



# COMPUTER AIDED DESIGNING AND MANUFACTURING



## CAD/CAM PRACTICAL FILE



### ASHISH

CAD/CAM practical course to fulfill the Design knowledge of the Mechanical Engineers as per the part of the syllabus using SOLIDWORKS and MASTERCAM for CAD/CAM practical training.

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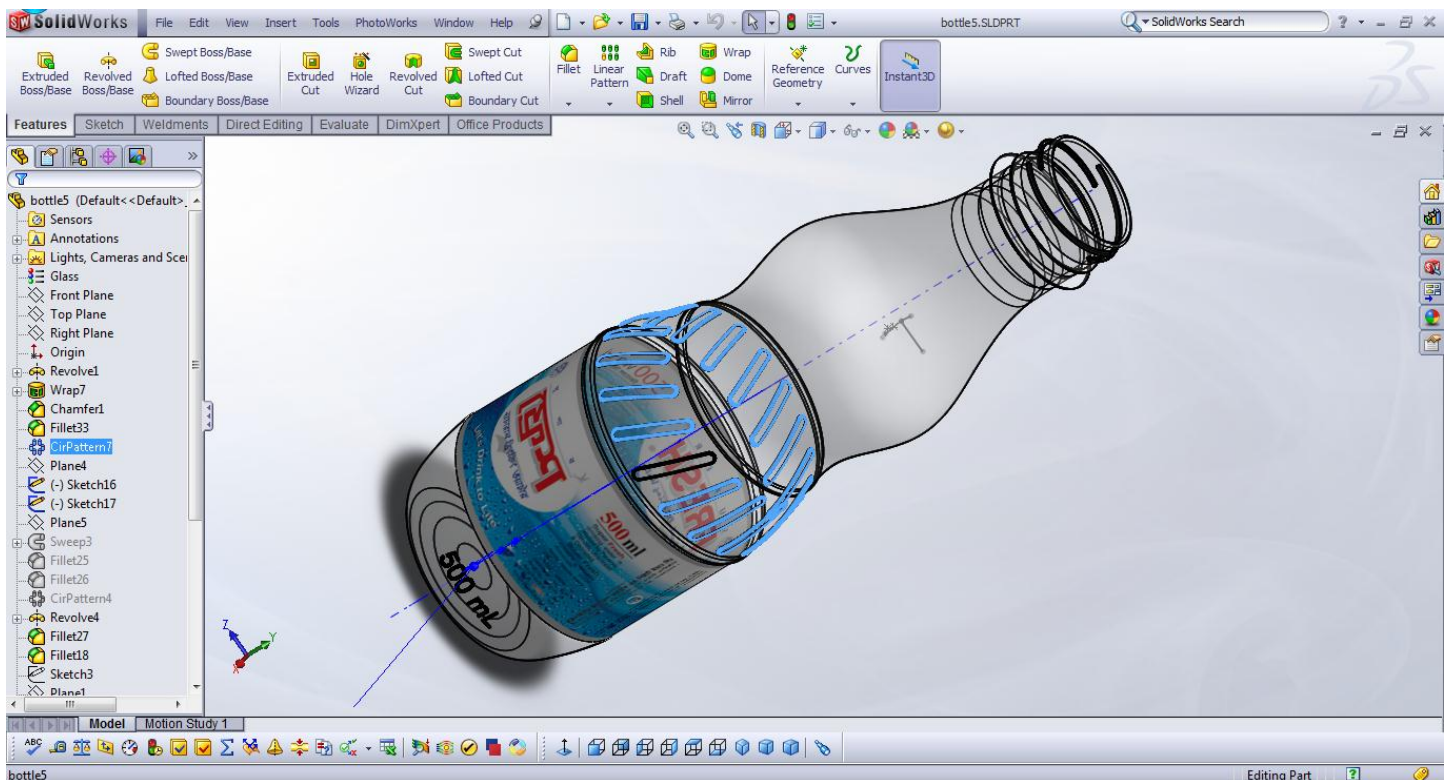
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## INTRODUCTION TO MODELING (USING CAD SOFTWARE)

### INTRODUCTION TO THE CAD (COMPUTER AIDED DESIGN) :-

A CAD system is a combination of hardware and software that enables engineers to design everything from furniture to airplanes. In addition to the software, CAD systems require a high-quality graphics monitor. CAD is the use of computer system to assist in the creation modification, analysis, or optimization of a design. CAD software is used to increase the productivity of the designer, improve the quality of design, and improve communication through documentation.



CAD software for Mechanical Design uses CAD system allows to create 3D models like feature modeling, surface modeling, and sheet metal modeling. It's also provide the Assembly of the all parts with proper mechanical mating relations. Also use of casting CORE and CAVITY, structural design in weldment toolbar, pipe and tubing design and Various design analysis e.g. static analysis, dynamic analysis, fluid flow analysis, thermal analysis is also available in the Solidworks.



## SOLIDWORKS

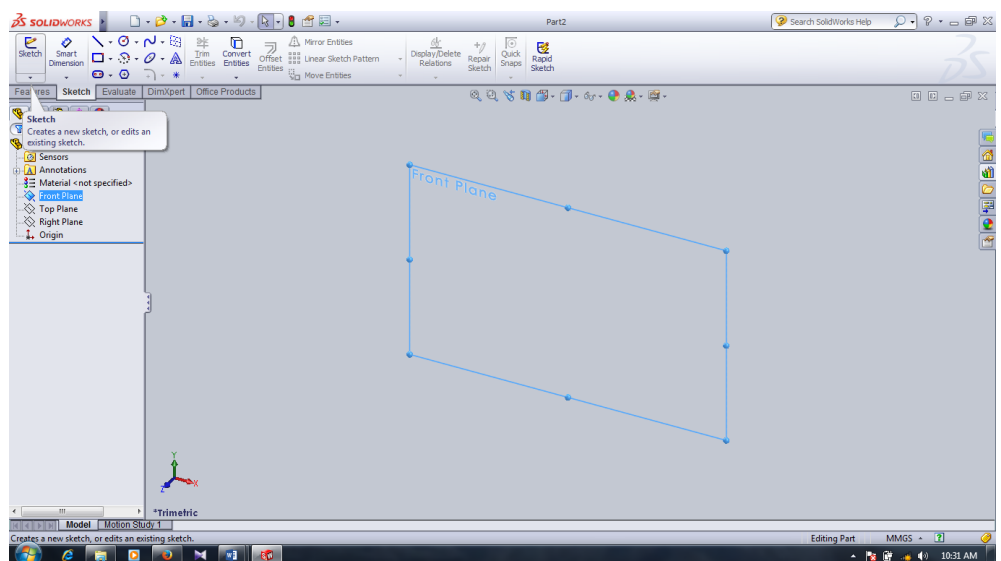
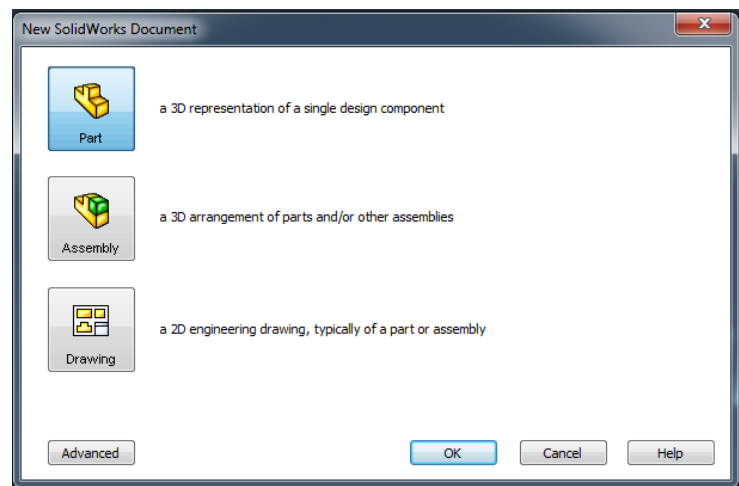
Solidworks is a 3d modeling software produced by Dessault system corp. ltd. Solid works is a CAD Tool and its Parametric Feature based system it's allows you to create 3d parts, assemblies and 2d drawings. Parametric modeling everything is control by either Parameter, relations and dimensions. The basic requirement for creating a solid model in every 3d CAD tool is a sketch. The sketch for the features is drawn in the sketch ribbon inside the solid works. Let's discuss each and every drawing and 3d tools with some practical examples.

