A person in a flight suit is working on the tail rotor assembly of a helicopter. The person is kneeling on the ground, holding a long metal rod that is part of the rotor system. The helicopter's tail rotor is visible in the background, and the person is wearing a flight helmet and a dark jacket. The scene is set outdoors on a grassy field.

# **ME119: Engineering Drawing & Graphics**

**CAD-3D - SolidWorks**

**Department of Mechanical Engineering  
Indian Institute of Technology Bombay**



### CAD packages:

- AUTO-CAD
- PRO-E
- CATIA
- UNI-GRAPHICS
- **SOLIDWORKS**

F1 racing car made by using CAD software

# Outline

- Introduction to 3D CAD
- Solidworks Package
- Exercise
- Conclusions

# Types of CAD

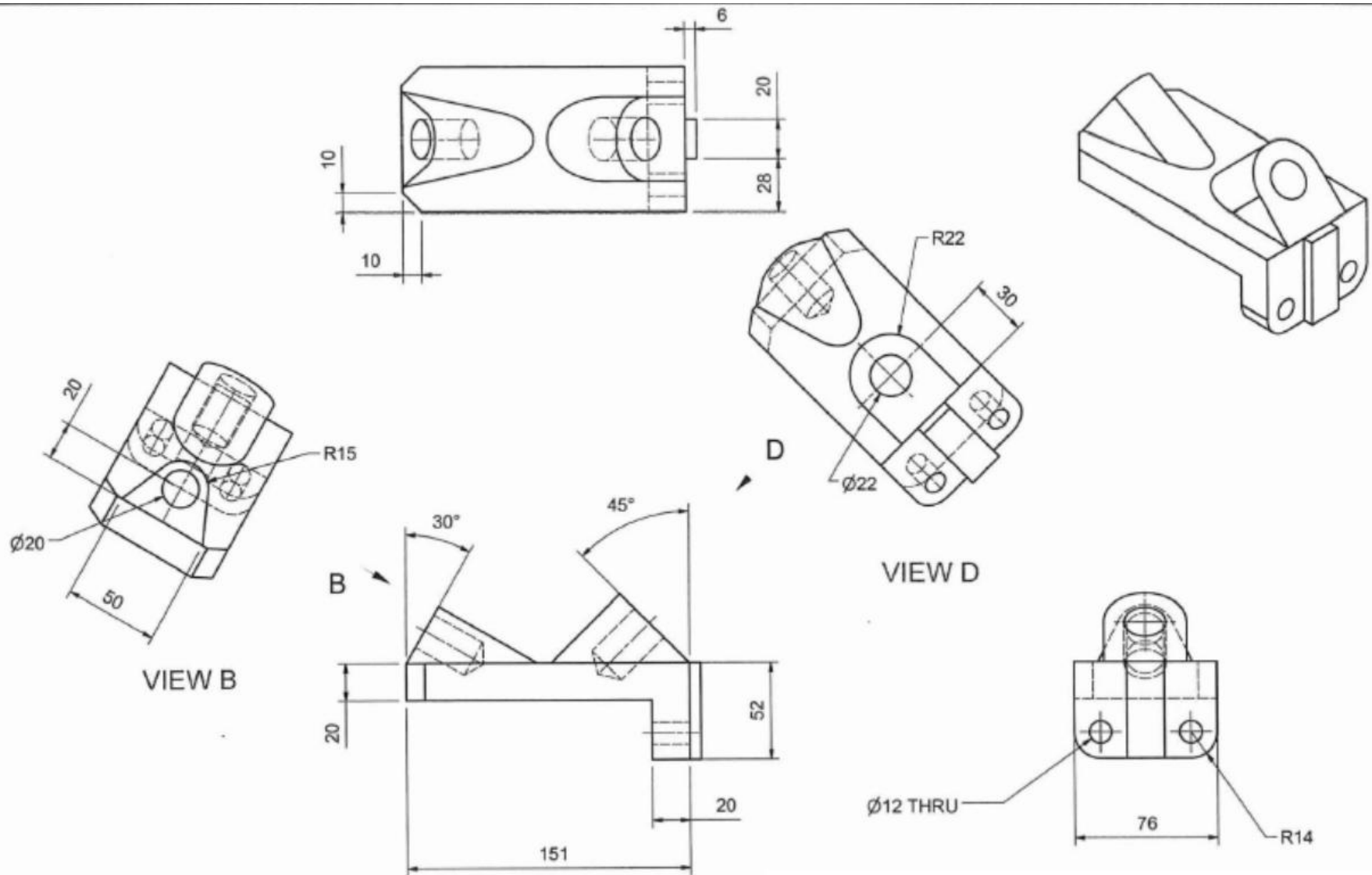
Computer-Aided Drafting	Computer-Aided Design
2D: Computer is used as a sophisticated drafting board.	3D: Design the objects in 3D (called Solid Modeling). Simply by specifying the viewing point, you can get all views including sectional and perspective views.

# Solid-Works

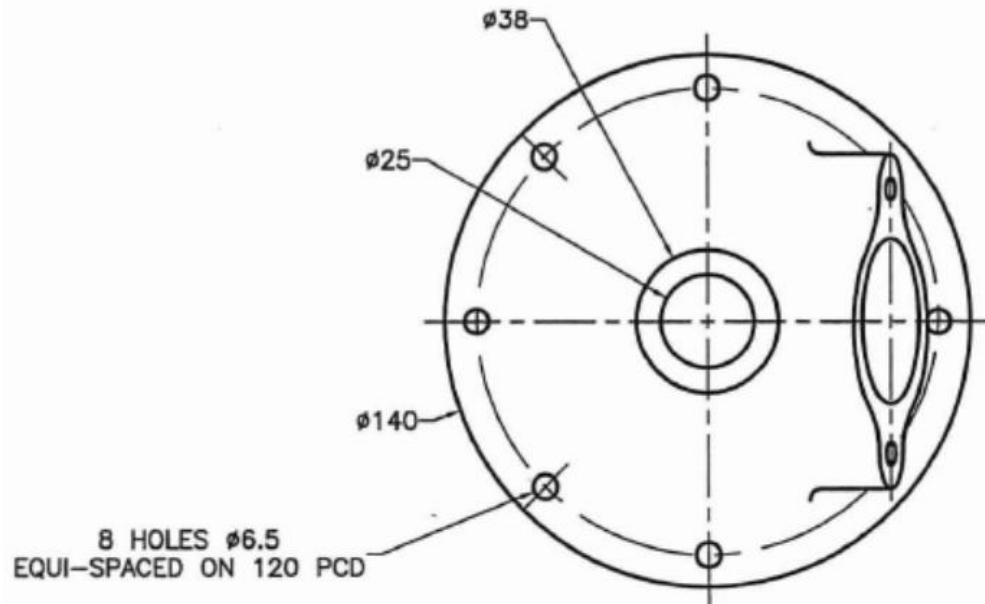
## Basics

- Solid-Works is a 3D solid modeling package which allows users to develop full solid models in a simulated environment for both design and analysis
- To produce simple and complex parts, assemblies, and drawings.
- It saves time, effort, and money that would otherwise be spent prototyping the design.

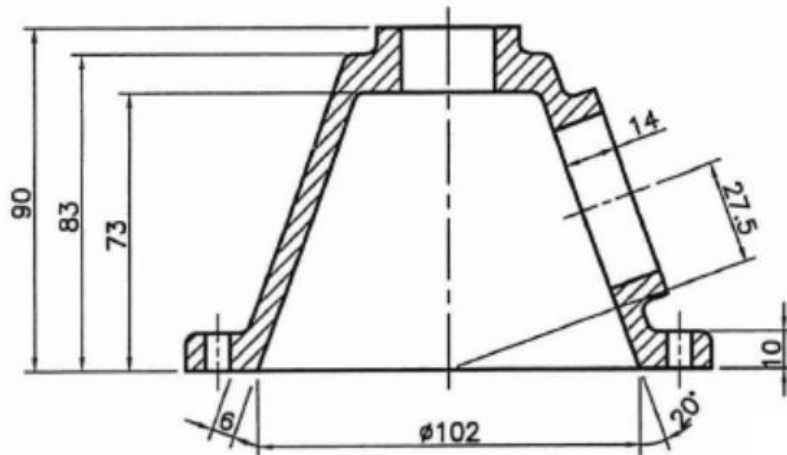
# Problem for Part Modeling



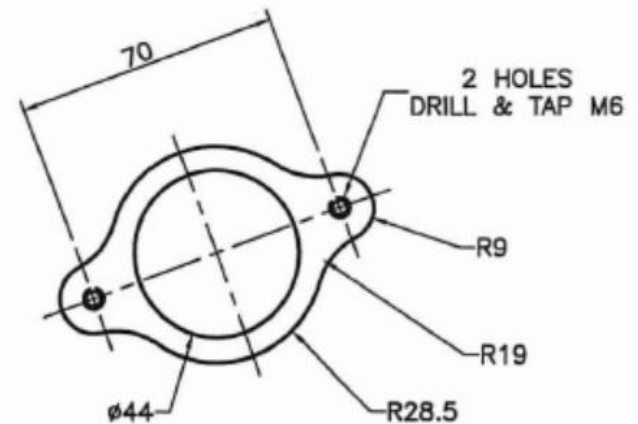
# Problem for Part Modeling



8 HOLES  $\phi 6.5$   
EQUI-SPACED ON 120 PCD

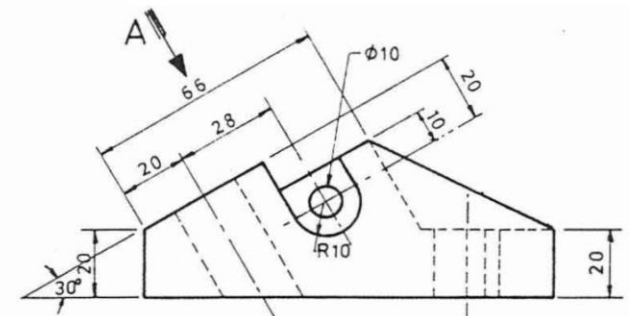
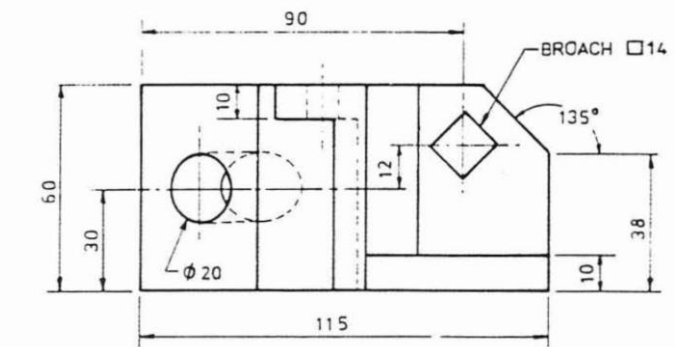
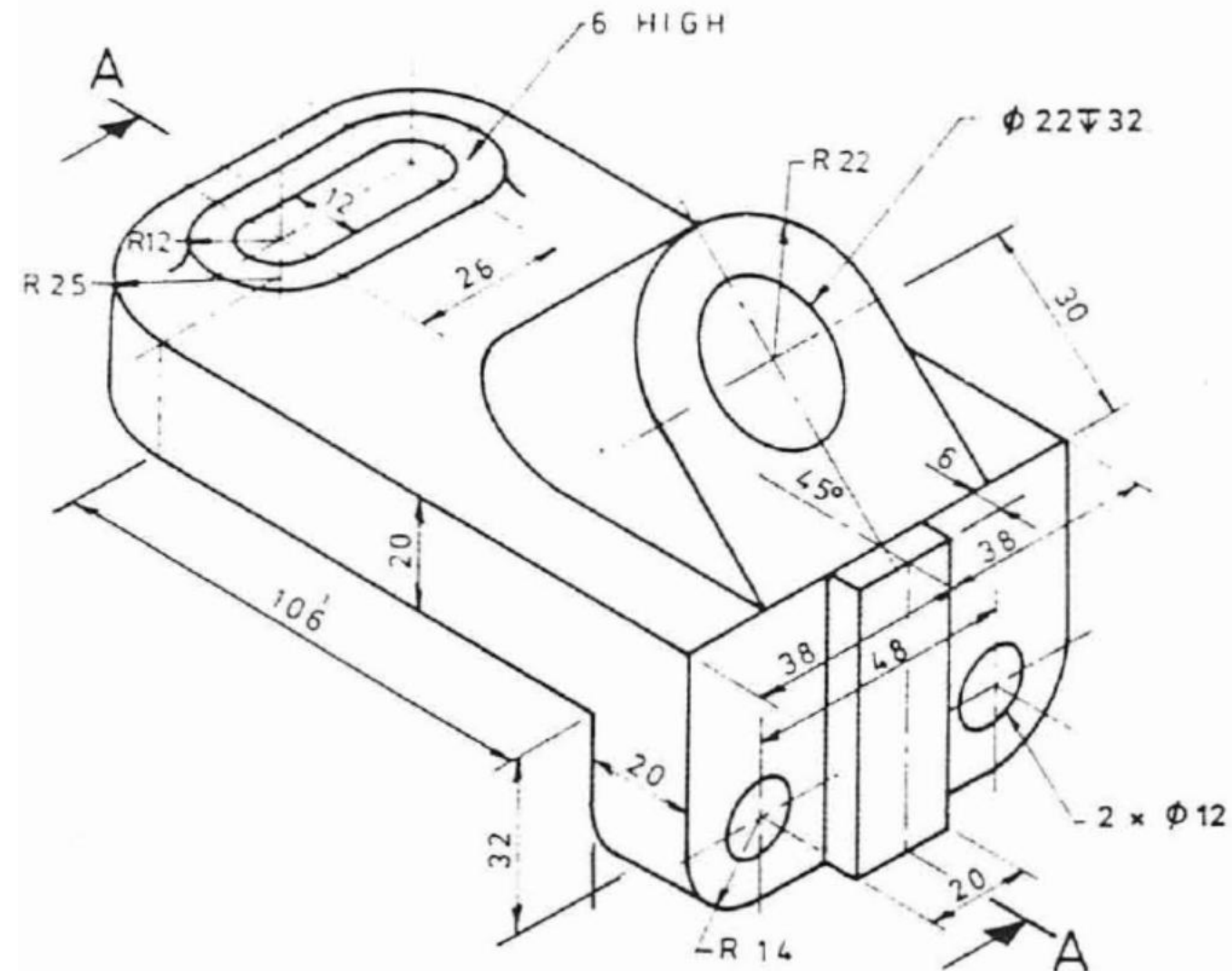


