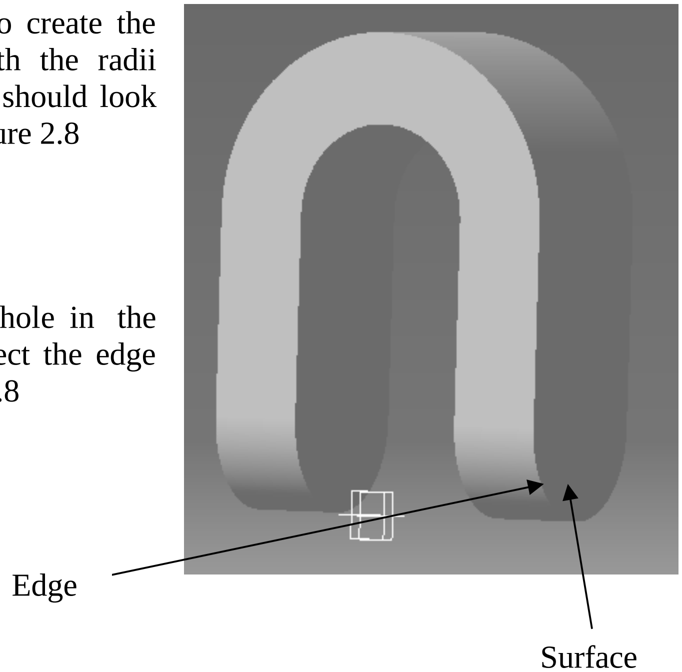


Figure 2.8

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.

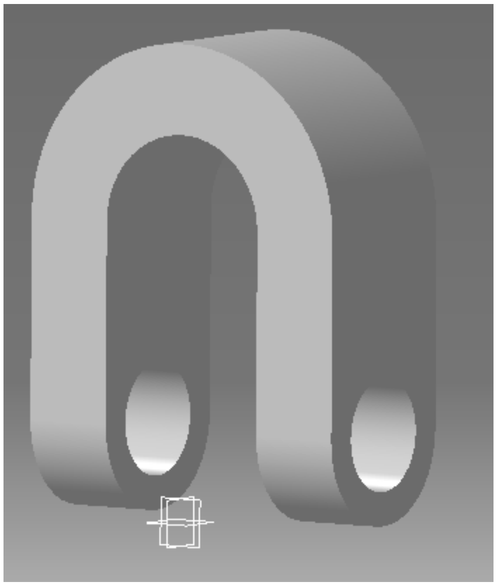
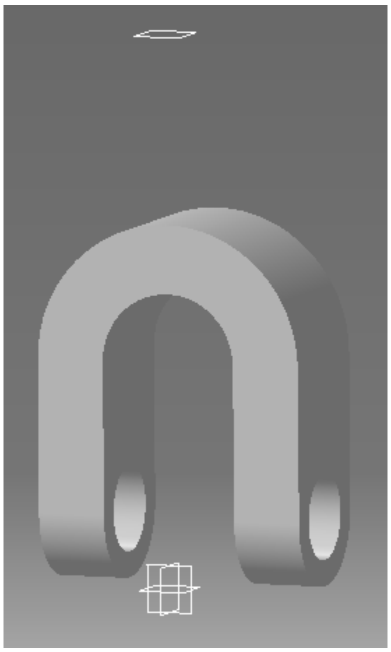


Figure 2.9

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**

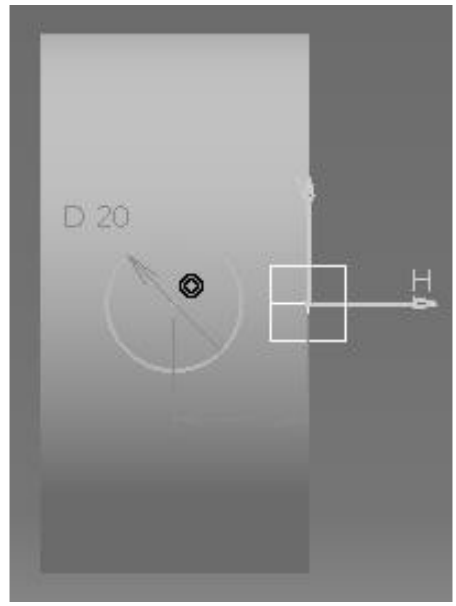
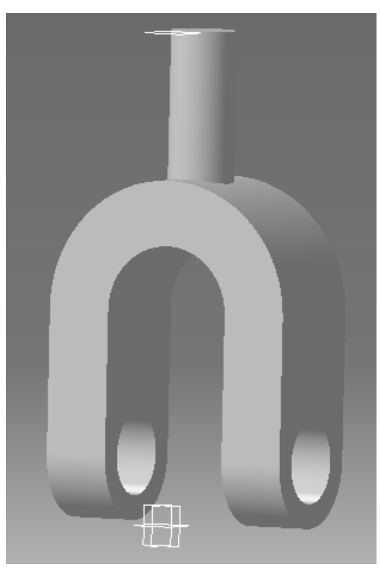


Figure 2.11

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."**

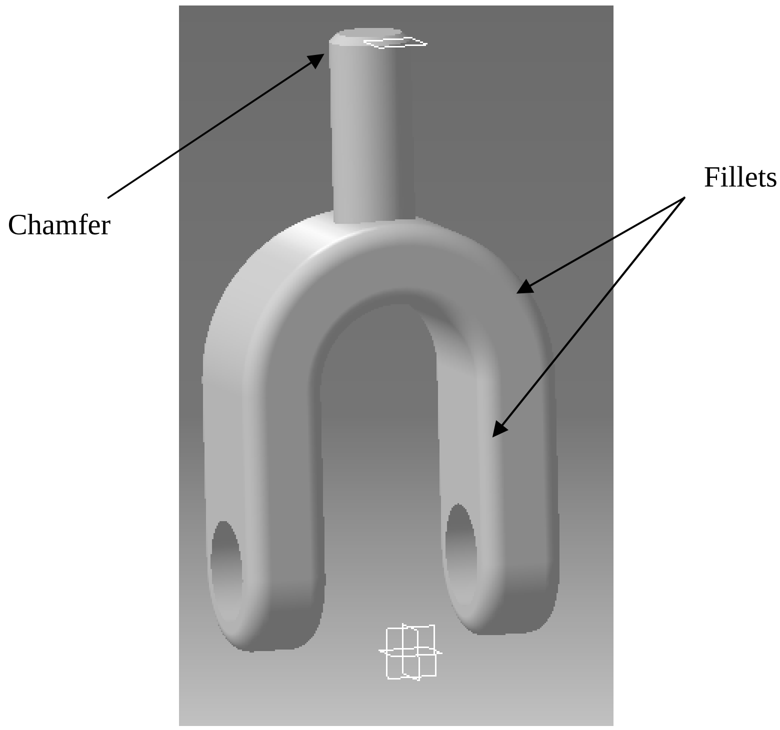


Figure 2.13

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

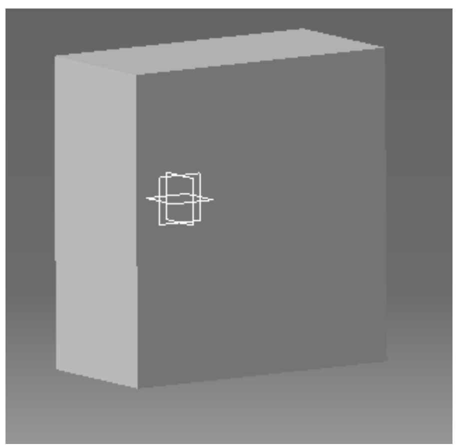


Figure 3.1

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.

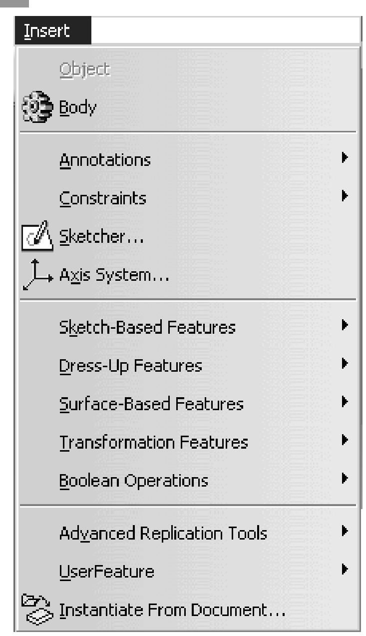
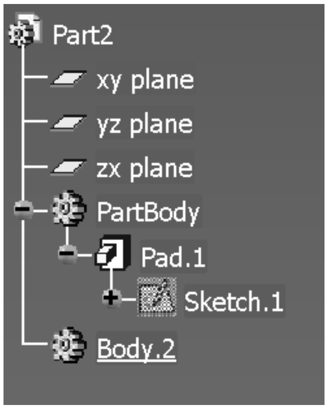


Figure 3.2

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



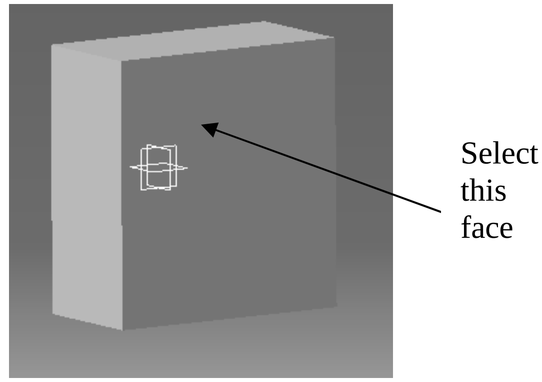


Figure 3.3 **Figure 3.4**

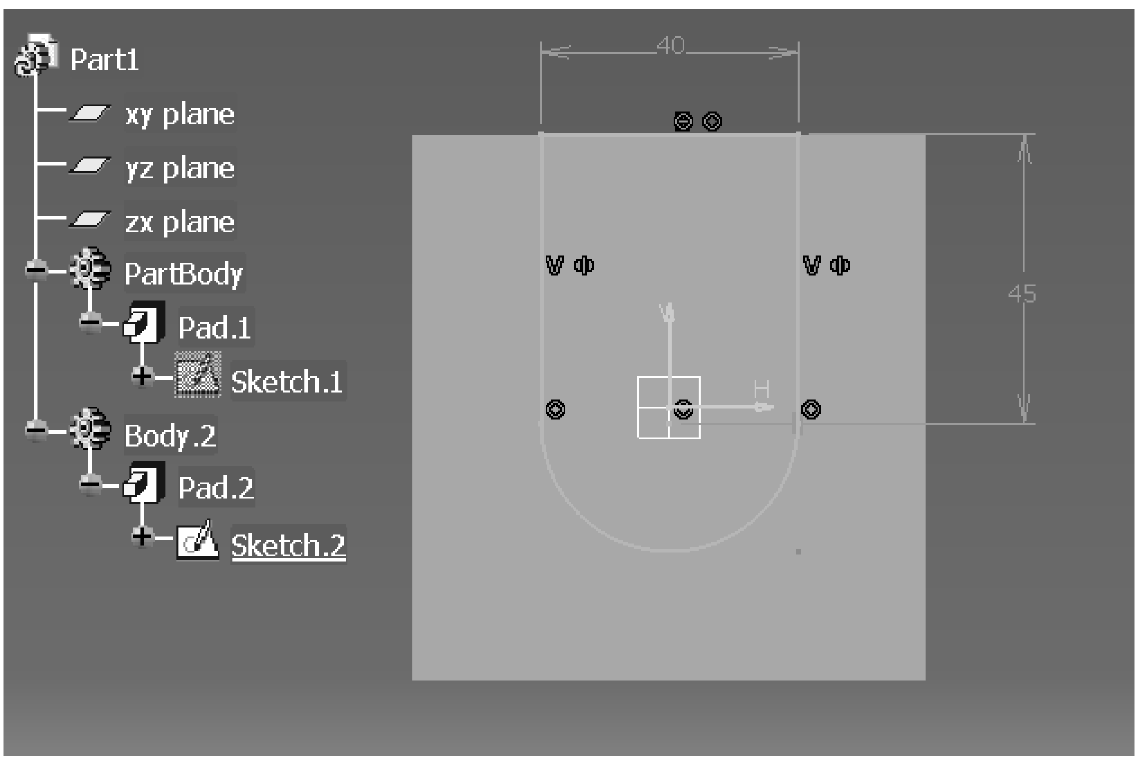


Figure 3.5

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.

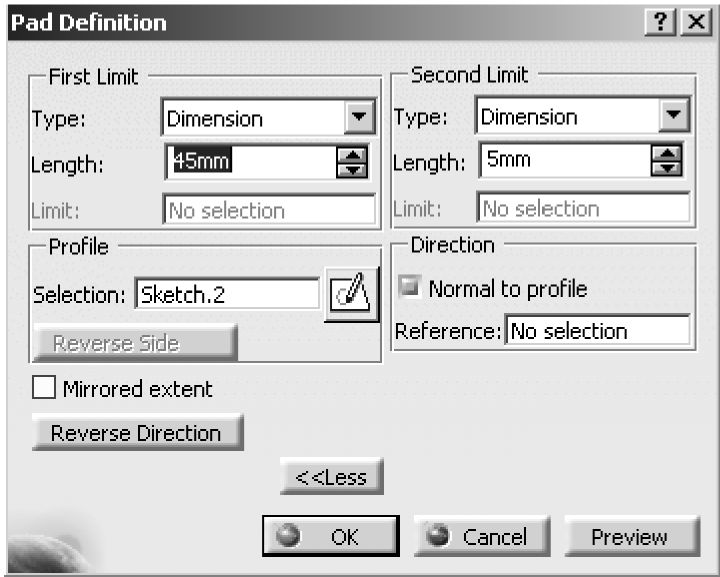


Figure 3.6

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.