Before stating solidworks let's familiar with the user interface of the software. When software starts choose "new" for the creating new drawing part file.



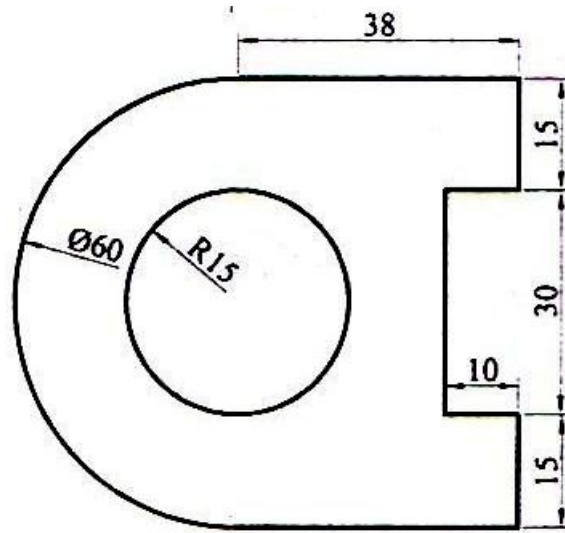
You have 3 option available to you on your solidworks screen 1. Part, 2. Assembly and 3. Drawing

- Choose part (either double click on in or just click once and press "OK").
- Choose sketch ribbon in the solid works screen and select first plane, front, top and right and click on the sketch icon.
- Sketch environment is highlighted and we can draw the various 2d drawing using 2d drawing sketching in solid works.
- For setting the initial units setting in solidworks go to "tools>options>click documents and properties > click on "UNITS" and choose your default units what you need to choose e.g. mmgs (millimeters, grams, second) IPS (inch, pound, second).



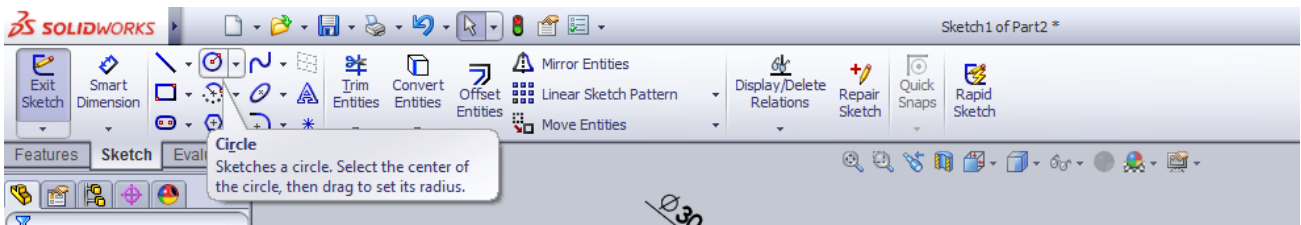
**PRACTICAL -1**

**2D DRAWING USING SKETCHER IN SOLIDWORKS – 2 DRAWING.**

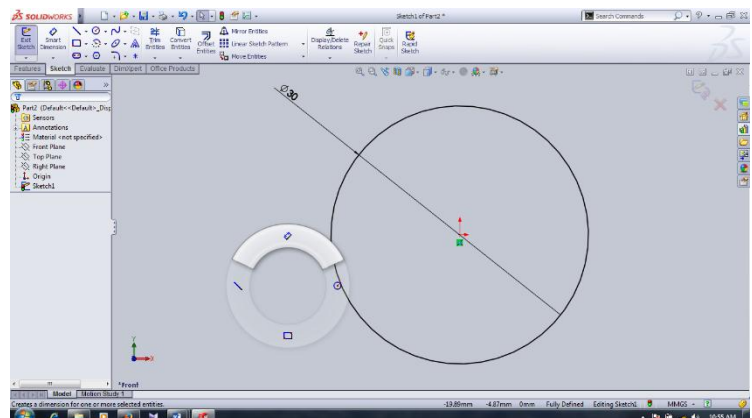


**Top View**

- Open sketch environment in solid works choose one of default plane (say front) and click on the circle icon on the sketch toolbar ribbon.

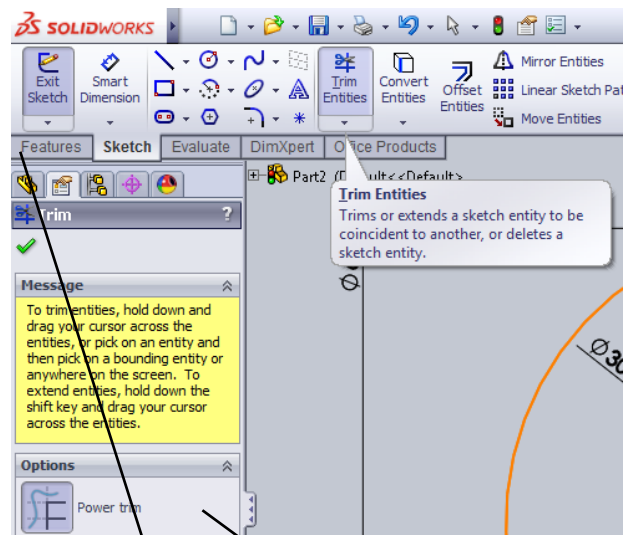
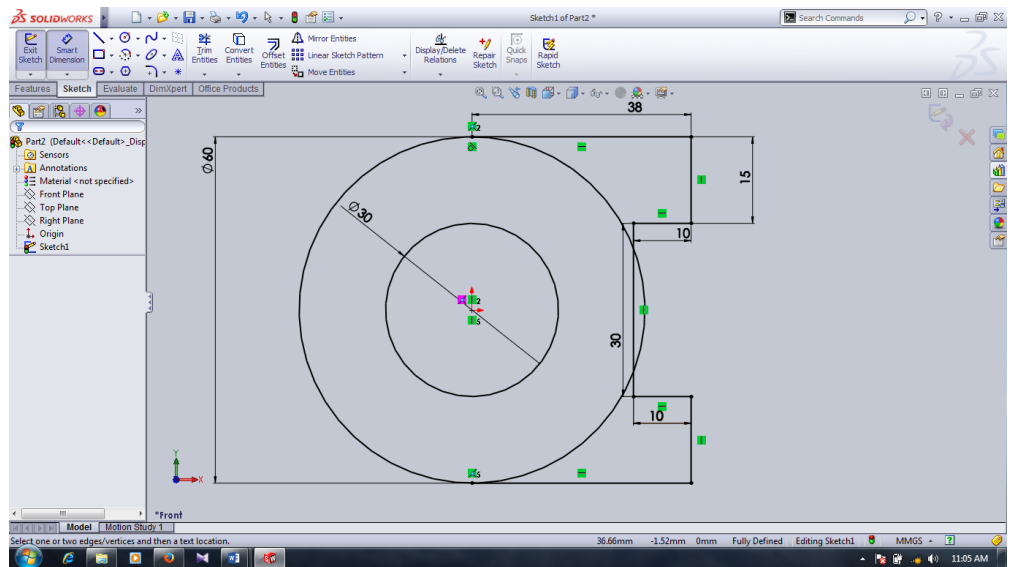


- Draw a circle inside the drawing area first then choose smart dimension icon on the sketch ribbon or right click and drag the mouse cursor to UP you've automatically choose smart dimension tool. Select the circle and give the proper dimension as per given drawing.
- Draw the same circle of diameter of 60 in same center point. Then draw the line as per given drawing and give dimensions with smart dimension tool.



