BHARTIYA INSTITUTE OF ENGINEERING & TECHNOLOGY, SIKAR

LAB MANUAL

VIII SEMESTER

CAD LAB

Subject Code - 8ME6A

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1. INTRODUCTION

Computer Aided Drafting is a process of preparing a drawing of an object on the screen of a computer. There are various types of drawings in different fields of engineering and sciences. In the fields of mechanical or aeronautical engineering, the drawings of machine components and the layouts of them are prepared. In the field of civil engineering, plans and layouts of the buildings are prepared. In the field of electrical engineering, the layouts of power distribution system are prepared. In all fields of engineering use of computer is made for drawing and drafting.

The use of CAD process provides enhanced graphics capabilities which allows any designer to

- Conceptualize his ideas
- Modify the design very easily
- Perform animation
- Make design calculations
- Use colors, fonts and other aesthetic features.

REASONS FOR IMPLEMENTING A CAD SYSTEM

1. **Increases the productivity of the designer**: CAD improves the productivity of the designer to visualize the product and its component, parts and reduces the time required in synthesizing, analyzing and documenting the design

2. **Improves the quality of the design**: CAD system improves the quality of the design. A CAD system permits a more detailed engineering analysis and a larger number of design alternatives can be investigated. The design errors are also reduced because of the greater accuracy provided by the system

3. **Improves communication:** It improves the communication in design. The use of a CAD system provides better engineering drawings, more standardization in the drawing, better documentation of the design, few drawing errors and legibility.

4. **Create data base for manufacturing:** In the process of creating the documentation for these products, much of the required data base to manufacture the products is also created.

5. **Improves the efficiency of the design:** It improves the efficiency of the design process and the wastage at the design stage can be reduced.

APPLICATION OF CAD:

There are various processes which can be performed by use of computer in the drafting process.

1. **Automated drafting**: This involves the creation of hard copy engineering drawings directly from CAD data base. Drafting also includes features like automatic dimensioning, generation of cross – hatched areas, scaling of the drawing and the capability to develop sectional views and enlarged views in detail. It has ability to perform transformations of images and prepare 3D drawings like isometric views, perspective views etc.,
2. **Geometric modeling**: concerned with the computer compatible mathematical description of the geometry of an object. The mathematical description allows the image of an object to be displayed and manipulated on a graphics terminal through signals from the CPU of the CAD system. The software that provides geometric modeling capabilities must be designed for efficient use both by computer and the human designer.

BENEFITS OF CAD:

The implementation of the CAD system provides variety of benefits to the industries in design and production as given below:

1. Improved productivity in drafiting
2. Shorter preparation time for drawing
3. Reduced man power requirement
4. Customer modifications in drawing are easier
5. More efficient operation in drafting
6. Low wastage in drafting
7. Minimized transcription errors in drawing
8. Improved accuracy of drawing
9. Assistance in preparation of documentation
10. Better designs can be evolved
11. Revisions are possible
12. Colours can be used to customize the product
13. Production of orthographic projections with dimensions and tolerances
14. Hatching of all sections with different filling patterns
15. Preparation of assembly or sub assembly drawings
16. Preparation of part list
17. Machining and tolerance symbols at the required surfaces

18. Hydraulic and pneumatic circuit diagrams with symbols
19. Printing can be done to any scale

LIMITATIONS OF CAD

1. 32 – bit word computer is necessary because of large amount of computer memory and time
2. The size of the software package is large
3. Skill and judgment are required to prepare the drawing
4. Huge investment

CAD SOFTWARES

The software is an interpreter or translator which allows the user to perform specific type of application or job related to CAD. The following softwares are available for drafting.

1. AUTOCAD
2. Pro – E
3. CATIA
4. MS OFFICE
5. PAINT
6. ANSYS
7. MSc.NASTRAN
8. IDEAS
9. SOLID WORKS
10. HYPERMESH
11. FLUENT – GAMBIT

The above software is used depending upon their application.

AUTO CAD

Auto CAD package is suitable for accurate and perfect drawings of engineering designs. The drawing of machine parts, isometric views and assembly drawings are possible in AutoCAD. The package is suitable for 2D and 3D drawings.

2. AutoCAD – BASICS

2.1 STARTING WITH ACAD

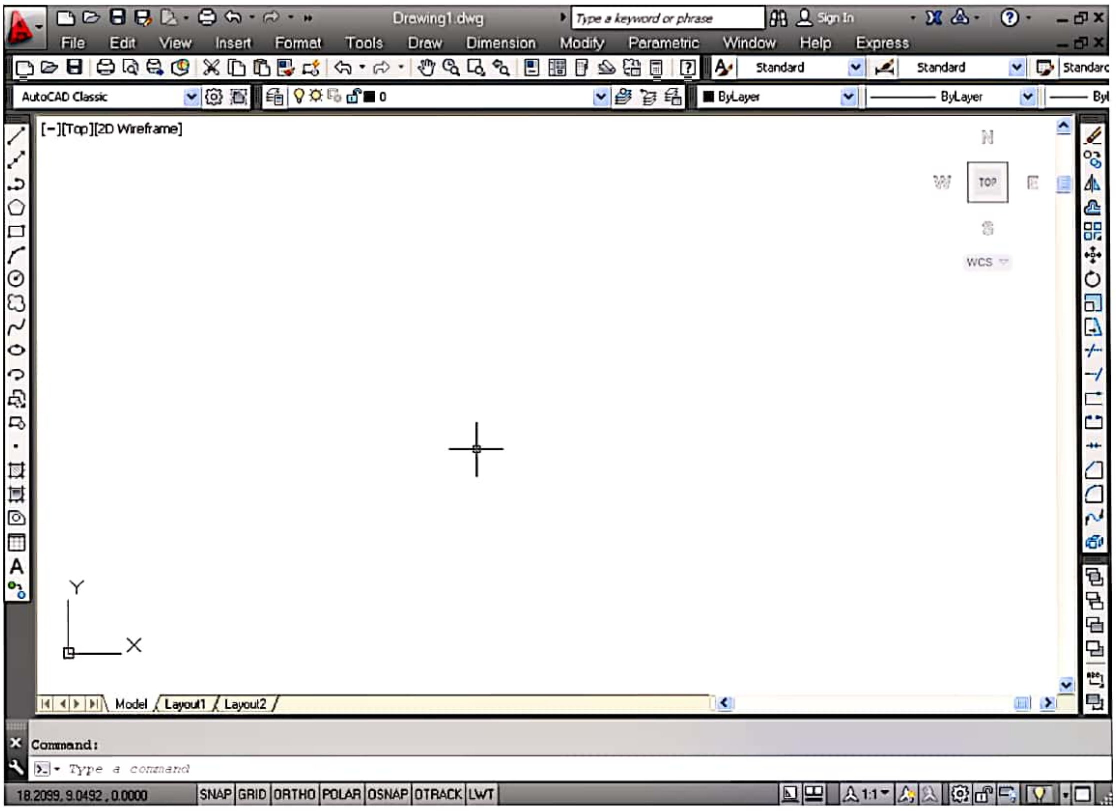
CAD uses four basic elements for preparation of any drawing:

1. Line
2. Curves
3. Text
4. Filling point.

Computer Aided Drafting is done by the operator by placing the mouse pointer by placing the mouse pointer at the desired location and then executing the command to draw the graphic elements using different methods.

Advanced computer aided drafting packages utilize four areas on the screen.

1. Drawing Area
2. Command Area
3. Menu Area
4. Tool boxes.



2.2 LAYOUT AND SKETCHING

The package provides various facilities for layout, sketching and borders for preparing a drawing. It provides facilities for display co-ordinates and measurement units.

a. **Units:** The format for display co – ordinates and measurement can be selected according to the requirement. Several measurement styles are available in ACAD. The main methods are engineering and architectural, having specific base unit assigned to them.

i. Decimal: select to enter and display measurements in decimal notation

ii. Engineering: Display measurements in feet and decimal inches.

iii. Architectural: Display measurements in feet, inches and fractional inches

iv. Fractional: Display measurements in mixed numbers notation

v. Scientific: Display measurements in scientific notation.

The precision that is specified controls the number of decimal places or fractional size to which we want linear measurements displayed.

b. **Angles**: Select the format in which we want to enter and display angles.

i. Decimal Degrees: Display partial degrees as decimals

ii. Deg/Min/Sec: Display partial degrees as minutes and seconds.

iii. Grades: Display Angles as grades

iv. Radians: Display angles as radians.

v. Surveyor: Displays angles in surveyor units.

c. **Angle measure:** Select the direction of the zero angle for the entry of angles:

i. East: Select to specify the compass direction east as the zero angle.

ii. North: Select to specify the compass direction north as the zero angle.

iii. West: Select to specify the compass direction west as the zero angle.

iv. South: Select to specify the compass direction south as the zero angle.

v. Other: Select to specify a direction different from the points of the compass as the zero angles.

d. **Area:** Enter the approximate width and length which is planned to draw in full scale units. This limits the area of the drawing covered by grid dots when the grid is turned on. It also adjusts several default settings, such as text height, line type scaling and snap distance to convenient values. It is possible to adjust these settings.

e. **Title block:** Select the description of an ACAD drawing file of a title block to insert as a symbol in the new drawing. It can add or remove drawing files of title blocks from the list with the Add or Remove buttons

f. **Layout:** Paper space is often used to create complex multiple view drawings. There are three types of paper spaces:

1. Work on the drawing while viewing the layout.
2. Work on the drawing without the layout visible
3. Work on the layout of the drawing.

The following procedure is used for this purpose

1. From the File menu or from the standard tool bar, choose New
2. In the start up dialog box, choose Use a wizard, and select Advanced wizard
3. Choose OK
4. In the Advanced Setup Dialog box, Select Title Block.
5. Select Title Block Description and Title Block file Name from the lists and then choose Add.
6. In the Select Title Block File dialog box, Select a title block, then choose open
7. In the Advanced Setup dialog box, a sample of that title is displayed.
8. Choose Done.

2.3 DRAWING ENVIRONMENT

ACAD provides two drawing environments for creating and laying out the drawing.

i. Model Space
ii. Layout Space.

ACAD allows creating drawing, called a model, in full scale in an area known as model space without regard to the final layout or size when the drawing is plotted on the paper.

In the space opened for the first time, it is possible to create floating viewports to contain different views of the model. In the paper space, floating viewports are treated as objects which can be moved and resized in order to create a suitable layout.

LIMITS

This sets and controls the drawing boundaries.

At the command prompt, enter **limits**

ON/OFF/<LOWER LEFT CORNER> <current>: Specify a point, enter on or off, or press

enter.

LTSCALE

This sets the line type scale factor. Use LTSCALE to change the relative length of the dash – dot line types per drawing unit

At the Command prompt, enter **ltscale**

New scale factor <current> : Enter a positive real value or press enter

Changing the line type scale factor causes the drawing to regenerate.

MEASURE

This places point objects or blocks at measured intervals on an object.

At the command prompt, enter **measure**

Select object to measure: Use an object selection method <segment length> / Block: Specify a distance.

PAN

This moves the drawing display in the current viewport.

At the command prompt , enter **pan**

Displacement: Specify a point (1)

The point which specify indicates the amount to move the drawing or the location of the drawing to be moved.

Second point: Press or specify a point (2)

If pressed, ACAD moves the drawing by the amount which is specified in the Displacement prompt. If we specify a point, ACAD moves the location of the drawing to that point.

2.4 ELEMENTS OF DRAWING

2.4.1 DRAW COMMANDS

LINE:

A line is specified by giving its two end points or first point and the distance of line along with its angle of inclination. A line can be drawn by using two commands.

Command: **line**

Specify first point: Specify a point (1)

Specify next point or [Undo]: Specify a point (2)

The second point can be indicated by **@d<a**

Where **d** is the distance of line and **a** is the angle of inclination in degrees.

PLINE:

This is a poly line which allows continuous segment of the line and it is drawn similar to the line command. The polyline allows changing the thickness of the line according to the requirement.

From the Draw tool bar choose the Polyline flyout.

Draw pull down menu: **Polyline**

At the command prompt, enter **pline**

Syntax

Specify start point: Specify a point (1)

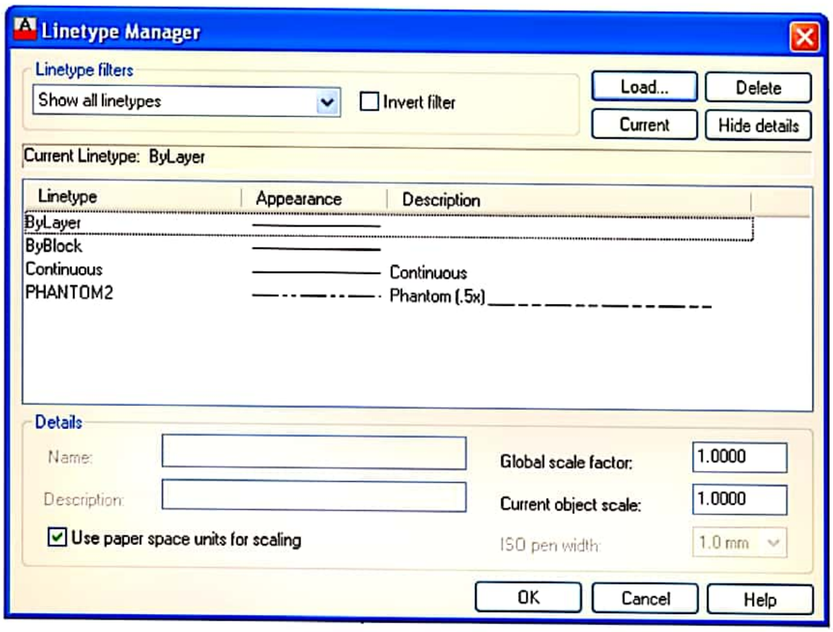
Current line-width is <current>

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: Specify a point (2) or enter an option

b. LINETYPE

Creates, loads, and sets linetypes. The LINETYPE command defines line characteristics consisting of dashes, dots, and spaces.

Format menu: Linetype or Command line: linetype



1. CURVES

Following are the various types of curves used in the drawings:

i. Circle

ii. Ellipse

iii. Arc

iv. Regular or any other type.

i. Circle: The circle can be drawn by using two types of commands

a. Circle

b. Donut

a. **CIRCLE:** This command draws the circle by using four methods:

i. Center point and radius

ii. Two point circle

iii. Three point circle

iv. Tangent circle

At the command prompt, enter **circle**

Specify center point for circle or [3P (Three Points)/2P (Two Points)/Ttr]: Specify a point or enter an option

b. **DONUT:** This draws filled circles and rings.