ALL FILLET RADII TO BE R3 U.N.O



2 HOLES  
DRILL & TAP M6

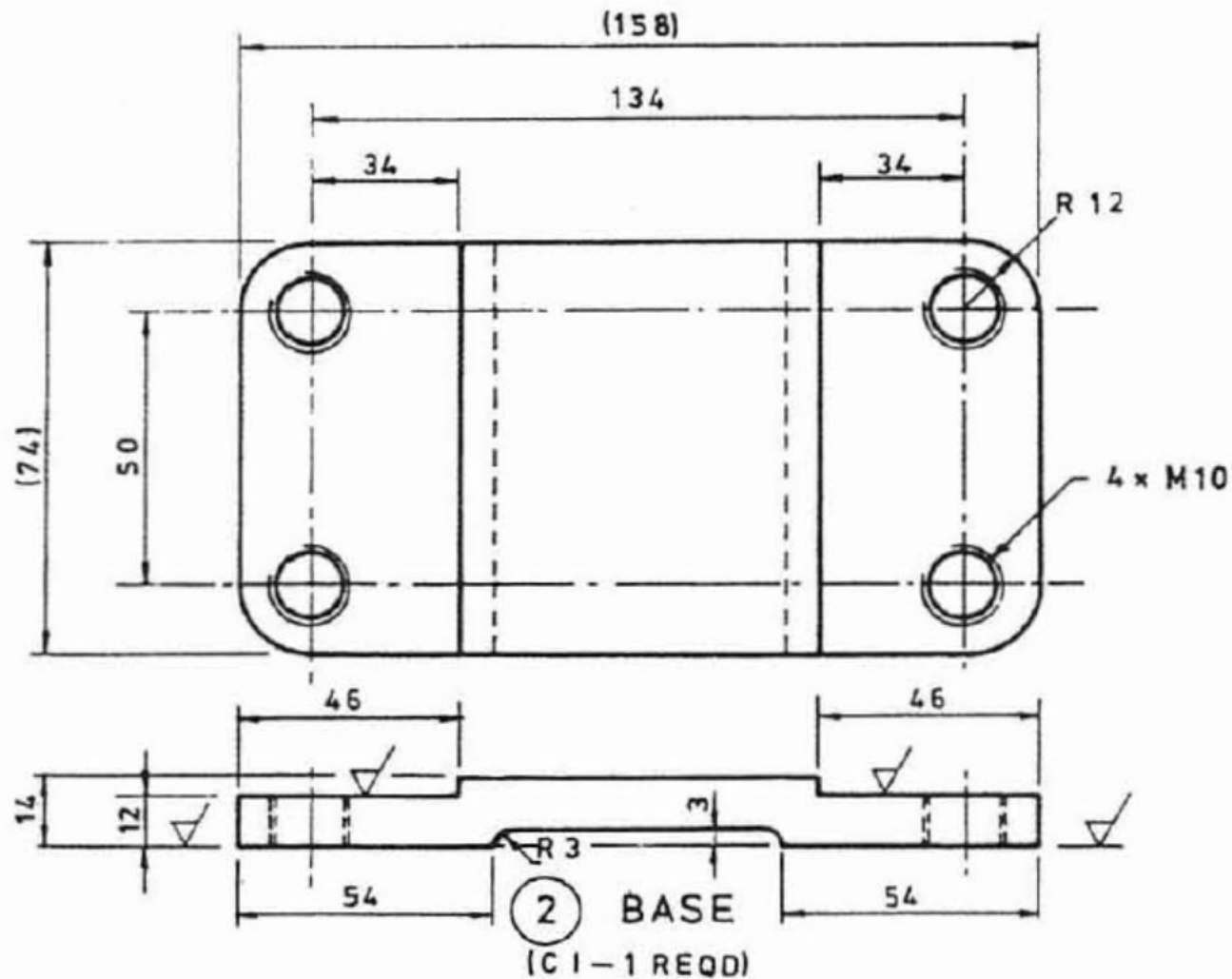
# Problem for Part Modeling





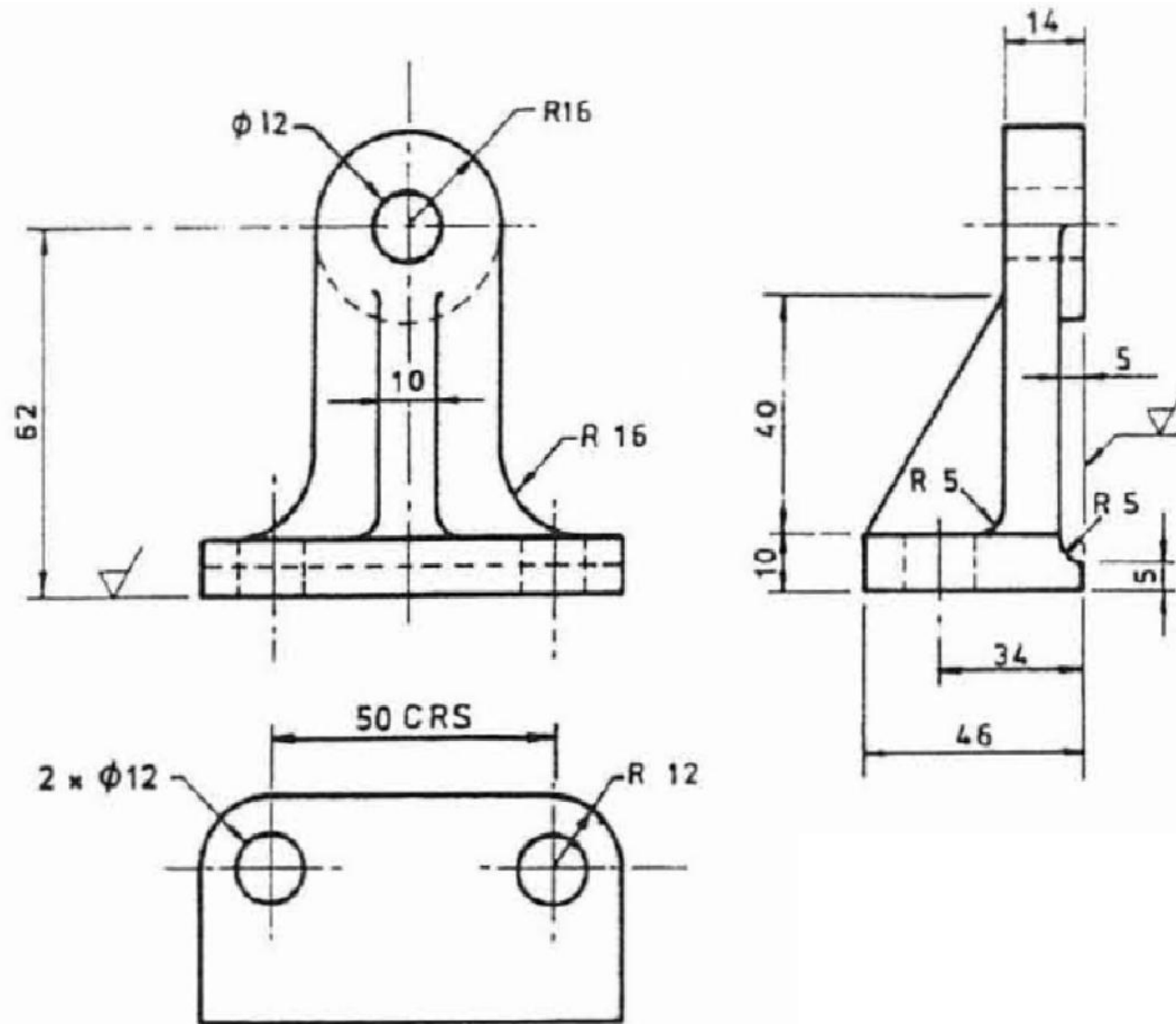
# Problem for Assembly Modeling

## Component-1: Base



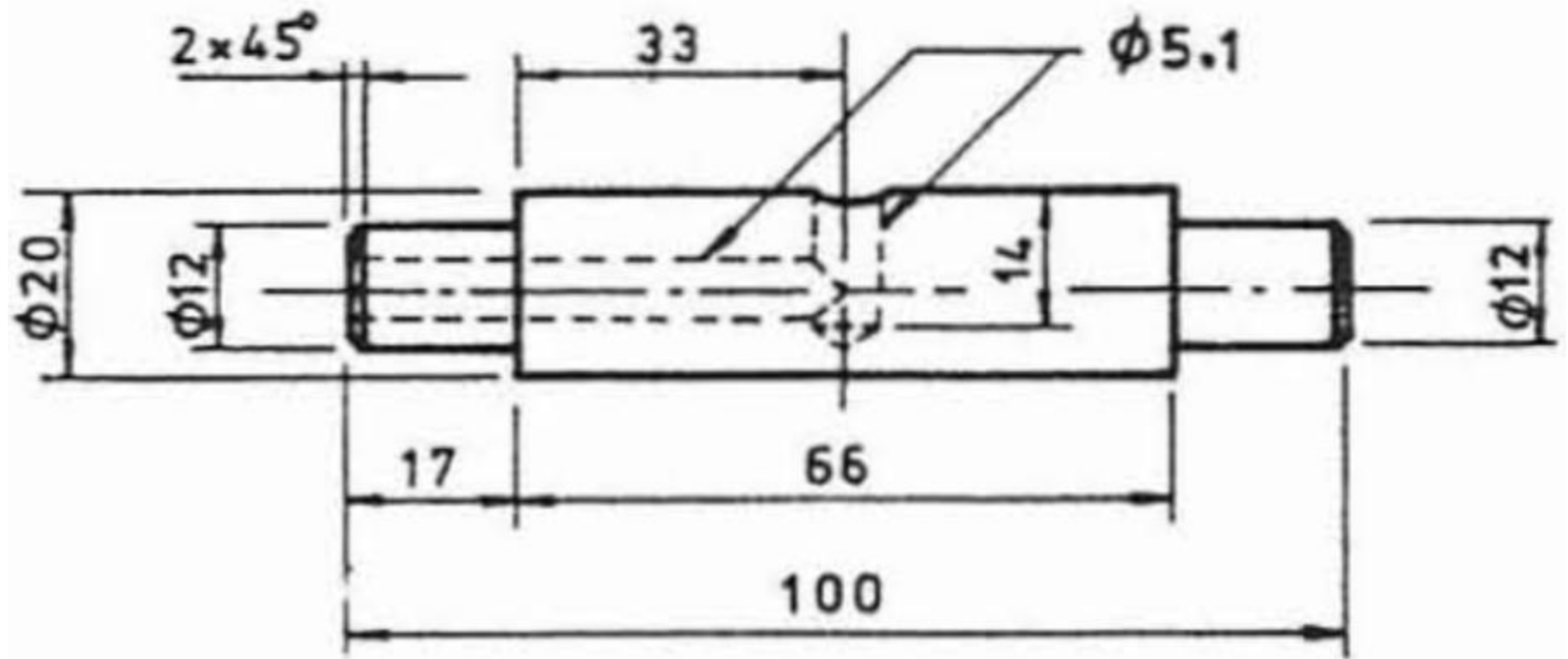
# Problem for Assembly Modeling

## Component-2: Bracket



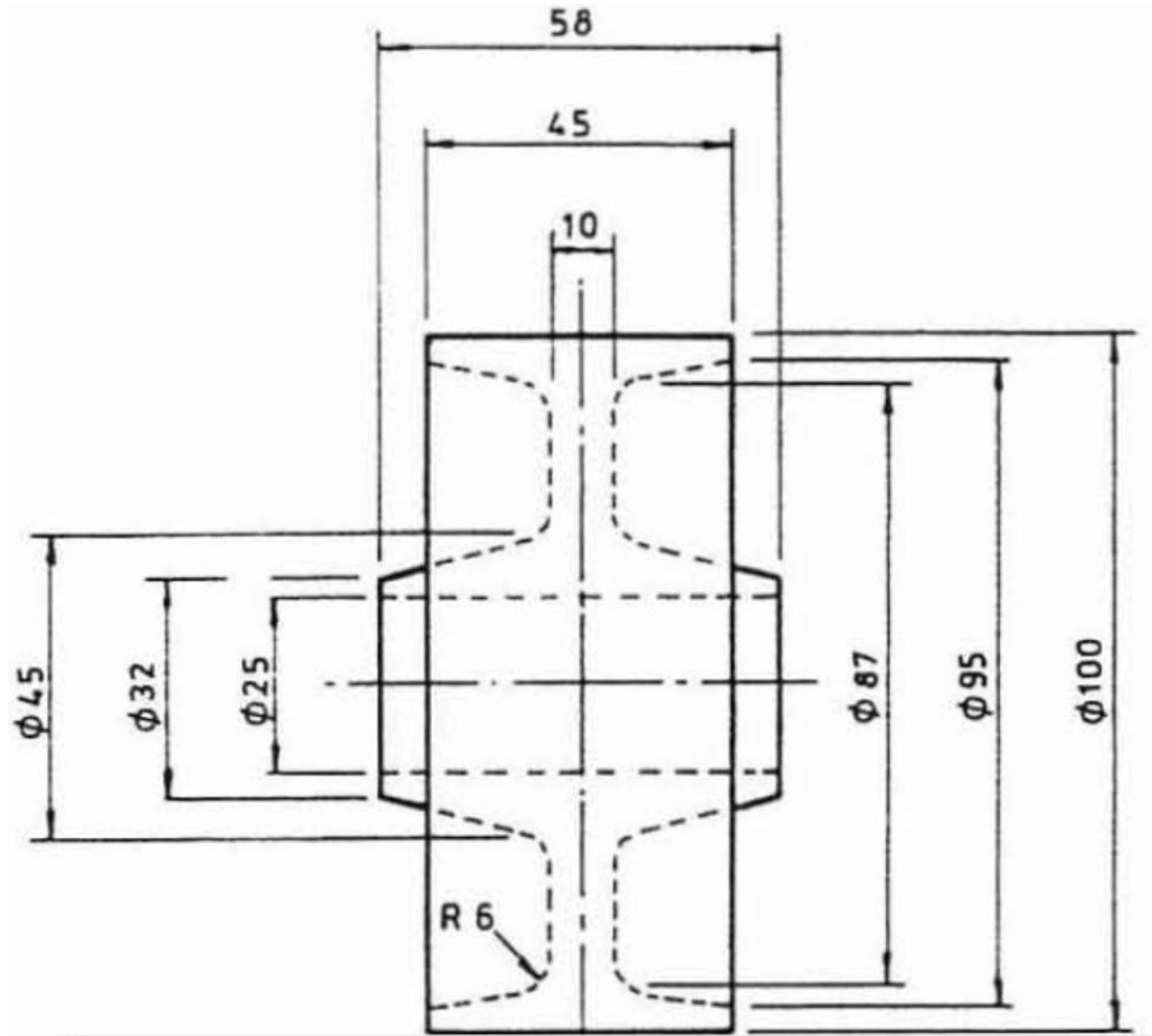
# Problem for Assembly Modeling

## Component-3: Spindle



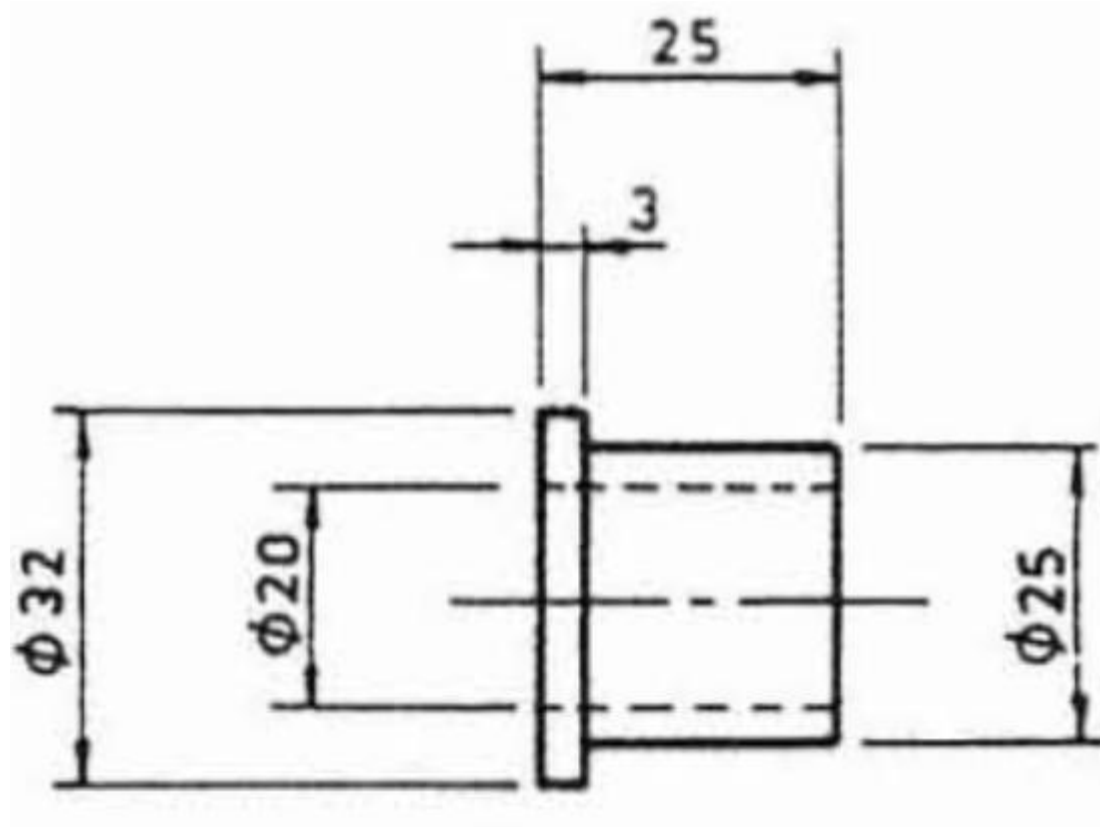
# Problem for Assembly Modeling

## Component-4: Roller



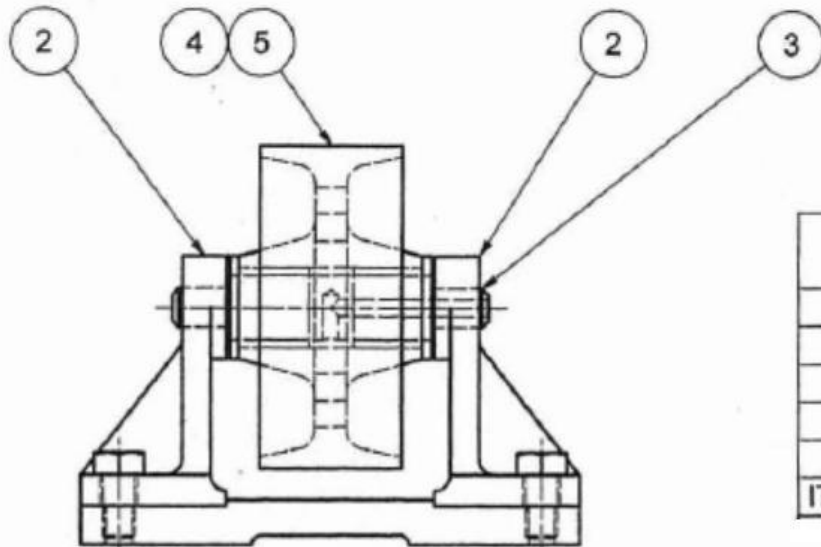
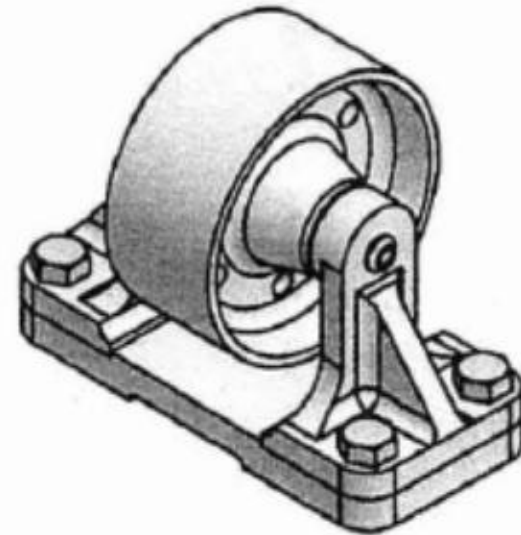
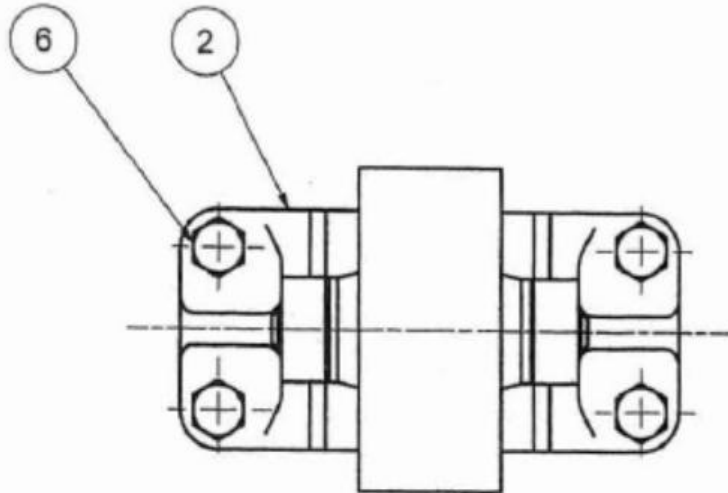
# Problem for Assembly Modeling

## Component-5: Bush



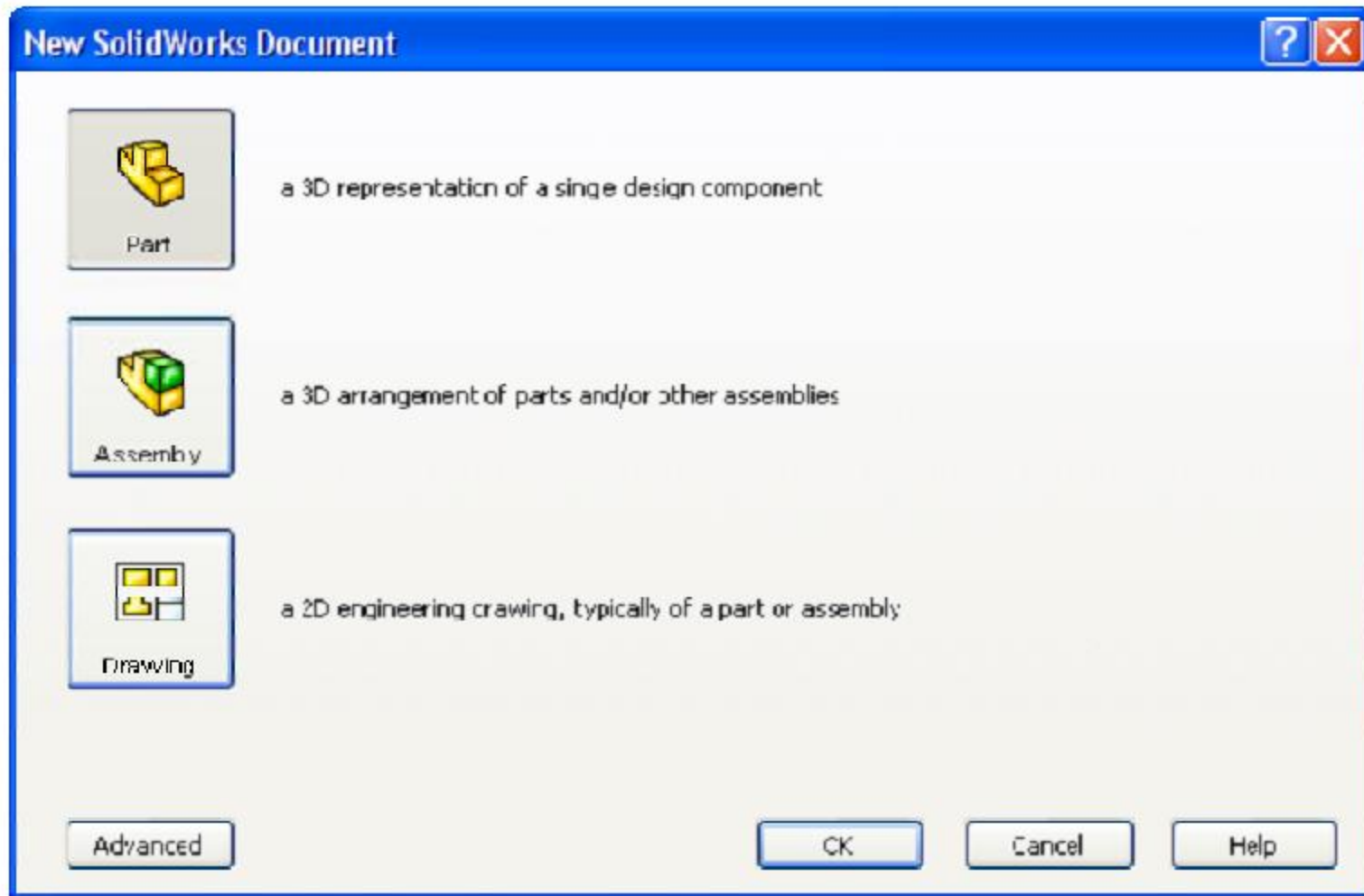
# Problem for Assembly Modeling

## Assembly

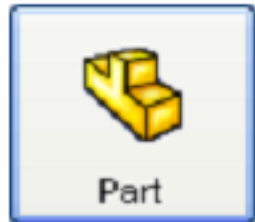


6	Hex Bolt	Steel, Mild	4	ANSI B18.2.3.5M - M10 x 1.5 x 20
5	ROLLER	Cast Steel	1	RA_01-3
4	BUSH	Bronze, Soft Tin	2	RA_01-5
3	SPINDLE	Alloy Steel	1	RA_01-4
2	BRACKET	Cast Iron	2	RA01-2
1	BASE	Cast Iron	1	RA_01-1
ITEM	DESCRIPTION	MATERIAL	QTY	PART NUMBER

# Solid-Works Components



# Solid-Works Components - PARTS



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