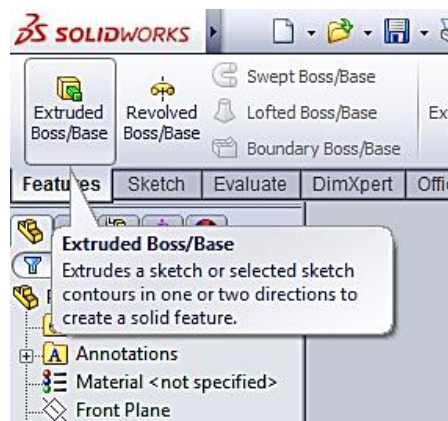


- You draw the object similar as you see in the screen to remove the extra unwanted objects choose trim command in the sketch ribbon and click on the power trim option available on the right side of the properties toolbar and select the line you need to remove it will automatically delete or disappear from the screen which object you don't need in this drawing.
- The final object is prepared as per your drawing.
- (You can also put the relationships in the object just click the first object and second object holding down the "ctrl" key the relation matched to the object will appear on the right side on the properties toolbar automatically and you can choose the appropriate relation as you want to put inside the object.)*
- Choose or click on the **"feature"** ribbon inside the solid works and choose **'Extruded Boss/Base'** and your object will automatically get extruded in the "Z" direction give them height you need to give say 40mm. click on OK .



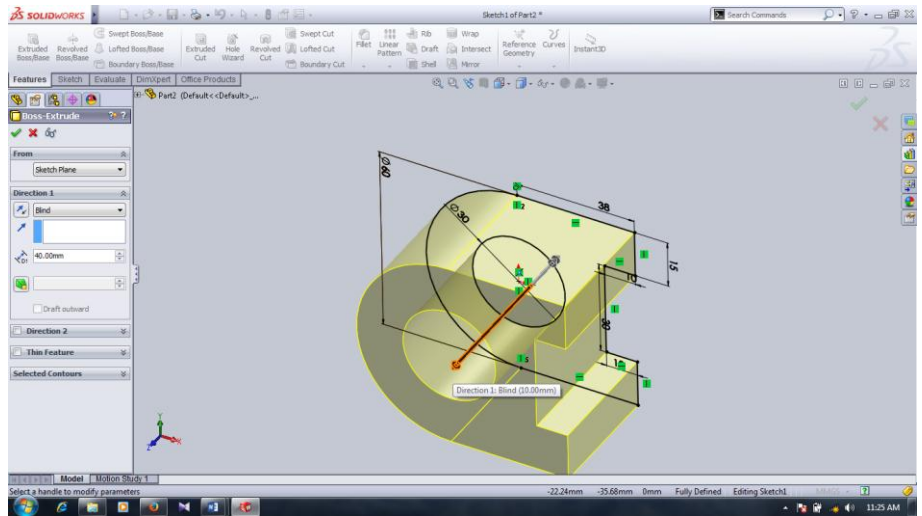
Feature

Power trim

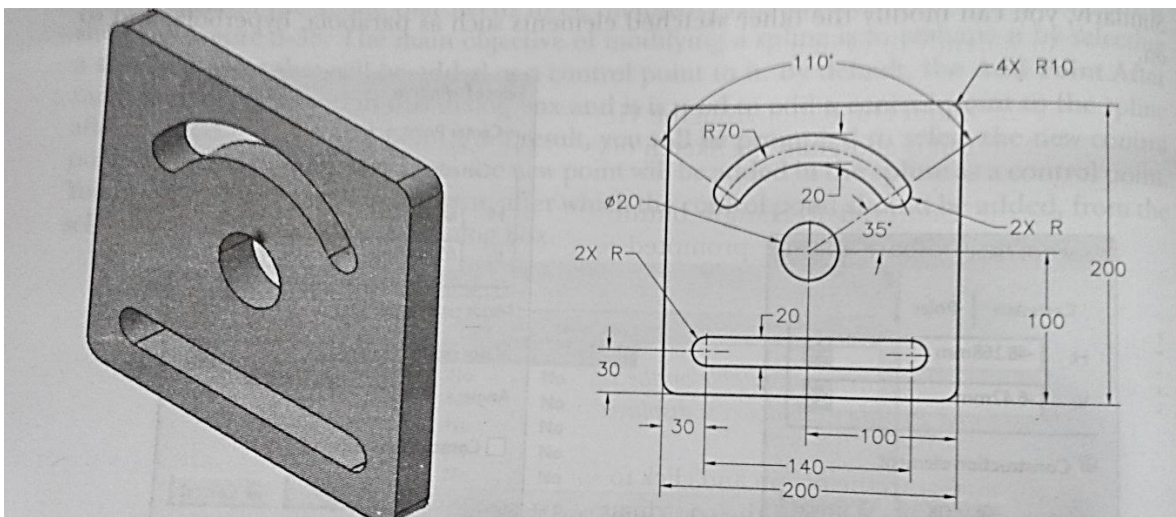
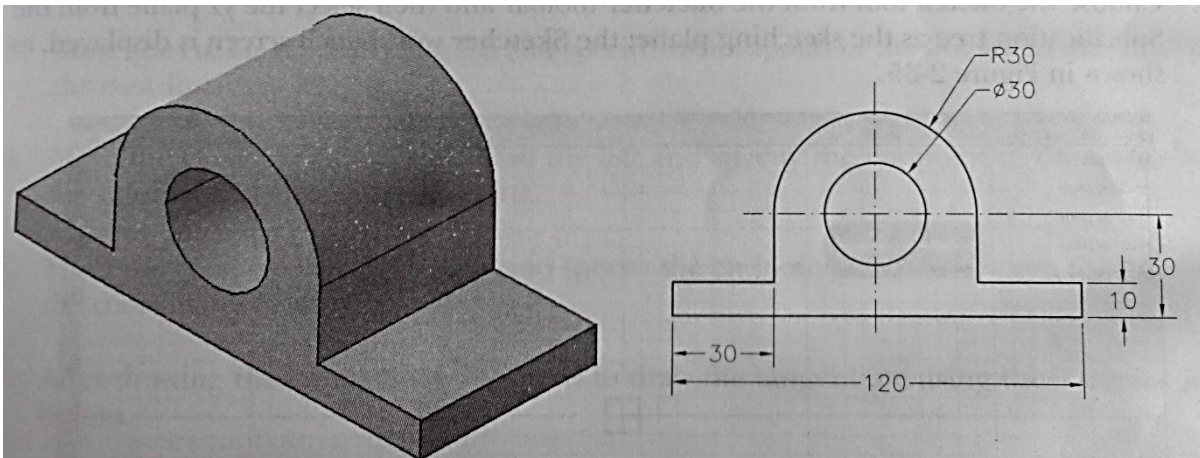


**Feature toolbar ribbon**

- First object is completed as per your drawing part and you also learn how to extrude the object using extruded boss command in the solid works.
- Click on OK or green check indicating on the left corner or in right corner in the properties manager.
- Save the object with proper name and close the solid works.