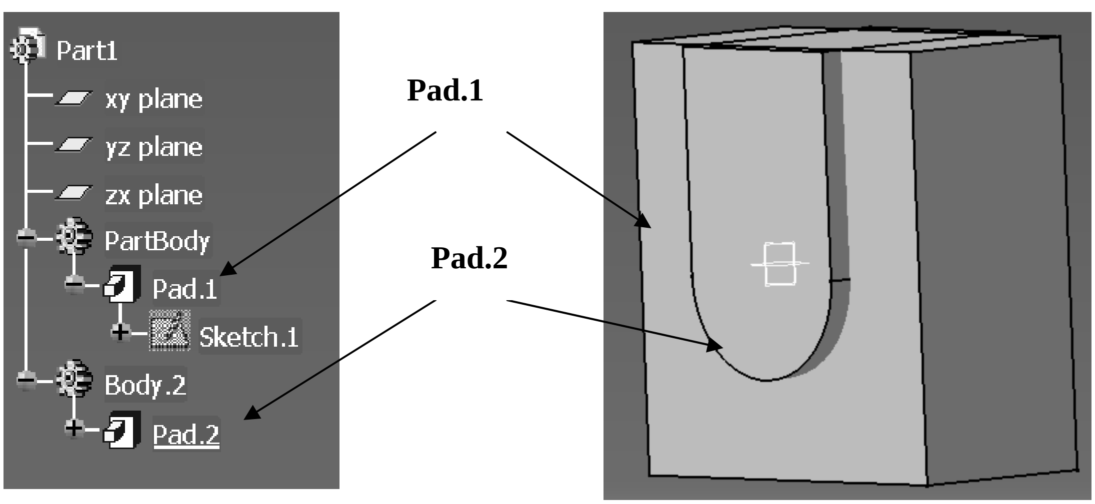


Figure 3.7

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.

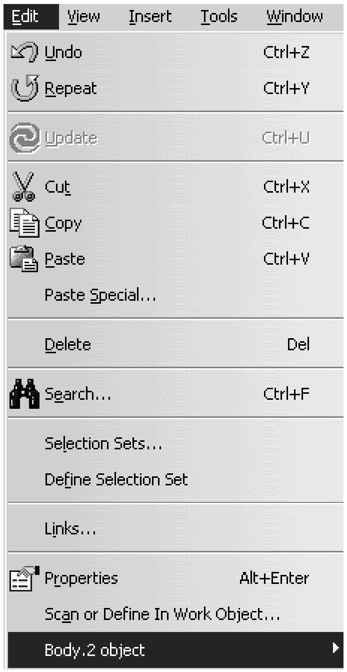


Figure 3.8

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.

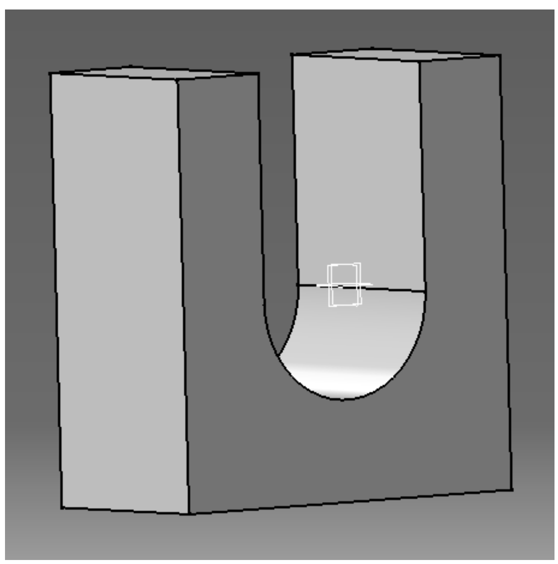


Figure 3.9

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

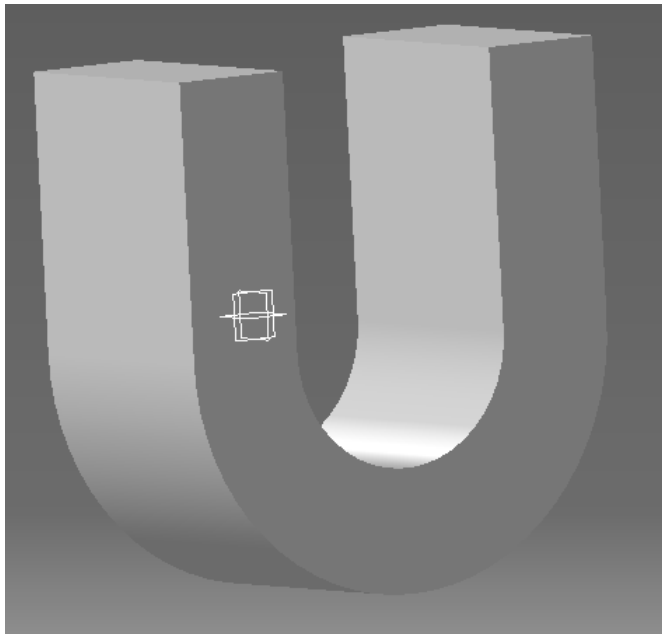


Figure 3.10

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

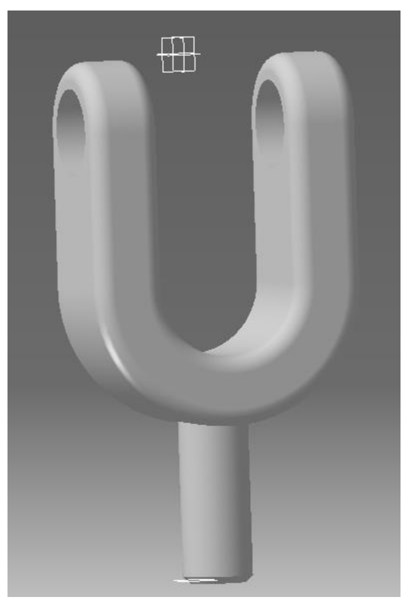


Figure 3.11

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

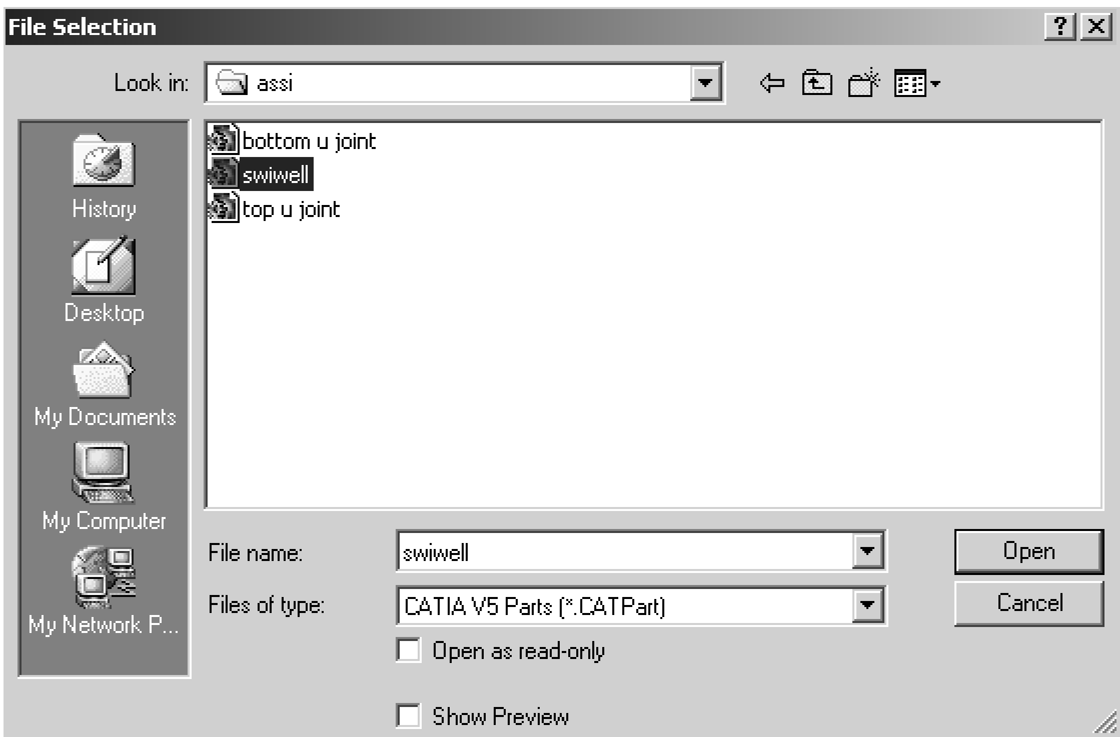


Figure 4.1

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.

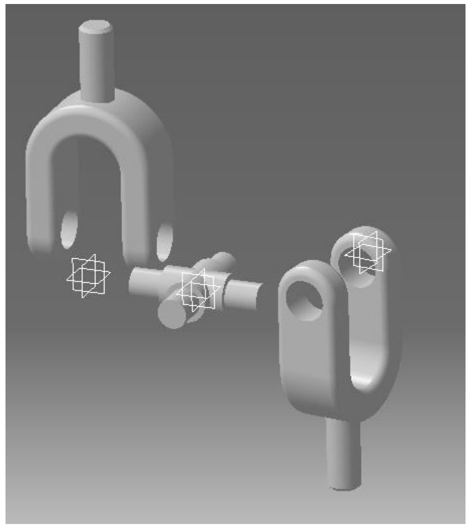


Figure 4.2

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

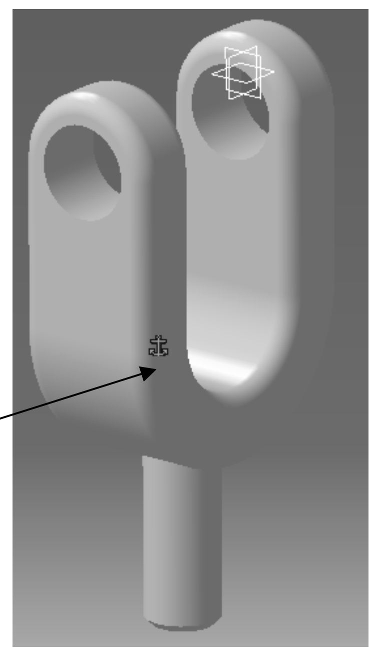
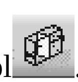


Figure 4.3

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

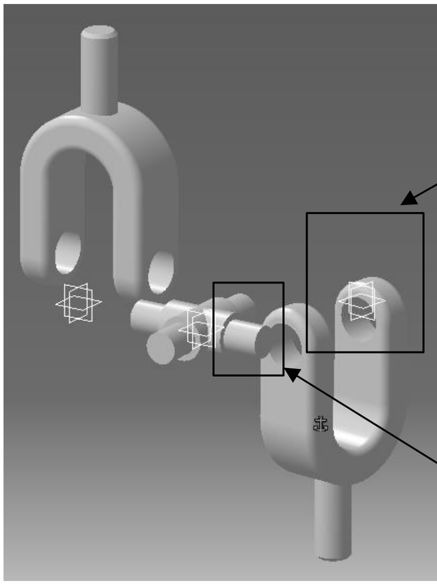
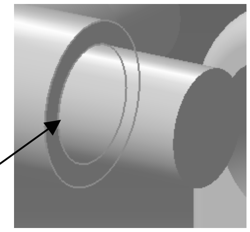
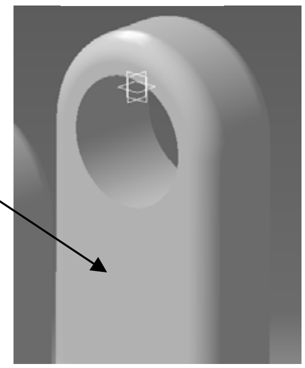


Figure 4.4



View A



View B

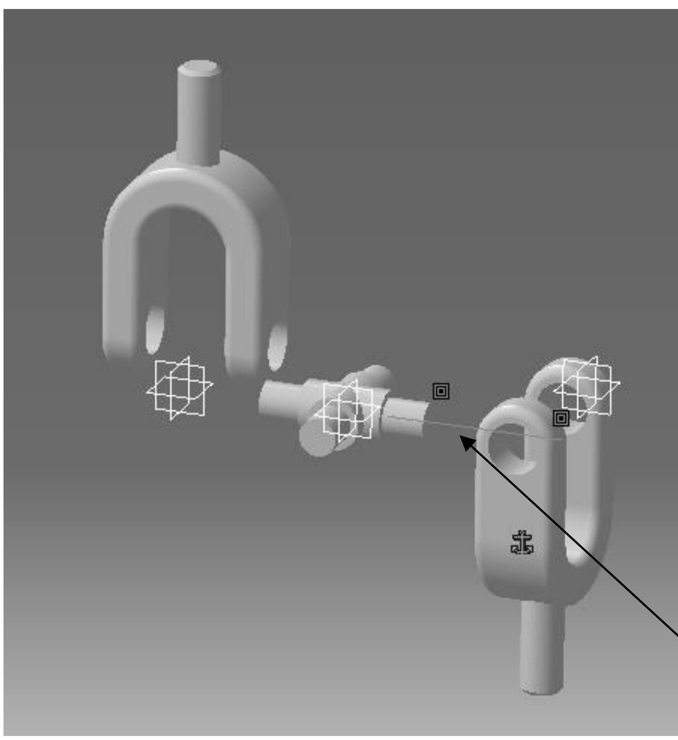


Figure 4.5

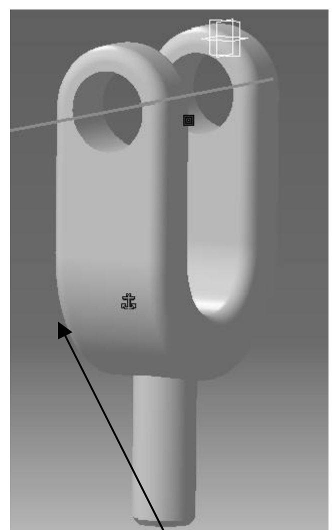


Figure 4.6

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.