# Solid-Works Components - Assembly



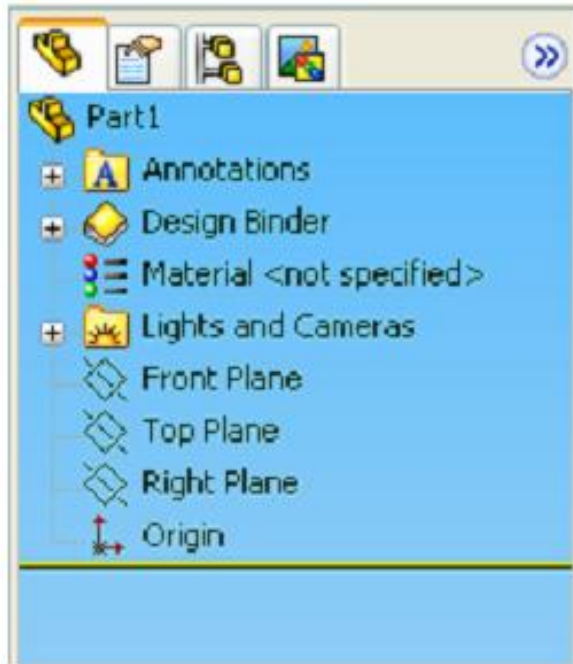
- The second component is the assembly. Assemblies are collections of parts which are assembled in a particular fashion using mates (constraints).
- Any complex model will usually consist of one, or many assemblies.

# Solid-Works Components - Assembly

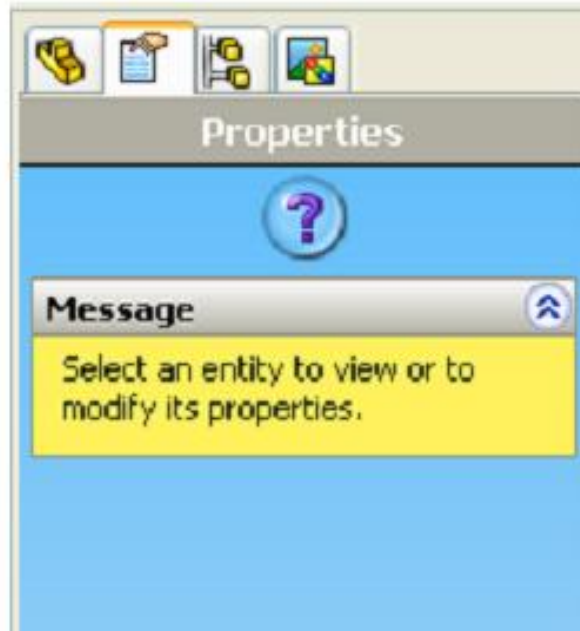
- The third, and final component in SolidWorks is the Drawing.
- A drawing is the typical way to represent a 3D model such that any engineer (or manufacturer) can recreate your part.
- Drawings are important because they provide a standard way of sharing your design.



# SolidWorks



Feature Manager, lists all features that have been created within your model

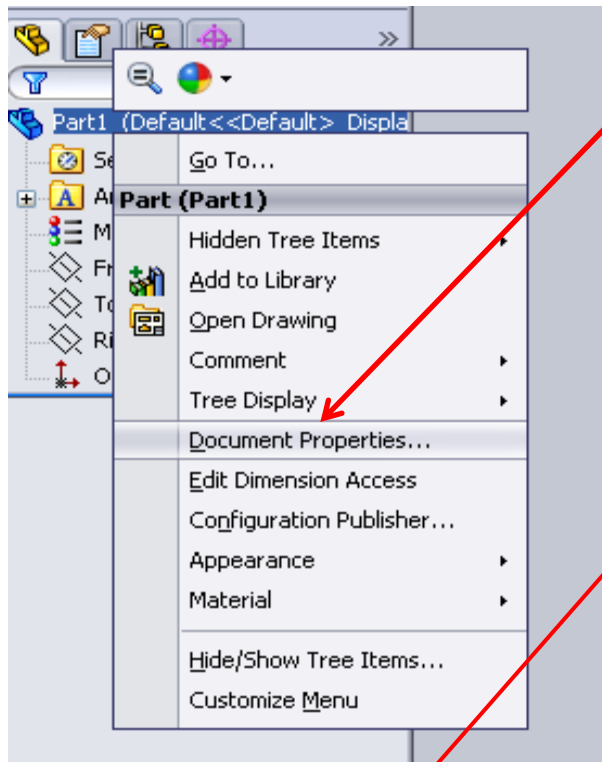


Property Manager, allows you to adjust the properties of various entities either during construction, or once it has been created

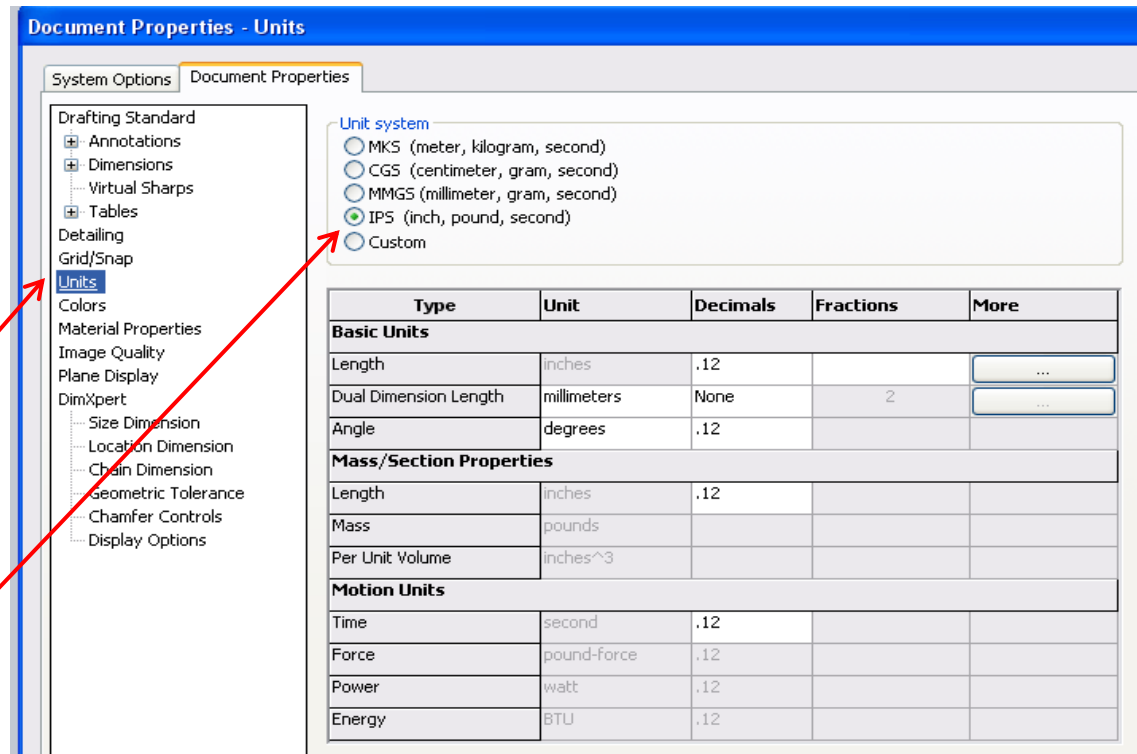


Configuration Manager and is used to set up different view configurations such as exploded views or 3D section views.

# Solid Works- Tool Bar



In feature manager, RIGHT CLICK on PART1, then choose DOCUMENT PROPERTIES. Always remember to do this FIRST when making a new part.