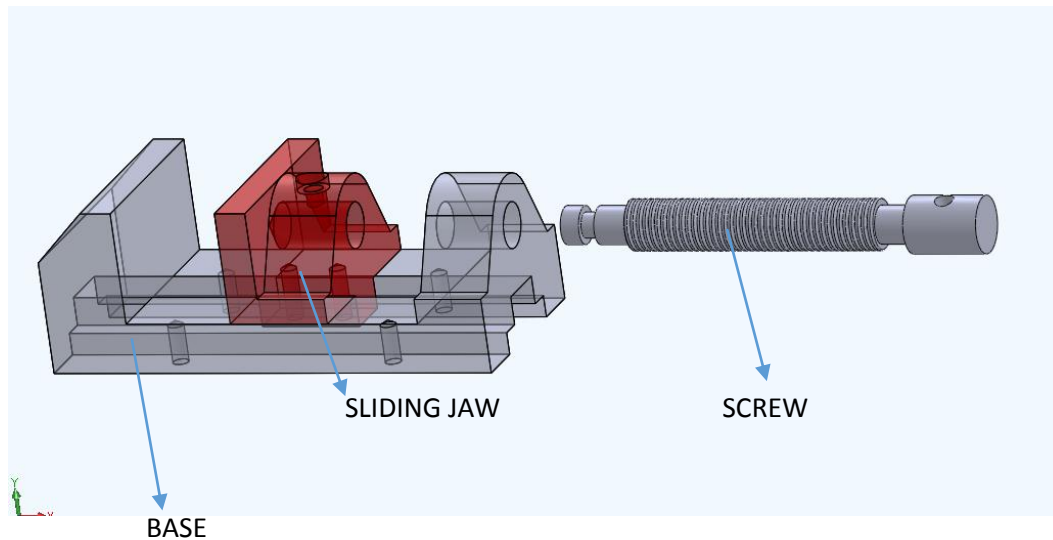


**PRACTICE DRAWINGS**



**PRACTICAL -2**

**3D MODELLING USING 3D FEATURES (MODELLING OF BENCH VICE, SCREW JACK)**

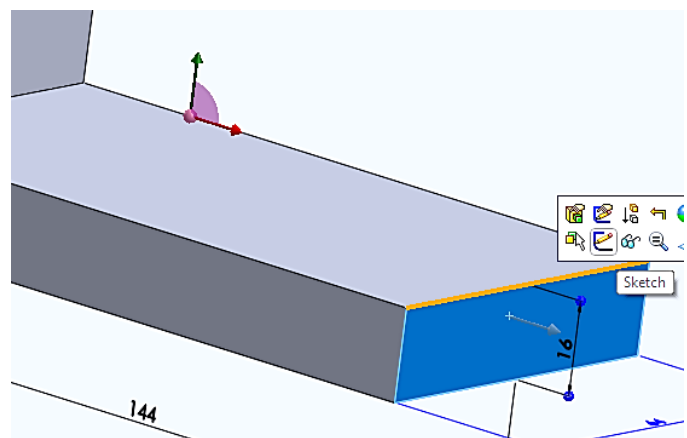
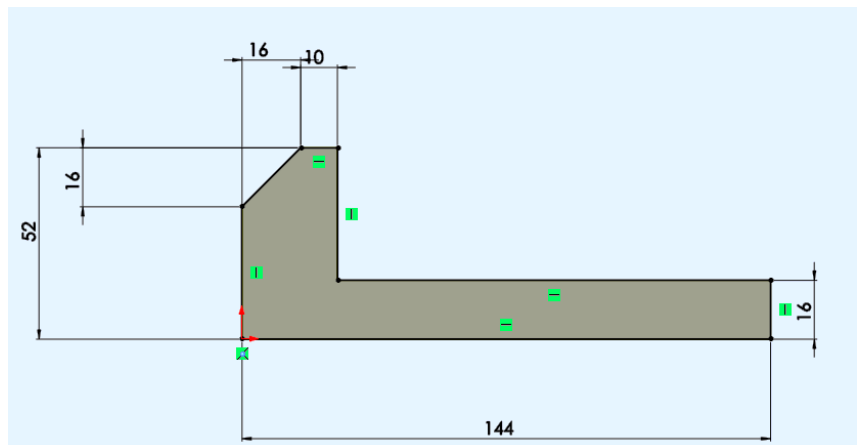


- Before drawing the 3d model we need to draw the each component of the screw jack one by one here is the simple model of the screw jack which contains 3 basic objects.

1. **Base**
2. **Screw**
3. **Sliding jaw**

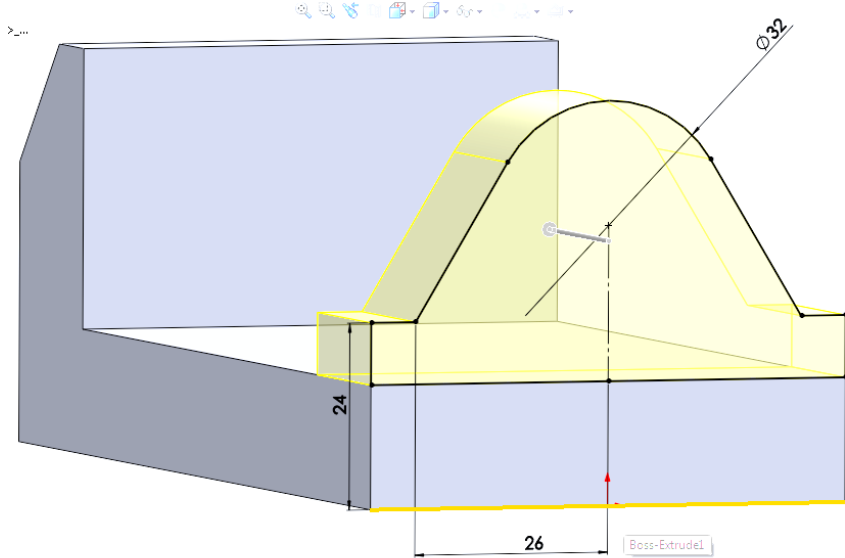
- **CREATING BASE FEATURE**

1. Open the solid works and draw the object as same as shown in the object with same dimensions and extrude this object using extrude boss to 64mm.
2. After extrude select the back face of the object and click on sketch tool shown in the sketch tool ribbon or highlight icon of the sketch tool. Click "ctrl+8" and draw the object shown below and after completion of the drawing of the object again choose extrude boss to extrude the sketch in desired direction in 22mm.



The object is shown in the next page.

3. Again select the back face of the base plate and draw the diagram shown in the figure 6. And this time choose “Extruded cut” and choose “through all” and cut all the material through to the object. And also draw a concentric hole of the diameter of 14.50 mm and also extruded cut this hole to through all.



4. Click on the green check and save the file named “base”.

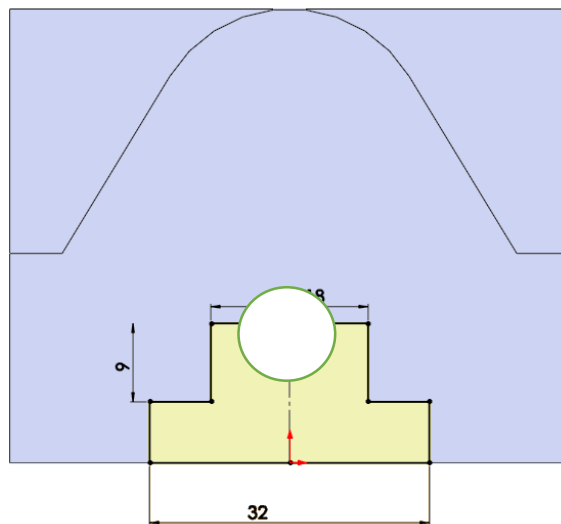
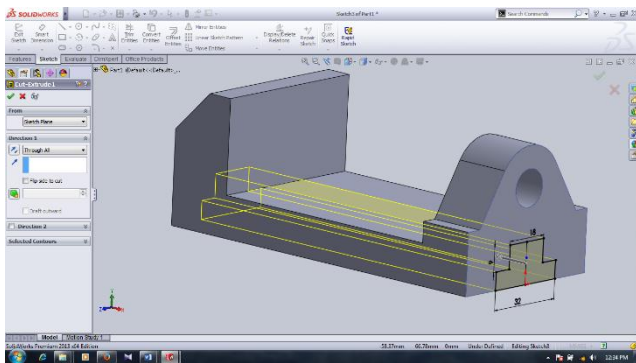
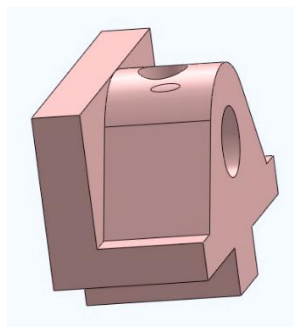