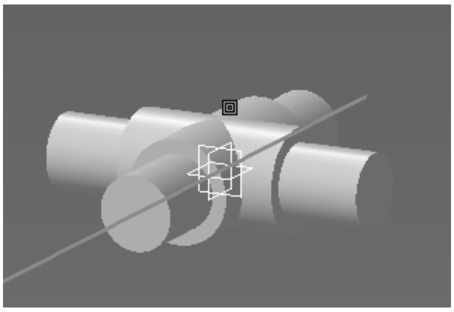
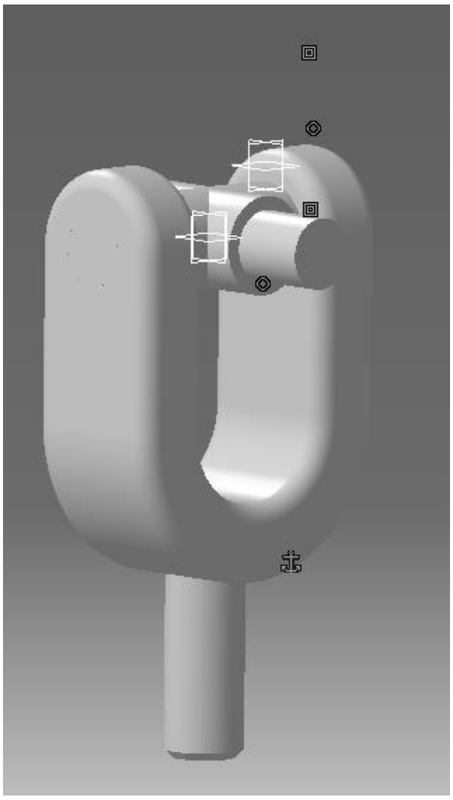


Figure 4.7



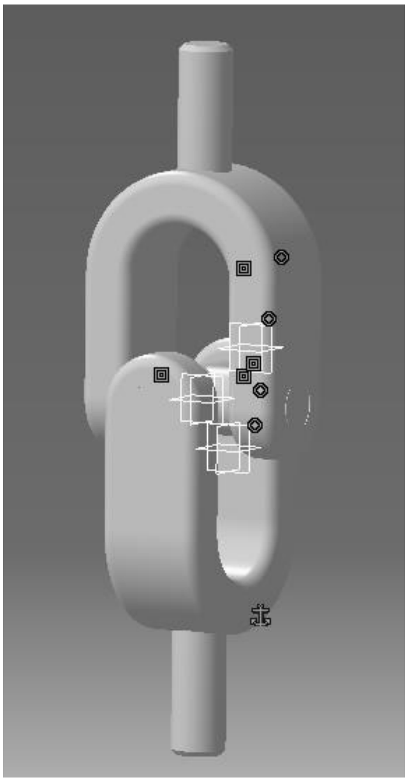


Figure 4.7 **Figure 4.8**

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:

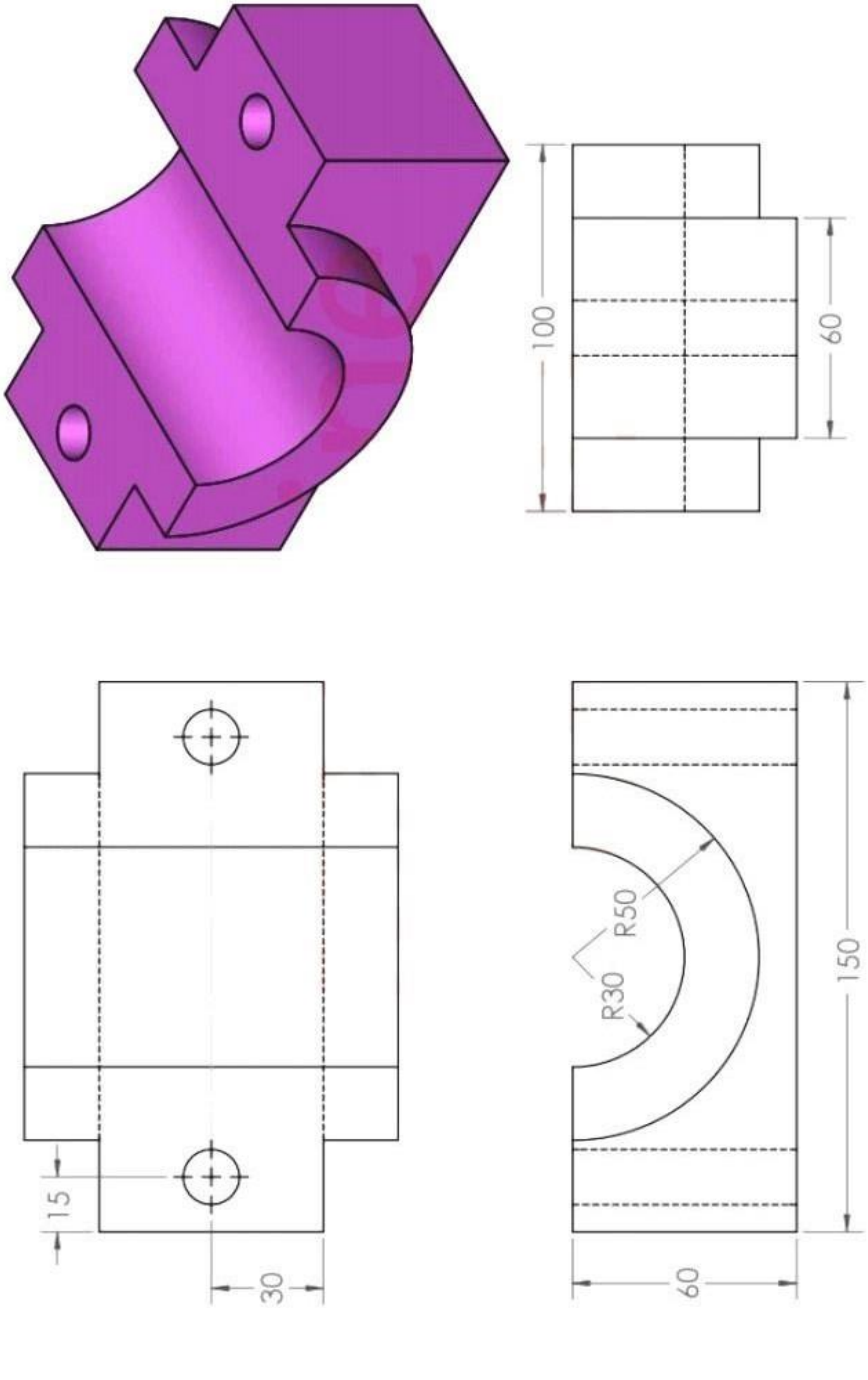
Preparation of 3D model using CATIA V5.18 software

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:

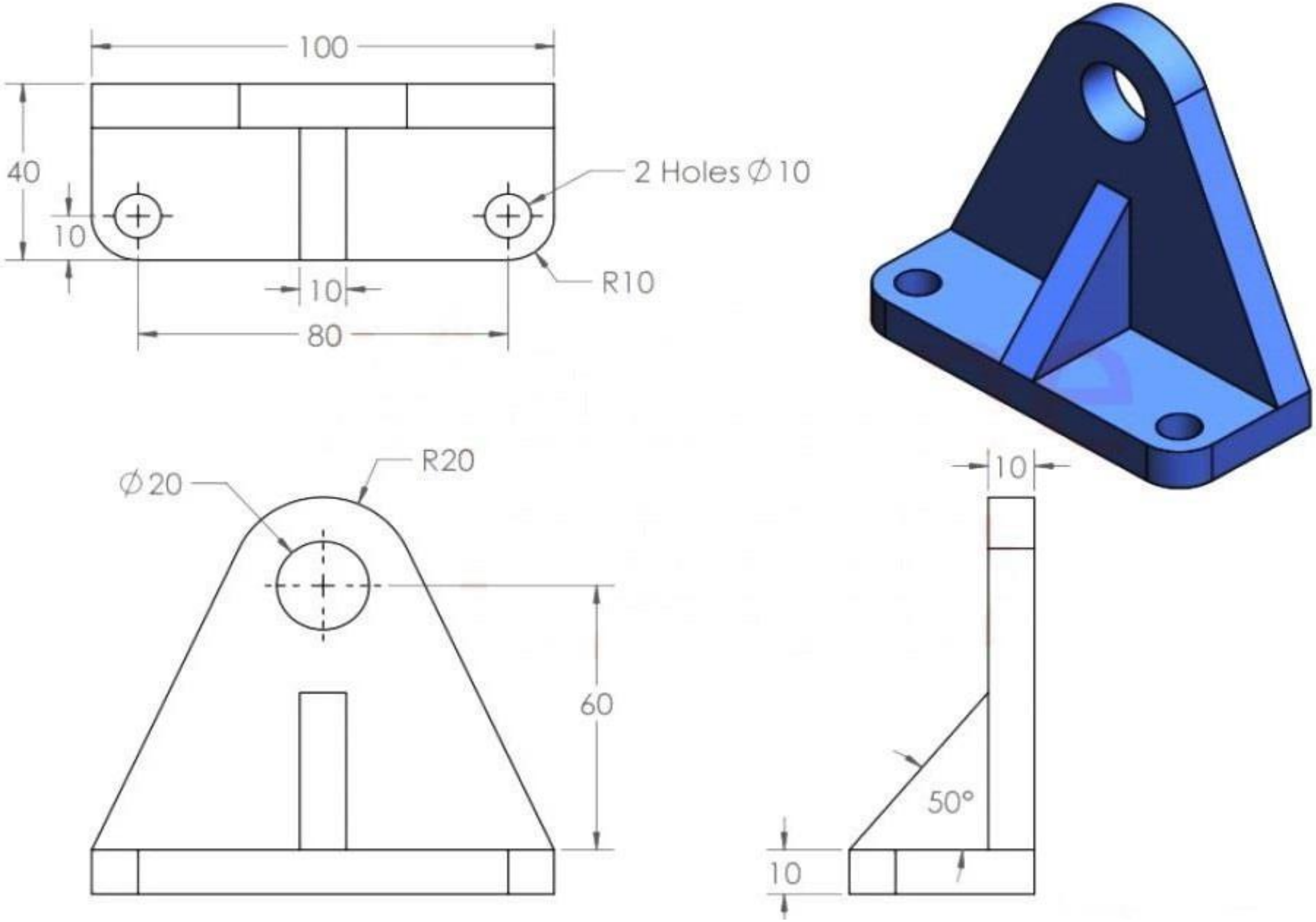
Preparation of 3D model using CATIA V5.18 software

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:

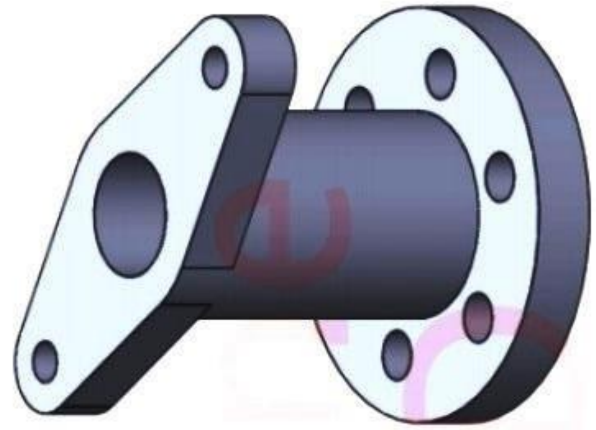
Preparation of 3D model using CATIA V5.18 software