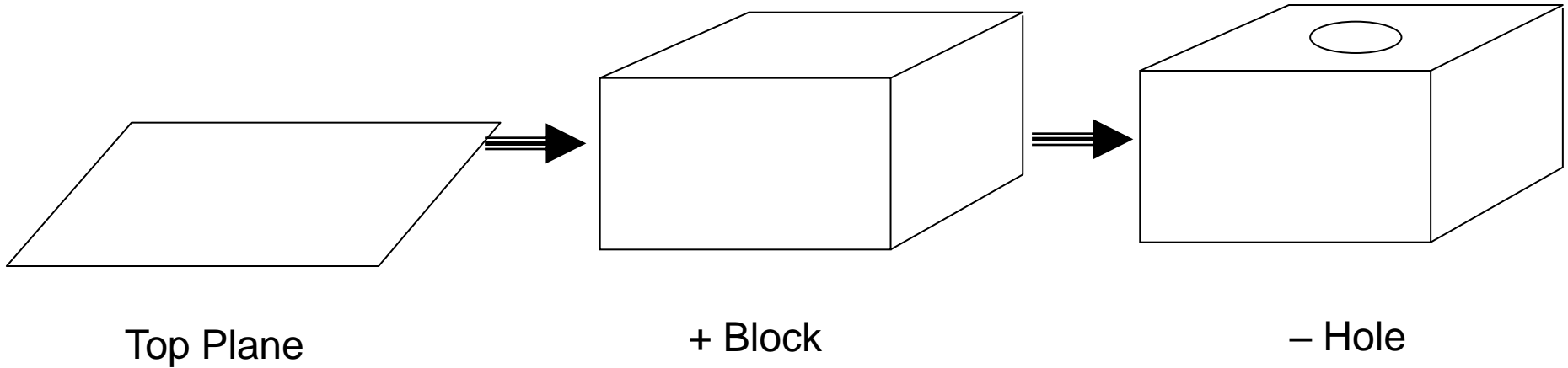


Click on UNITS, then choose IPS for inches.

Then choose OK

## (2) 3-D Object Creation Procedure

By Creating Features



Each Feature:

- 2-D Sketching
- 3-D Formation

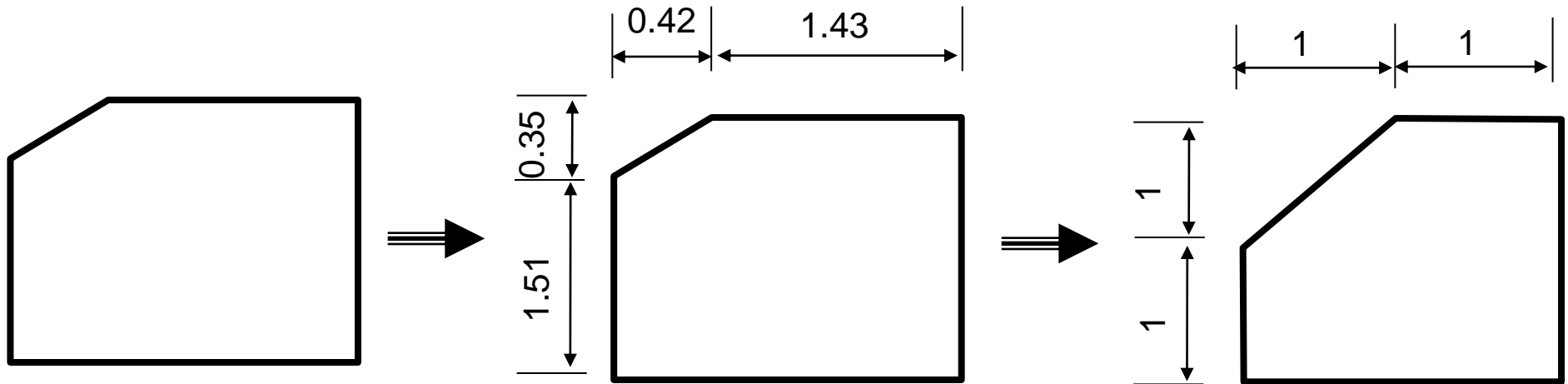
### (3) 2-D Sketching

#### Parametric Modeling

##### (a) Procedure

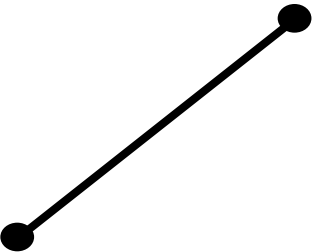
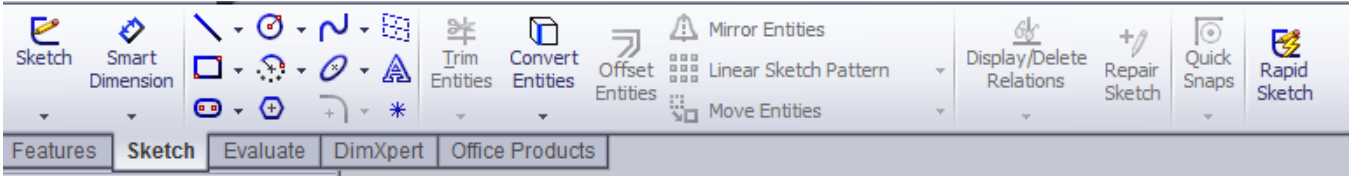
- Sketch the geometry
- Dimension the geometry
- Modify the dimension values

e.g.,

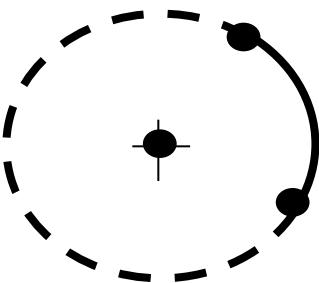


**(b) 2-D Object Creation Methods**

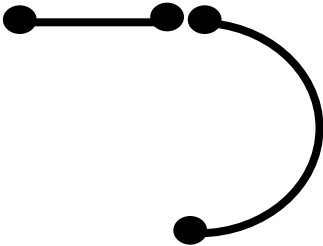
Menu: Tools->Sketch Entities



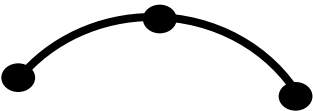
Line



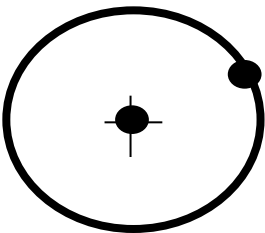
Centerpoint Arc



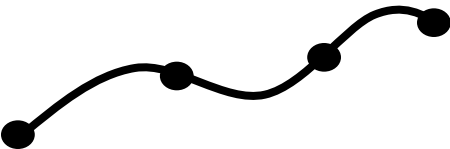
Tangent Arc



3 Point Arc



Circle



Spline



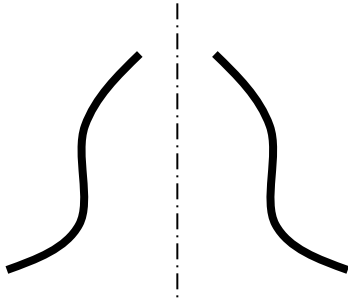
Rectangle



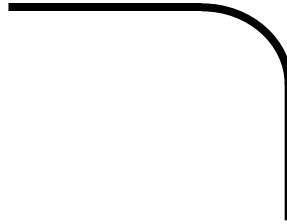
Point

## (c) Additional 2-D Object Creation Methods

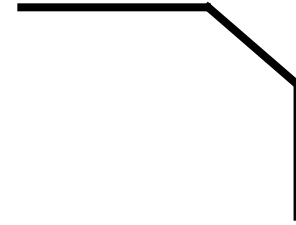
Menu: Tools->Sketch Tools



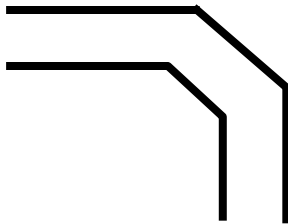
Mirror



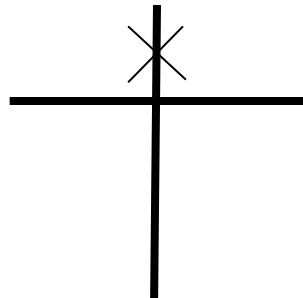
Fillet



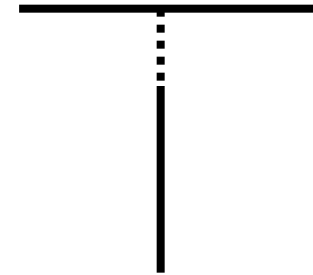
Chamfer



Offset Entities



Trim

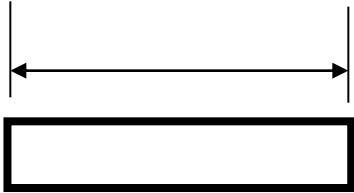


Extend

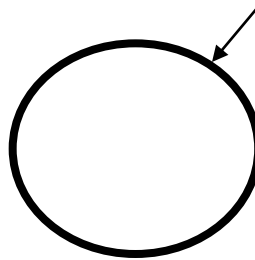


### (d) Dimensioning

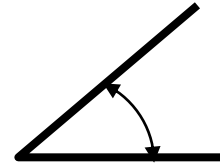
Menu: Tools->Dimensions->Smart



Linear



Radial



Angular

### (e) Relations

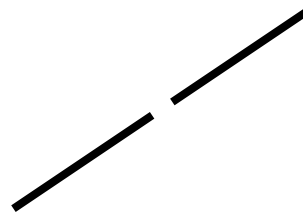
Menu: Tools->Relations



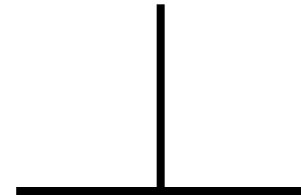
Horizontal



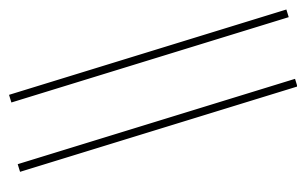
Vertical



Collinear



Perpendicular



Parallel



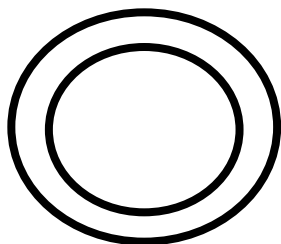
Tangent



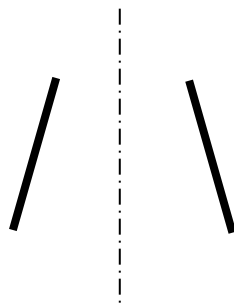
Midpoint



Coincident



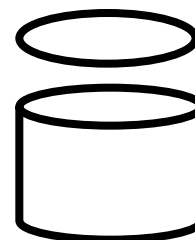
Concentric



Symmetric

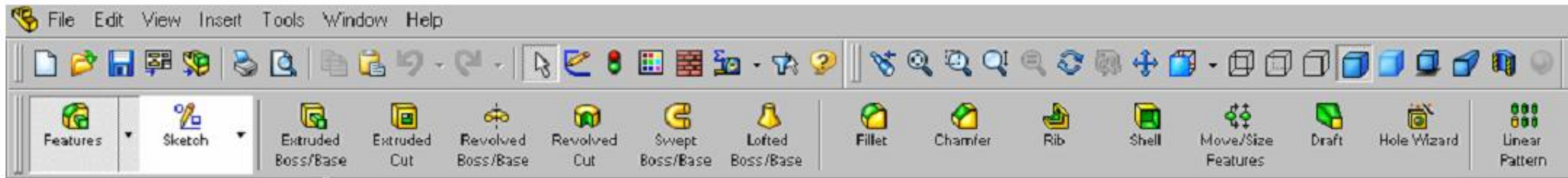


Equal



Coradial

# 3-D features tool bar

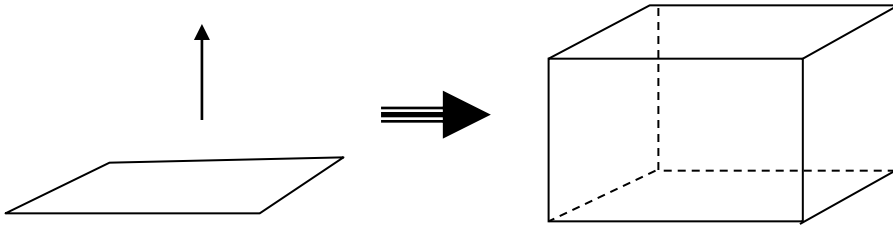


## Features:

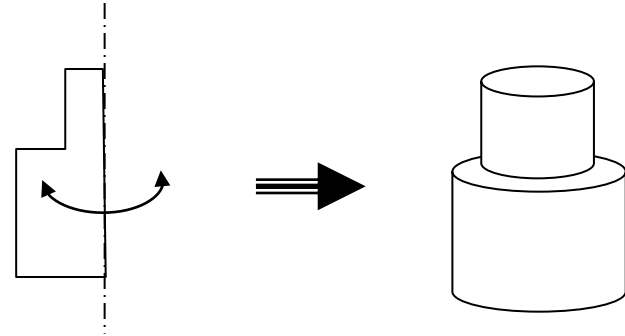
Extrude Boss/Base - Cut  
Revolved Boss/Base - Cut  
Swept Boss/Base - Cut  
Lofted Boss/Base – Cut  
Fillet, Chamfer, Rib, Shell  
Reference Geometry  
Curves