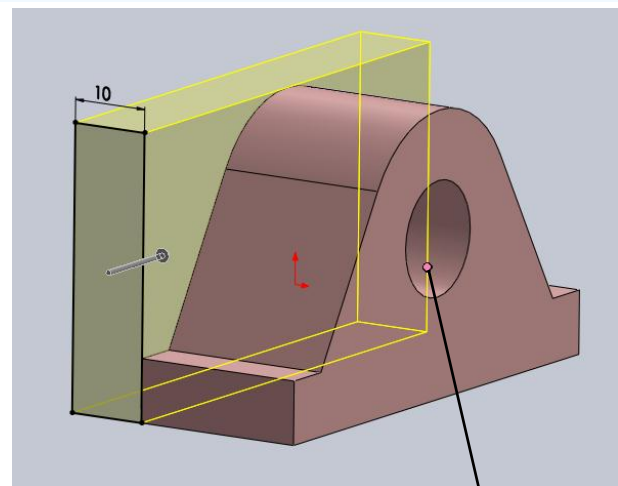
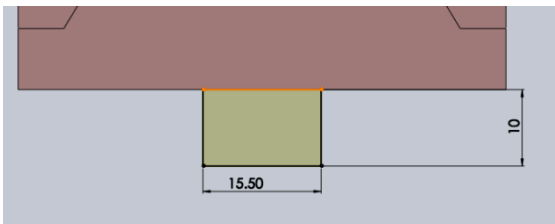
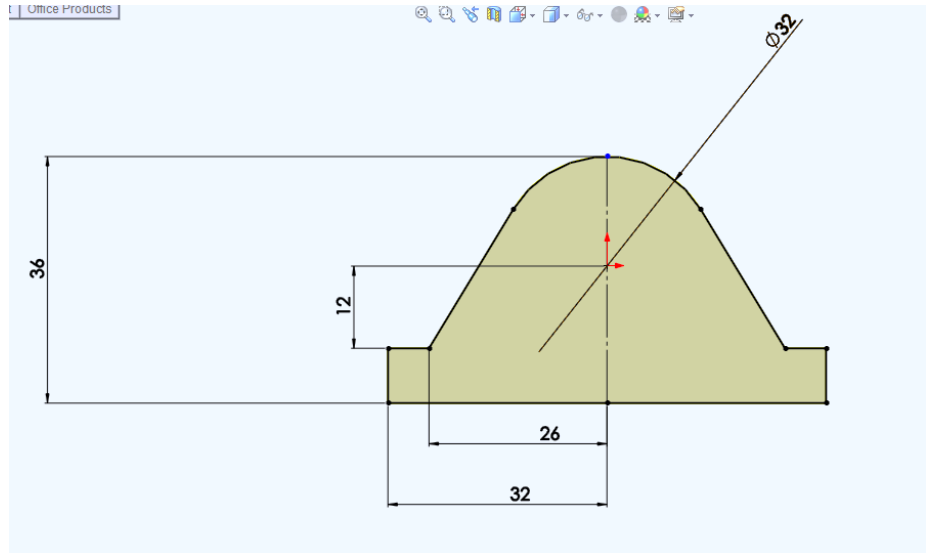
Fig- 6

**2<sup>nd</sup> PART (sliding jaw)**



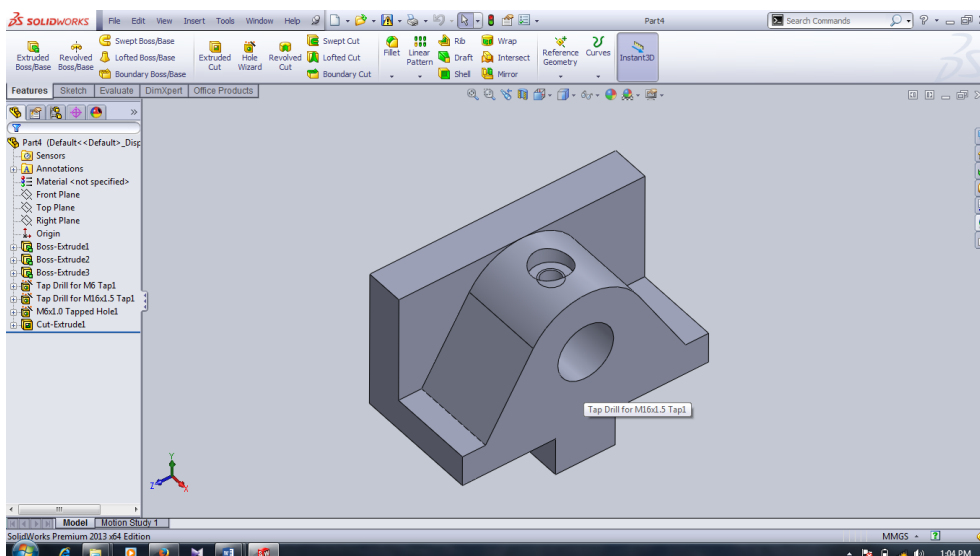


1. Create a new drawing file and draw the 2d object same as shown in the figure. And extrude this drawing to 22mm.
2. Select the corner face of the object and draw a 1mm thickness rectangle same as shown in below and extrude it up to surface or give value 52mm.
3. Same draw a rectangle at the bottom of the sliding jaw as per given dimensions and extrude them to 22mm or up to end.

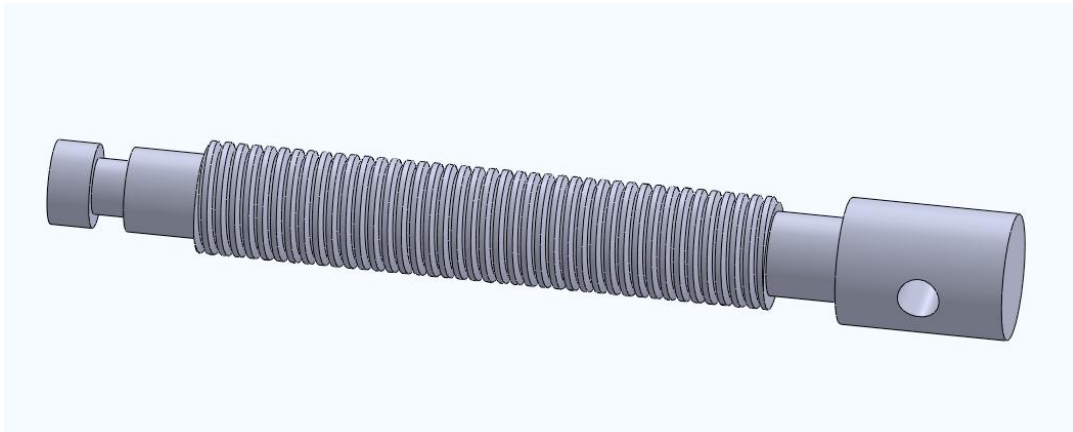


4. Create a circle on the top of the plane and draw a circle using hole wizard or circle command to create a top circular hole to complete the sliding jaw object.
5. Save the file named sliding jaw and open the new file.