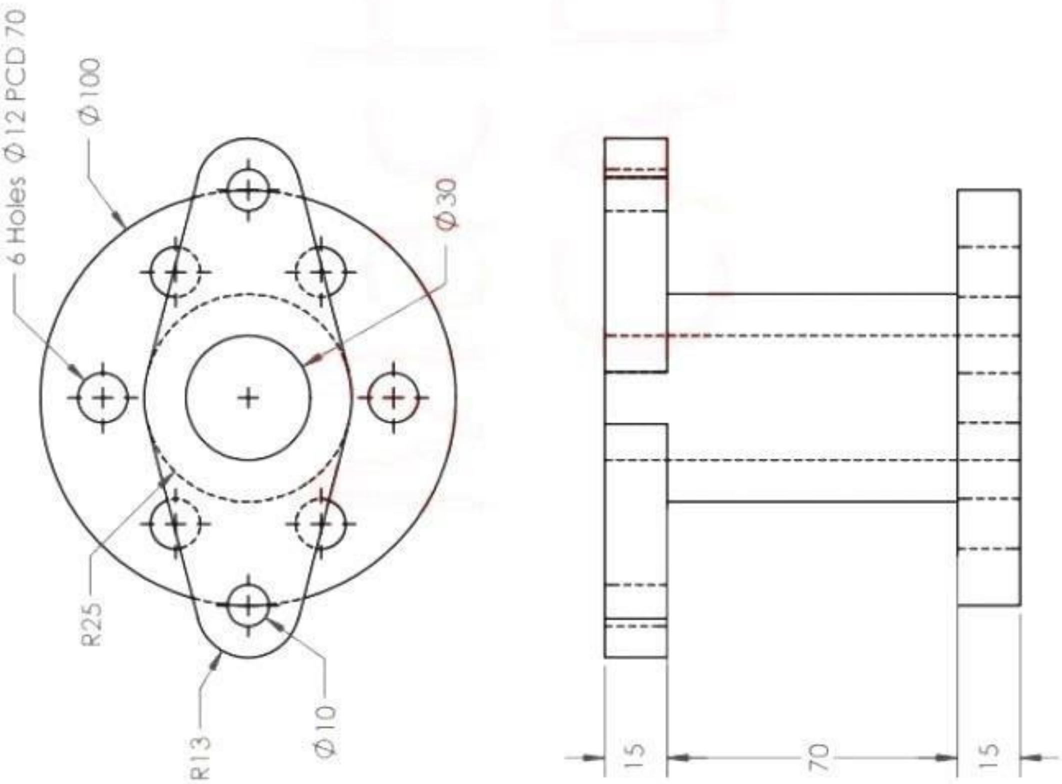
TOOLS USED:

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:

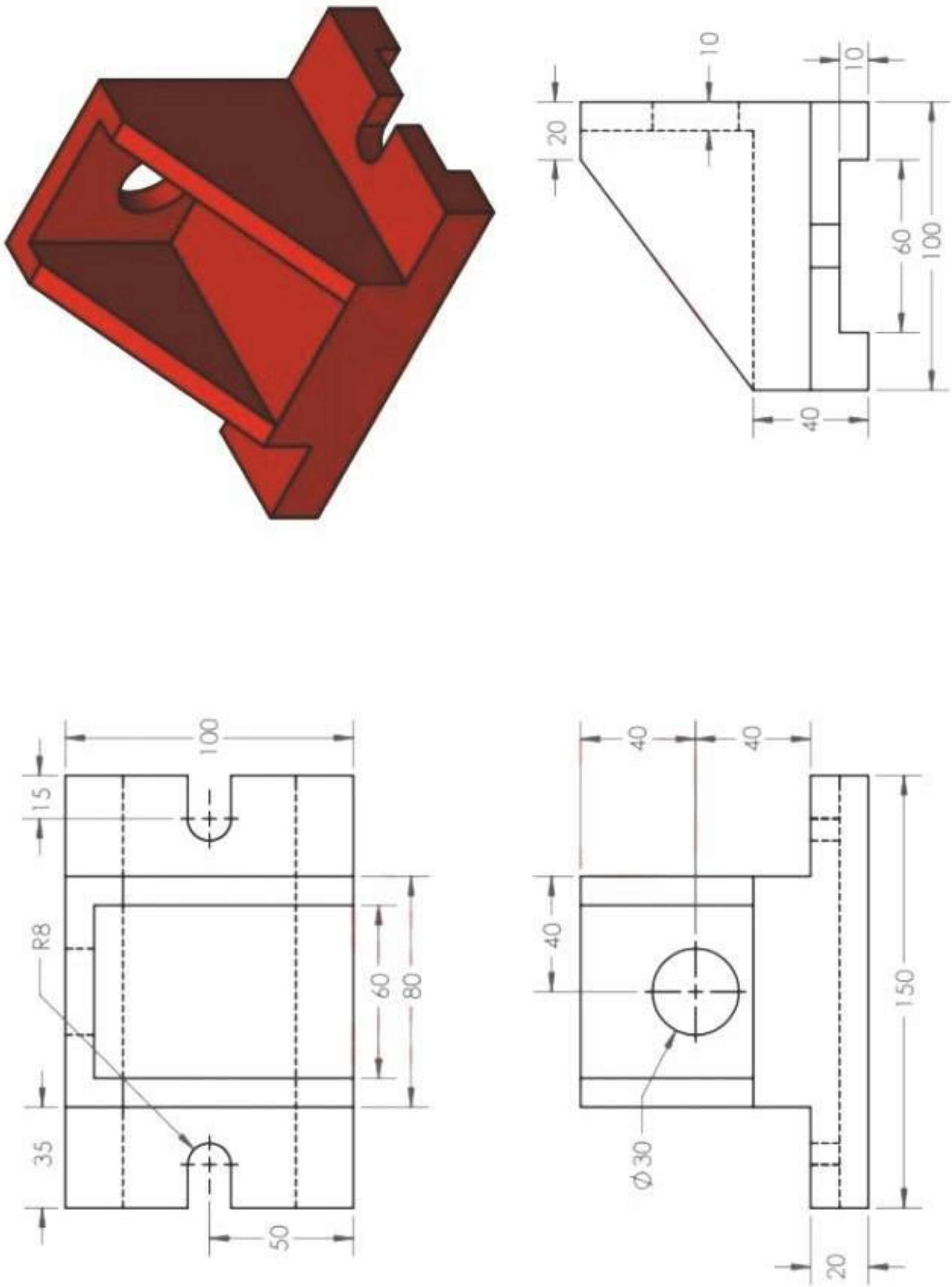
Preparation of 3D model using CATIA V5.18 software

TOOLS USED:

Pad, Pocketing etc.,

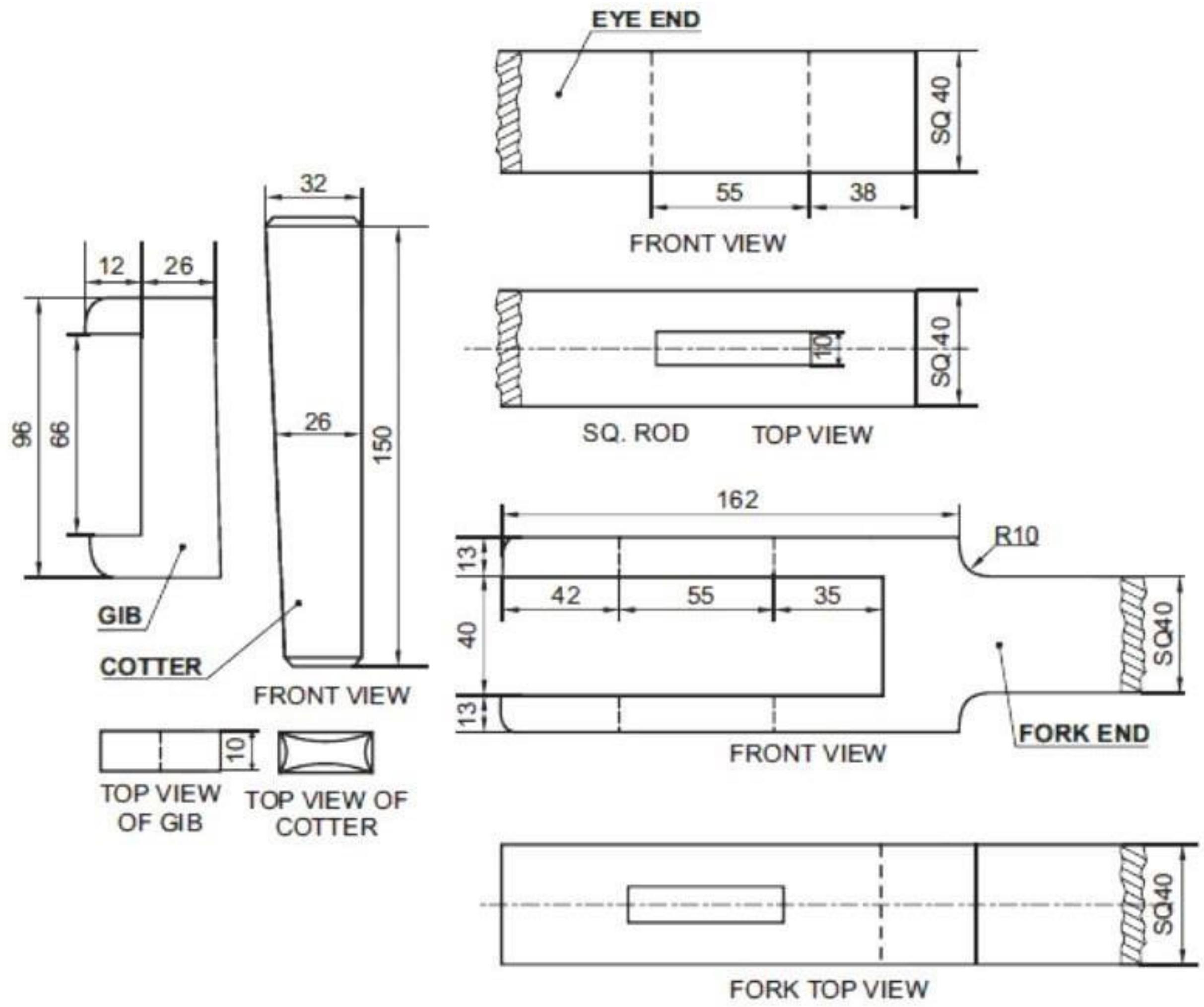
PROCEDURE:

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
60
80
35
50
40
40
40
150
Ø 30
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:

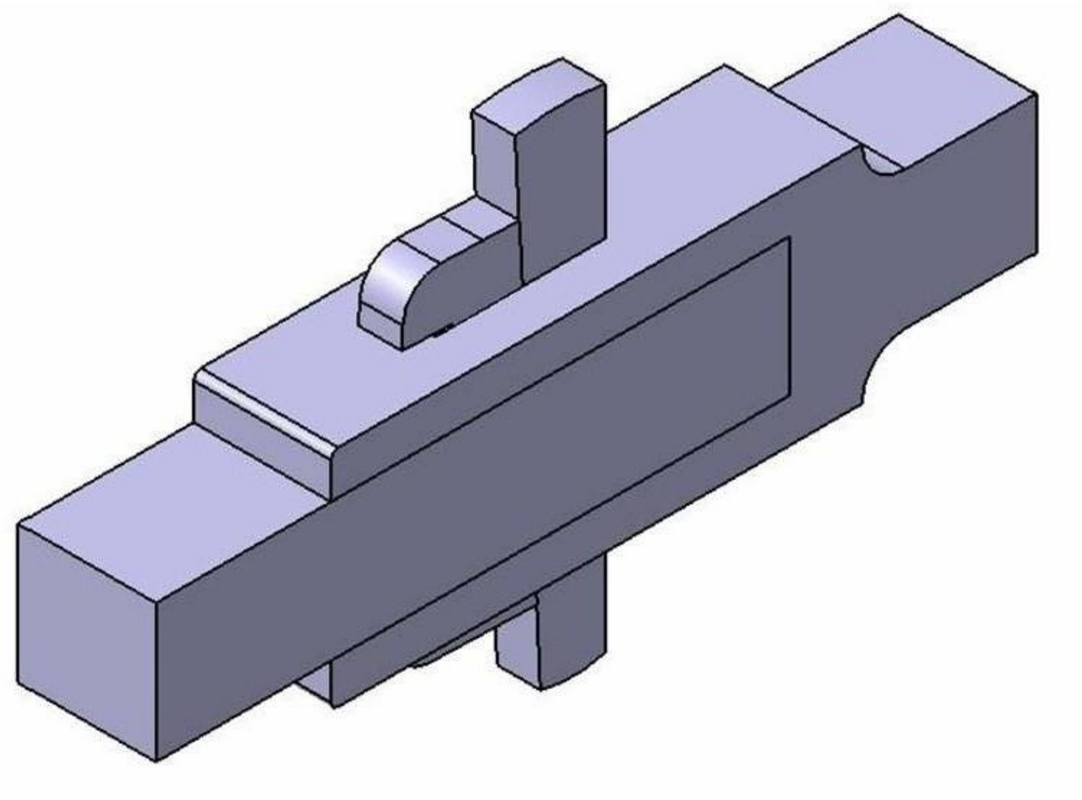
Preparation of 3D Assembly model using CATIA V5.18 software

TOOLS USED:

Existing Component icon, Smart move, Constraining etc.,

PROCEDURE:

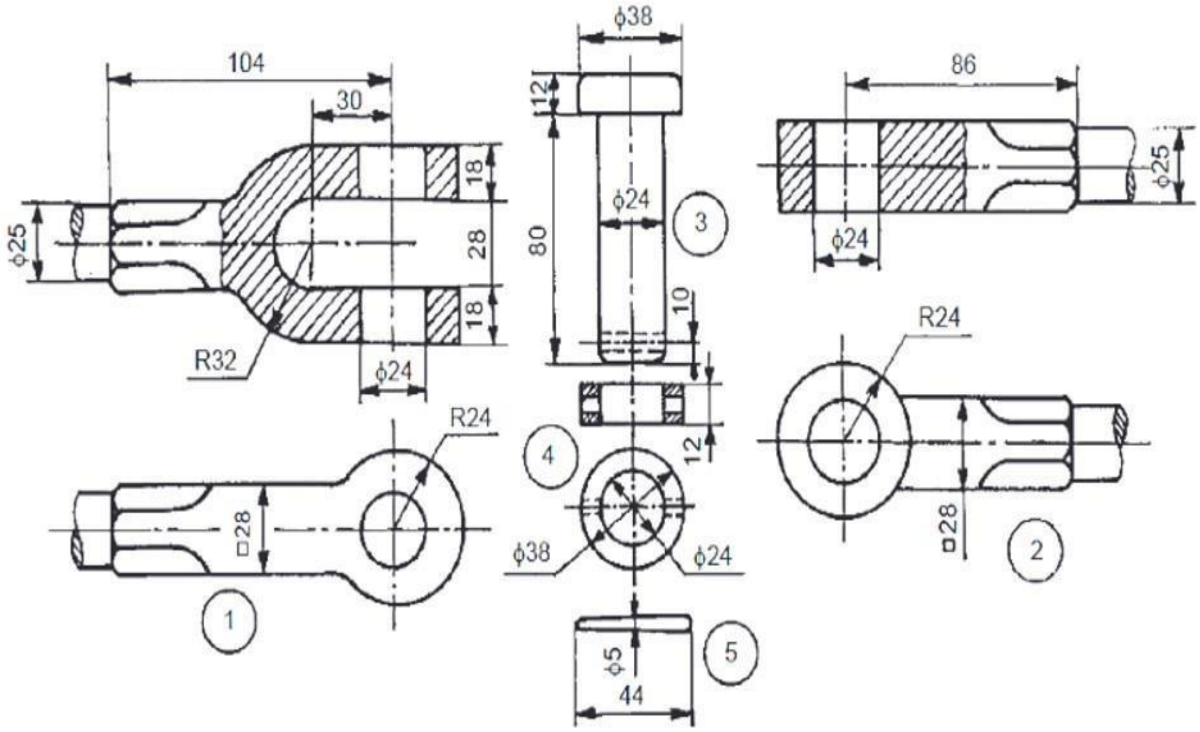
1. *Assembly Design* command is activated to launch the required workbench.
2. The commands for assembling parts are available in the toolbar on the right side of the application window.
3. Click the *Existing Component icon* in the Product Structure toolbar, then the File Selection dialog box is displayed 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:

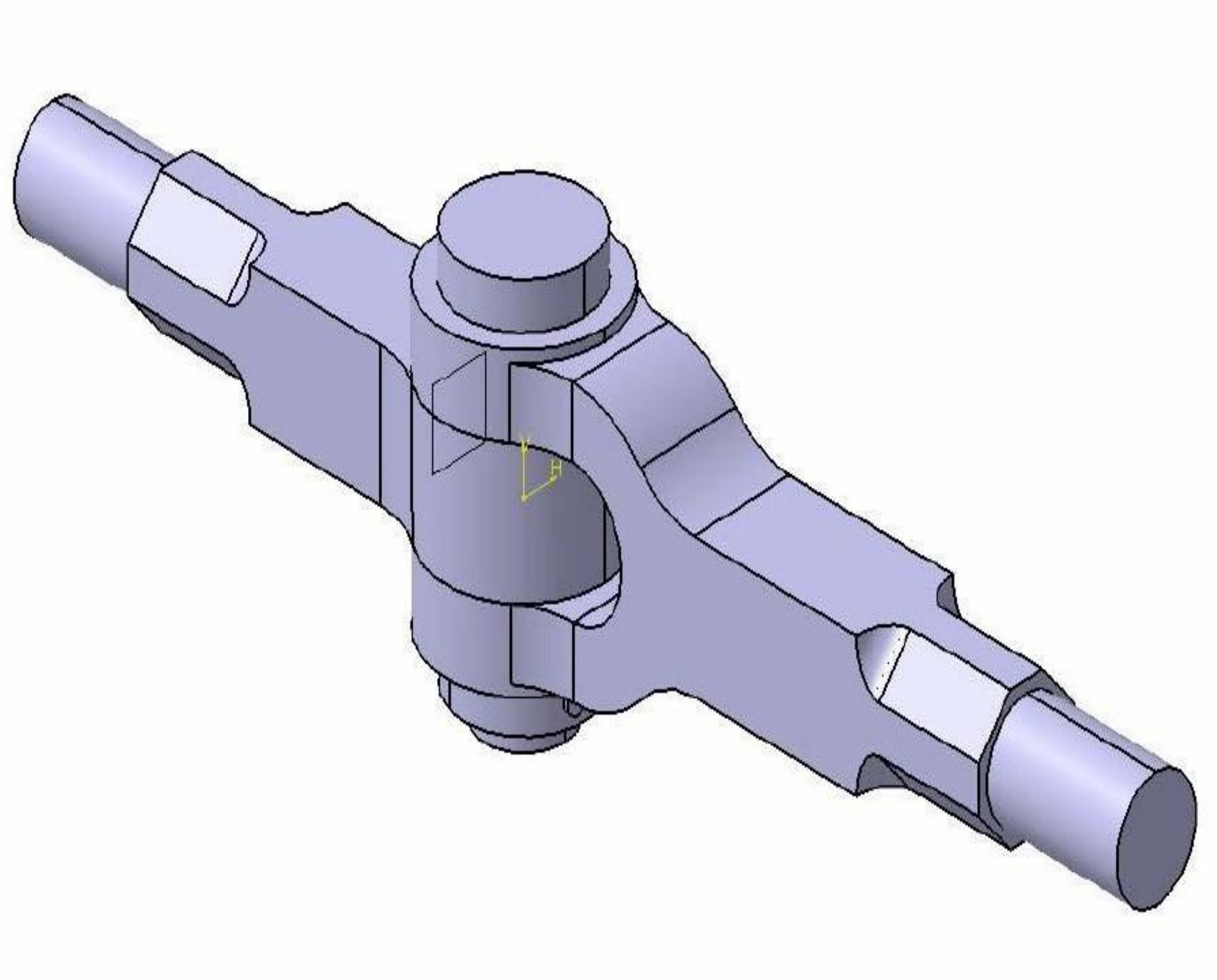
Preparation of 3D Assembly model using CATIA V5.18 software

TOOLS USED:

Existing Component icon, Smart move, Constraining etc.,

PROCEDURE:

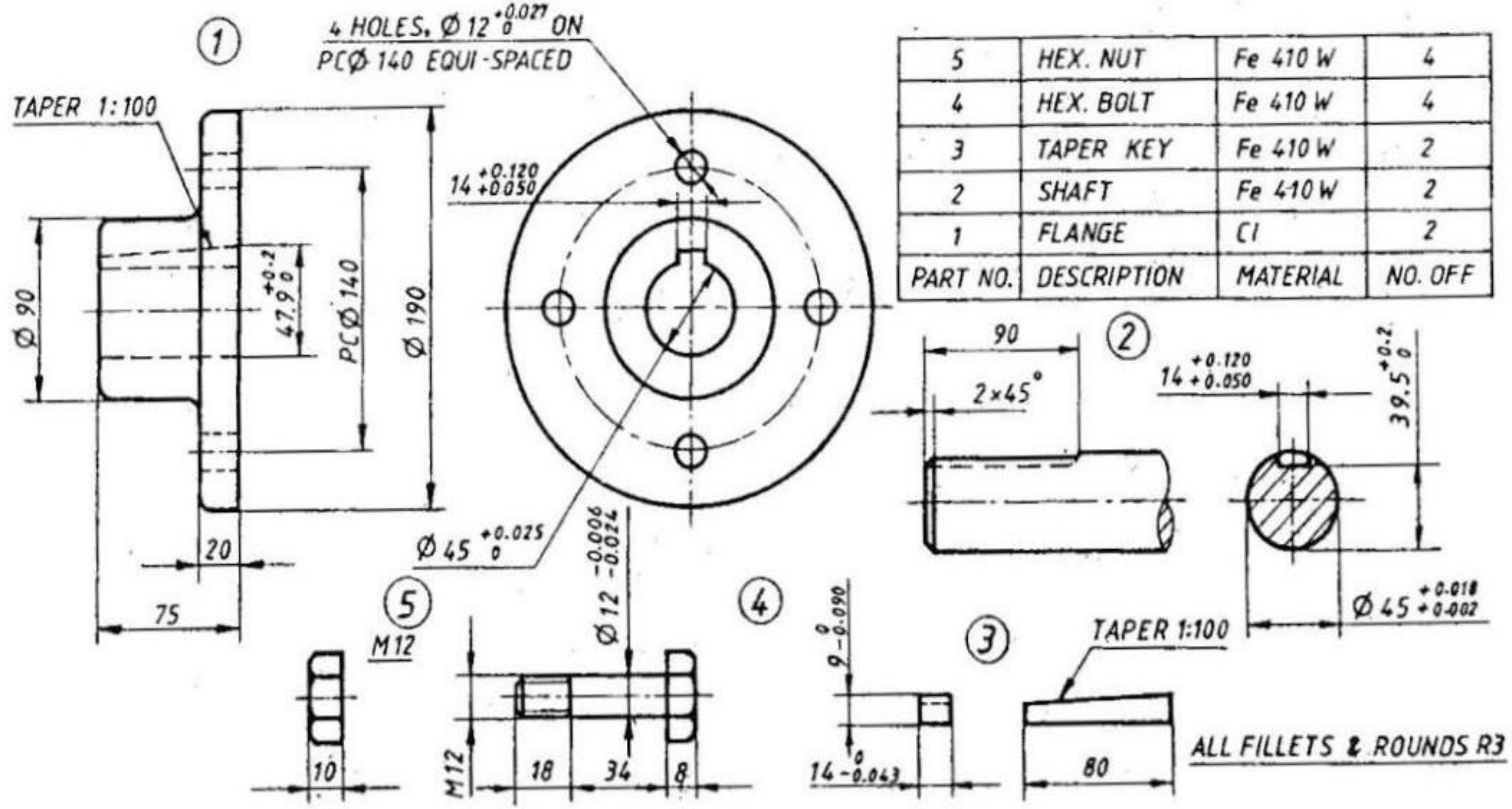
1. *Assembly Design* command is activated to launch the required workbench.
2. The commands for assembling parts are available in the toolbar on the right side of the application window.
3. Click the *Existing Component icon* in the Product Structure toolbar, then the File Selection dialog box is displayed 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:

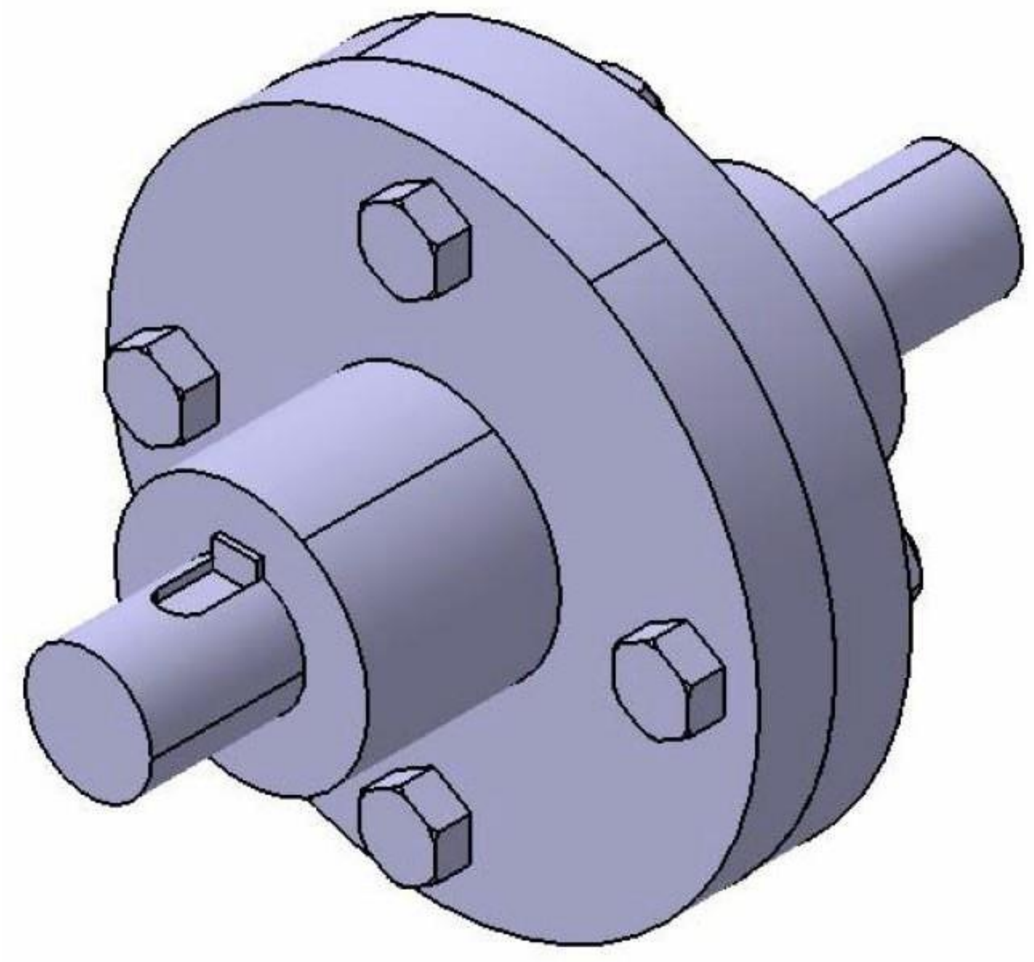
Preparation of 3D Assembly model using CATIA V5.18 software.