Donuts are constructed of a closed polyline composed of wide arc segments.

At the command prompt, enter **donut**

Specify inside diameter of donut <current>: Specify a distance or press ENTER

If you specify an inside diameter of 0, the donut is a filled circle.

Specify outside diameter of donut <current>: Specify a distance or press ENTER

Specify center of donut or <exit>: Specify a point (1) or press ENTER to end the command

ii. **ELLIPSE:** It is a curve having major and minor axis with a center.

The ellipse can be prepared by four methods.

Axis endpoint

Arc

Centre

Iso circle

ELLIPSE

Creates an ellipse or an elliptic arc.

Axis end point: Defines the first axis by two specified endpoints. The angle of the first axis determines the angle of the ellipse. The first axis can define either the major or the minor axis of the ellipse.

Arc: Creates an elliptical arc. The angle of the first axis determines the angle of the elliptical arc. The first axis can define either the major or the minor axis of the elliptical arc.

Center: Creates the ellipse by a specified center point.

Isocircle: Creates an isometric circle in the current isometric drawing plane.

At the command prompt, enter **ellipse**

iii. **Arc:** The arc is a curve specified by center and radius as well as the start angle and end angle . There are seven method used for drawing an arc.

1. Three point method
2. Start point-center point –end point
3. Start point-center point-length of chord
4. Start point-end point –angle of inclusion
5. Start point-end point-direction
6. Start point-center point-angle of inclusion
7. Start point-end point-radius

These methods can be used by executing the arc command

- **ARC:** creates an arc.

At the command prompt, enter arc

Center/<start point>: specify a point, enter c, or press enter

- **Polyarc:** the second method of the drawing the arc is poly arc by use of pline command. This command allows drawing of filled arc of any width .it also allows for drawing of a regular or irregular curve.

2. Drawing of Rectangle: A rectangle can be drawn by LINE command or by Rectangle command. The **PLINE** command also allows for drawing of hollow or filled rectangle .A **SOLID** command is also used for drawing of filled rectangles.

1. **RECTANGLES:** draws a rectangular polyline

At the command prompt, enter **rectangle**

First corner: specify point (1)

Other corner: specify point (2)

2. **SOLID:** creates solid –filled polygons .solids are filled only when fill system variable is set to on view is set to plan.

At the command prompt, enter **solid**

First corner: specify point (1)

Other corner: specify point (2)

The first two points define one edge of the polygon.

Third point: specify a point (3) diagonally opposite the second

Forth point: specify a point (4) or press enter

3. DRAWING OF POLYGON

Creates an equilateral closed polyline .A polygon is a polyline object. AUTOCAD draws polyline with zero width and no tangent information.

At the command prompt enter **polygon**

number of sides **<current>**: enter a value between **3 and 1024** or press enter

Edge/<center of polygon>: **specify a point (1) or enter**.

4. POINT

Creates a point object .points can act as nodes to which you can snap objects .you can specify a full 3D location for a point.

At the command prompt, enter **point**

Point: specify a point

5. ERASING OF OBJECT:

The object can be removed or erased by use of erase command

ERASE

This removes object from drawing

At the command prompt, enter **erase**

Select objects: use an object selection method.

6. COLOURING OF OBJECT:

The object can be drawn with any variety of colour which ranges from 0 to 256.

The setting of colour can be done by color command

COLOR

Sets the colour for new objects.

At the command prompt, enter color <current>:enter a value (1-255),color name ,by block, or by layer

7. FILLING OF OBJECT: the object can be filled with different colors and patterns by use of hatch command

This command allows selection of various patterns, scale of pattern and angle of pattern.

HATCH

This fills an area with a pattern.

HATCH fills the specified hatch boundary with non-associative hatch

A non –associative hatch is not updated when its boundaries are modified .a hatch boundary consists of an object or objects that completely enclose an area

At the command prompt, enter hatch

Pattern (? Or name/ U, style) <current>: enter a predefined pattern name, enter u, enter? Or press enter.

8. **SCALING OF DRAWING:** zoom command displays the object at a specified scale factor. The value entered is relative to the limits of the drawing .for example, entering 2 doubles the apparent display size of any objects from what it would be if it were zoomed to the limits of the drawing. If you enter a value followed by xp, auto CAD specifies the scale relative to paper scale units for example; entering 0.5xp displays model space at half the scale of paper space units. The following illustration shows a number of viewports arranged in paper space. the view in each view port is scaled relative to paper space .the first view is scaled 1=1 relative to paper space (1xp) ,the second is scaled 0.5=1 relative to paper space (0.5xp),and so on.

ZOOM

This increases or decreases the apparent size of objects in the current view port

At the command prompt, enter zoom

All/center/dynamic/extents/left/previous/vmax/window/<scale(x/xp)>:enter an option or value ,specify a point ,or press enter

9. **TEXT:** The text in software is indicated by font's .the fonts define the shapes of the text characters that make up each character set. In AUTOCAD, you can use true type fonts in addition to AUTOCAD's own compiled shape (SHX) fonts.
A font is indicated by various parameters like

i. Style :these are four types: normal,bold,italic,underline

ii. Size: this is the size of characters

iii. Colour: there are facilities to colour the characters selecting layer.

iv. Type: different types of fonts may be used:

Mono text: COMPUTER AIDED DESIGN

Romans: COMPUTER AIDED DESIGN

Romand: **COMPUTER AIDED DESIGN**

Dtext: This displays text on the screen as it is entered .AutoCAD can create text with a variety of character patterns, or fonts. These fonts can be stretched, compressed, oblique, mirrored, or aligned in a vertical column by applying a style to the font .text can be rotated, justified, and made any size.

At the command prompt, enter **text**

Justify/style/<start point>: specify a point or enter an option

TEXT: This creates a single line of text .AutoCAD can create text with a variety of character patterns, or fonts. These fonts can be stretched, compressed, oblique, mirrored, or aligned in a vertical column by applying a style to the font.

At the command prompt, enter **text**

Justify/style/<start point>: specify a point or enter an option

QTEXT: This controls the display and plotting of text and attribute of objects.

At the command prompt, enter **text**

ON/OFF <current>: enter on or off, or press enter

10. TRANSFORMATIONS: These are the modifications in the drawn objects.

There are different types of transformations used

1. MOVE: This allows to move or displace objects a specified distance in a specified direction

At the command prompt, enter **move**

Select objects: use an object selection method

Base point or displacement: specify a base point (1)

Second point of displacement: specify a point (2) or press enter

2. **COPY:** This is used for producing a duplicate copy of the drawing.

At the command prompt, enter **copy**

Select objects: use an object selection method

<Base point or displacement >/multiple: specify a base point(1)

For a single copy or enter m for multiple copies

3. **ROTATE:** It moves objects about a base point

At the command prompt, enter **rotate**

Select objects: use an object selection method

<Rotate angle >/reference: specify an angle or enter r

4. **STRETCH:** This moves or stretches objects .AutoCAD stretches lines, arcs, elliptical arcs, splines, rays and polyline segments that cross the selection window.

At the command prompt, enter **stretch**

Select objects: use the CPOLYGON or cross object selection method(1,2)

Base point or displacement: specify a point (3) or press

Second point of displacement: specify a point ($) or press

5. **EXTEND:** This extends an object to meet another object. Objects that can be extended include arcs, elliptical arcs, lines, open 2D, and 3Dpolylines and rays.

At command prompt, enter **extend**

Select boundary edges

(projmode=UCS, edge mode=no extend)

Select objects: use an object selection method

6. **SCALE:** This enlarges or reduces selected objects equally in X and Y directions

At the command prompt, enter **scale**

Select objects: use an object selection method

Base point: specify a point (1)

<Scale factor>/reference: specify a scale or enter r

7. **TRACE:** This creates solid lines.

From the miscellaneous tool bar choose

At the command prompt, enter **trace**

Trace width<current>: specify a distance, enter a value ,or press enter

From point: specify point (1)

To point: specify a point (2)

To point: specify a point (3) or press to end the command

8. **EXTRUDE:** This creates unique solid primitives by extruding existing two-dimensional objects extrudes also creates solids by extruding two-dimensional objects along a specified path .we can extrude multiple objects with extrude

At the command prompt enter, **extrude**

Select objects: use an object selection method

Path/<height of extrusion>: specify a distance or enter p

9. **MIRROR:** This is used to producing mirror image of the object

At the command prompt enter, **mirror**

Select objects: use an object selection method

First point of the mirror line: specify a point (1)

Second point: specify a point (2)

10. OFFSET: This creates concentric circles ,parallel lines and parallel curves, offset creates a creates a new object at a specified distance from an existing object or through a specified point

At the command prompt enter, **offset**

Offset distance: specify a distance, enter t or press enter

11. ARRAY: This creates multiple copies of objects in pattern. Arrays are three types.

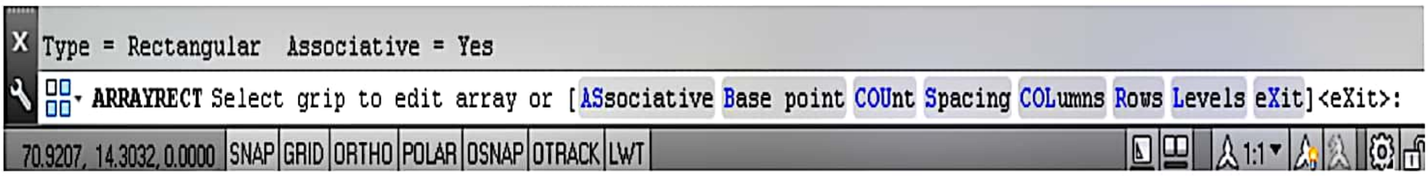
a) Rectangular Array

b) Path Array

c) Polar Array

Rectangular Array: In this, the object is arranged in an array of rows and columns.

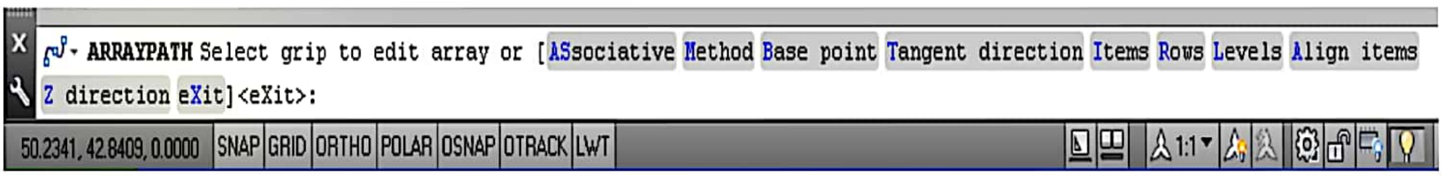
At the command prompt: type **ARRAYRECT** or select the option from **MODIFY** toolbar. It asks you to select objects. Select the object and press enter. By default it shows an array of 3 rows and 4 columns. The no. of rows and columns can be changed by selecting the **COUnt**



option (OR) by selecting COLUMNS and ROWS options separately

Path Array: In this, an object is arranged in a specified path.

At the command prompt: type **ARRAYPATH** or select the option from **MODIFY** toolbar. Then select object to be arrayed. Then select the path through which the object is made to be arrayed.



12. CUTTING OF OBJECTS

The drawn objects can be cut or trimmed by using following commands

1. **TRIM:** Trims objects at a cutting object defined by other objects. Objects that can be trimmed include arcs ,circles, elliptical arcs, lines, open 2D and 3Dpolylines,rays and splines

At the command prompt, enter **trim**

Select cutting edges:

Select objects: use object selection method

<Select object to trim>/project/edge/undo: select an object, enter an option, or press enter

2. **BREAK:** This erases an object or splits the object in to two parts

From the modify toolbar select break flyout

At the command prompt, enter **break**

Select objects: use an object selection method

First point of the mirror line: specify a point (1) on an object

Enter second point: specify the second break point (2) or enter F

13. DIMENSIONING IN DRAWINGS:

The dimensions are inserted in the drawing by use of DIM command. There are various types of dimensions used in AutoCAD.

1. Linear dimensions:

Horizontal- this allows horizontal dimensions

Vertical- this allows vertical dimensions

Aligned- this allows inclined dimensions

Rotated- this allows inclined dimensions

2. Angular dimensions:

This allows angular dimensioning of objects

3. Radial dimensions:

This allows radial dimensioning of arc or circle

4. Diametric dimensions:

This allows diameteral dimensions of the circle

For dimensioning of objects, the first point and second point has to be specified. The dimension text must be written and then the position of dimension must be specified

at the command prompt ,enter **dim**

Dim: Enter a dimensioning mode command

14. AREA:

This allows calculation of the area and perimeter of objects or of defined areas

From the object properties toolbar, choose the inquiry flyout, then

At the command prompt, enter **area**

<First point>/object/add/subtract: specify a point or enter option

15. FILLET

Rounds and fillets the edges of the object

At the command prompt enter **fillet**

Polyline / Radius / Trim / <Select first object>: use an object selection method or enter an option

Select first object

Select second object: use an object selection method

Enter radius <current>: specify a distance or press

Chain / Radius <Select edge>: Select edges or enter **c** or **r** their intersection

16. CO-ORDINATE SYSTEM

The co- ordinate system can be modified in the AutoCAD. There are two types of co-ordinate systems used. The WCS (World co- ordinate system) is a universal system in which its origin is at the fixed position. The UCS (User co- ordinate system) is a system in which user can fix his origin at any point.

1. UCS : This manages user co- ordinate systems

At the command prompt enter **ucs**

Origin / z axis/ 3 point/ object/ view/ X/Y/Z / Prev/ Restore/Save/ Del/?/< world>: enter an option or press enter

2. WCS: This manges world co- ordinate system

17. EXPLODE:

This breaks a compound object into its component objects

At the command prompt enter **explode**

Select objects: use an object selection method.

18. UNION:

This measures the distance and angle between two points.

At the command prompt, enter **union**

Select object: Use an object selection method

19. DIST: This measures the distance and the angle between two points .

At the command prompt area enter **dist**

First point : Specify a point (1)

Second point : Specify a point (2)

Distance = calculated distance

Angle in XY plane = angle from XY plane = angle

Delta X = change in X

Delta Y = change in Y

Delta Z = change in Z.

20. REGENERATION OF DRAWING:

ACAD provides afacility of regenerating a drawing to clear the cross points or marks on the screen.