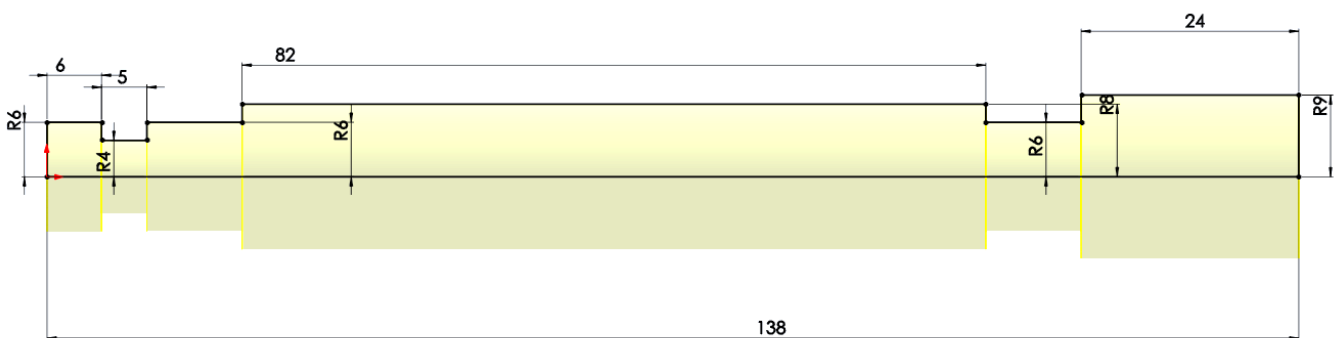
Hole 14.5mm



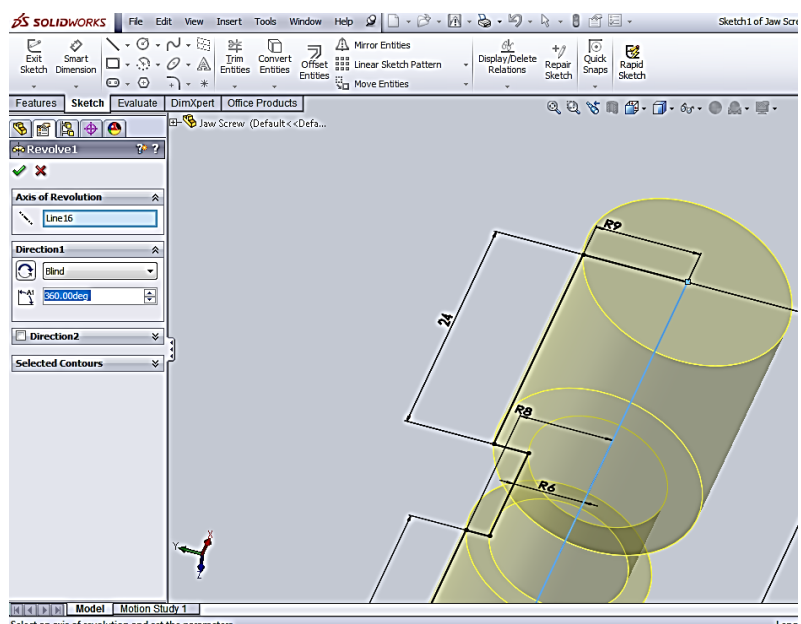
SCREW



- To create a screw First draw a diagram as shown in figure below as same as given dimension.
- After draw this diagram go to the feature ribbon toolbar and choose the **Revolved**



**Boss/Base** and choose the reference line and the remaining object to revolve and give them the degree of rotational angle say 360 for full round and our object is completed. Now we have the three basic object and

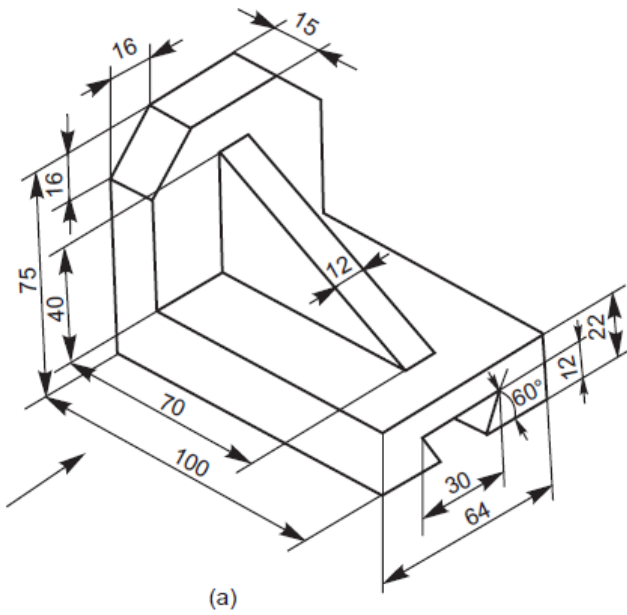


- We cleared some of the main basic feature tools. Now we move on to the assembly drawings.

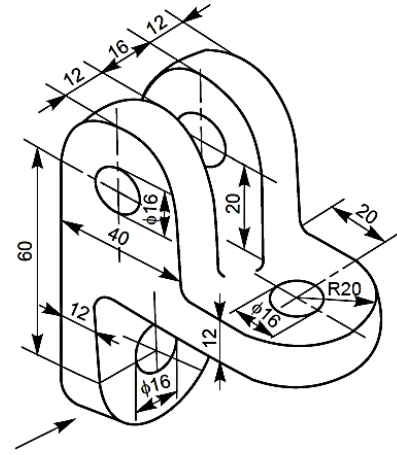


# BABA FARID GROUP OF ENGINEERING AND TECHNOLOGY

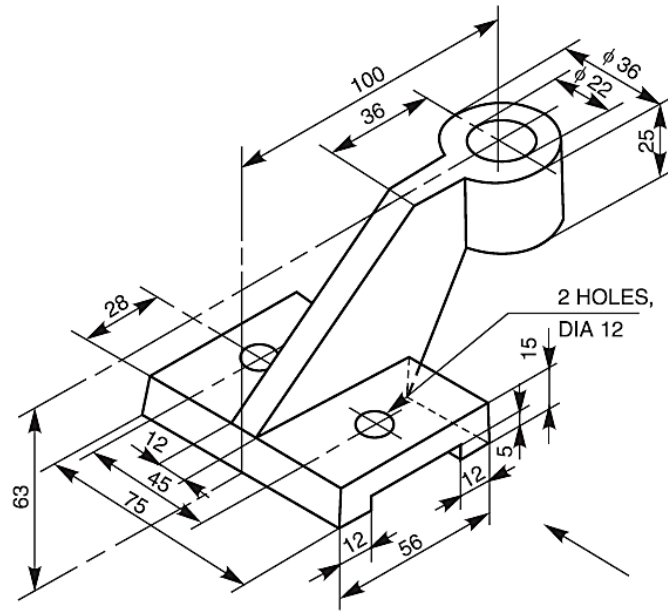
## PRACTICE DRAWING



(a)



(a)



(a)

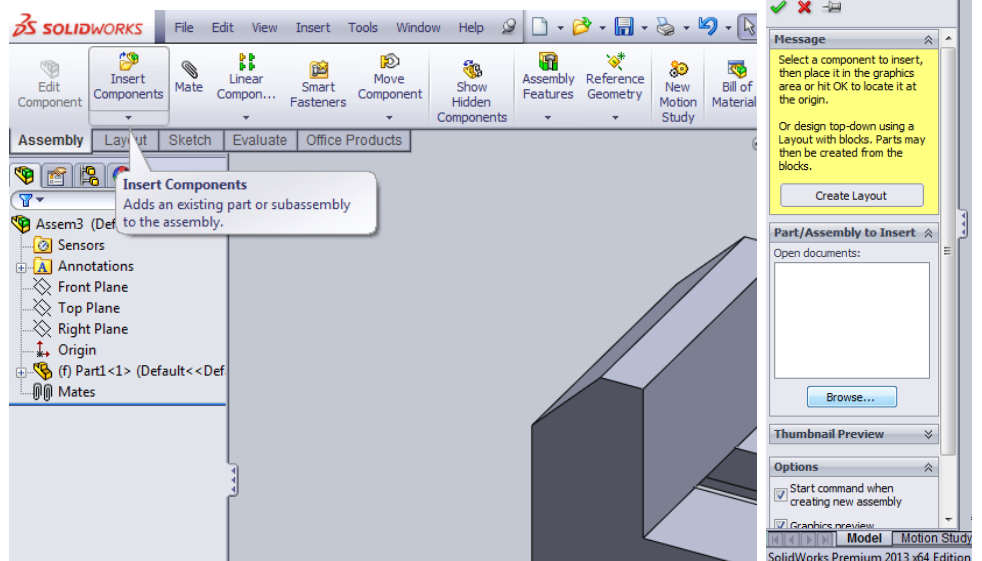


**PRACTICAL 3**

**ASSEMBLY AND DRAFTING WITH PROPER MATING CODITION AND INTERFARANCE CHECKING**

- We are try to assemble the machine vice with proper mating condition on this practical
- Open Solidworks software and choose new file and now click on “assembly” twice or just click once and press **OK**.