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.

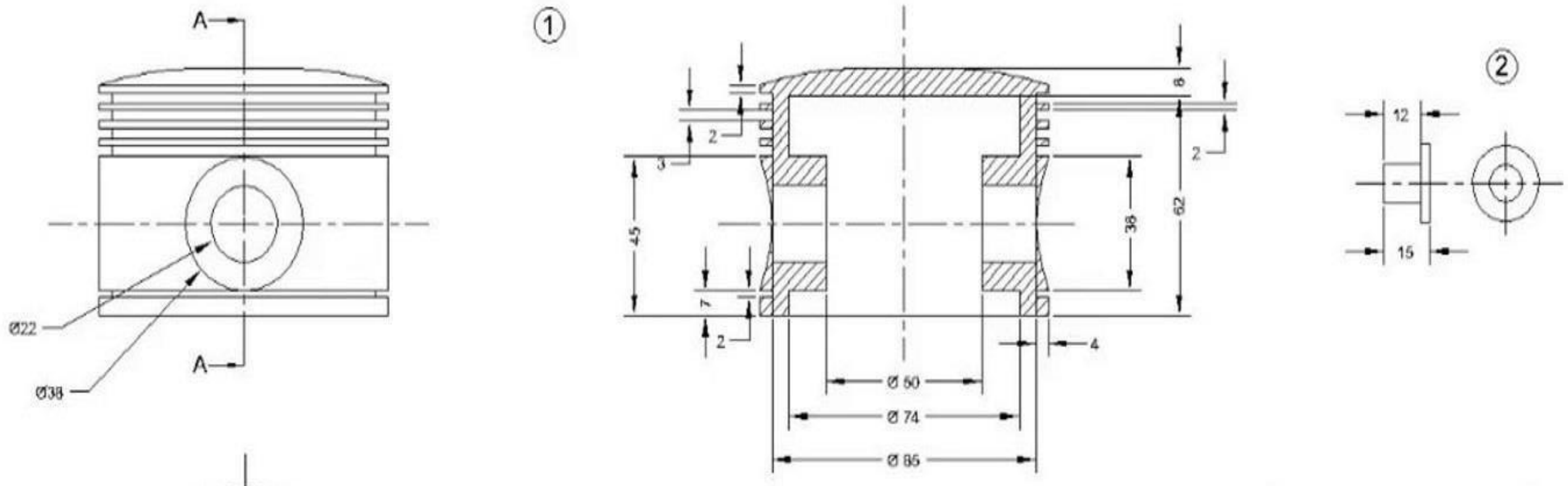


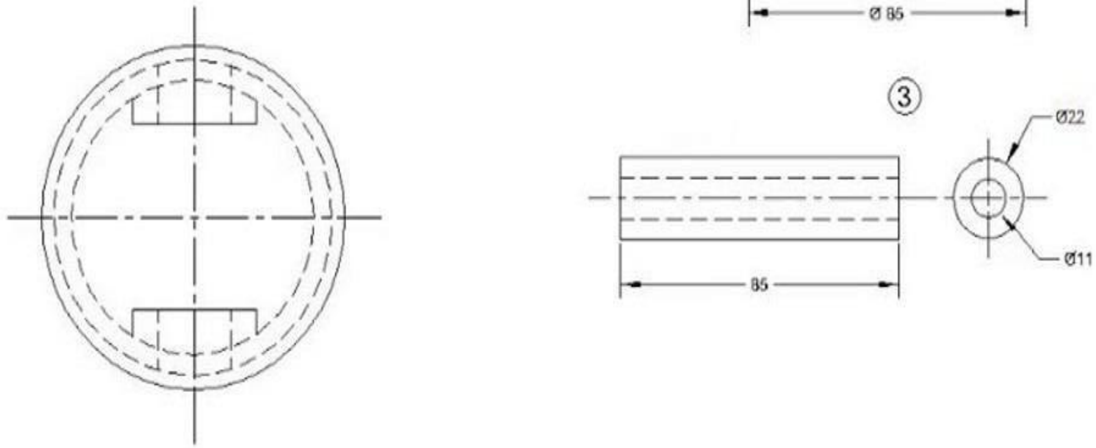
Assembly of Unprotected type Flange Coupling

RESULT:

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

Piston with Piston Pin





PARTS LIST		
Sl.No.	Name	Qty.
1	Piston	1
2	Piston Pin	1
3	Piston Pin Ring	2

EX. NO. 8 **ASSEMBLY OF PISTON WITH PIN**

DATE:

AIM:

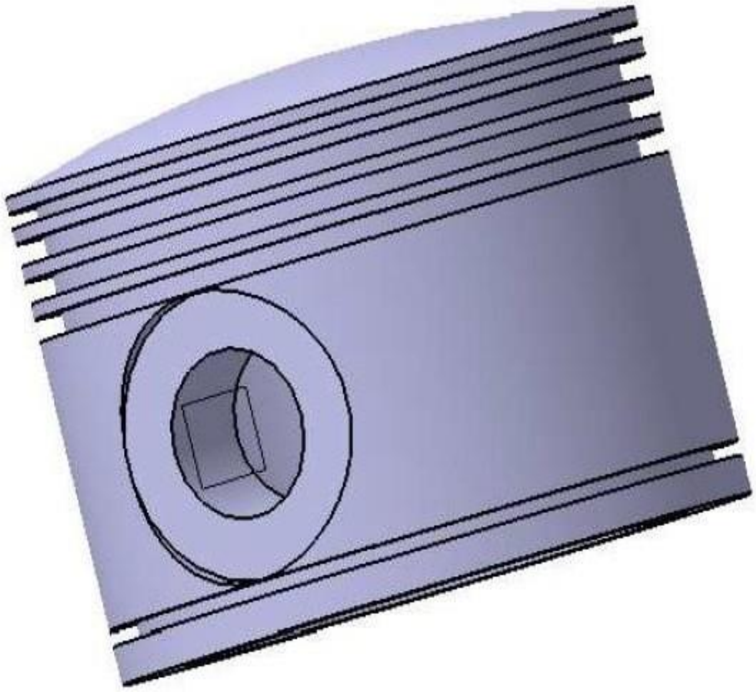
Preparation of 3D Assembly model using CATIA V5.18 software

TOOLS USED:

Existing Component icon, Smart move, Constraining etc.,

PROCEDURE:

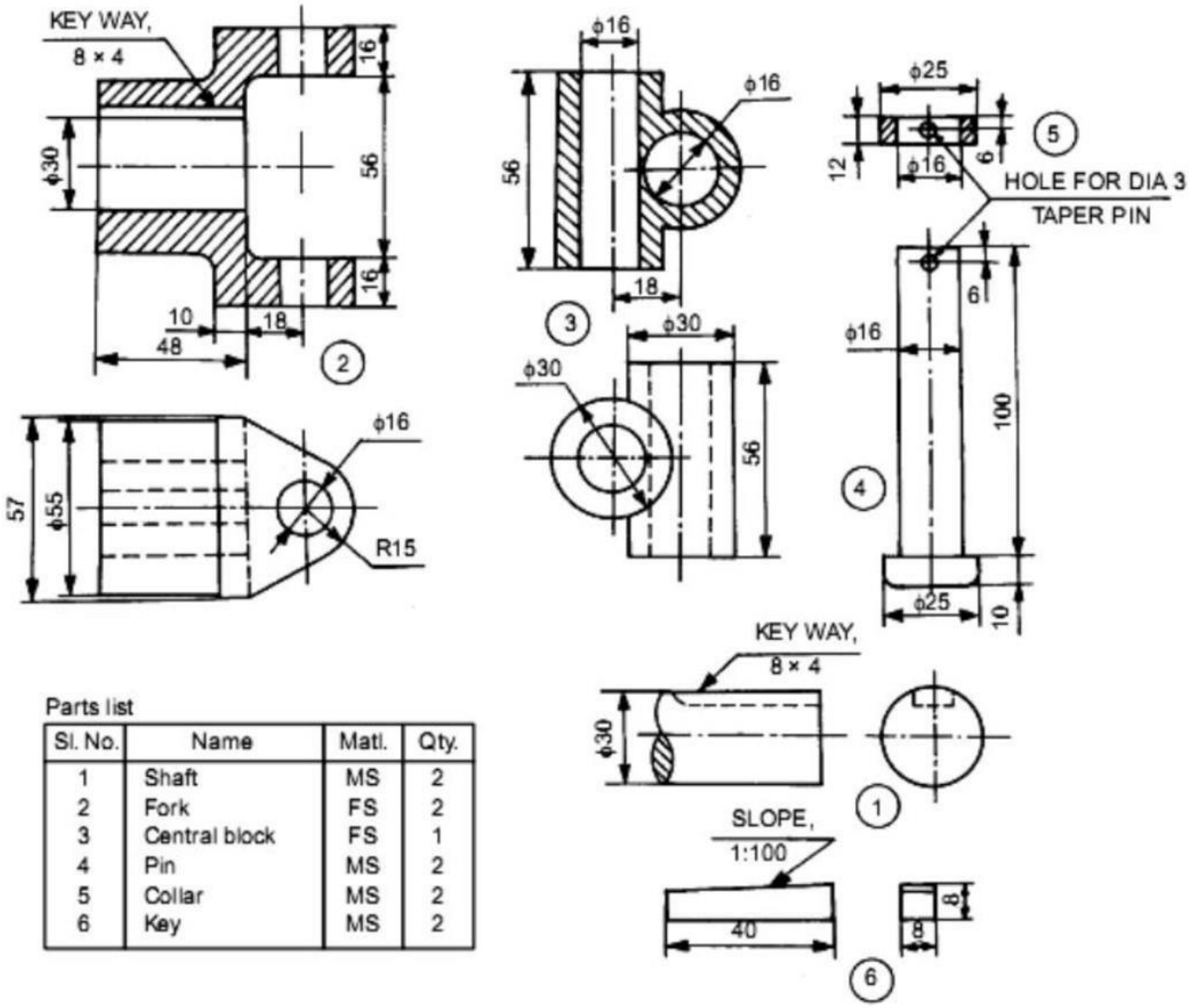
1. *Assembly Design* command is activated to launch the required workbench.
2. The commands for assembling parts are available in the toolbar on the right side of theapplication window.
3. Click the *Existing Component icon* in the Product Structure toolbar, then the File Selection dialog box is displayed 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 ***Piston, piston pin and piston ring*** 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 Piston with piston pin

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	Shaft	MS	2
2	Fork	FS	2
3	Central block	FS	1
4	Pin	MS	2
5	Collar	MS	2
6	Key	MS	2

Details of Universal Coupling

EX. NO. 9 **ASSEMBLY OF UNIVERSAL COUPLING**

DATE:

AIM:

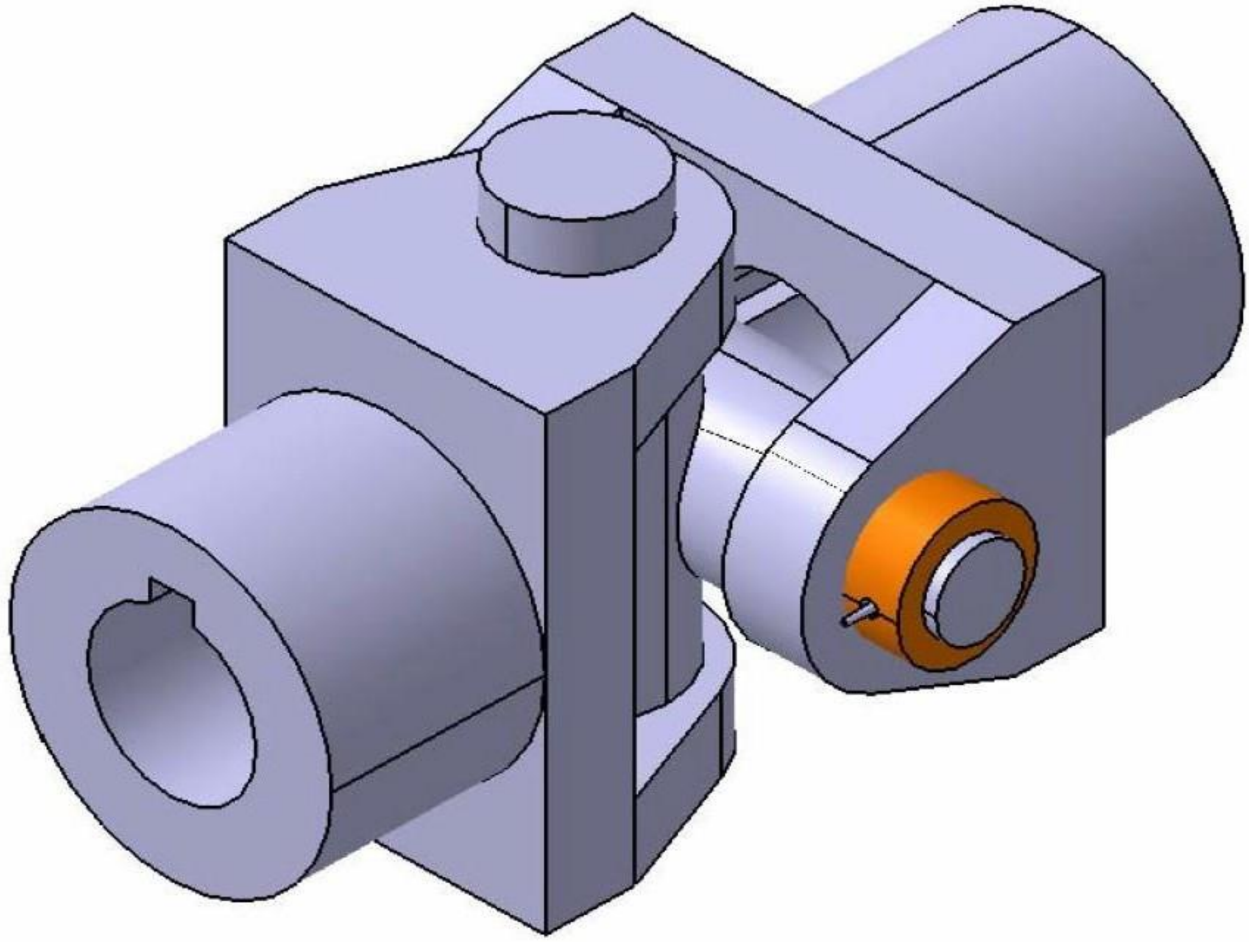
Preparation of 3D Assembly model using CATIA V5.18 software

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 parts are assembled in the following order ***Flange, Centre block, collar, Taperpin*** by adopting above methods
6. Select the update icon for assembly to be updated, all assembly design models areensured that it is in constraints condition before the models are finished.