## (4) Features

Menu: Insert->Boss/Base

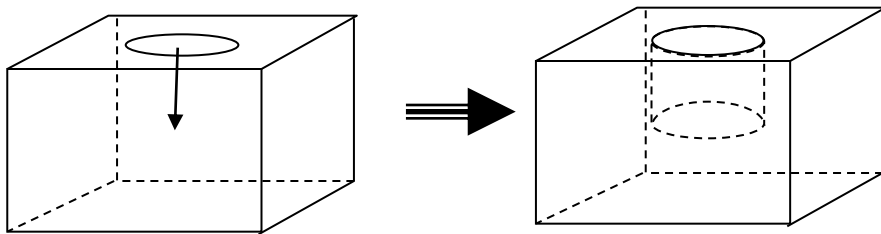


Extruded Boss/Base

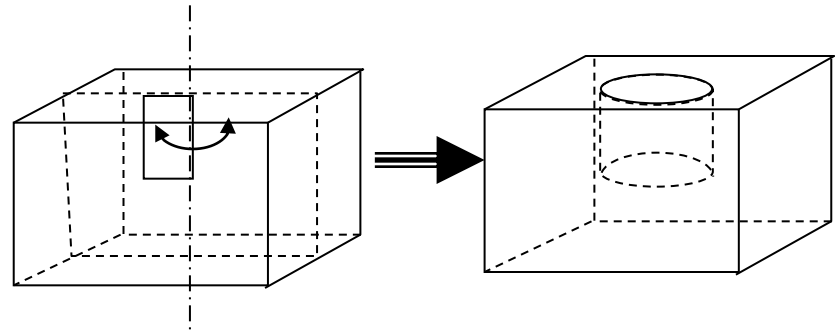


Revolved Boss/Base

Menu: Insert->Cut

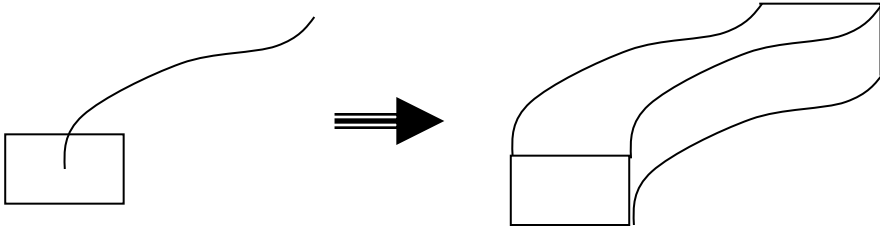


Extruded Cut

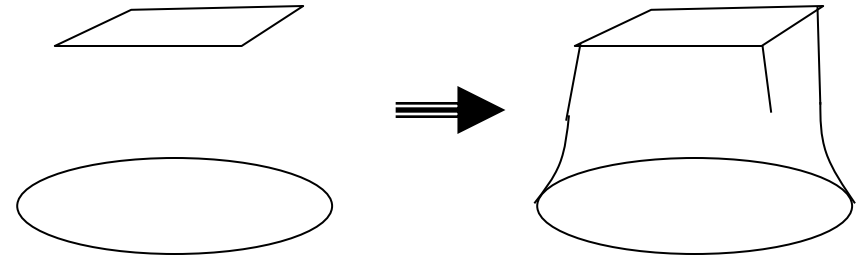


Revolved Cut

Menu: Insert->Boss/Base

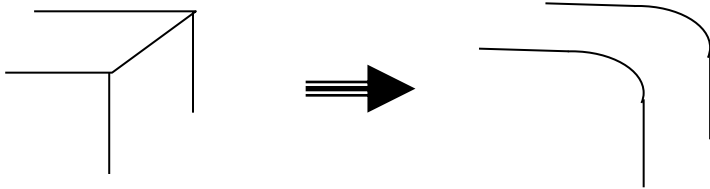


Sweep Boss/Base

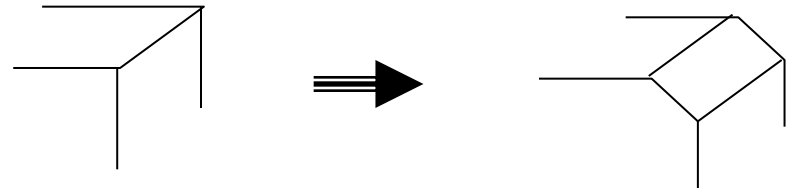


Lofted Boss/Base

Menu: Insert->Features

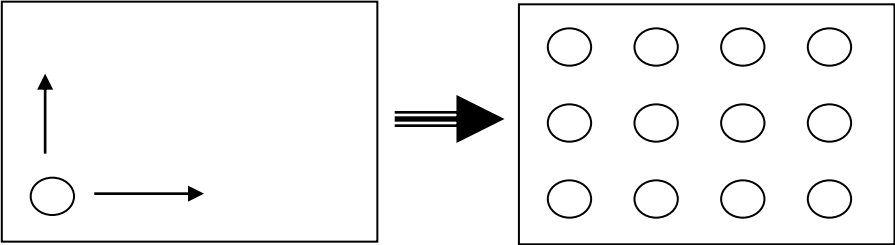


Fillet

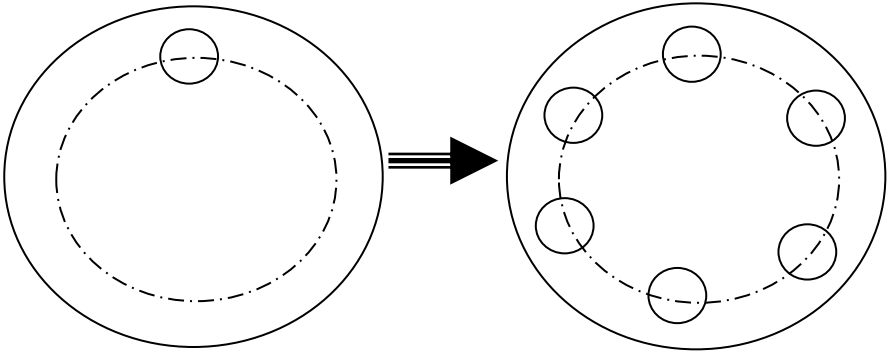


Chamfer

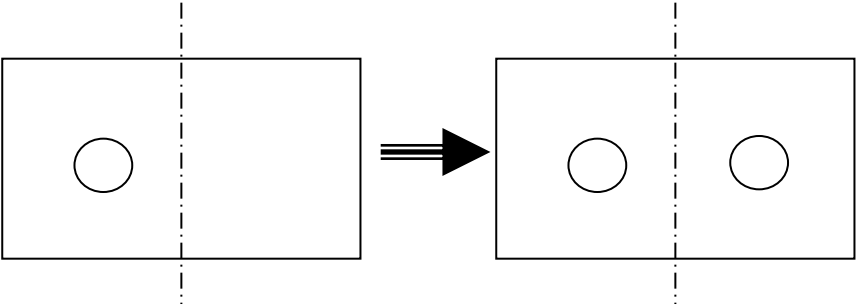
Menu: Insert->Pattern/Mirror



Linear Pattern



Circular Pattern



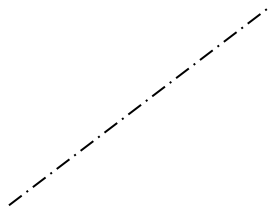
Mirror

## (5) Reference Geometry

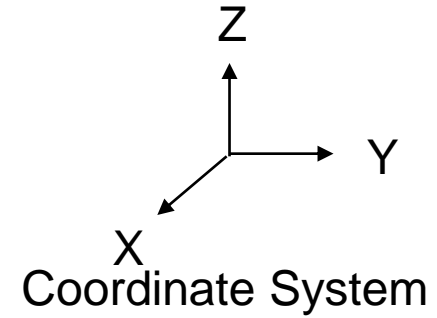
Menu: Insert->Reference Geometry



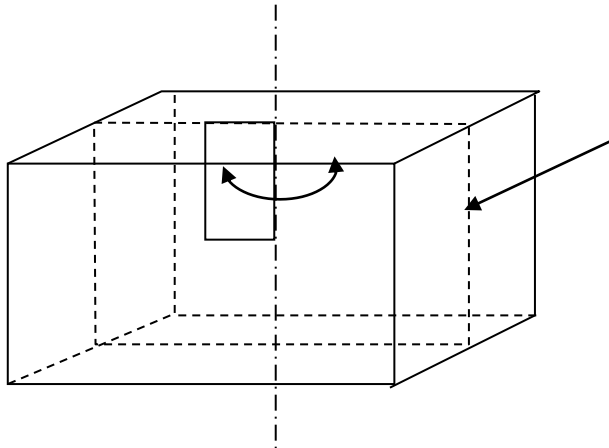
Plane



Axis



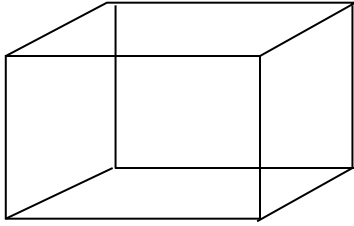
e.g.,



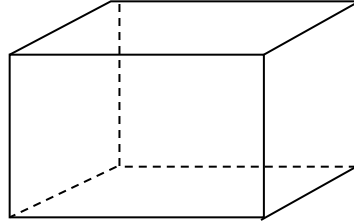
A reference plane for creating a sketch of  
revolved cut feature

## (6) Viewing

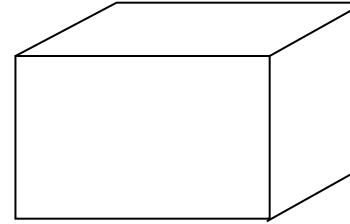
Menu: View->Display



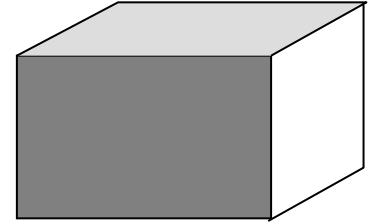
Wireframe



Hidden Lines  
Visible

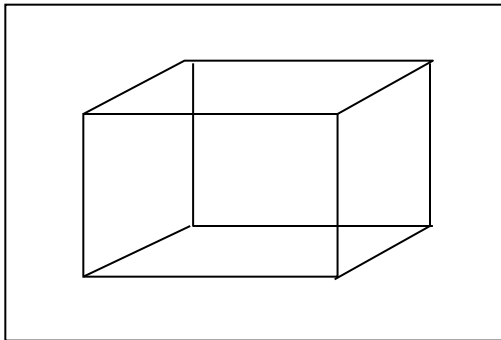


Hidden Lines  
Removed

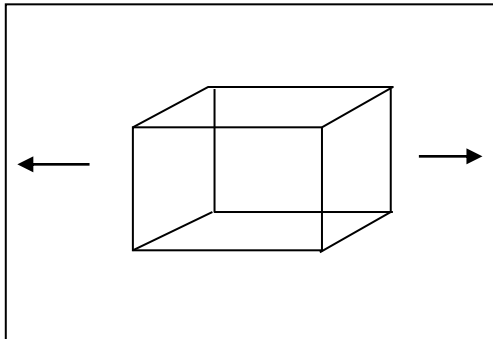


Shaded With  
Edges

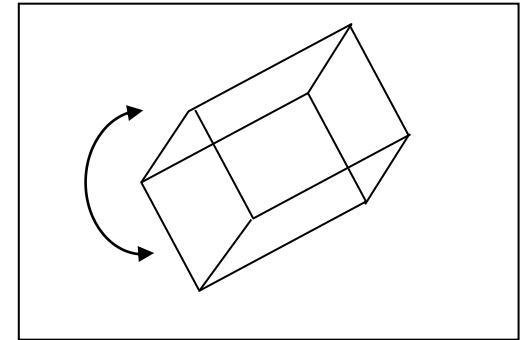
Menu: View->Modify



Zoom to Fit



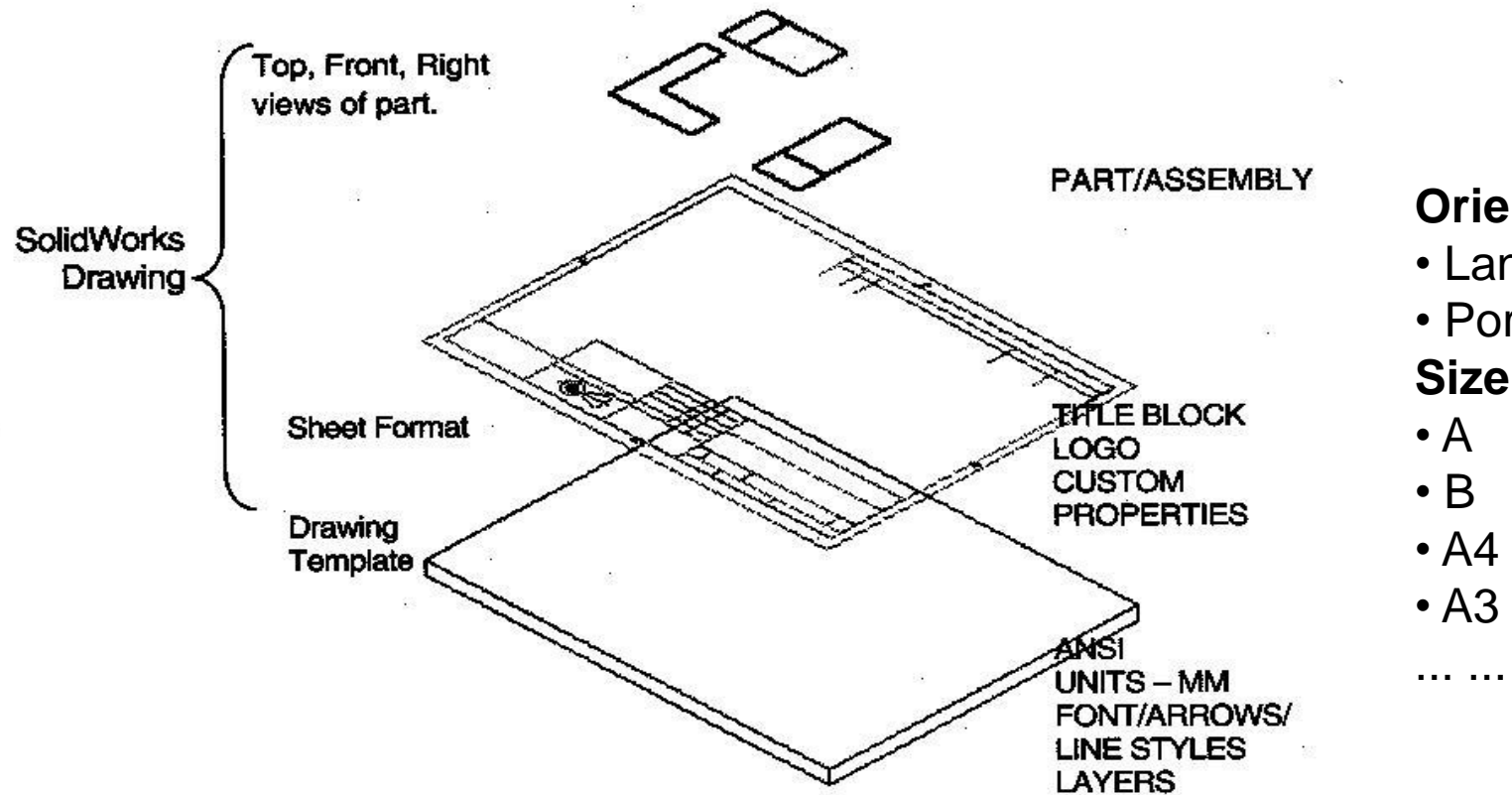
Pan



Rotate



## (1) Drawing Template and Drawing Format



Menu: File->New->Draw

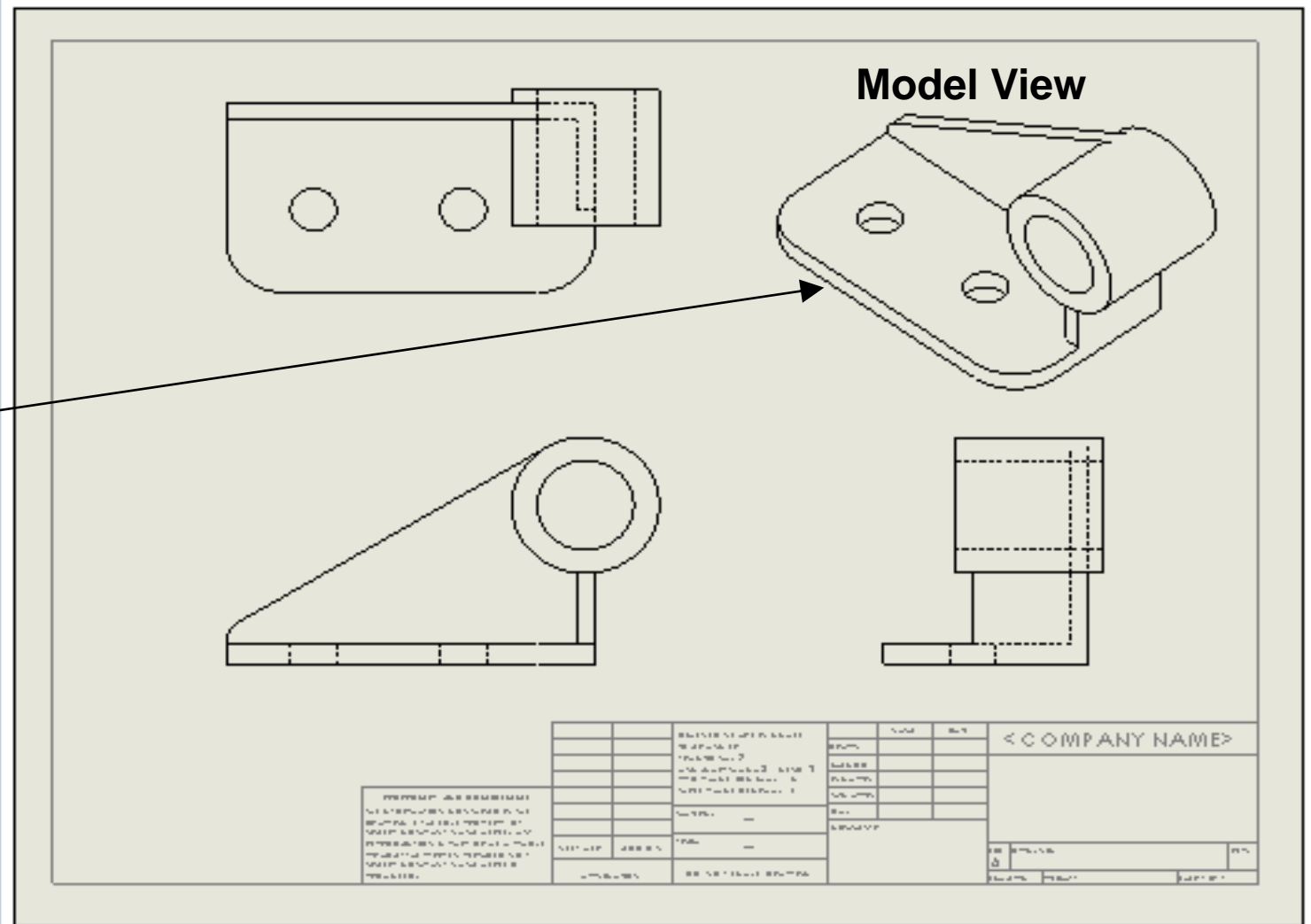
Menu: Insert->Drawing View

- [illegible]