- You will open a assembly environment window.
- Click on the Browse.. button to choose the assembly file one by one. First select the base.
- Insert the first component named BASE you saved before in part design and click on the screen object is automatically align and fix on that position because the first object we choose in the assembly in the solidworks is considered as a fix object so that the remaining part assemble properly with this base part.

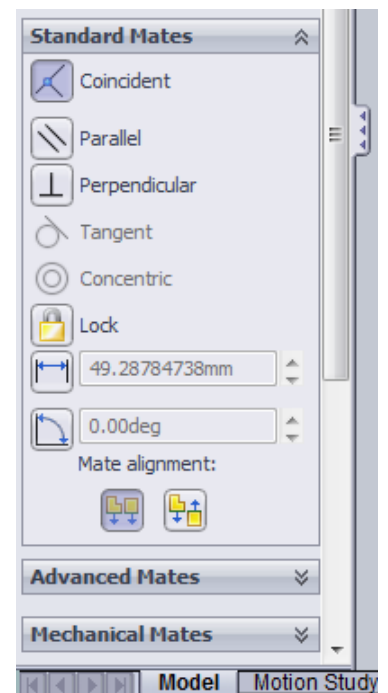


- Again select “insert component” on the toolbar ribbon and choose the second object and place the second object on anywhere on the assembly screen.
- Click on the “Mate” icon and select the both corner face of the base and sliding jaw in the assembly the object is automatically align with the second object with appropriate relation fits on that mate automatically but if you won't satisfy with that mate you can change the mate relations.

*There are so many types of mate available in the solidworks.*

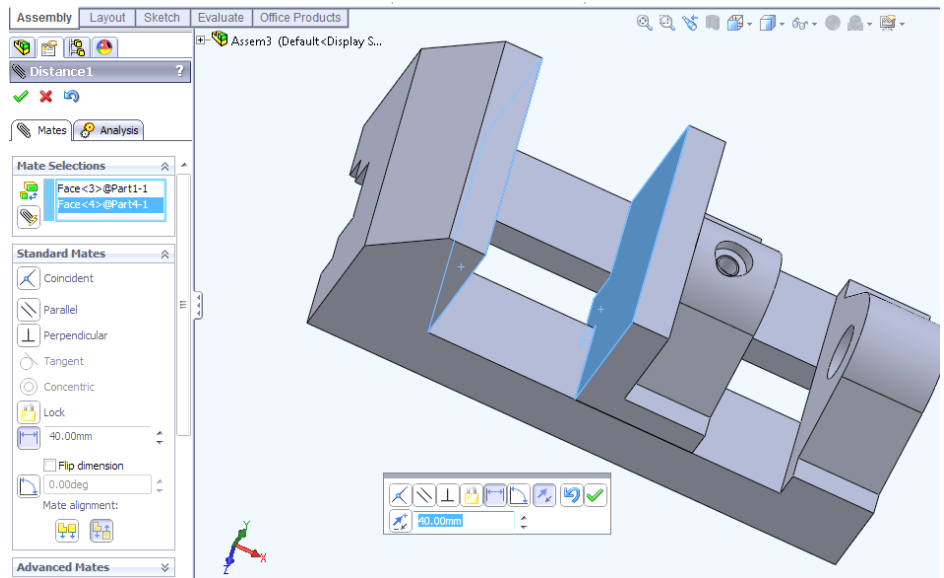
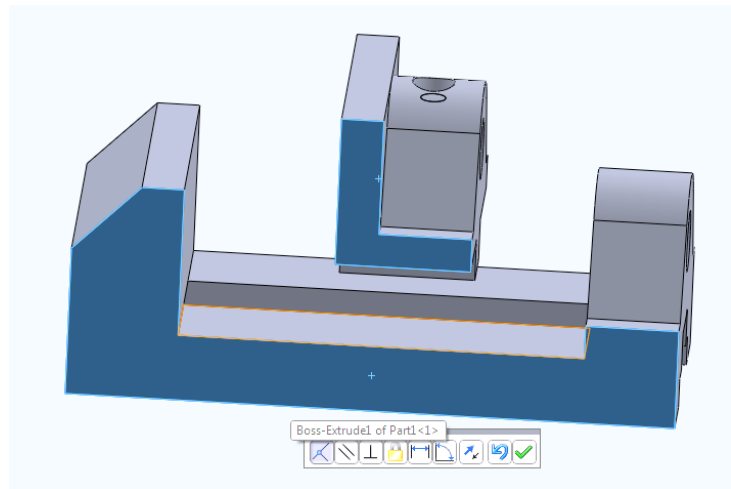
1. **Coincident mate**
2. **Parallel mate**
3. **Perpendicular**
4. **Tangent**
5. **Concentric mate**
6. **Distance mate**

now select the final screw component and choose the circular face of screw and hole of the cast iron base.

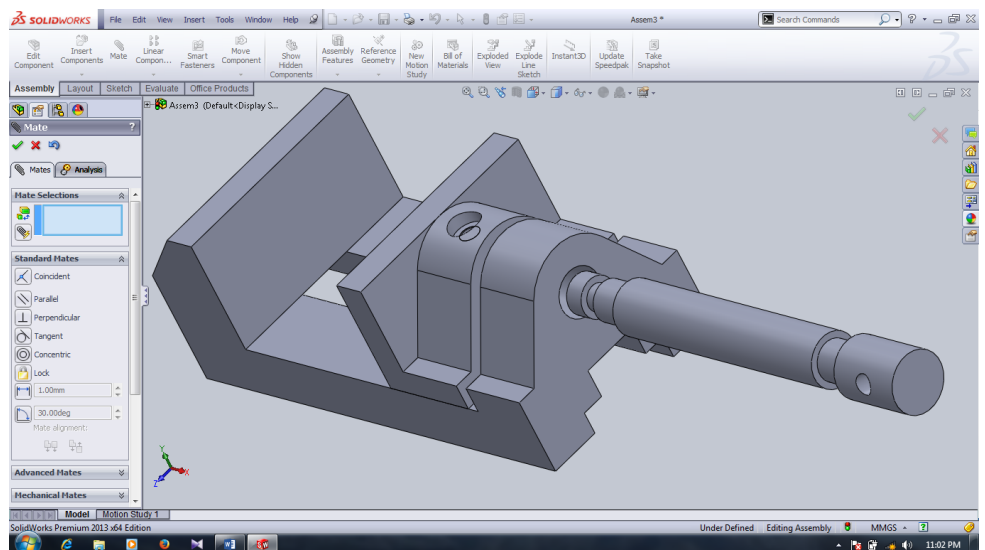




- first select the two corner face of the base and sliding jaw as shown in figure and choose coincident as a mating condition of the both.
- Now choose the bottom grooved face of base and sliding jaw as similar as the first one and again choose coincident.
- Select the two mating face and choose the “distance mate” and give a distance. Say 50mm
- Now choose the screw component and click on the “mate” on the toolbar ribbon and choose the circular face of the rod and hole of the base.
- Your assembly of the simple vice is completed now we move on to the collision detection part in the solid works.



for the collision detection first choose the move tool in property manager choose the collision detection and choose the object you want to study of the collision between them. The object is highlighted and creates sound when some interference or collision occurs.