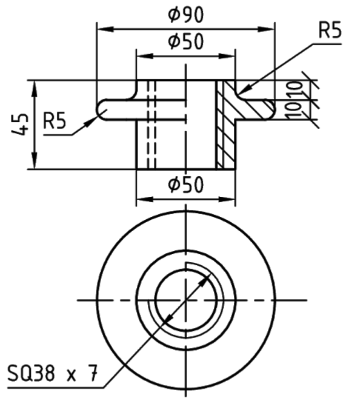


Assembly of Universal Coupling

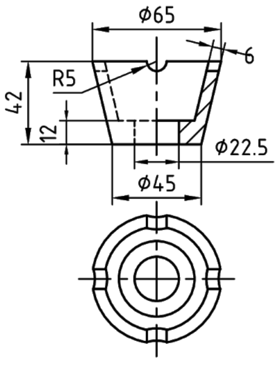
RESULT:

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

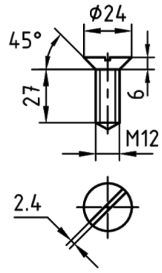
DETAILS OF SCREW JACK



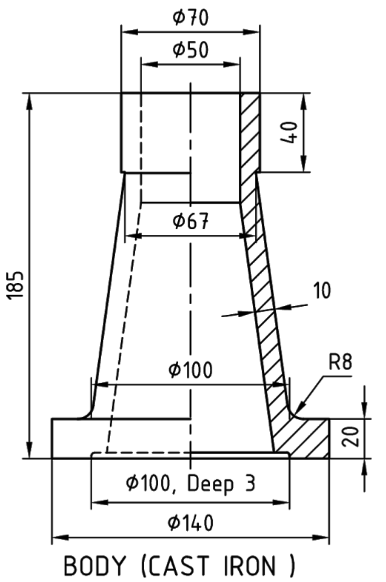
NUT (GUN METAL)



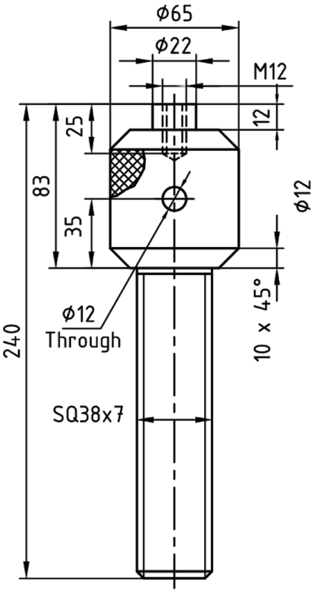
CUP (CAST STEEL)



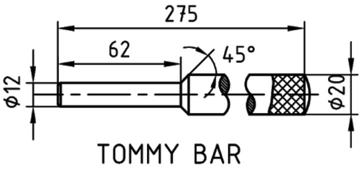
SCREW
(MILD STEEL)



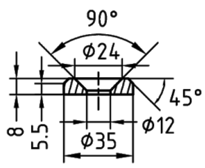
BODY (CAST IRON)



SCREW SPINDLE
(MILD STEEL)



TOMMY BAR
(MILD STEEL)



SPECIAL WASHER
(MILD STEEL)

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

DATE:

AIM:

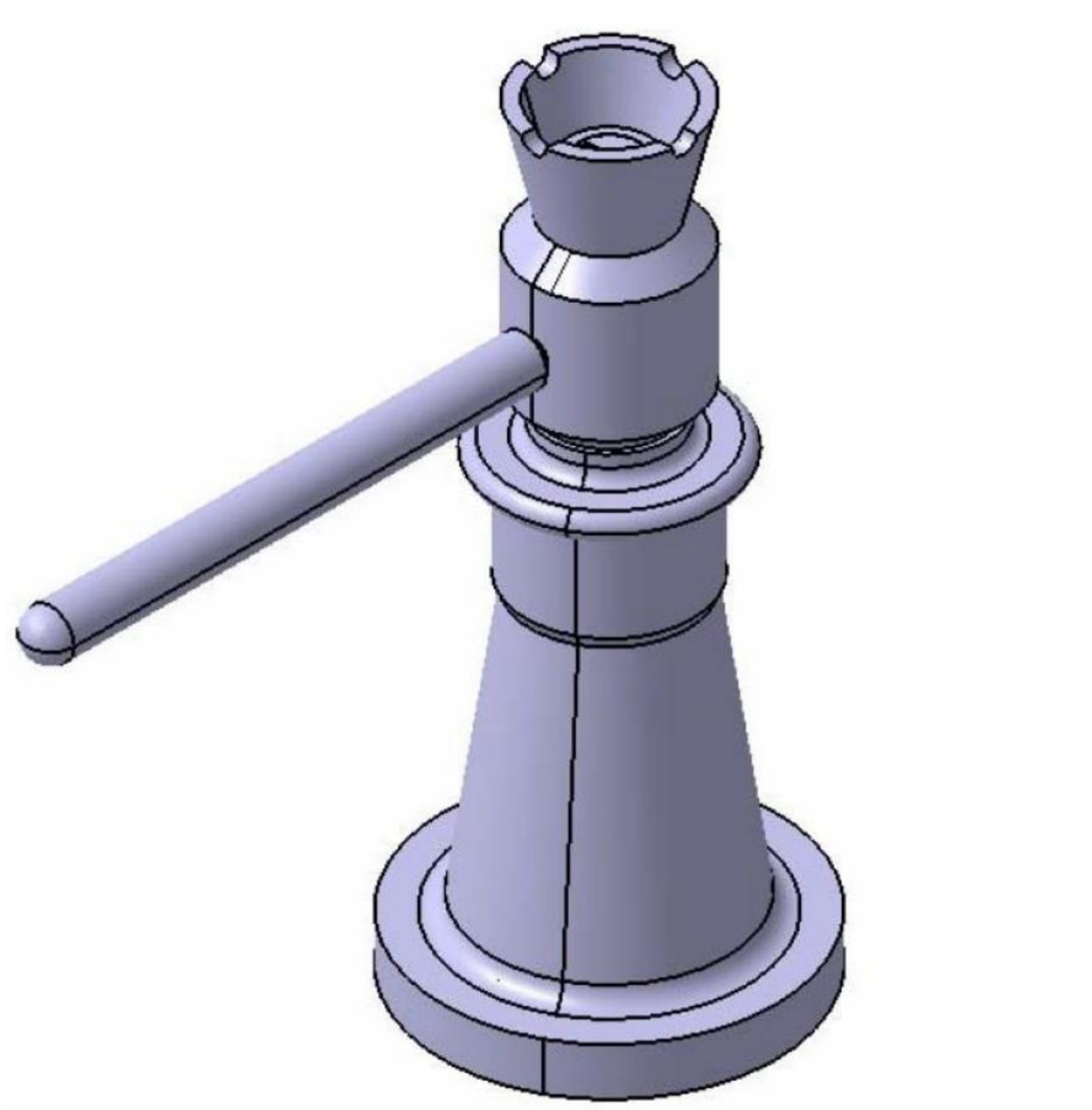
Preparation of 3D Assembly model using CATIA V5.18 software

TOOLS USED:

Existing Component icon, Smart move, Constraining etc.,

PROCEDURE:

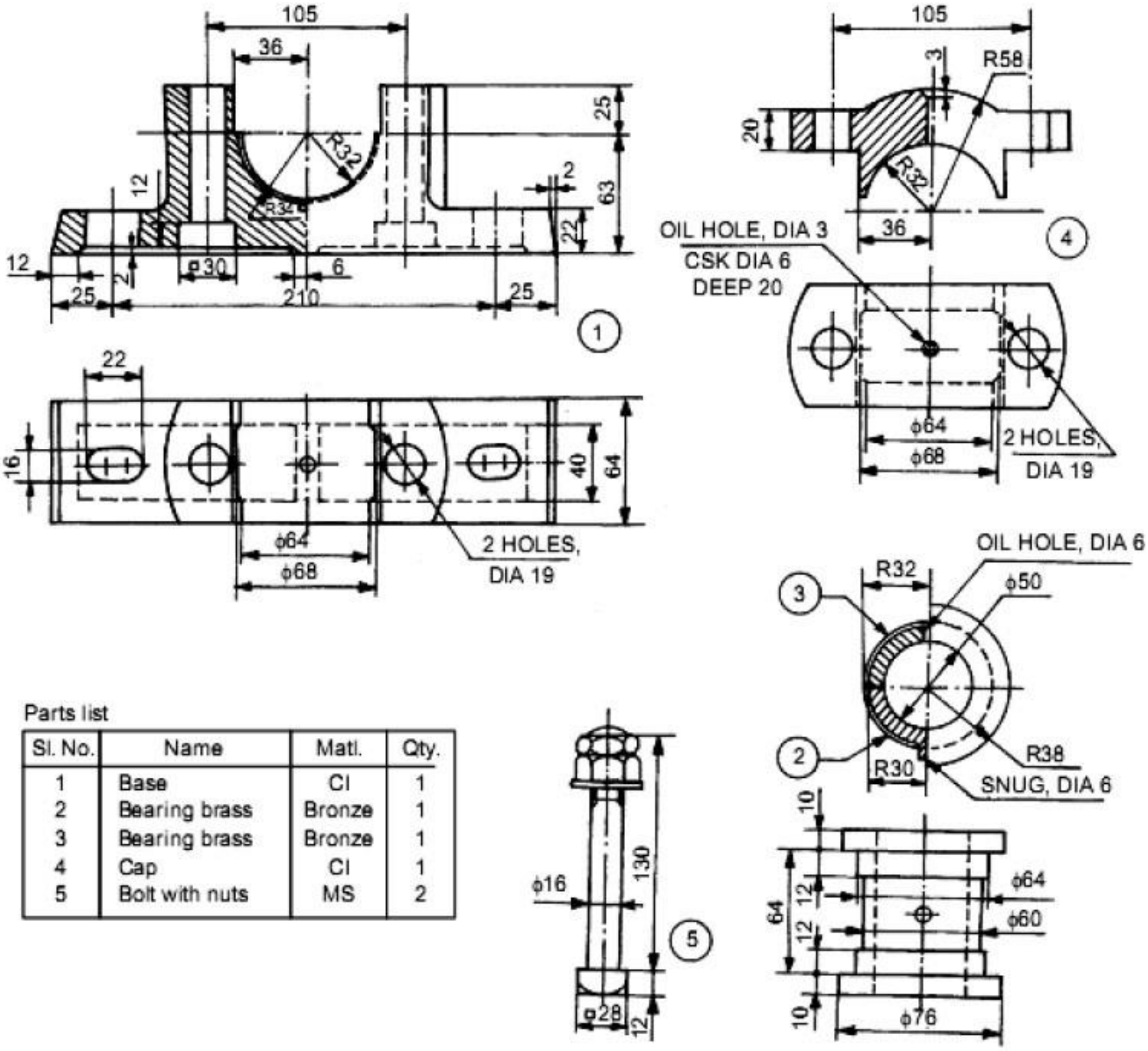
1. *Assembly Design* command is activated to launch the required workbench.
2. The commands for assembling parts are available in the toolbar on the right side of theapplication window.
3. Click the *Existing Component icon* in the Product Structure toolbar, then the File Selection dialog box is displayed 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:

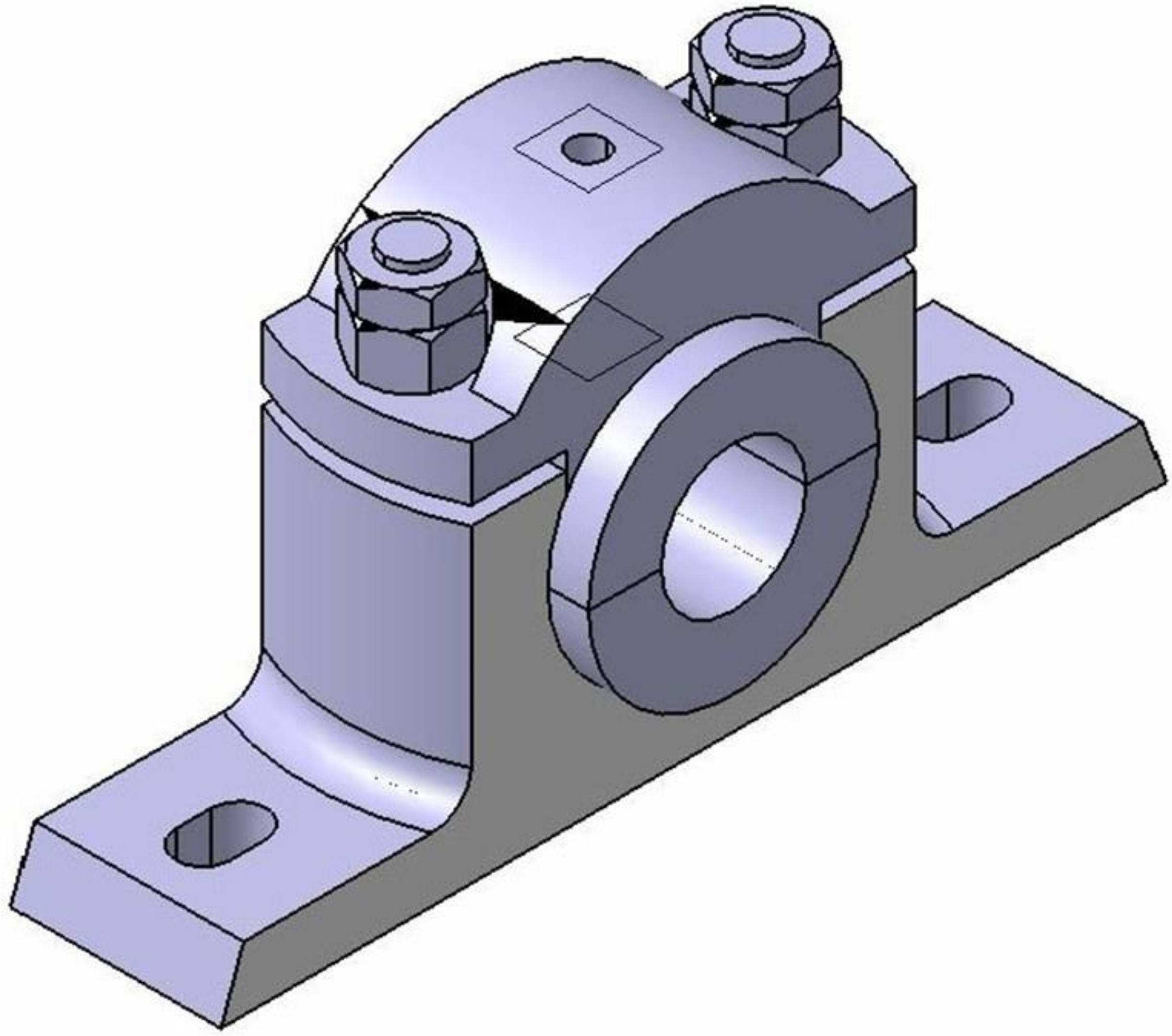
Preparation of 3D Assembly model using CATIA V5.18 software

TOOLS USED:

Existing Component icon, Smart move, Constraining etc.,

PROCEDURE:

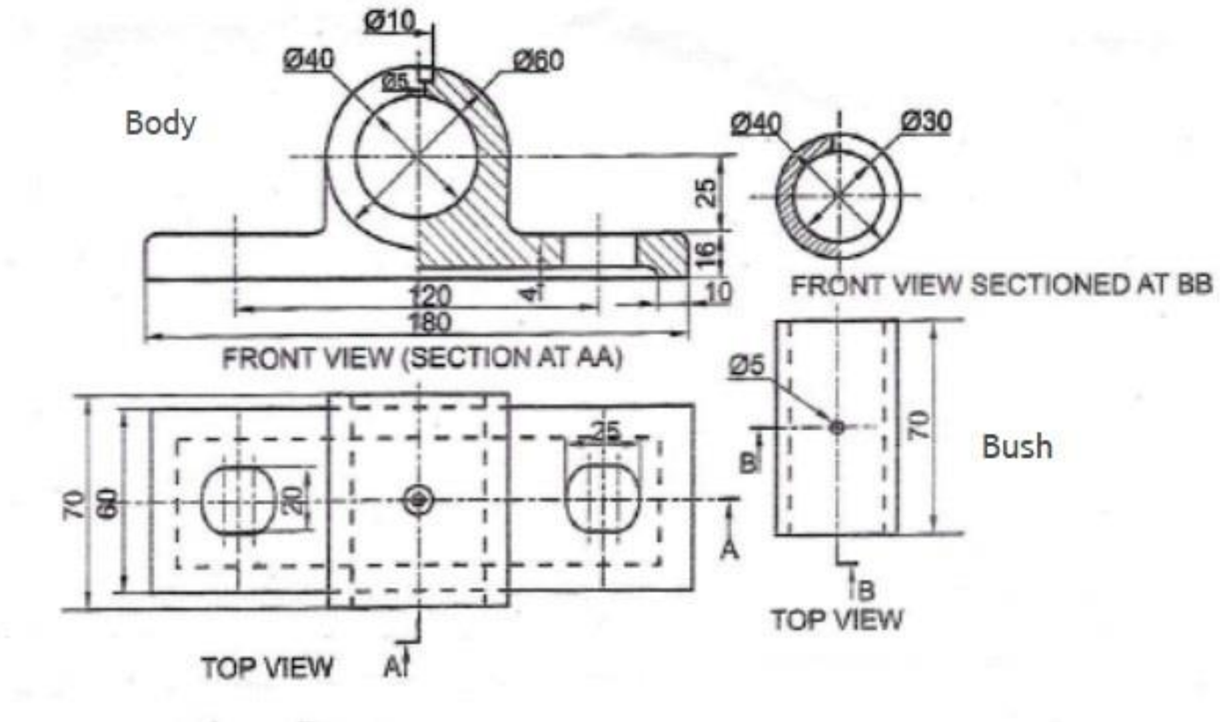
1. *Assembly Design* command is activated to launch the required workbench.
2. The commands for assembling parts are available in the toolbar on the right side of the application window.
3. Click the *Existing Component icon* in the Product Structure toolbar, then the File Selection dialog box is displayed 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:

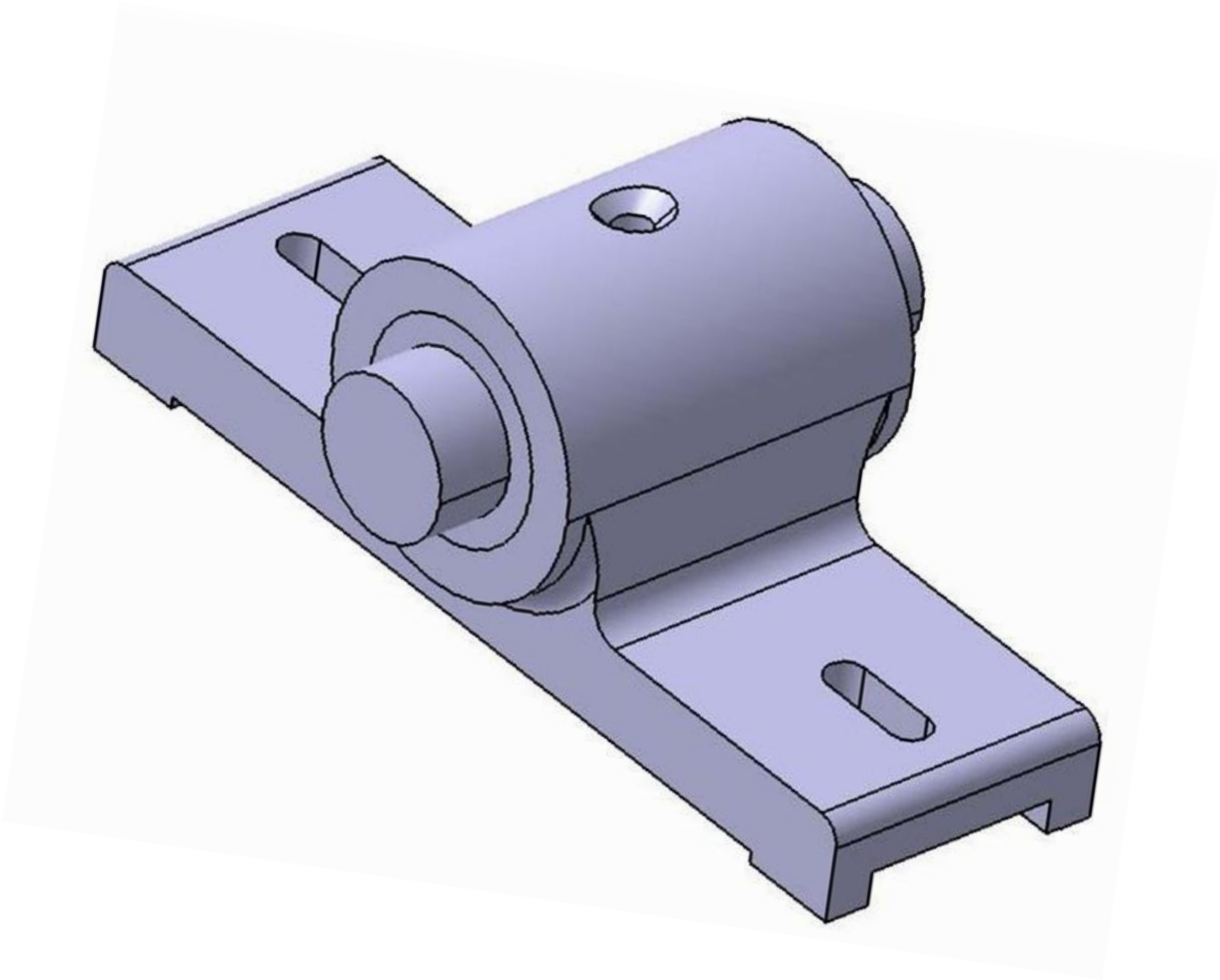
Preparation of 3D Assembly model using CATIA V5.18 software

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.

****

VIVA QUESTIONS

1. **Why tolerances are given to the parts?**

   Because it's impossible to make perfect settings✓

   To reduce weight of the component

   To reduce cost of the assembly

   To reduce amount of material used

2. **What is bilateral tolerance?**

   Total tolerance is in 1 direction only

   Total tolerance is in both the directions✓

   May or may not be in one direction

   Tolerance provided all over the component body

3. **Which type of tolerance provided in drilling mostly?**

   Bilateral

   Unilateral✓Trilateral

   Compound

4. **What is mean clearance?**

   Maximum size of hole minus maximum size of shaft Minimum

   size of hole minus minimum size of shaft

   Mean size of hole minus mean size of shaft✓

   Average of both size of shaft and hole

5. **Which of the following is incorrect about tolerances?**

   Too loose tolerance results in less cost✓

   Tolerance is a compromise between accuracy and ability

   Too tight tolerance may result in excessive cost

   Fit between mating components is decided by functional requirements

6. **Bilateral tolerances are used in**

   Unitary production

   Mass production✓

   Both Unitary and mass production

   None of the above

7. **The overall width of a part is dimensioned as 3.00 ± 0.02. What is the upper limit?**

   3.00

   +0.02

   3.02✓

   0.04

8. **How may types of Assemblies are used in CATIA**

   Top down

   Bottom up

   Both top down and bottom up✓

   None of the above

9. **Abbreviation of CATIA**

   Computer Aided Three Dimensional Interactive Animation Computer

   Aided Three Dimensional Interactive Application✓ Computer Assisted

   Three Dimensional Interactive Application Computer Aided Three

   Dimensional Internet Application

10. **Polar coordinates are used mostly for drawing**

    Circles

    Arcs

    Vertical lines Angled

    lines✓

11. **When the interior of an object is complicated, which of the following view is used?**

    Front view

    Side view

    Top view

    Sectional view✓

12. **When the cutting plane cuts the entire object the section is known as**

    ____

    Full section✓

    Half section

Revolved section Removed

section

13. **Inclined and offset cutting planes can be used if_____________**

all the hidden objects are not in one line

all the hidden objects are in one line

the single line nor offset sectioning is useful and shape of the object is inclined

it is used for combined objects✓

14. **Crane hook is to drawn by____________method.**

full section

half section

removed section revolved

section✓

15. **The section which cuts the object at an angle is called**

removed section

broken out section

auxiliary section✓

assembly section

16. **The command which is used to set a new layer is called**

LAYOFF

LAYVPI

LAYDEL

LAYER✓

17. **Modifying a layer consist of**

Line weight and line type✓

Thawing

Freeze

Deleting a layer

18. **Thin parts like stiffeners, webs, bolts, rivets, etc. are___________if they are cut by the cutting plane along their axis.**

Not hatched✓ hatched

sectioned

Not sectioned

19. **Which command is used to divide the object into segments having predefined length?**

Divide

Chamfer

Trim

Measure✓

20. **Status bar do not contain**

Snap

Grid

Erase✓

Polar

21. **Which mode allows the user to draw 90° straight lines**

Osnap

Ortho✓

Linear

Polar tracking

22. **To obtain parallel lines, concentric circles and parallel curves;____________is used.**

Array

Fillet

Copy

Offset✓

23. **The default grid spacing in both X and Y directions is:**

10✓

20

5

15

24. **Scale command can be accessed easily by typing**

SL

S

SC✓

C

25. **What is the save extension of the sketcher file in CATIA?**

CATPart✓ CATIAPart

CPart

None of the above

****