- REDRAW
- REGEN
- REGENALL
- REGENAUTO

21. TOLERANCE

This creates geometric tolerances. Geometric tolerances define the maximum allowable variations of form or profile, orientation, location and run out from the exact geometry in a drawing. They specify the required accuracy for proper function and fit the objects drawn in AutoCAD

22. SKETCH

This creates a series of free hand line segments.

From the miscellaneous toolbar, choose

At the command prompt enter **sketch**

Follow the prompting

2.5 3D FUNCTIONS

1. BOX

This creates a three dimensional solid box.

At the command prompt enter **box**

Center/<corner of the box><0,0,0> :

Specify a point (1), enter c, or press enter

Corner of a box

Specifying a point or pressing defines the first corner of the box.

Cube/length /<other corner>: specify a point (2) or enter an option **center**

Creates the box by a specified center point

2. CONE

This creates a 3D solid cone. A cone is solid primitive with a circular or elliptical based tapering symmetrically to a point perpendicular to its base.

At the command prompt enter **cone**

Elliptical /<center point> <0,0,0> : specify a point , enter e or press enter

3. CYLINDER

This creates a 3D solid cylinder. A cylinder is solid primitive with a circular or elliptical based to a point perpendicular to its base without a taper.

At the command prompt enter **cylinder**

Elliptical /<center point> <0,0,0> : specify a point , enter e or press enter

4. SPHERE

This creates a 3D solid sphere. A sphere is positioned so that its central axis is parallel to the Z-axis of the current UCS. Latitudinal lines are parallel to the XY plane.

At the command prompt enter **sphere**

Center of the sphere <0,0,0> : specify a point , enter e or press enter

5. WEDGE

This creates a three dimensional solid with a sloped face tapering along X axis.

At the command prompt enter **wedge**

Center <corner of the wedge> <0,0,0> : specify a point , enter e or press enter

Follow the prompting

6. ELEV

This sets an elevation and extrusion thickness of new objects. The current elevation is the Z value that is used whenever a 3D point is expected but only X and y values are supplied.

At the command prompt enter **elev**

Follow the prompting

7. SHADE

This displays a flat shaded image of the drawing in the current view port. SHADE removes hidden lines and displays a shaded picture of the drawing.

From the render toolbar, choose

At the command prompt, enter **shade**

8. REGION

This creates a region object from a selection set of existing objects. Regions are 2Dimensional areas you create from closed shapes.

9. REINIT

This reinitializes the input/output ports, digitizer, display and program parameters file.

10. REPLAY

This displays a GIF, TGA or TIFF image.

From the tools menu, choose image, then view.

11. REVOLVE

This creates a solid by revolving a two – dimensional object about an axis. From the solids toolbar, choose

At the command prompt, enter **revolve**

12. SHAPE

This inserts a shape. Before inserting a shape, you must load the file containing the desired shape.

13. ROTATE 3D

This moves objects about a three dimensional axis

From the modify toolbar, choose the rotate flyout then

Follow the prompting

14. SECTION

This uses the intersection of a plane and solids to create a region.

AutoCAD creates regions on the current layer and inserts them at the location of the cross – section. Selecting several solids creates separate regions for each solid.

15. SLICE

This slices a set of solids with a plane.

16. SHELL

This accesses operating system commands.

17. REVOLVE

This creates a solid by revolving a two dimensional object about an axis.

18. RENDER

This creates a realistically shaded image of a three dimensional wireframe or solid model. RENDER produces an image using information from a scene, the current selection set, or the current view.

2.6 Starting the drawing

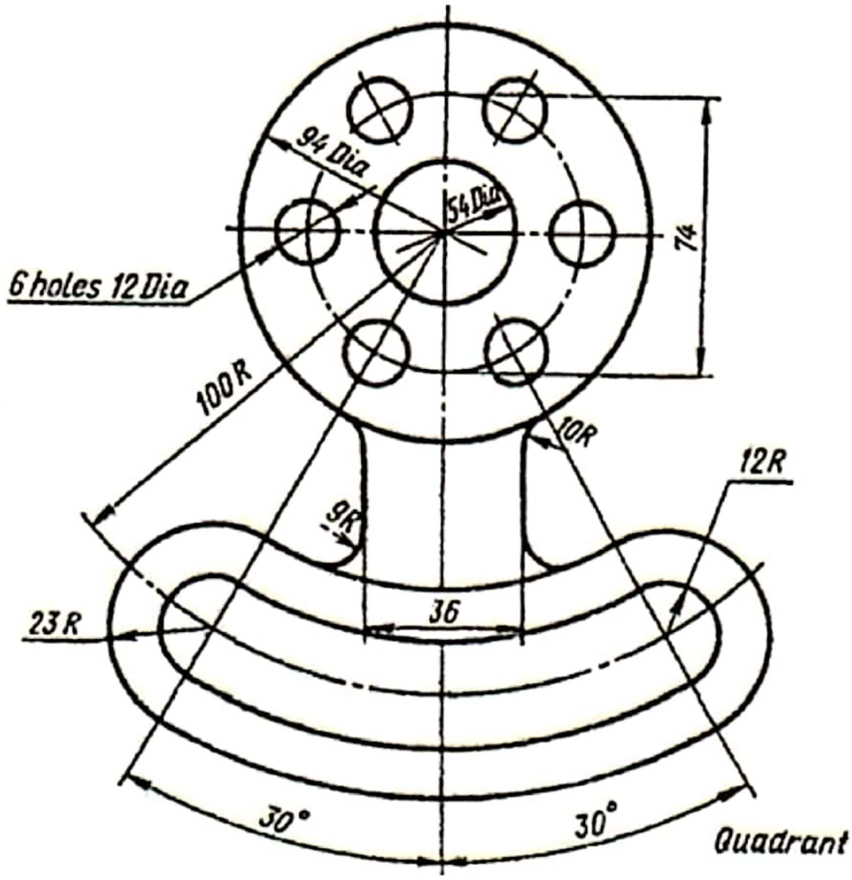
The figures we do in engineering are fitted into a template. In ACAD we manually draw a template known as *Drawing sheet* in two different formats.
The size of the drawing sheet is ISO A4 210 X 297.
The format is as given in the following figures

Polar Array: In this, an object is arranged in a circular shape.
At the command prompt: type **ARRAYPOLAR** or select the option from **MODIFY** toolbar. Then select object to be arrayed. Then select the center point of array. By default, a six items array is created. The No. of items can be changed by selecting the **Items** option. Angle between the two items can also be changed.

3. 2D DRAWINGS

FIGURE 1

Aim: to draw the following figure using ACAD

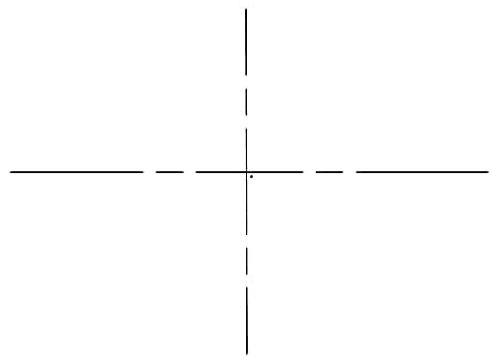


PROCEDURE

Set the limits of the drawing screen

STEP 1: Draw axis lines in the respective format with their intersection point at **(0,0)**

- Go to **PROPERTIES** tool bar
- Load line type as **ISO LONG DASH SHORT DASH** in the line type area.
- Select line type ISO LONG DASH SHORT DASH in the line type area.



STEP 2 a: Draw circles of given dimensions using circle command with their centre as the intersection of the axis lines.

- 3 circles of diameters 94, 74 and 54 are to be drawn
- The circle with 74 diameter is of **ISO LONG DASH SHORT DASH** format

STEP 2 b: Using **POLAR ARRAY** draw the 6 holes on the circle of diameter 74 each of 12 dia.

STEP 3: Draw two construction lines at an angle of 30° to the vertical axis line

STEP 4: With A as center an radius 100 draw an arc between the above lines

STEP 5: Offset the arc on the either side by the distances as mentioned in the figure.

STEP 6: Complete the figure by using fillet command.

STEP 7: Give dimensions to the completed figure.

Command: _qsave

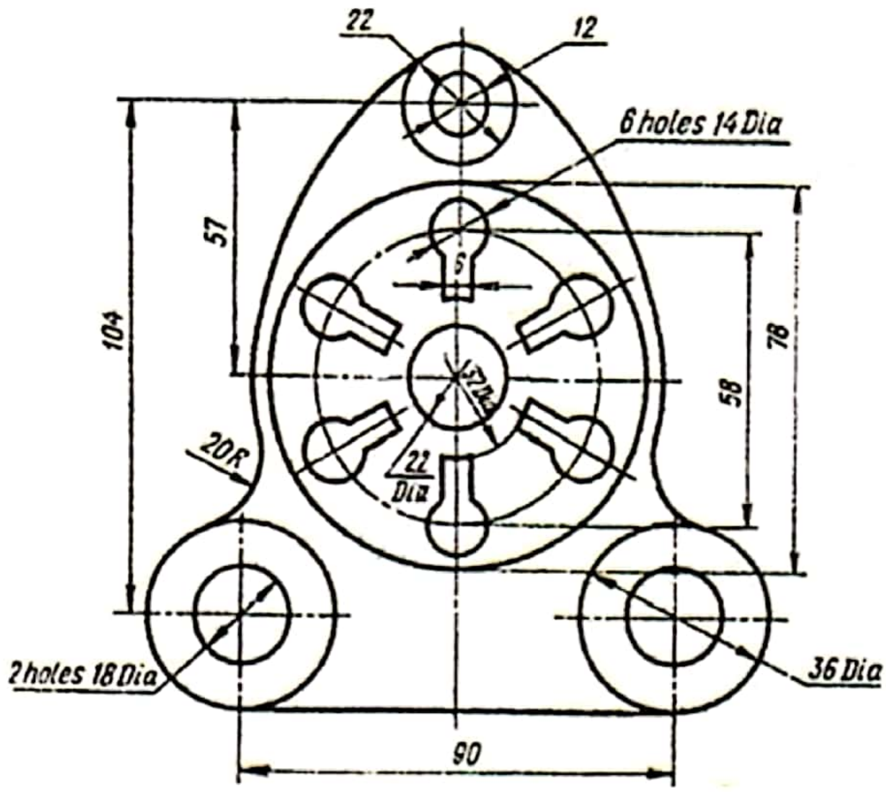
PRECAUTIONS:

Put **ORTHO ON** where ever necessary.

Use the required modify tool bar commands like **TRIM, ERASE, COPY, MIRROR** ETC.,

FIGURE 2

Aim: to draw the following figure using ACAD



PROCEDURE

Set the limits of the drawing screen

STEP 1: Draw axis lines in the respective format with their intersection point at **(0,0)**

STEP 2: Draw circles of given dimensions using circle command with their centre as the intersection of the axis lines.

STEP 3: Using **POLAR ARRAY** draw the 6 key holes on the circle of diameter 58 of given dimensions

STEP 4: For the outer cover use **CIRCLE** command and the in command prompt area type ***TAN TAN RADIUS***. This gives the idea of drawing the outer cover

STEP 5: Give dimensions to the completed figure.

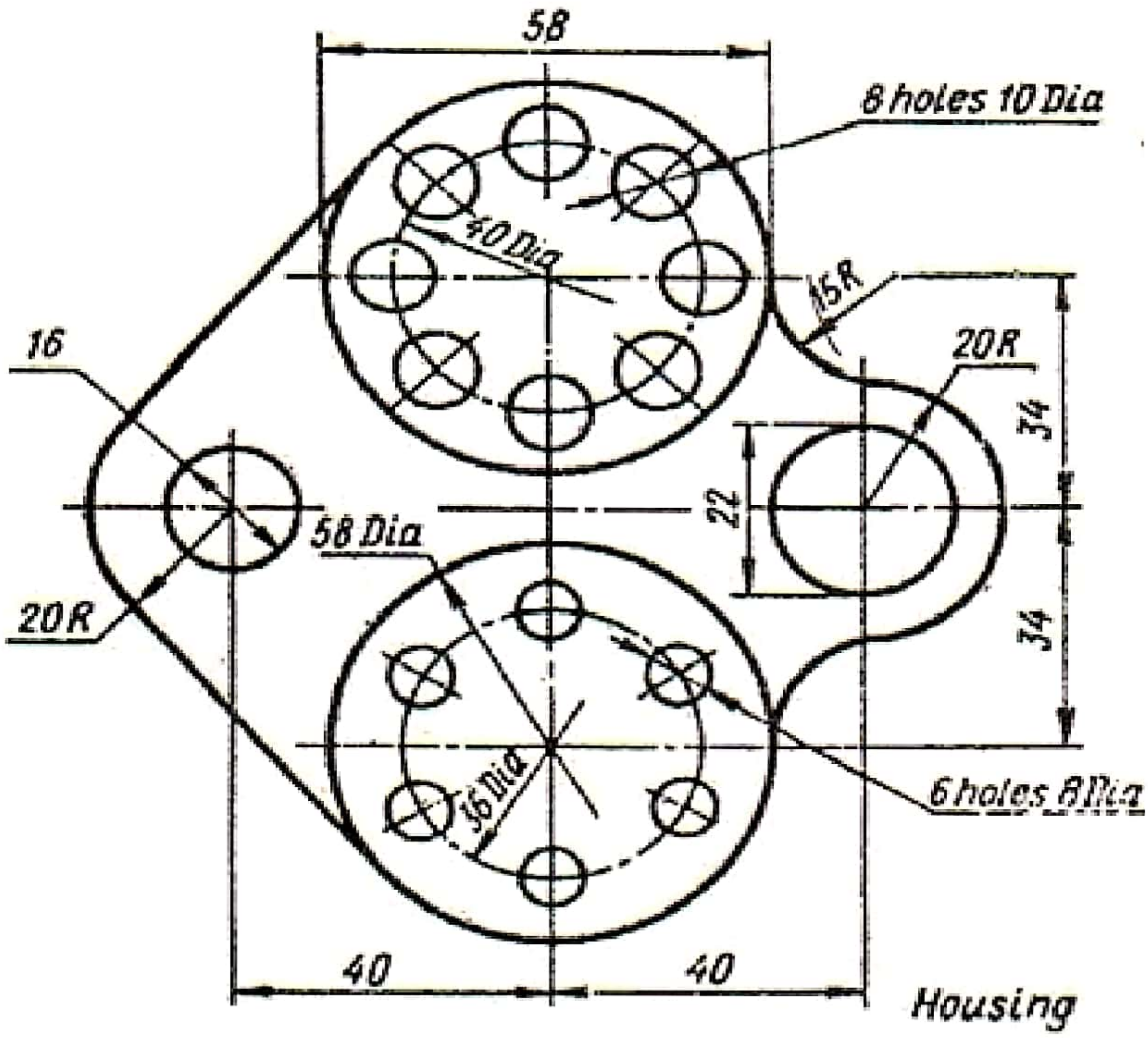
Command: _qsave

PRECAUTIONS:

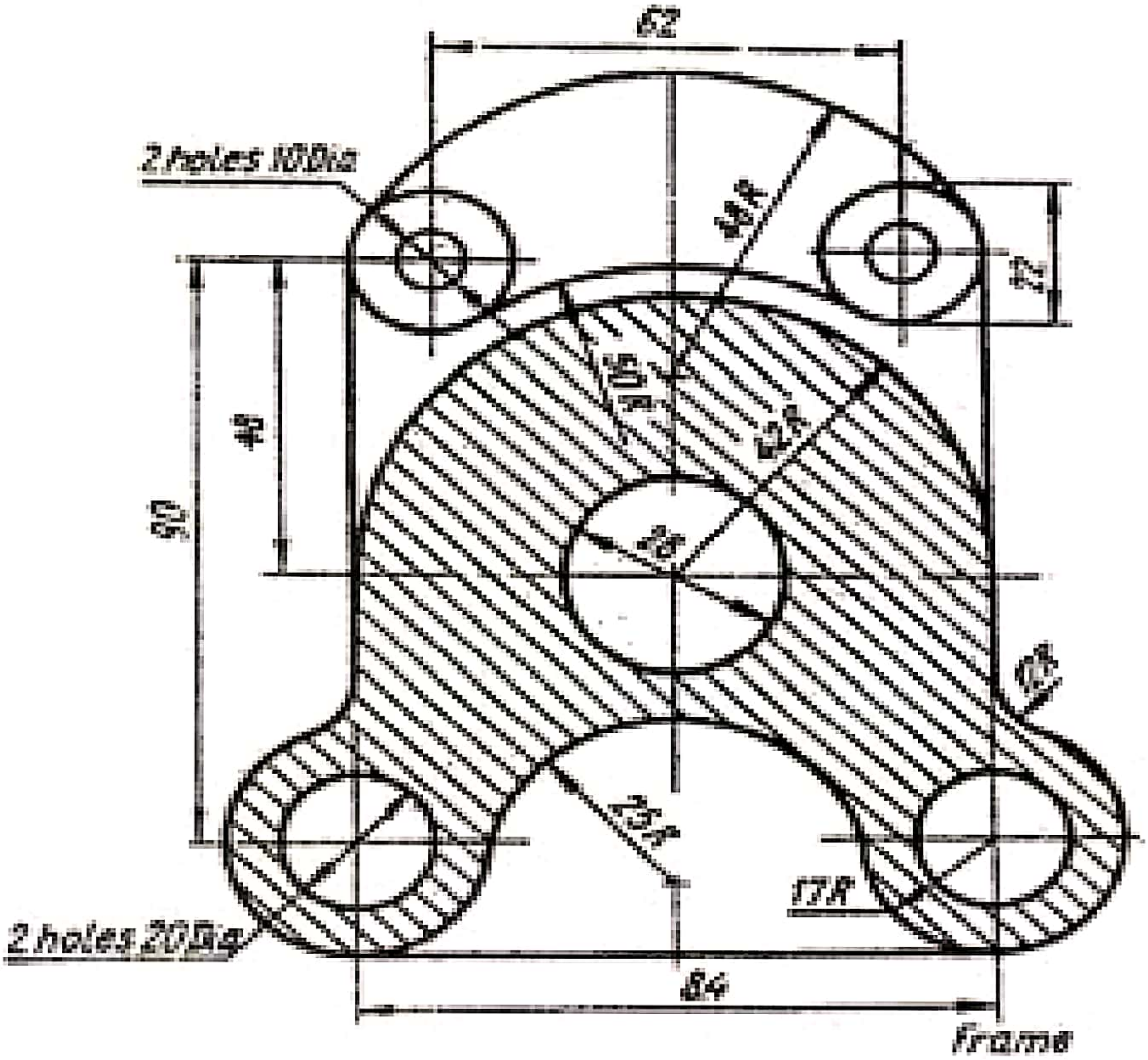
Put **ORTHO ON** where ever necessary.

Use the required modify tool bar commands like **TRIM, ERASE, COPY, MIRROR** ETC.,