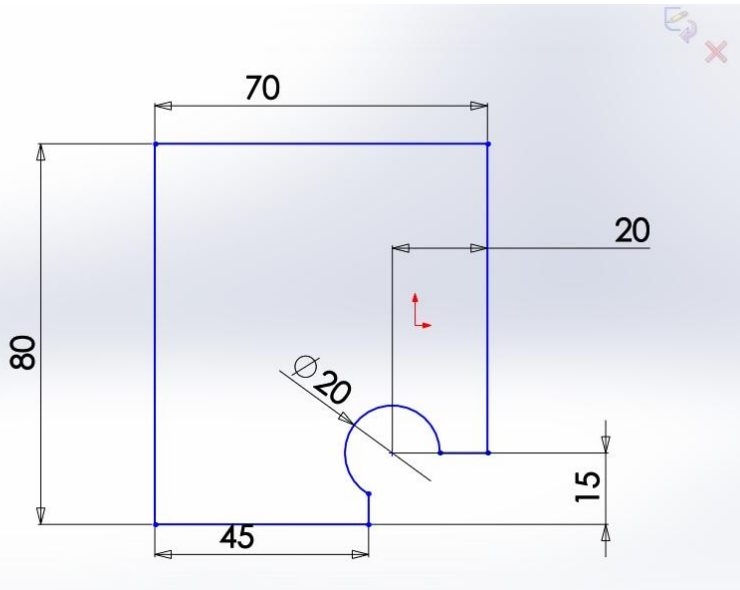
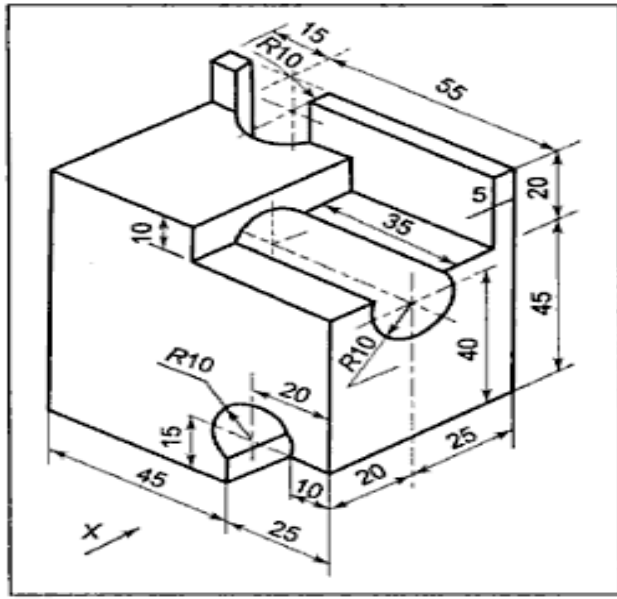
- Model View



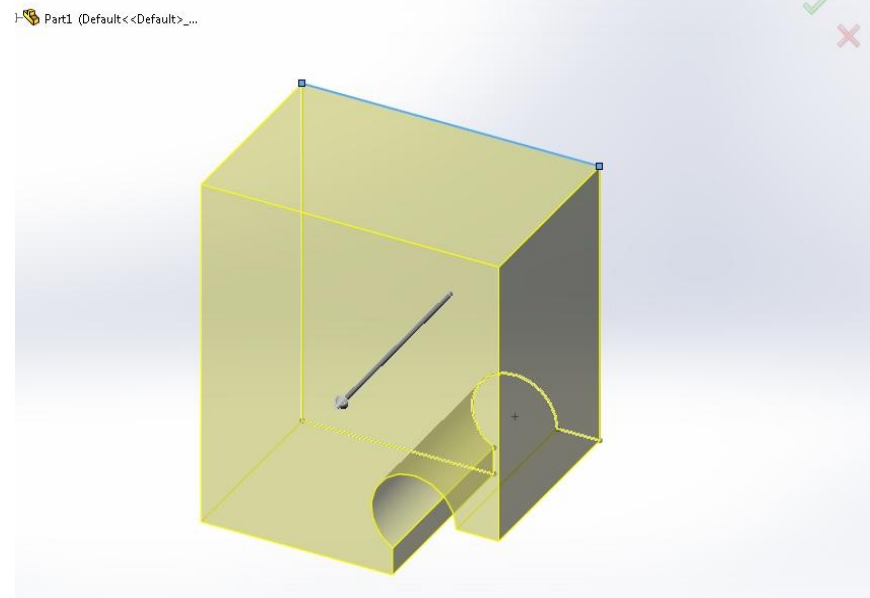
Orientation



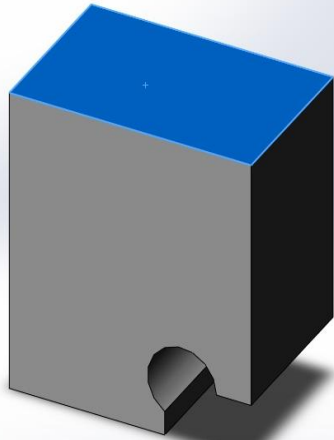
## Exercise 1:



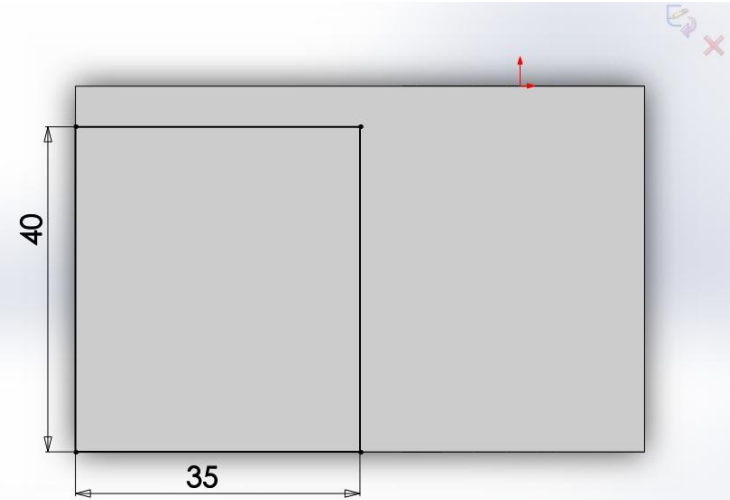
## Creating the base geometry sketch



## Extruding this sketch

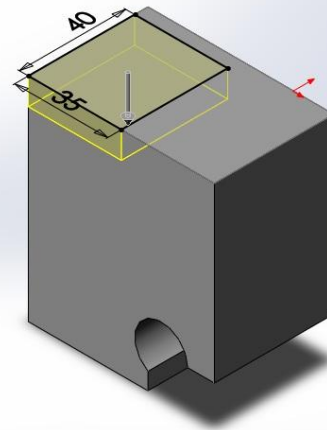


Selecting the top surface  
to make next feature

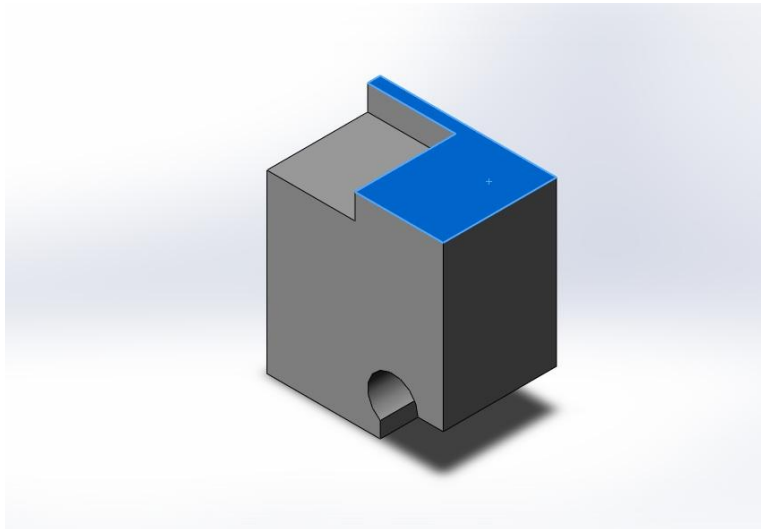


Drawing the next feature on this  
surface

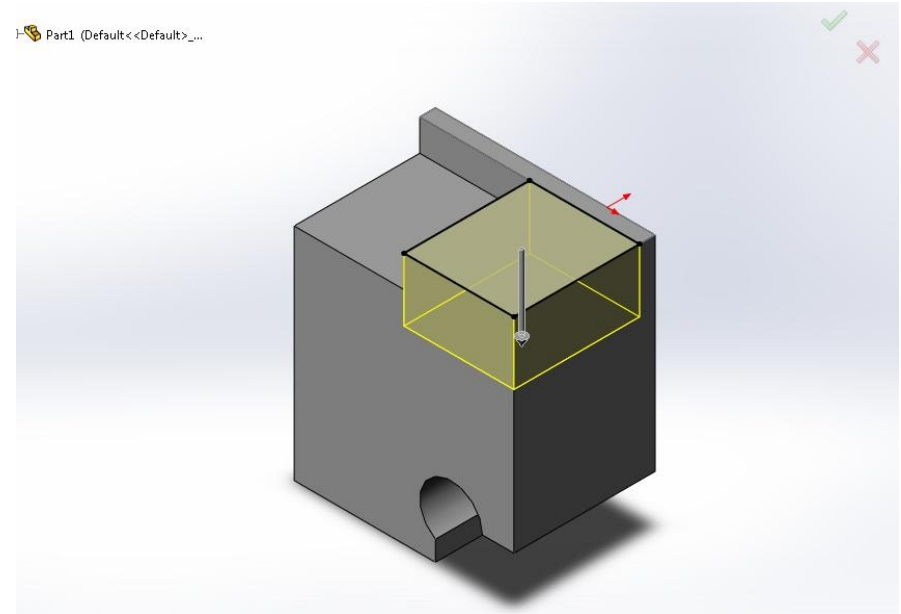
Part1 (Default<<Default>>\_...



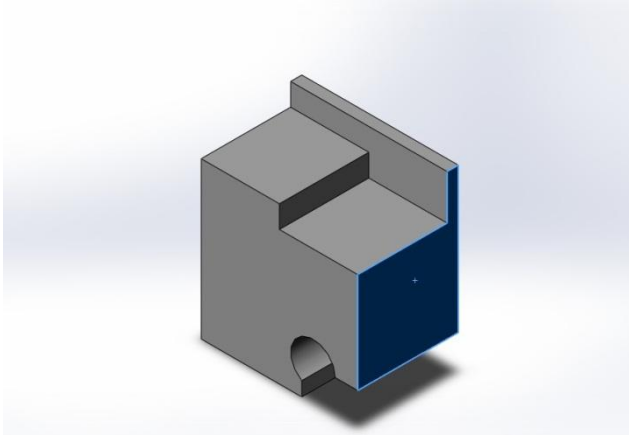
Making use of the Extrude cut



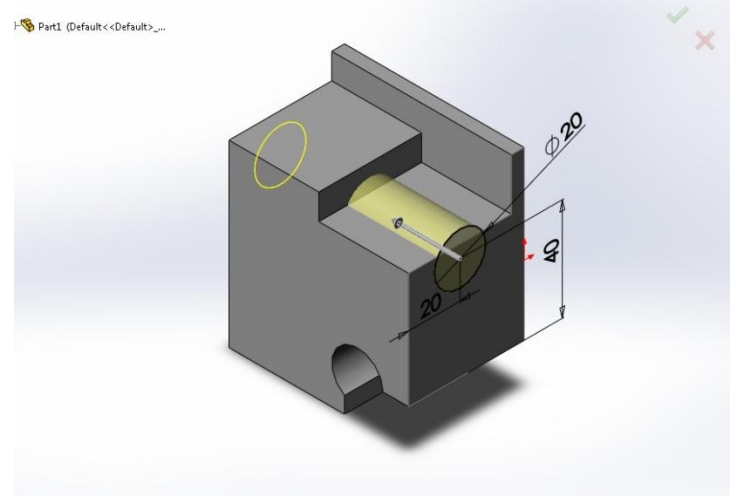
Selecting the top surface for next feature



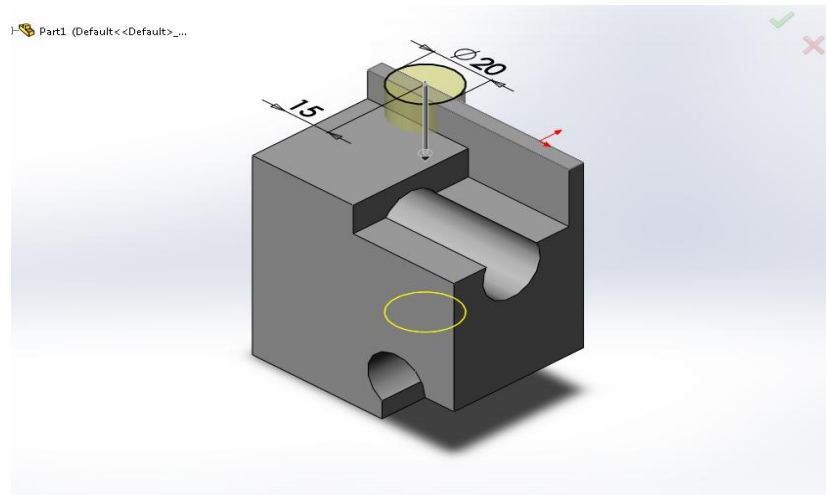
Making use of extrude cut on rectangle sketched on this surface