## **CAM (COMPUTER AIDED MANUFACTURING)**

**INTRODUCTION – CNC (Computer Numeric Control system, a dedicated computer is used to perform all the basic NC functions as per the control program stored in the memory of the computer. Thus the machine controlled data comes directly from the computer memory and not for the continuous tape as in case of head-wired NC system. The use of computer makes the system flexible so that the system can operate on a different type of machine tool.**

### BASIC SIMPLE WAY TO TRY THE CNC PROGRAMMING

1. For starting machine you type the code as given follows.

T01D1;  
G90G71G95;  
G96LIMS=1800;  
M08M03;

T01=Tool no.	G96= Constant cutting speed
D1= offset no.	LIMS= max. Spindle speed
G90= absolute system	M08= Coolant on
G71= dim. Data in mm	M03= spindle start clockwise
G95=feed rate mm/rev	

2. For the end of the machining process type the code as given follow.

M05M09;  
M30;

M05= absolute system	M09= Coolant on
M30= spindle start	

3. Some important CNC codes.

S250= minimum spindle speed  
M04= spindle start at anticlockwise  
G00= for rapid transverse of the tool  
G01= for linear interpolation as per feed rate.  
G02= circular interpolation at clockwise direction.  
G03= circular interpolation at anticlockwise direction.  
G04= dwell  
M68,69= clamp and declamp workpiece  
M00= program stop  
M02= end of program  
M03= spindle start clockwise  
M05= spindle off  
M06= tool change  
M08= coolant on  
M09= coolant off  
M30= end of the program  
M83= hardware on .



## PRACTICAL-1

### MANUAL PART PROGRAMMING OF CNC LATHE AND CNC MILLING

#### WORKPIECE INFORMATION

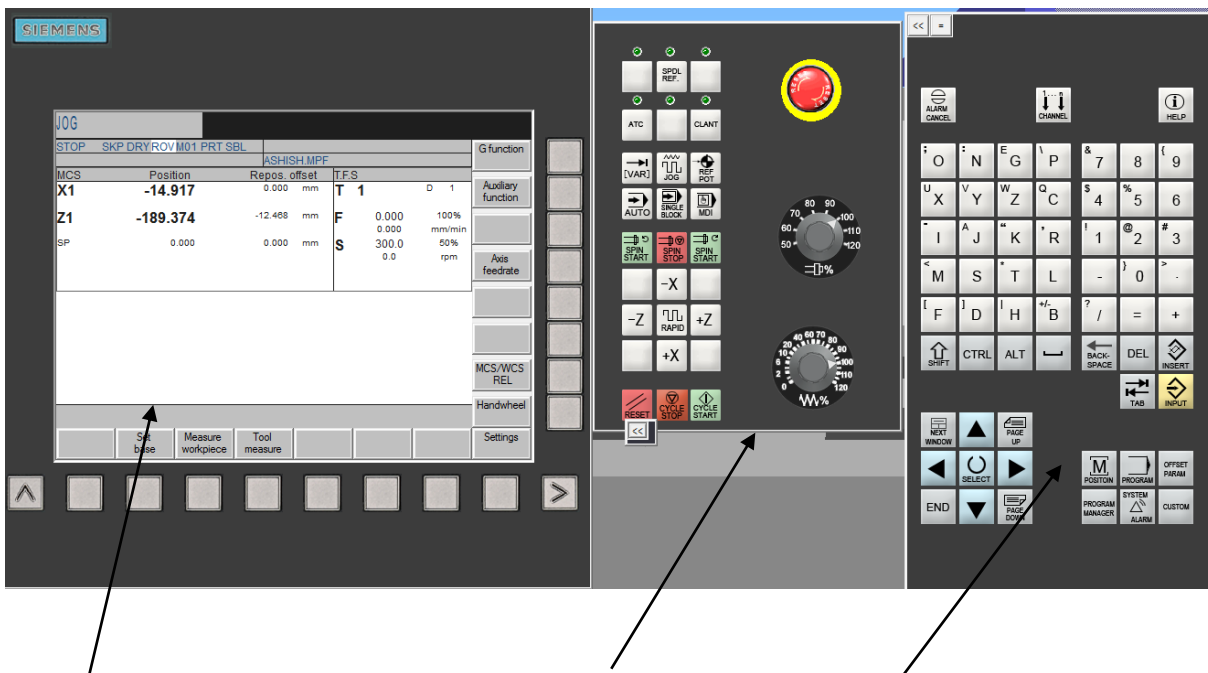
Bar

Diameter: 50.000

Length: 150.000

MATERIAL :08F Low carbon steel

#### OPERATING SYSTEM (SIEMENS CONTROL SYSTEM)



MAIN SCREEN

MANUAL OPERATING PANEL

PROGRAM PANEL

#### HOW TO SET THE MACHINE

1. Switch on the power supply of the machine.
2. Put the machine in JOG mod and choose the MCS (Machine Co-ordinate system).
3. Touch the tool with the work piece and set to the reference point.
4. Click on the tool measure and measure manual.
5. First set the X zero co-ordinate as per stock and Click “set length”
6. Second set the Y zero co-ordinate as per stock and Click “set length 2”
7. Switch to the REFERENCE mode and set the tool to the machine zero point.
8. Repeat this setup for each tool you mount in the turret.





**HOW TO FEED THE PROGRAM**

1. For the safety purpose first you need to close the machine door to avoid any accident.
2. Click on the MDI (manual data insert) mode.
3. Click on the program manager.
4. Select the NEW button.
5. Give a specified name of your program.
6. And then type the following program in you program screen.

**CNC PROGRAM (FACING, TURNING, TAPER TURNING, CIRCULAR INTERPOLATION AND THREADING)**

```
T01D1;  
G90G71G95;  
G96S1200;  
LIMS=1800;  
M03M08;  
G00X50Z6;  
G01F.8X50;  
G01X0;  
G00X51Z5;  
G00X50;  
G01X0;  
G00X51;  
G00X50Z6;  
G01X0;  
G00X51Z6;  
G00X50Z4;  
G01X0;  
G00X51Z5;  
G00X50Z2;  
G01X0;  
G00X51Z3;  
G00X50Z0;  
G01X0;  
G00X51Z2;  
G01X48;  
G01Z-70  
G00X51Z2;  
G00X46Z0;  
G01Z-70;  
G00X51Z2;  
G01X44Z0;  
G01Z-70;  
G00X51Z2;  
G01X42Z0;  
G01Z-70;  
G00X51Z2;  
G01X40Z0;
```



## BABA FARID GROUP OF ENGINEERING AND TECHNOLOGY

G01Z-70;  
G00X51Z2;  
G00X38Z2;  
G01Z0F.5  
G01X40Z-2;  
G00X41Z1;  
G00X36Z0;  
G01X40Z-4;  
G00X48Z-69;  
G01Z-71;  
G03X50Z-72CR=2;  
G00X51Z-69;  
G01X46Z-70;  
G03X50Z-74CR=4  
G00X55Z15;  
T2D2;  
G00Z0X40;  
CYCLE97 (5.000 , , 3.000, -70.000,  
40.000, 40.000, 0.000, 0.000, 2.500,  
0.030, 60.000, 0.000, 10.000,  
2.000, 1, 1.000)  
M05M09M30;

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### **HOW TO RUN THE PROGRAM IN THE MACHINE.**

1. Select the program you type and saved before with your specified name.
2. Click on RENUMBER.
3. Press EXECUTE.
4. Close the machine door.
5. Click on the MAIN POSITION in program panel.
6. Choose the auto mode.
7. And click on the START CYCLE.
8. PROGRAM RUNS.