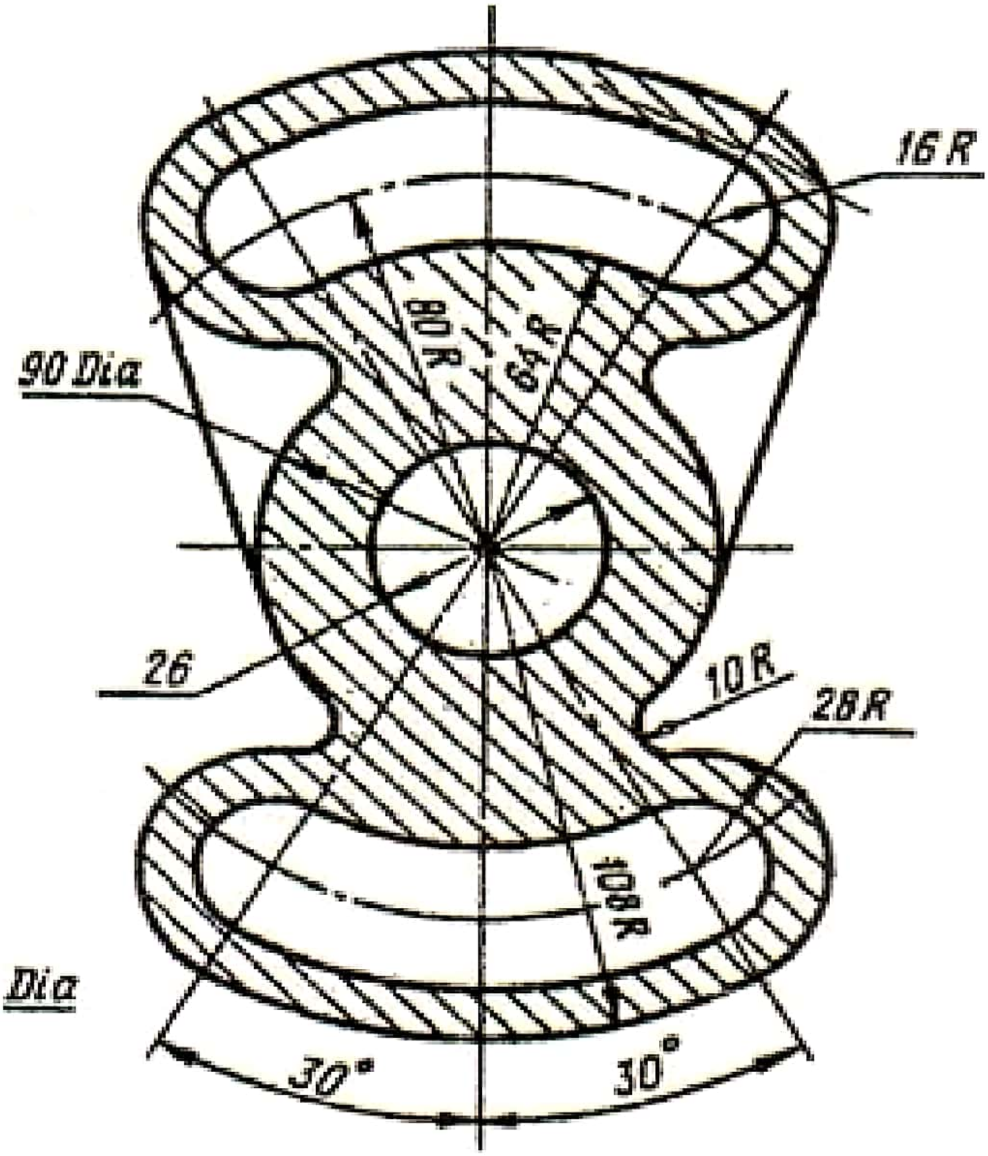
PRACTICE FIGURE 3



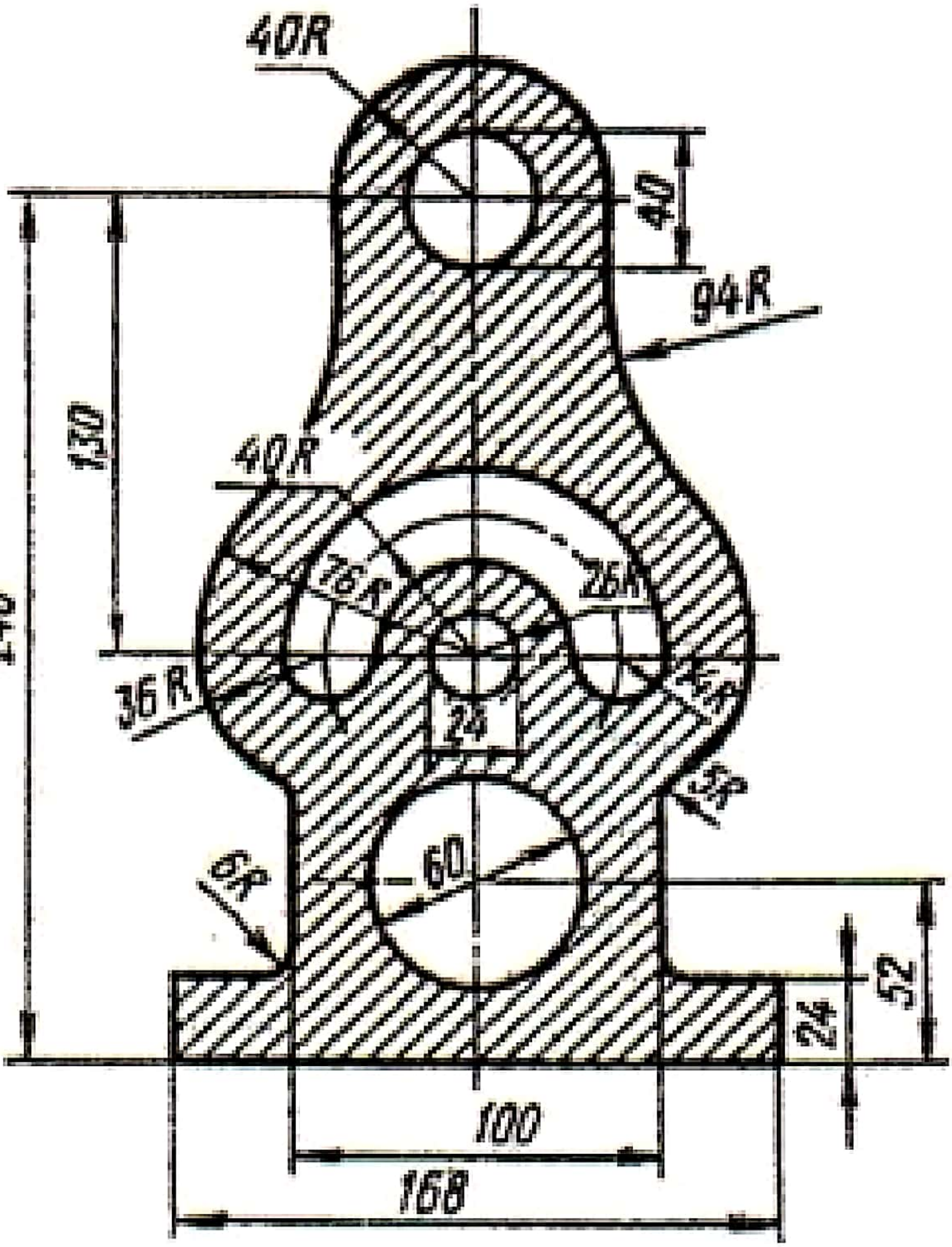
PRACTICE FIGURE 4



PRACTICE FIGURE 5

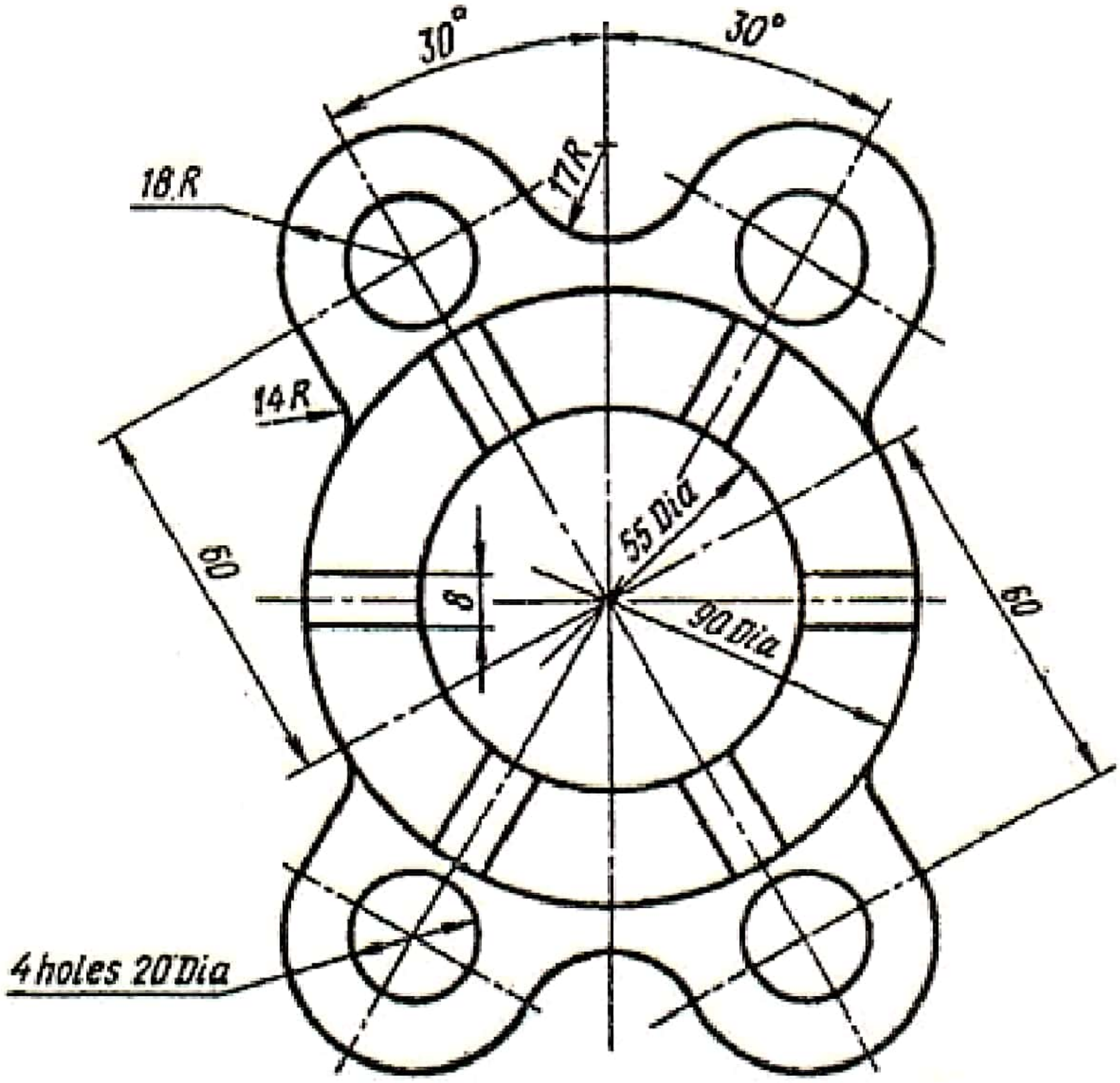


PRACTICE FIGURE 6



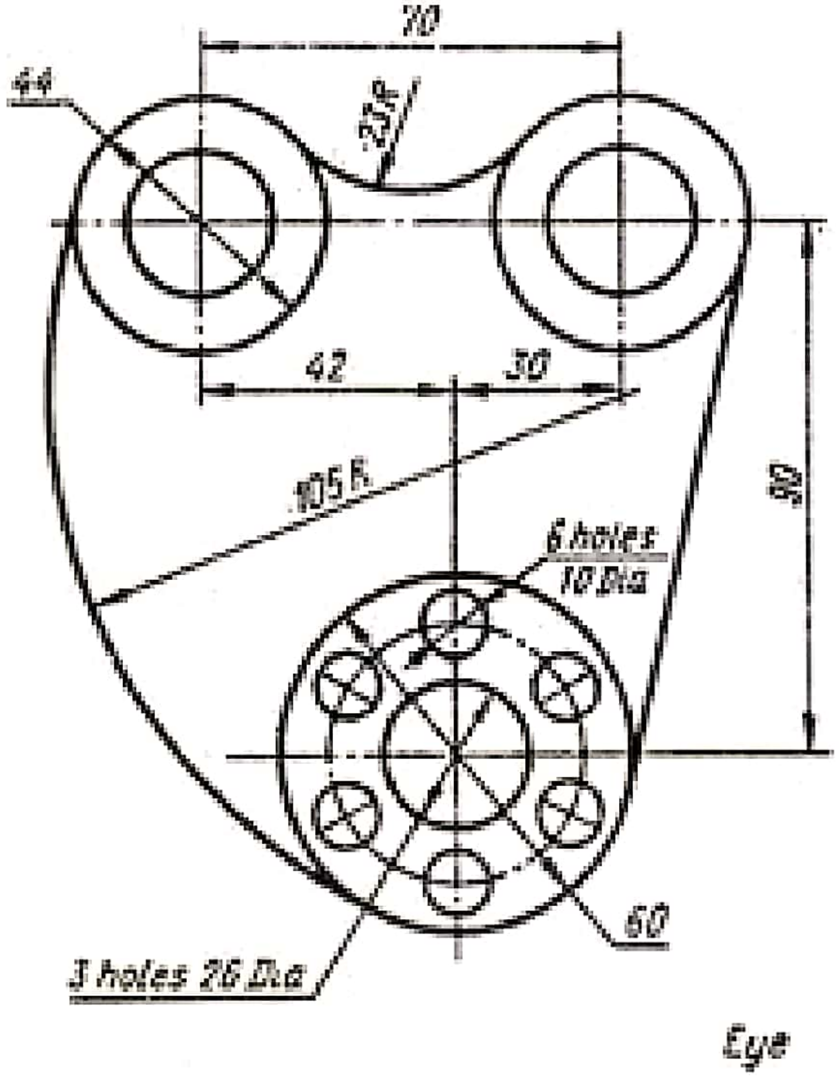
Scanned with CamScanner

PRACTICE FIGURE 7

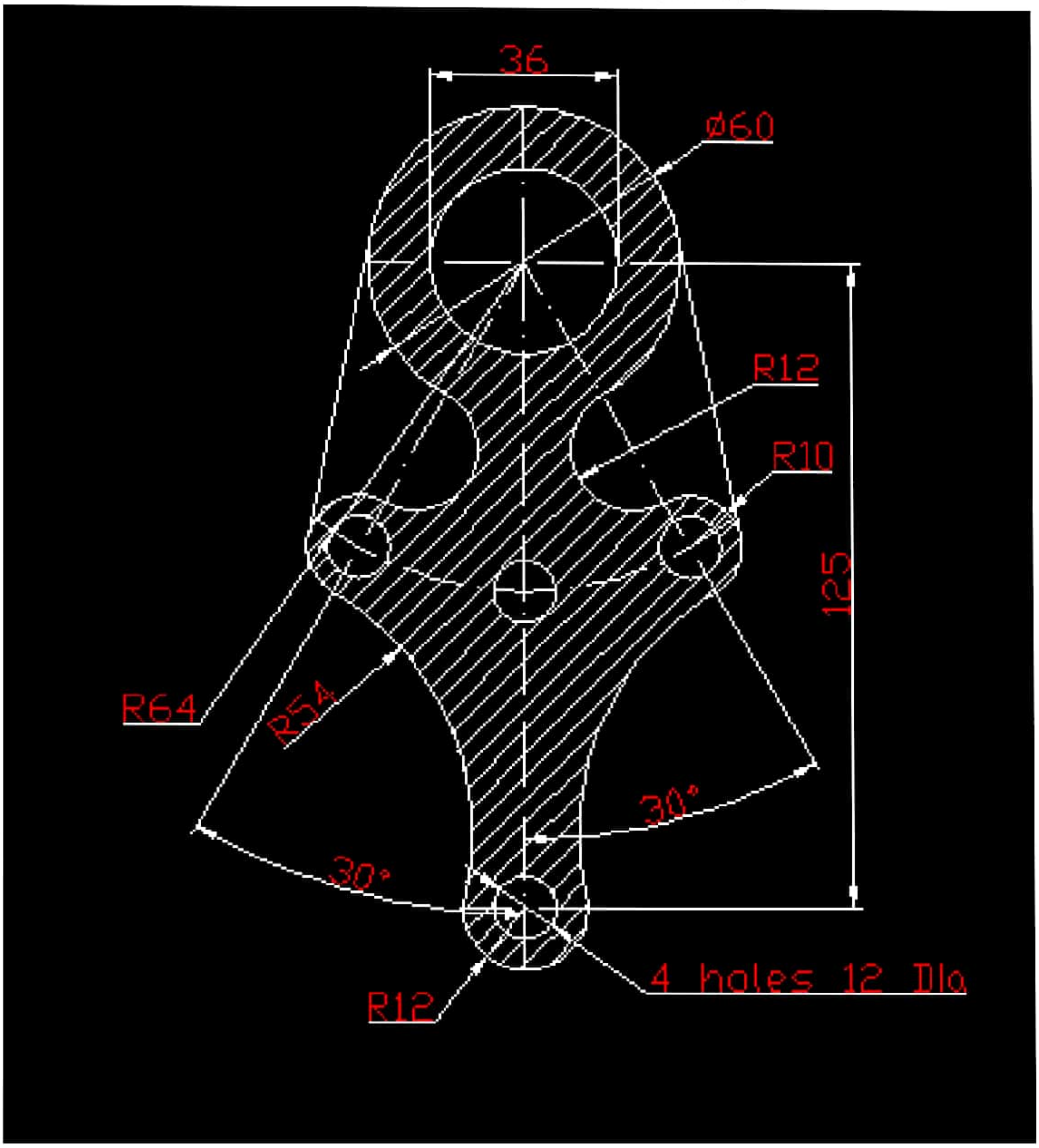


Scanned with CamScanner

PRACTICE FIGURE 8



PRACTICE FIGURE 9



4. ISOMETRIC DRAWINGS

For all isometric figures right click **GRID** in drafting tool bar <setting> change grid snap to **ISOMETRIC SNAP**. And check **ORTHO ON**

F5 – TOGGLE KEY BETWEEN ISOPLANE TOP, ISOPLANE LEFT AND ISOPLANE RIGHT

FIGURE 1

Aim: to draw the following figure using ACAD

COMMANDS USED

Line, Dimensions, Drafting commands

PROCEDURE

<Ortho on> <Isoplane Top> <Osnap on>

Command: _line Specify first point:

Specify next point or [Undo]: **104**

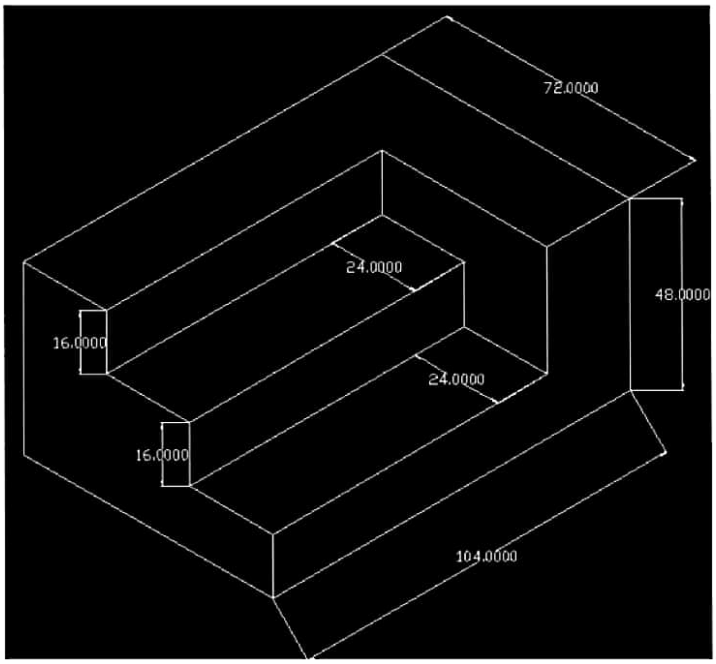
Specify next point or [Undo]:

Command: _qsave

Command: _dimaligned

Specify first extension line origin or <select object>:

Specify second extension line origin:



Command: _dimlinear

Specify first extension line origin or <select object>:

Specify second extension line origin:

Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]:

Dimension text = 48.0000

Command: _dimedit

Enter type of dimension editing [Home/New/Rotate/Oblique] <Home>: **_o**

Select objects: 1 found

Enter obliquing angle (press ENTER for none): **30**

Command: _qsave

FIGURE 2

Aim: to draw the following figure using ACAD

COMMANDS USED

Line, Drafting commands, Dimension aligned, Dimension linear, Dimension oblique, Layers

Command: _line

Specify first point: <Isoplane Left>

Specify next point or [Undo]: 12

Specify next point or [Undo]:

<Isoplane Top> 25

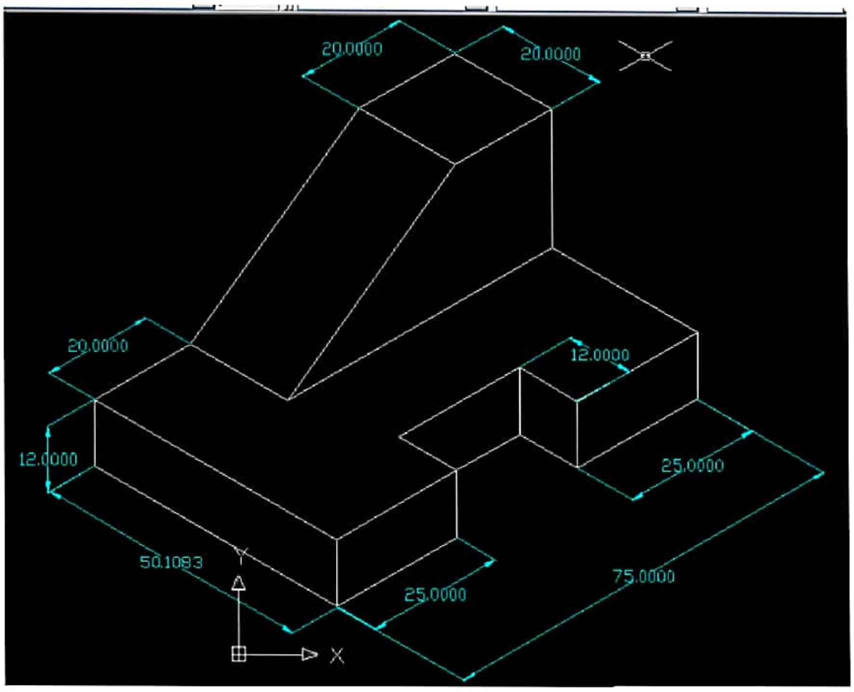
Command: _qsave

Command: _dimlinear

Specify first extension line origin or <select object>:

Specify second extension line origin:

Specify dimension line location or



[Mtext/Text/Angle/Horizontal/Vertical/Rotated]:

Dimension text = 12.0000

Command: _dimaligned

Specify first extension line origin or <select object>:

Specify second extension line origin:

Specify dimension line location or [Mtext/Text/Angle]:

Dimension text = 25.0000

Command: _dimedit

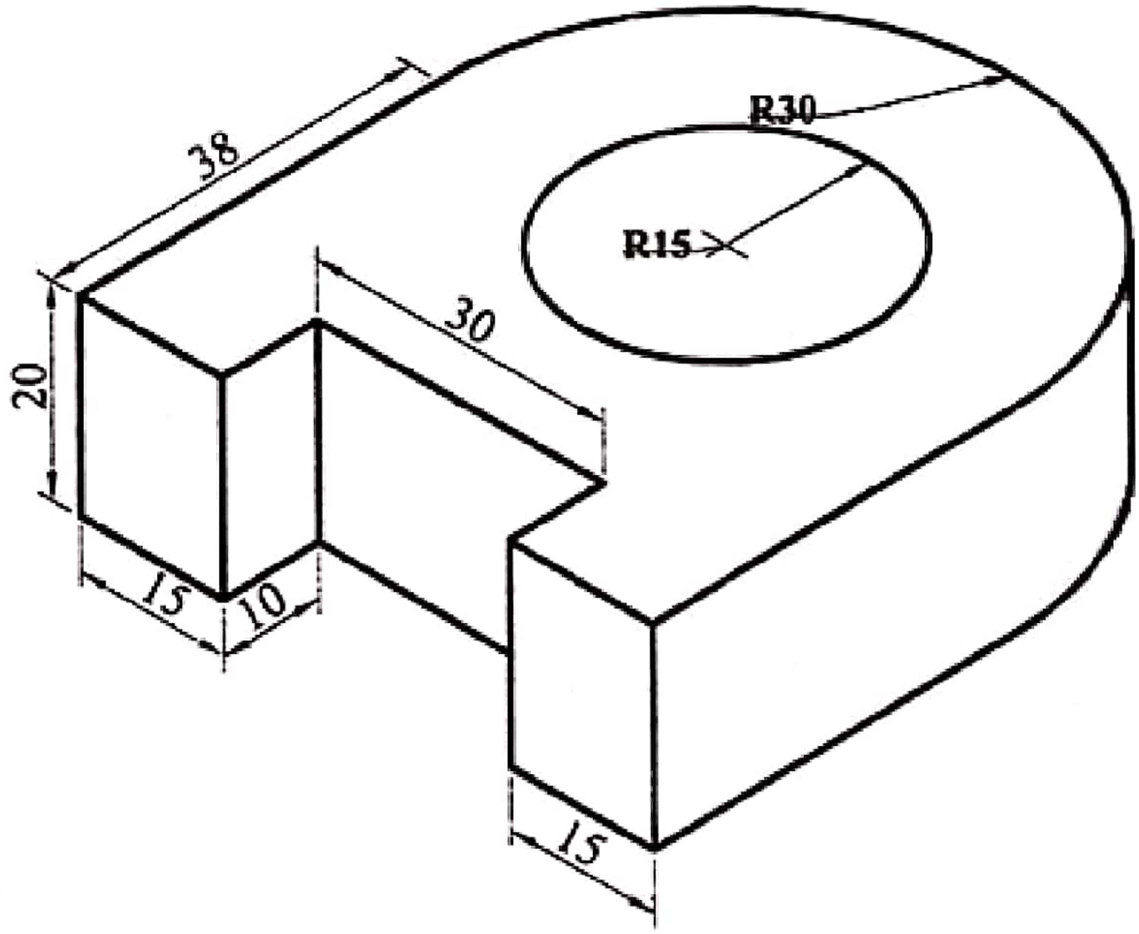
Enter type of dimension editing [Home/New/Rotate/Oblique] <Home>: **_o**

Select objects: 1 found

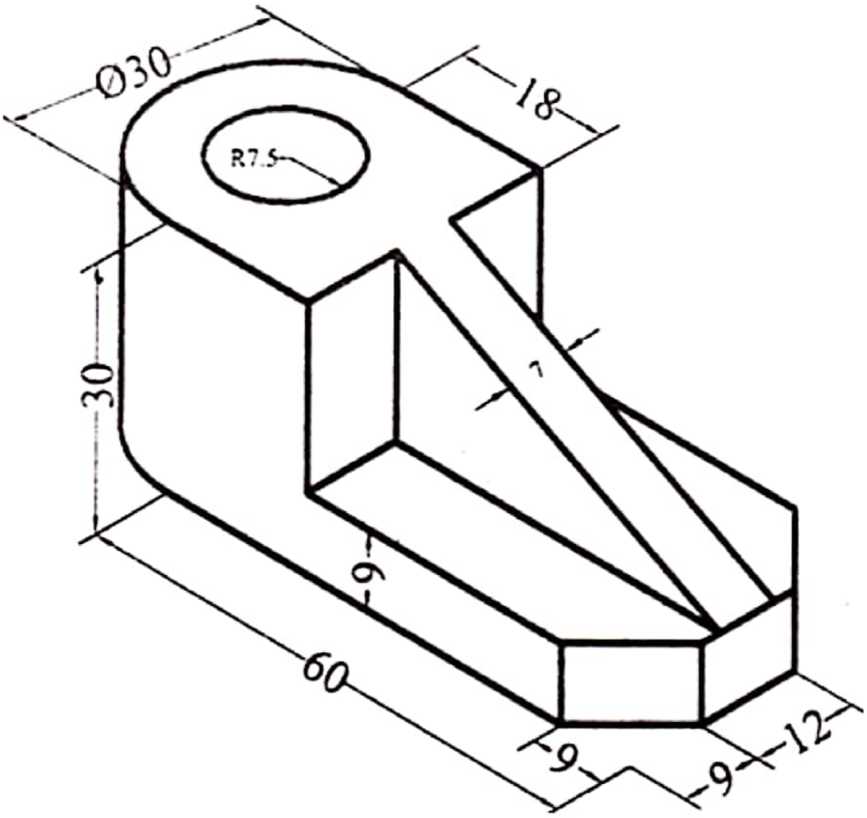
Enter obliquing angle (press ENTER for none): **30 or -30**

Command: _qsave

PRACTICE FIGURE 1



PRACTICE FIGURE 2



*****APAPRT FROM THESE PRACTICE OTHER FIGURES GIVEN IN THE TEXT BOOK. THIS IMPROVES YOUR SKILL OF DRAWING***

5. 3- D DRAWINGS

The commands used while draing 3D solids are briefly discussed in the beginning. The major commands used are:

- **Pedit** – To join the individual poly line segments
- **Explode -** To break the joined part into individual entities
- **Extrude** – To add thickness to the object and create a solid
- **Union –** To unite two solids and form an union
- **Intersect –** To form an intersection of two solids
- **Shade –** This displays a flat – shaded image of the drawing in the current viewport.

SHADE removes hidden lines and displays a shaded picture of the drawing

- **Rotate 3D** – This moves objects about a three dimensional space
- **Revolve –** This creates a solid by revolving a two – dimensional object about an axis
- **Region** – This creates a region object from a selection set of existing objects. Regions are two dimensional areas.
- **View ports –** Splits the screen into a number of ports. Enables us to have a multi viewing of the object at the same time.

VIEW pull down menu → **View ports**

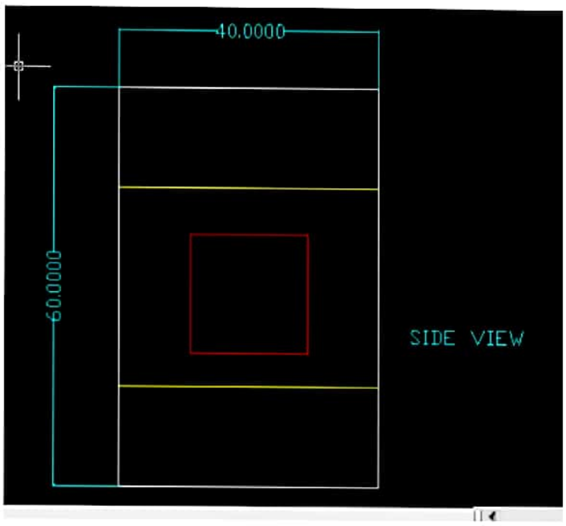
- **3D ORBIT -** Allows us to rotate the object on a 3 dimensional axis.

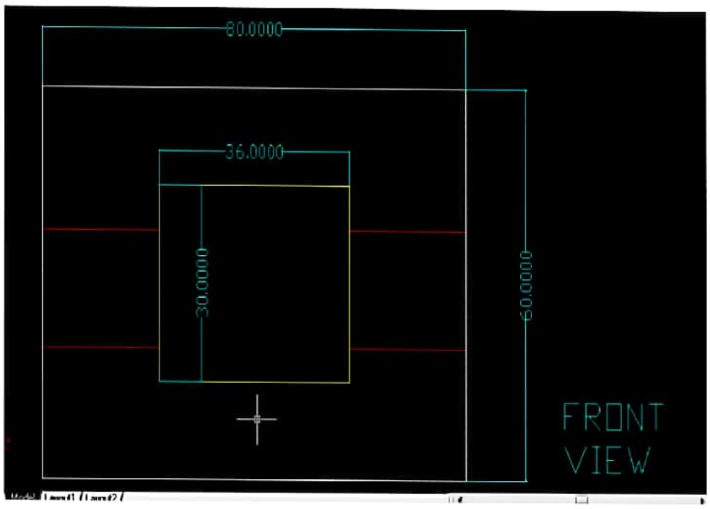
VIEW pull down menu → **3D orbit**

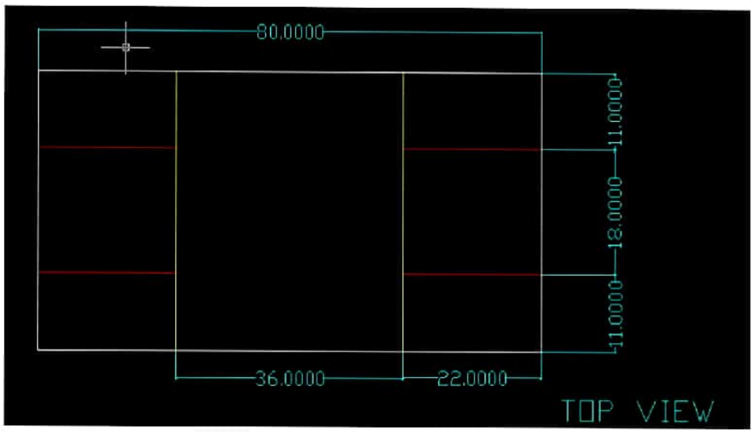
- **3D VIEWS -** Allows us to visualize the views of object object

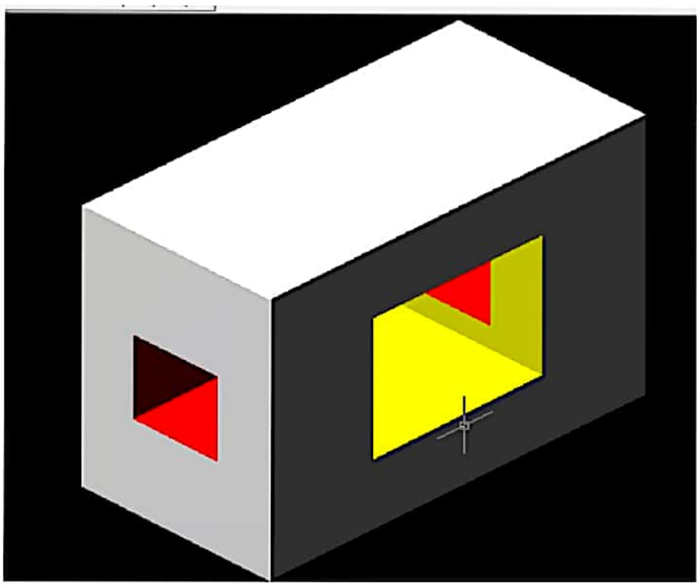
VIEW pull down menu → **3D view**

PRACTICE FIGURE – 1









PROCEDURE:

STEP 1: Set your screen to **SW ISOMETRIC VIEW**.

Go to ***VIEW*** pull down menu.

Select **3D VIEWS** and from the **sub menu select SW ISOMETRIC VIEW**.

STEP 2: Using RECTANGLE command draw the bottom rectangle of dimensions 80 X 60 and **EXTRUDE** it to a height of 40. We get the outside solid

To put the holes required on the front and the side of the solid,

STEP 3: Draw the inner rectangles and extrude them as per give dimensions

STEP 4: Substract the inner solids from the outer one

We can use different colours to distinguish between the solids, so that it is easier to substract.

STEP 5: Shading can be done to the wire frame model using **commands in the SHADE tool bar.**

The front, top and the side views of the figure can be obtained from the

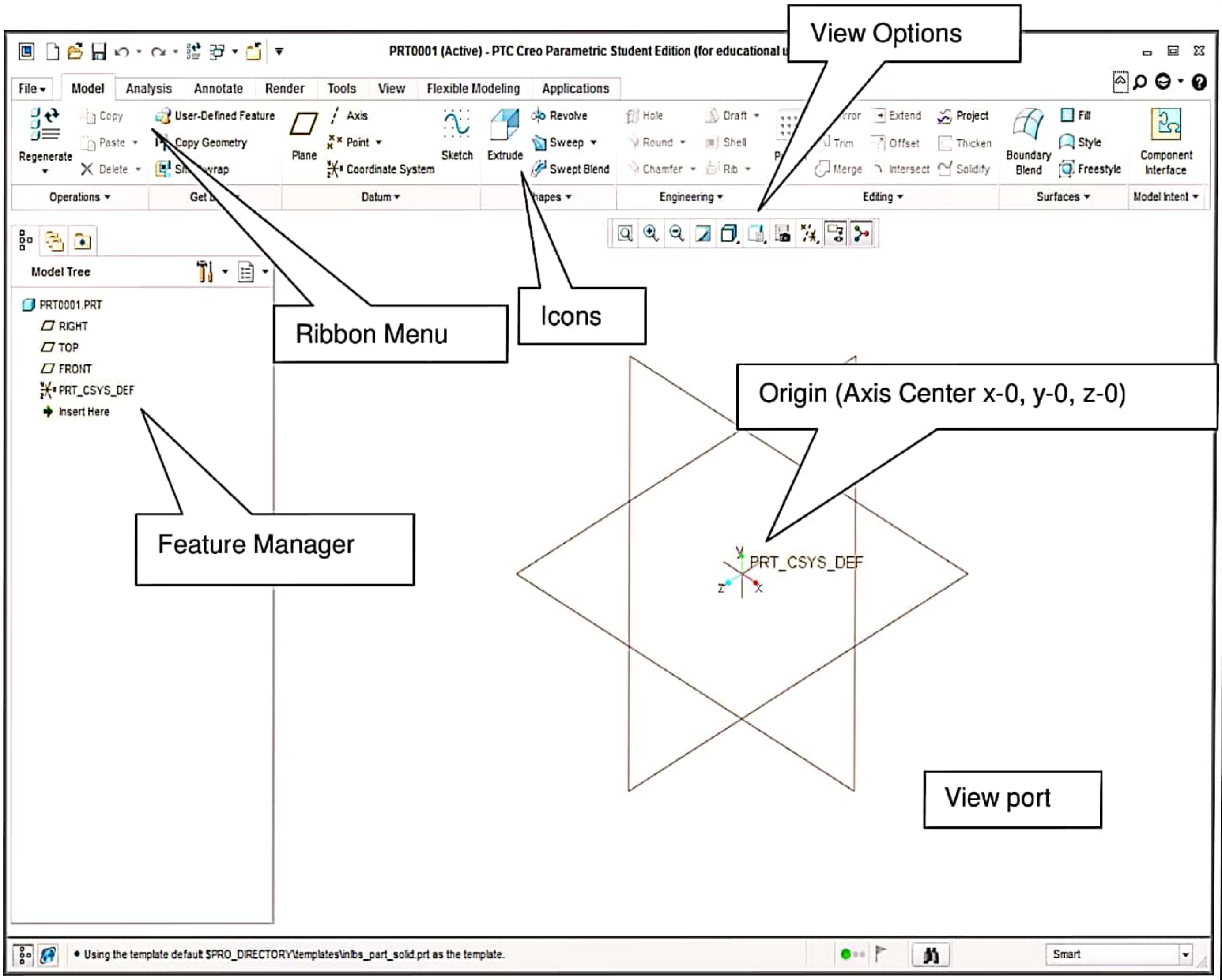
View pull down menu → **3D views** → **Front view** etc.,

The model can be made to orbit using

View pull down menu → **3D ORBIT**

6. Introduction to Pro/E - creo

CreoParametric 3.0 Interface



Mouse Buttons

Left Button - Most commonly used for **selecting** objects on the screen or sketching.