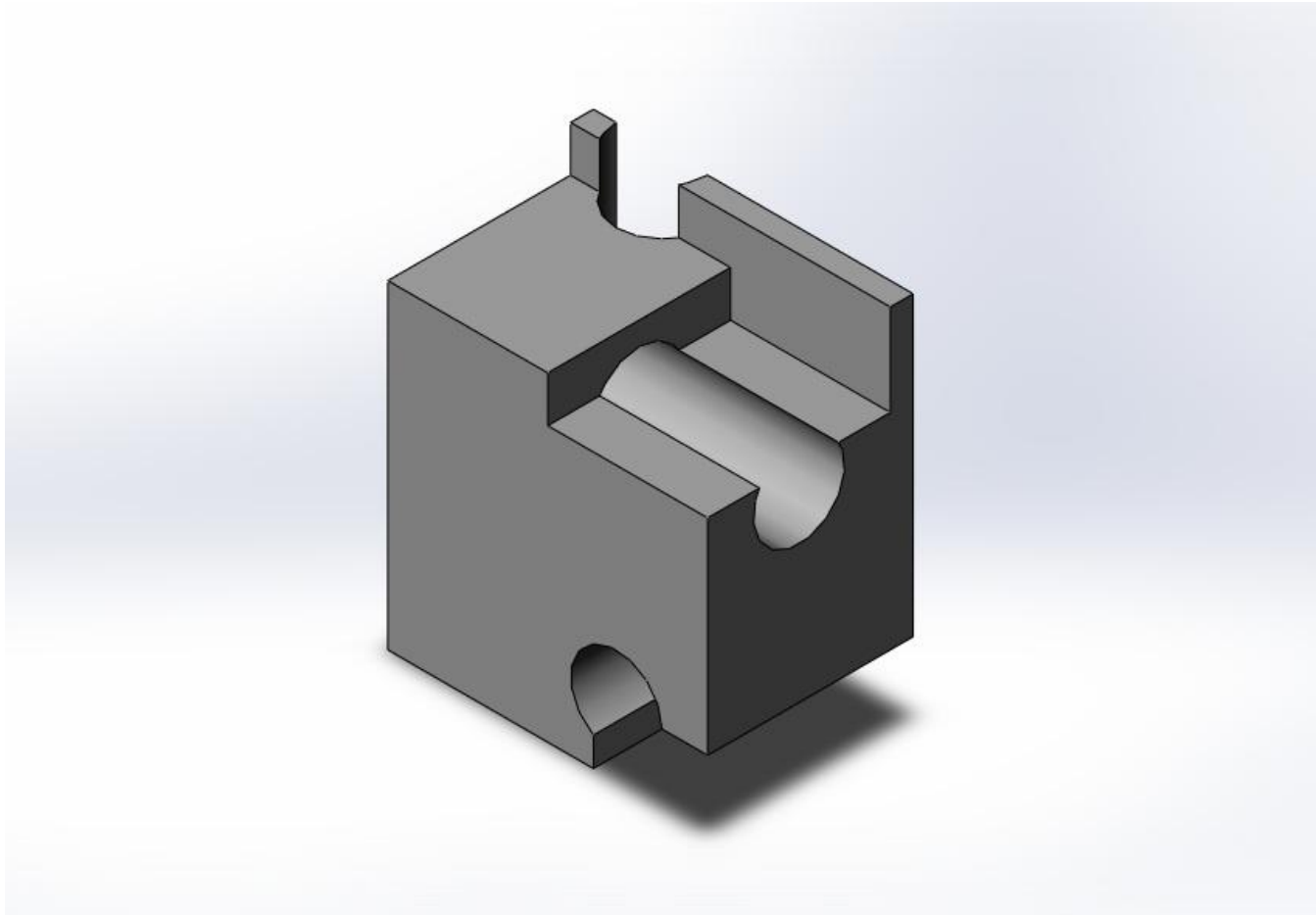
Selecting the surface for next feature



Drawing circle on the surface and creating a through hole



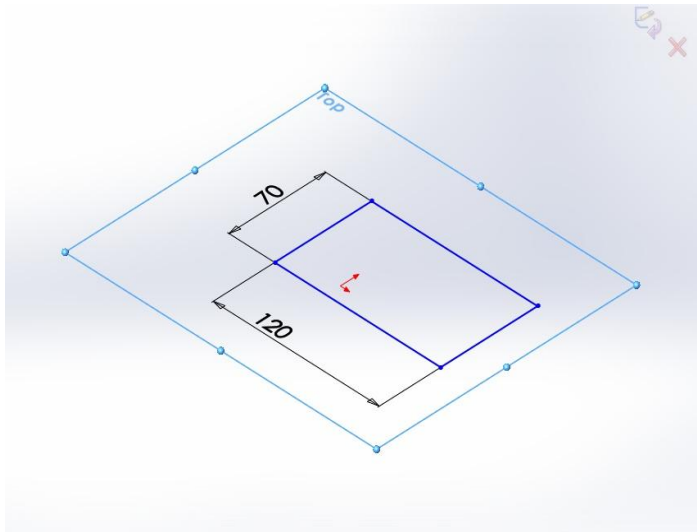
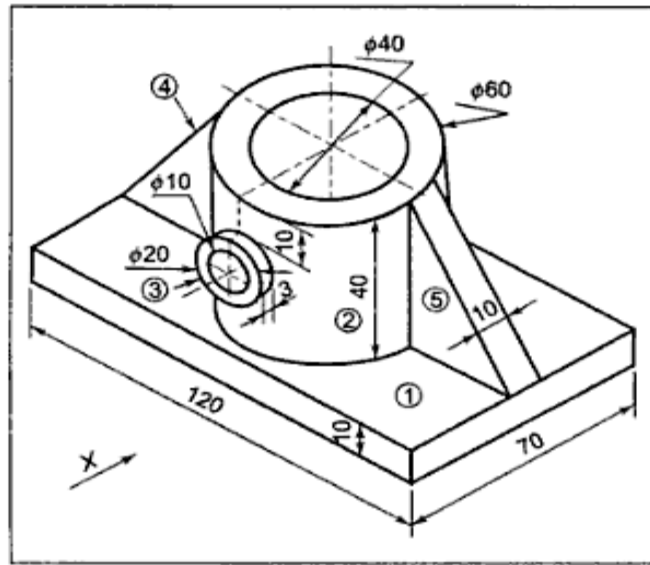
Similarly making another hole on top surface



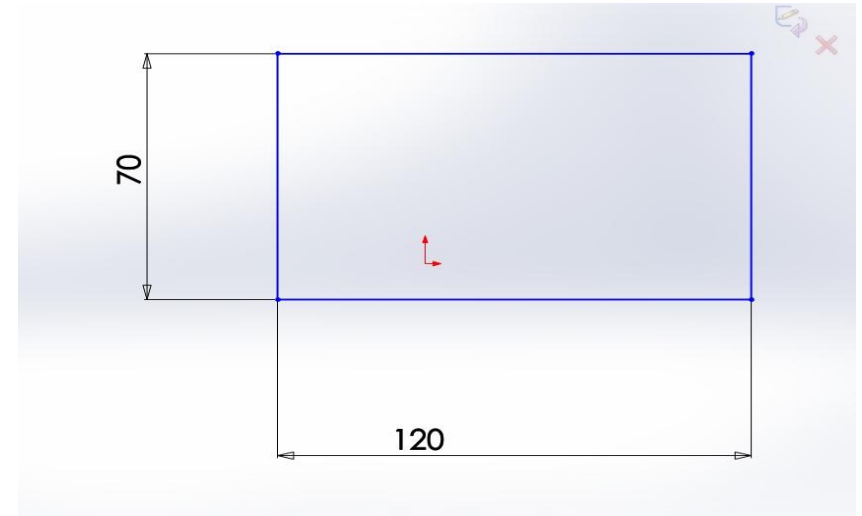
Final component



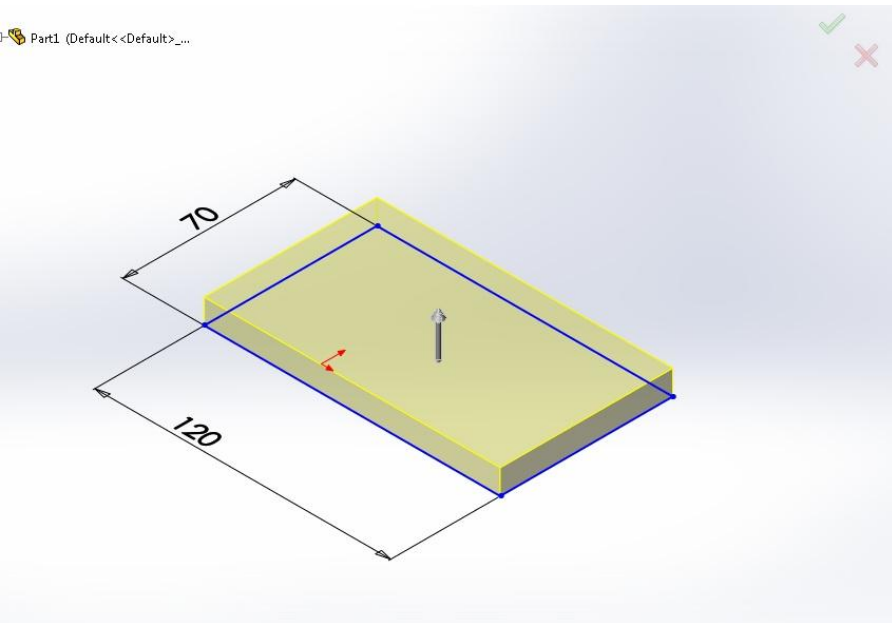
## Exercise 2:



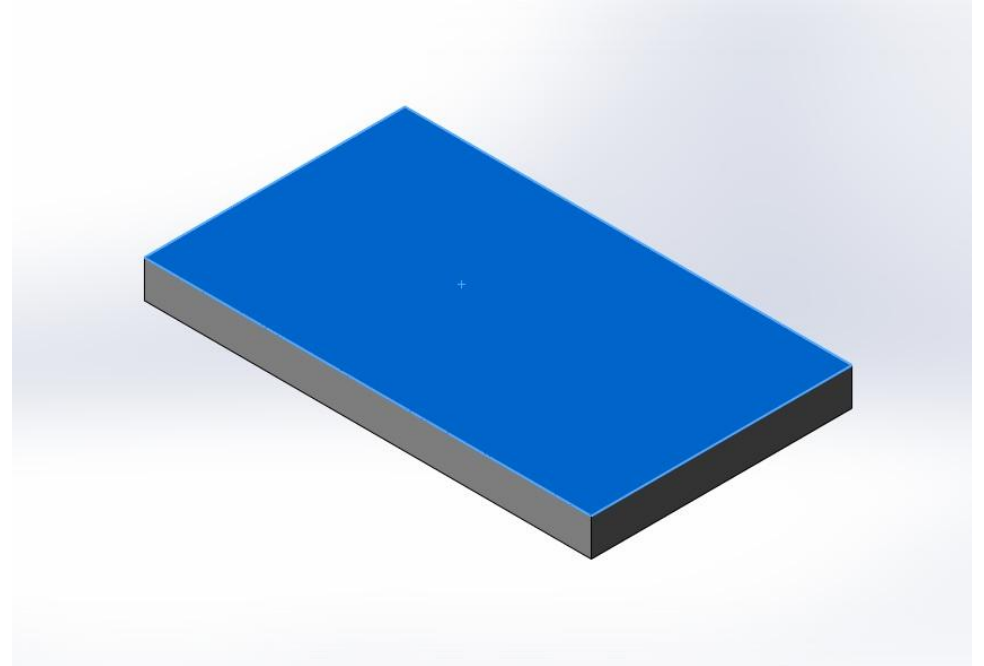
Drawing a base feature on Top plane



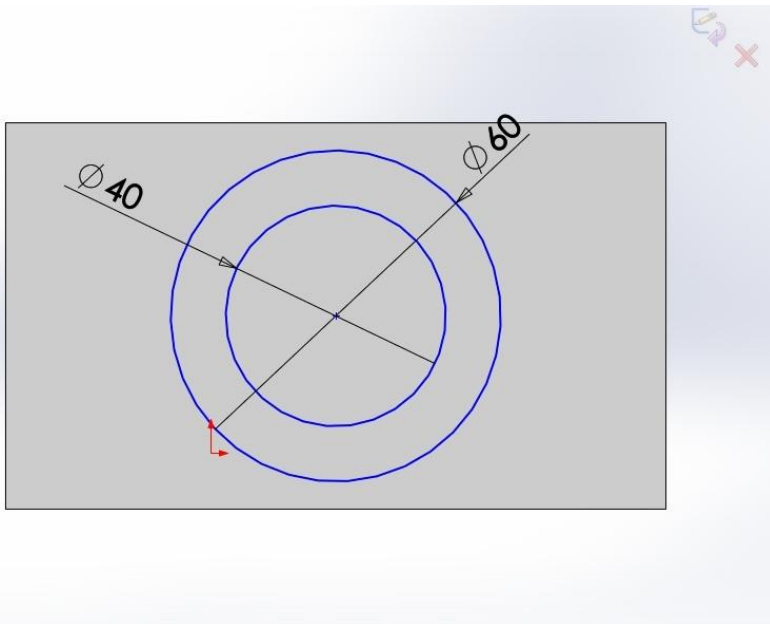
Highlighting top plane



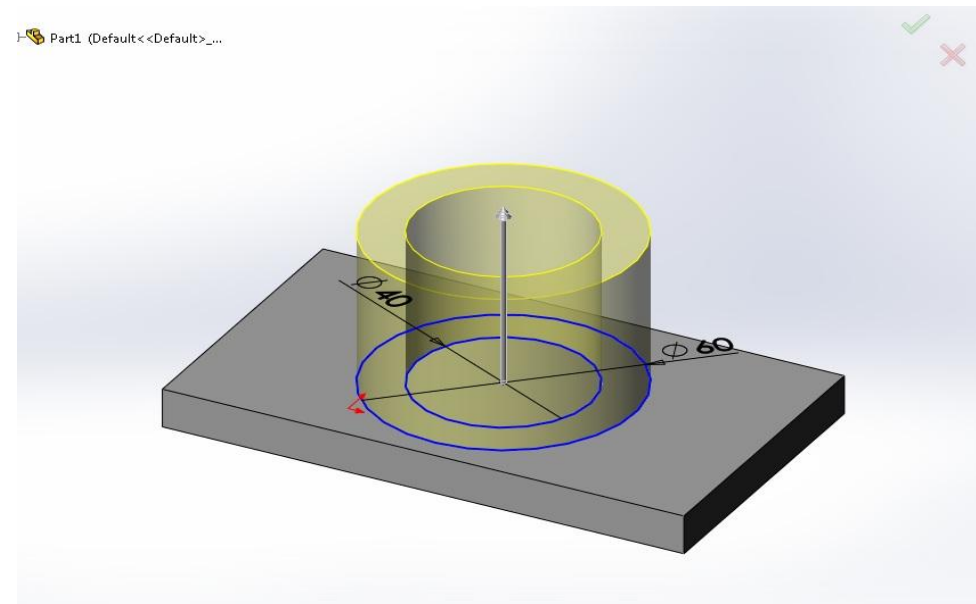
Extruding the base feature



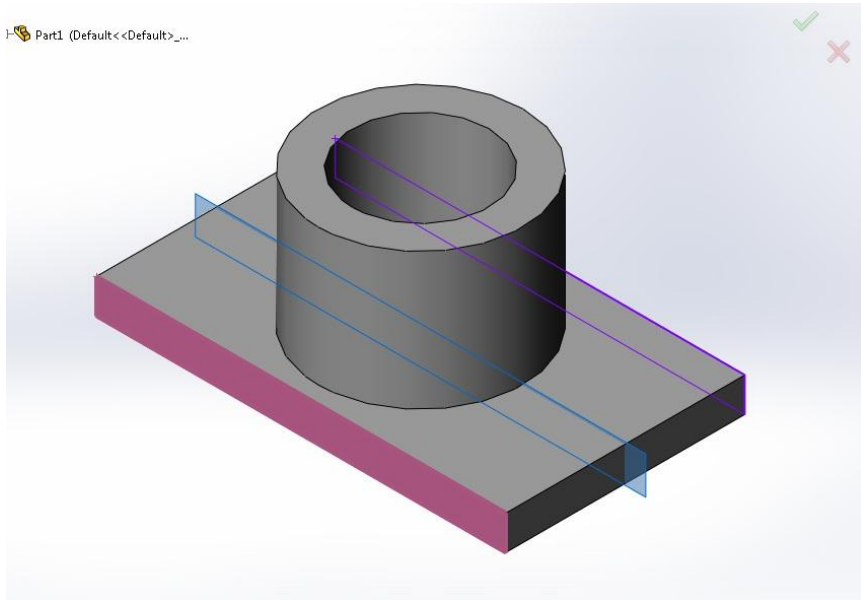
Base feature after extrusion, blue region shows selection of plane on which next feature to be drawn



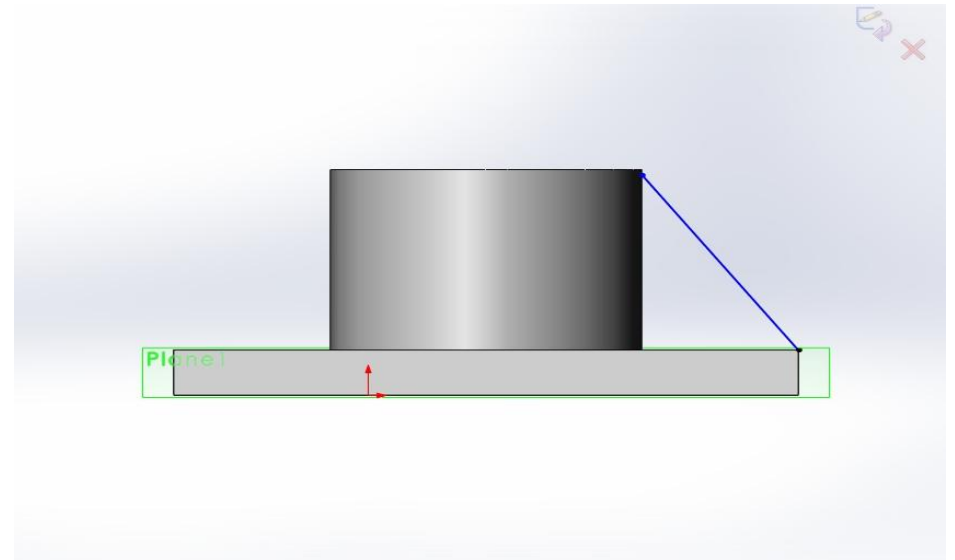
Two concentric circles for making  
a hollow cylinder