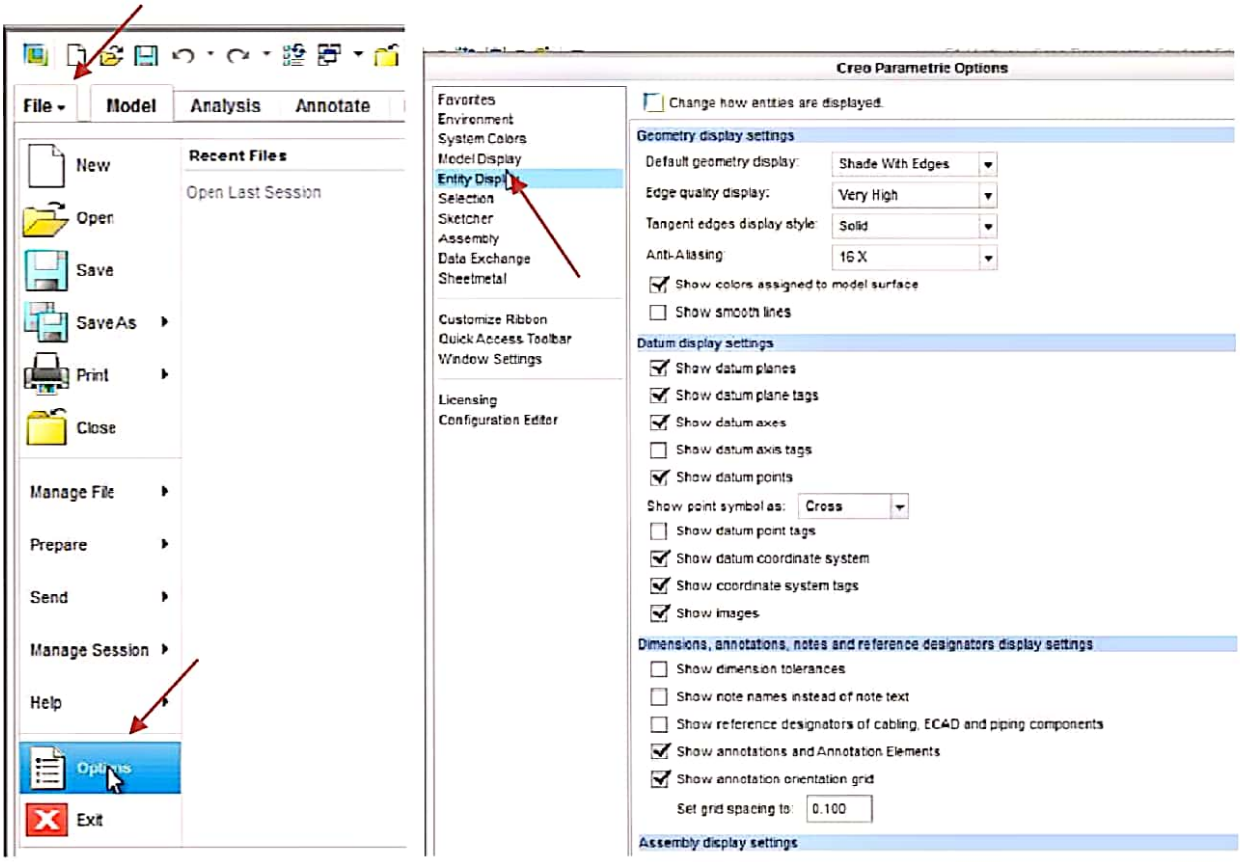
Right Button – Used for activating pop-up **menu** items, typically used when editing. (*Note: you must hold the down button for 2 seconds*)

Center Button – (option) Used for model **rotation**, **dimensioning**, **zoom** when holding Ctrl key, and **pan** when holding Shift key. It also **cancels** commands and line chains.

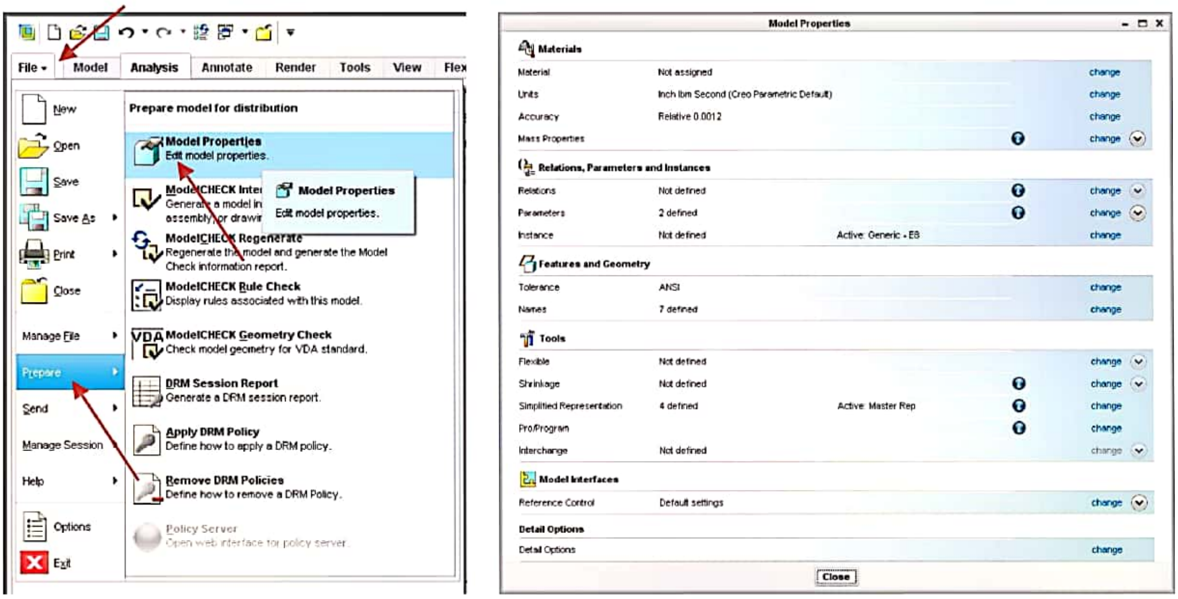
Center Scroll Wheel – (option) same as Center Button when depressed, only it activates **Zoom** feature when scrolling wheel

Options & Properties, menus, *The heart of* creo

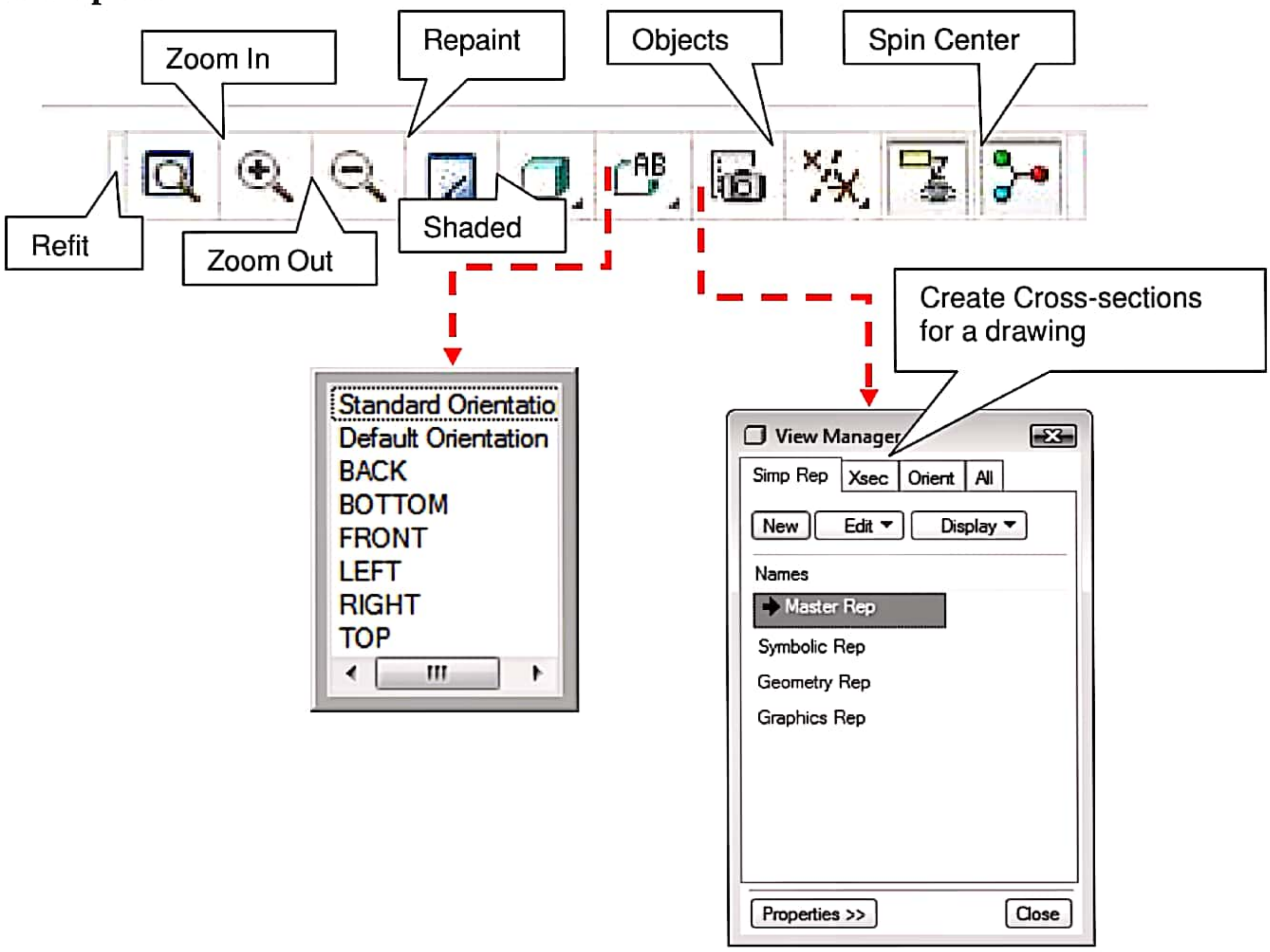
Selecting the File—Options --pull down- *(located at the top left side of the screen)* **opens the active documents Options.**