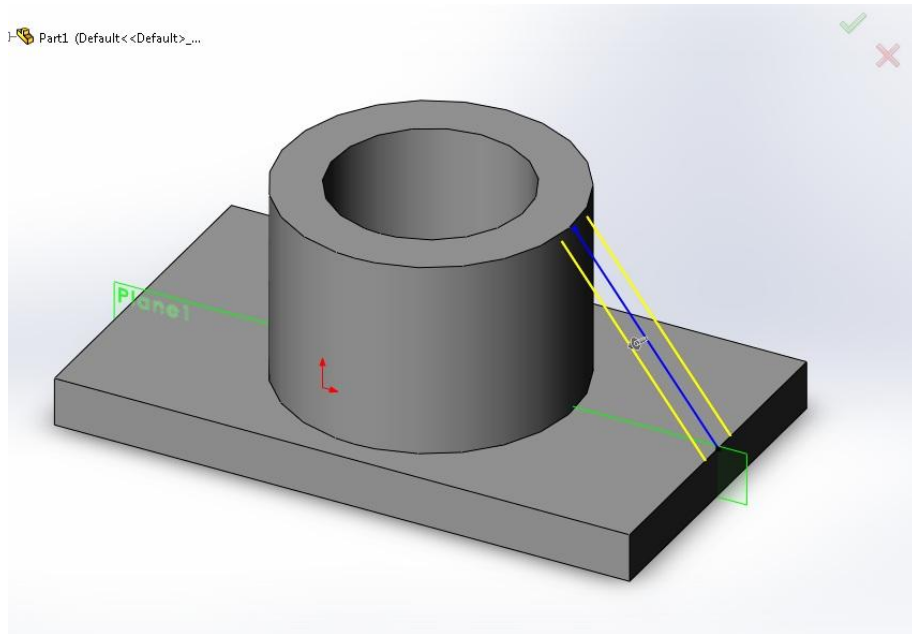
Extruding this feature



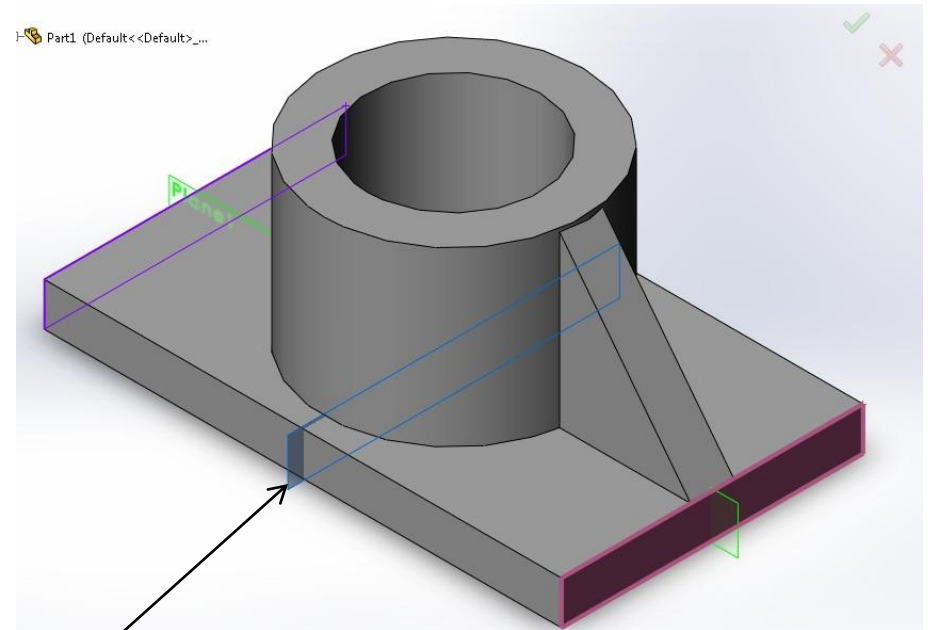
Creating a midplane for drawing Rib



Drawing a line on midplane connecting the top edge of cylinder and corner edge of base feature

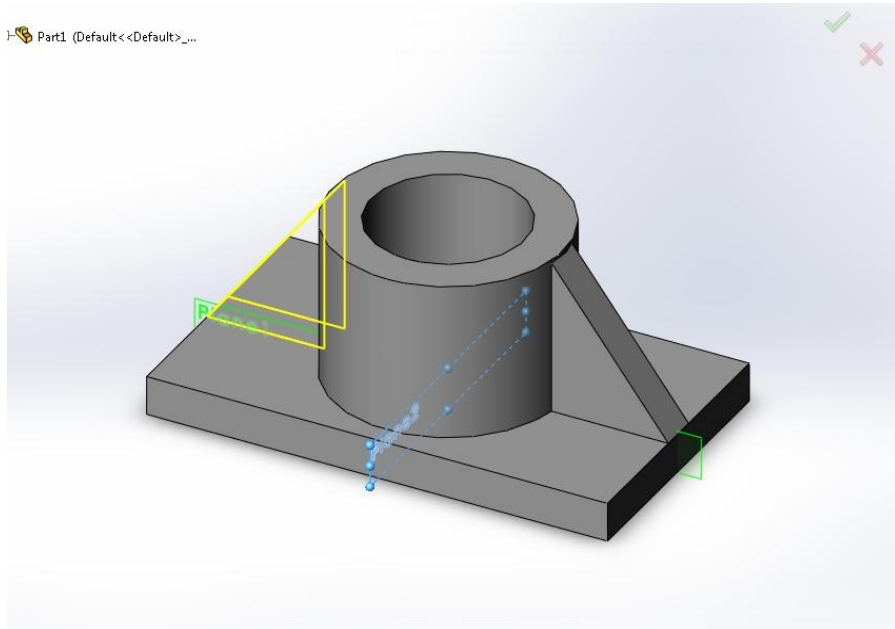


Extruding the rib line on both sides

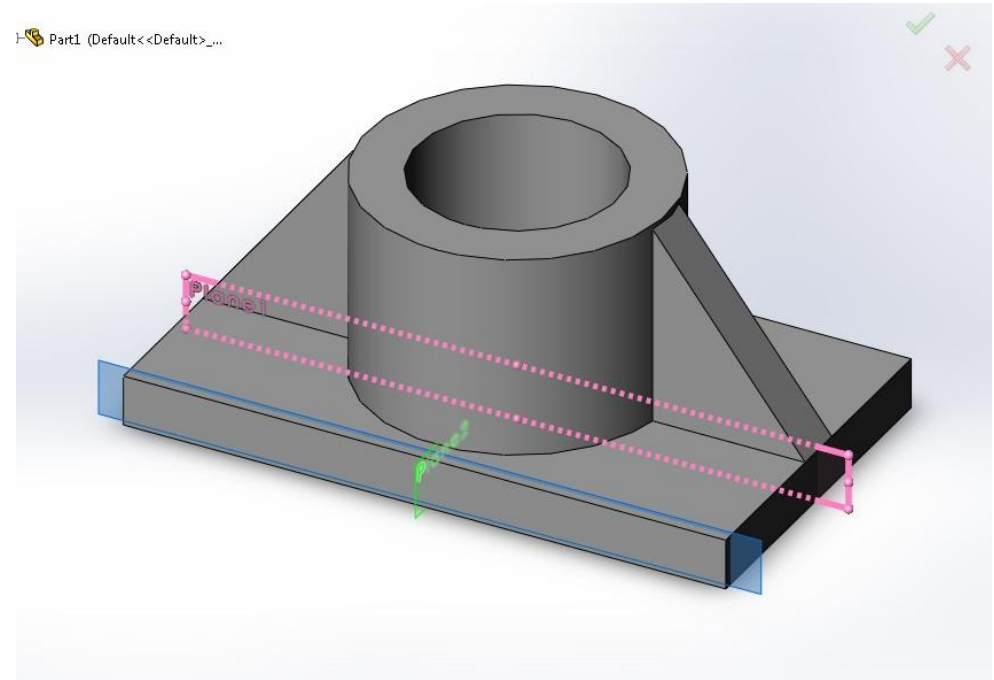


Mirror plane

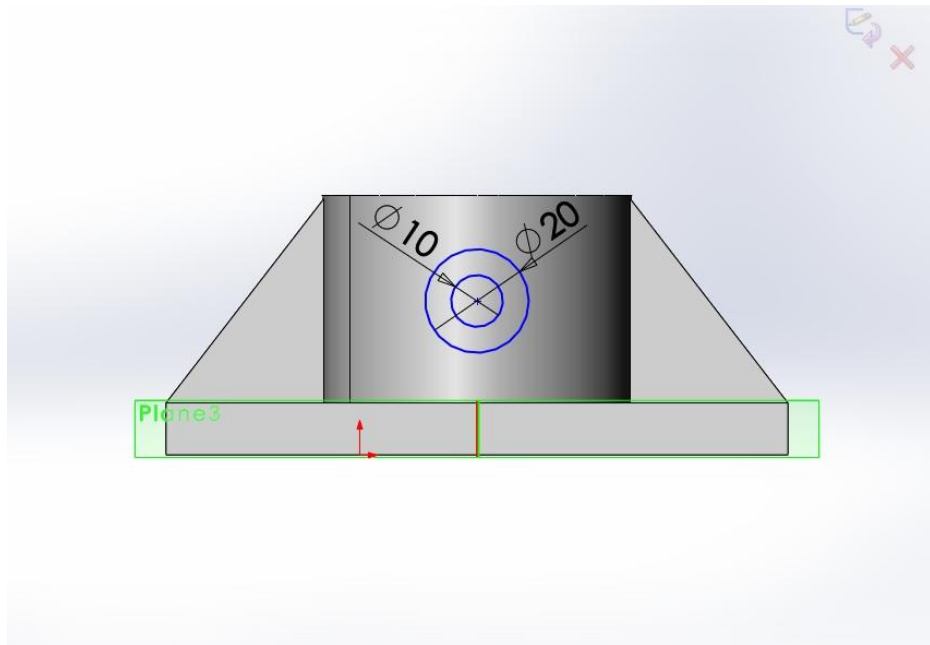
Creating a mirror plane to get the rib on other side



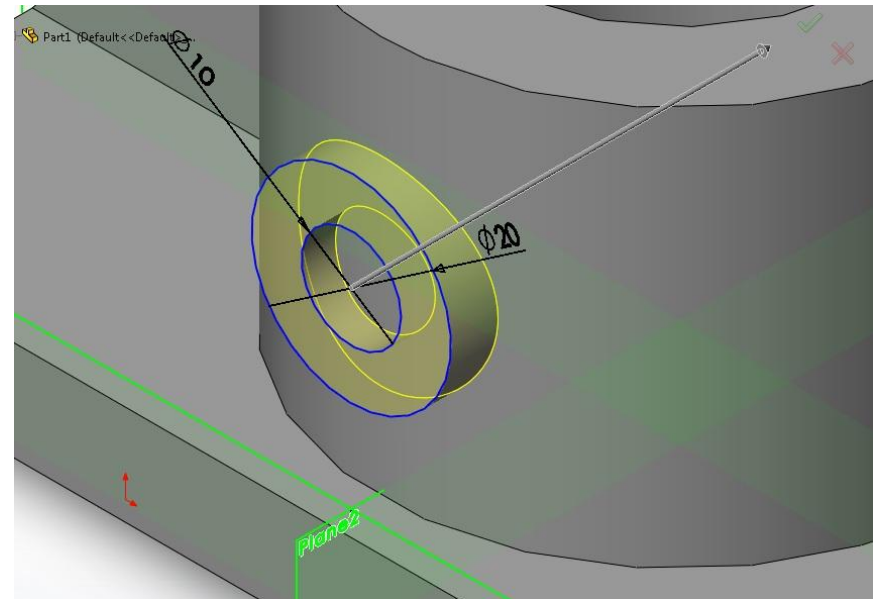
Selection of mirror entities and mirror plane



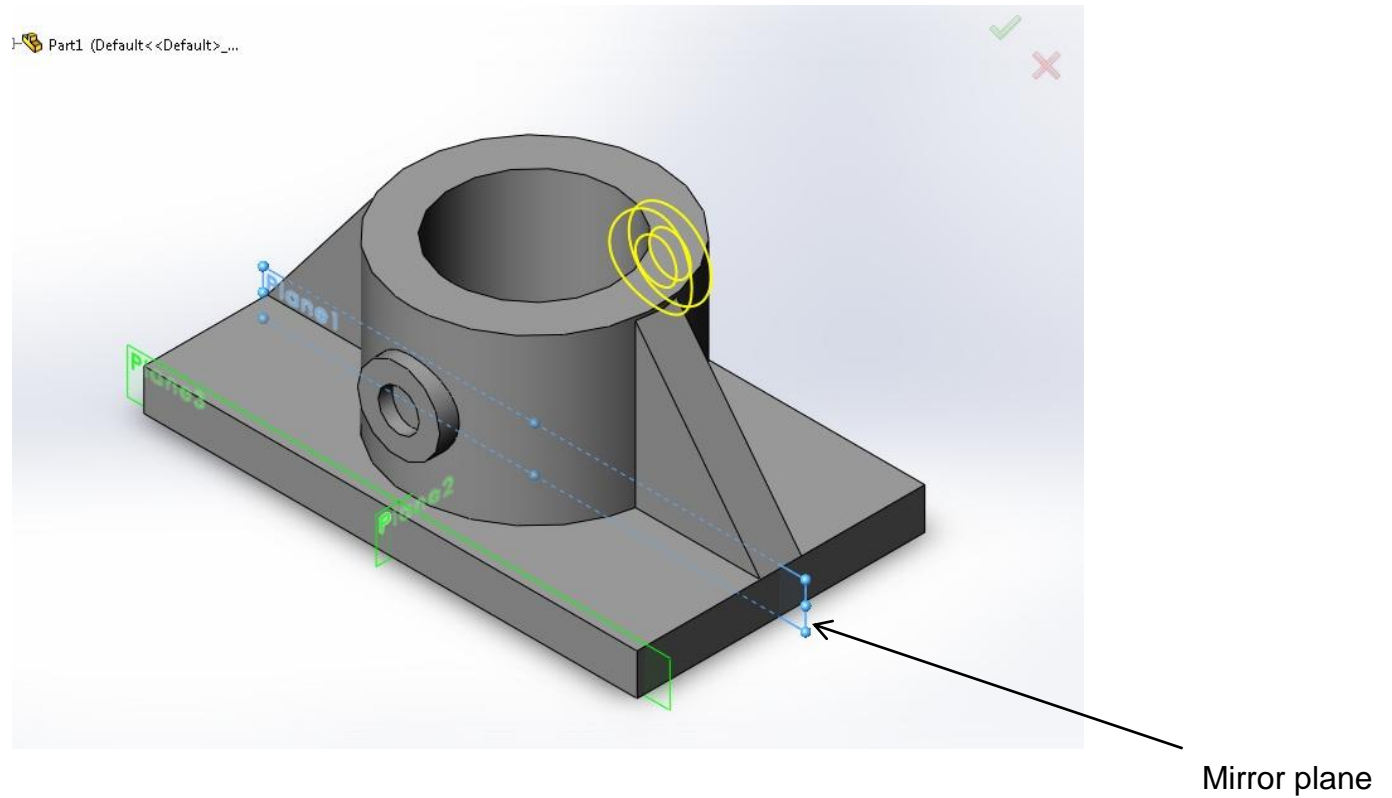
Creating a plane to draw next feature



Drawing concentric circles on this plane



Extruding this hollow cylinder up to next feature



Creating a mirror plane to make this feature on other side