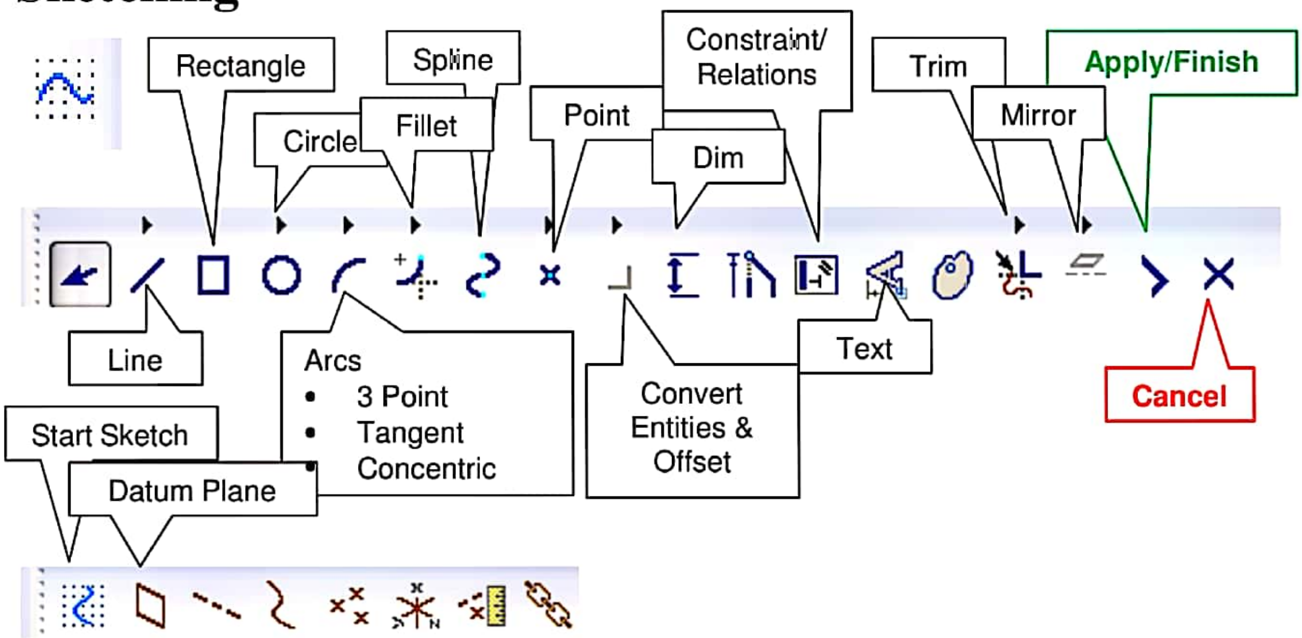
Model Properties



View options



Sketching

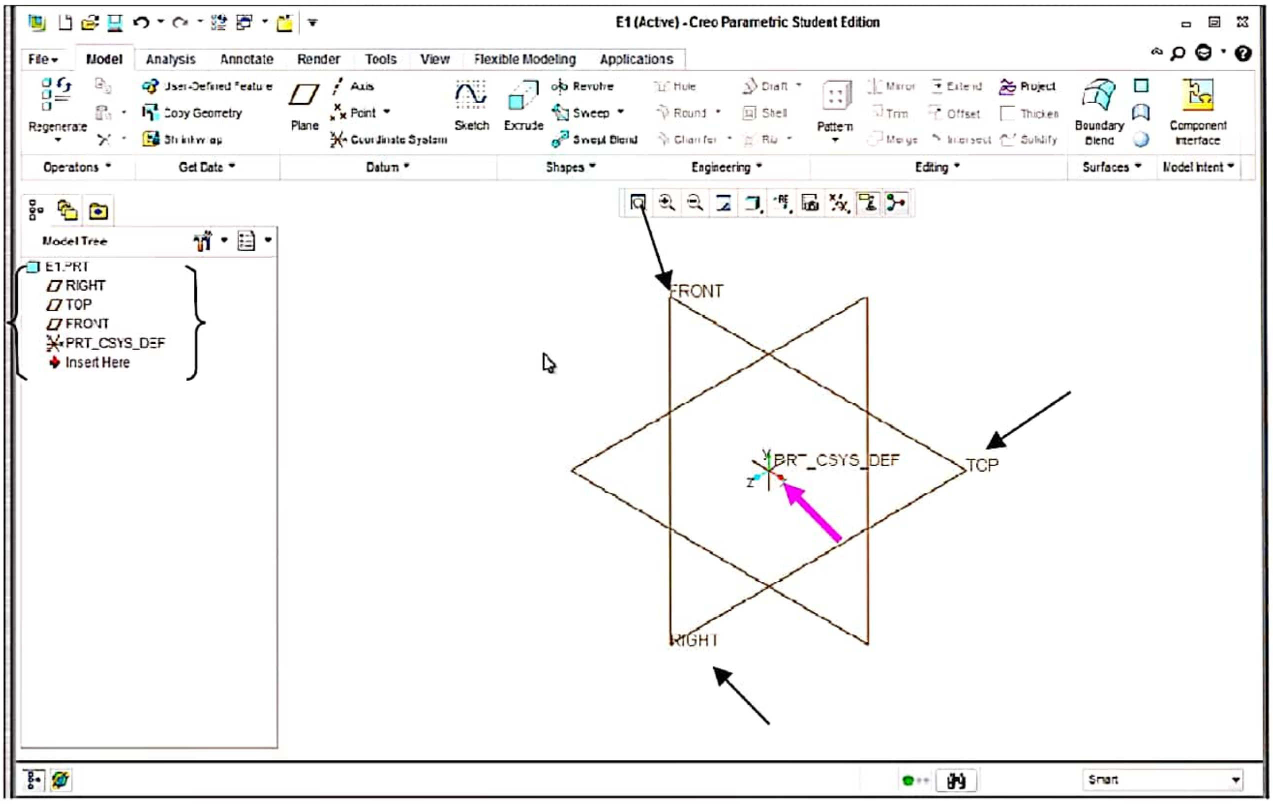


NOTE: If you do not see all of these icons on your interface you can customize the toolbars to bring them up.

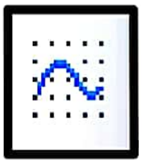
Where do you start a sketch?

Sketches can be created on any Datum Plane or Planar Face or Surface. Pro/E provides you with three datum planes centralized at the Origin (your zero mark in space)

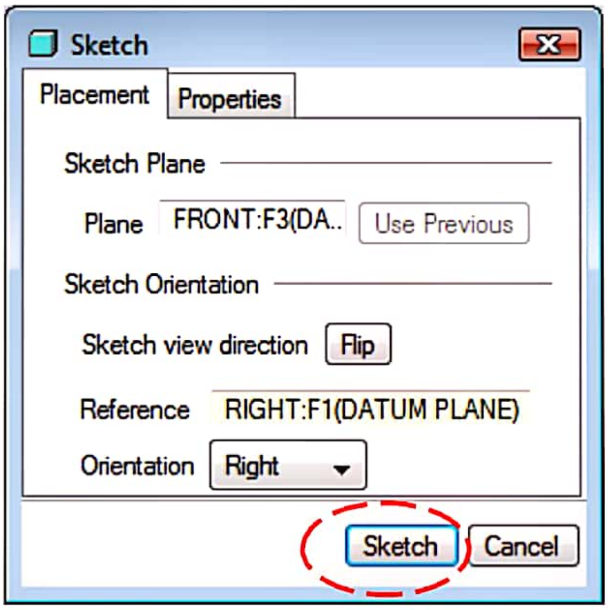
NOTE: Planes can also be created and will be discussed in more detail in the future. Also after completing a sketch always select the Apply/Finish check mark on the sketch toolbar, this will activate the extrude or revolve feature tools.



To start a sketch Pre-select the plane or face you desire to sketch on and then select the Sketch Icon.



Sketch Options

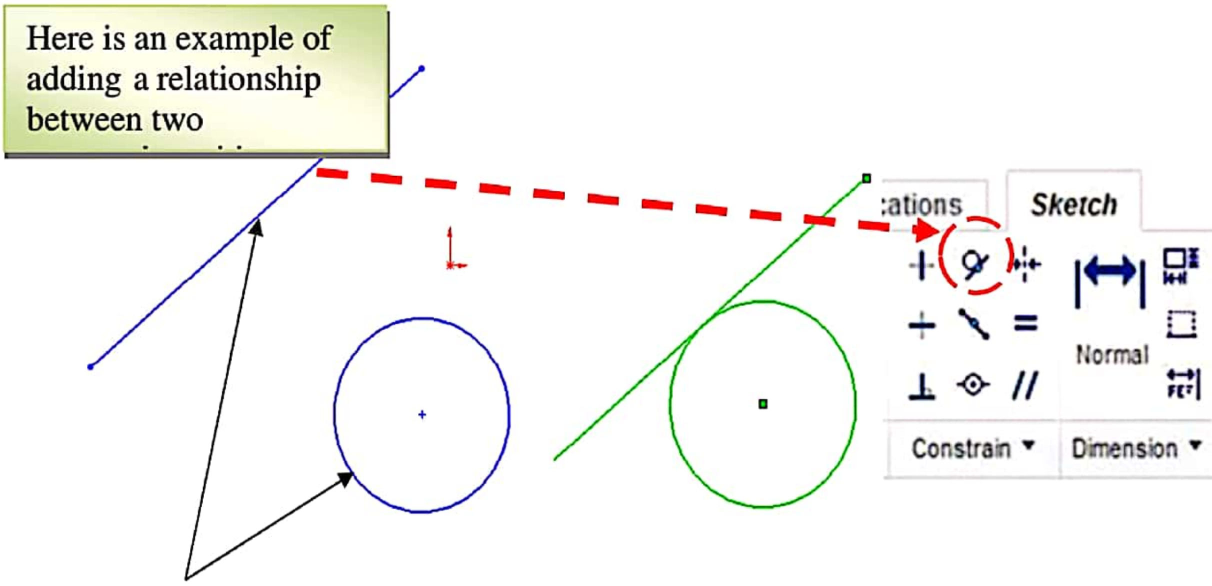


Controlling your geometry

Pro/E uses two methods for constraining geometric entities.

Constraints and **Dimensions**

Constraints can be referred to as common elements of geometry such as Tangency, Parallelism, and Concentricity. These elements can be added to geometric entities automatically or manually during the design process.



Cautious sketching can save time

There are 3 primary file types in Creo, which iŶclude…

1. ***Part*** (.prt)

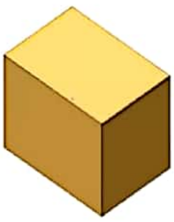
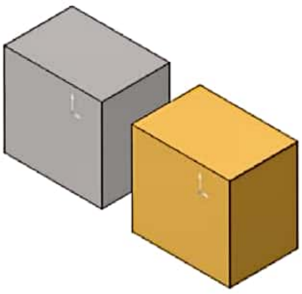
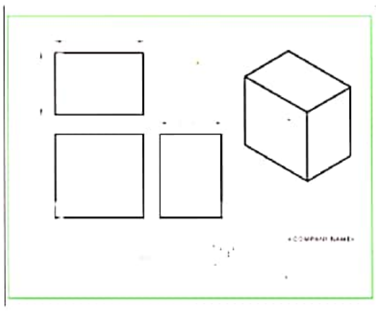
Single part or volume.

2. ***Assembly*** (.asm)

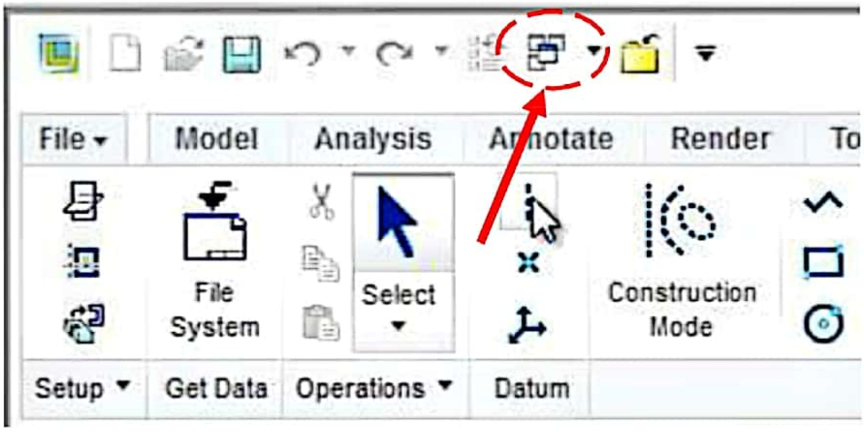
Multiple parts in one file assembled.

3. ***Drawing*** (.drw)

The 2D layout containing views, dimensions, and annotations.

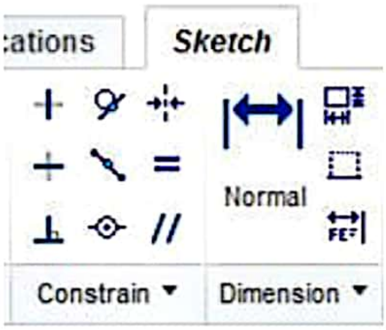




Switching between documents (Activating a document)



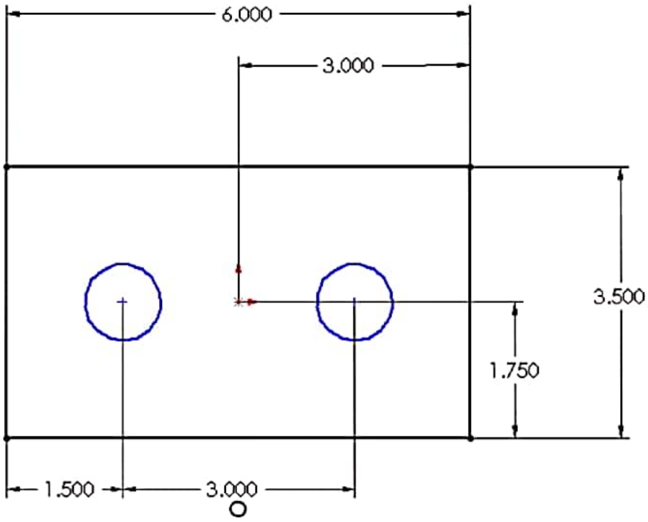
Select the Window pull-down menu and you will see the available documents. Click on the document you wish to work on from the list to update it.

Sketch Constraints (Relations)



Constraint	Geometric entities to select	Resulting Constraint
Horizontal or Vertical	One or more lines or two or more points.	The lines become horizontal or vertical (as defined by the current sketch space). Points are aligned horizontally or vertically.
Collinear	Two or more lines.	The items lie on the same infinite line.
Perpendicular	Two lines.	The two items are perpendicular to each other.
Parallel	Two or more lines. A line and a plane (or a planar face) in a 3D sketch.	The items are parallel to each other. The line is parallel to the selected plane.
Tangent	An arc, ellipse, or spline, and a line or arc.	The two items remain tangent.
Concentric	Two or more arcs, or a point and an arc.	The arcs share the same centerpoint.
Midpoint	Two lines or a point and a line.	The point remains at the midpoint of the line.
Coincident	A point and a line, arc, or ellipse.	The point lies on the line, arc, or ellipse.
Equal	Two or more lines or two or more arcs.	The line lengths or radii remain equal.
Symmetric	A centerline and two points, lines, arcs, or ellipses.	The items remain equidistant from the centerline, on a line perpendicular to the centerline.

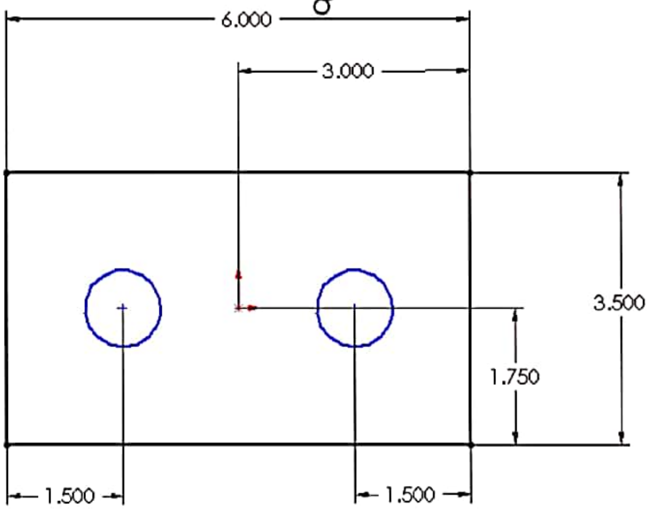
Controlling your geometry with dimensions…

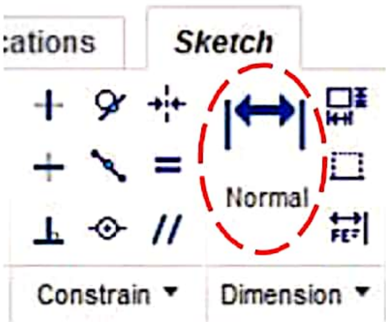


Strong versus **Weak** Dimensions - Double click and change to make them Strong!

Dimensioning this way will enable the length of the bracket to change but the holes will always remain positioned to 1.5" off each

Dimensioning this way will enable the length of the bracket to change but the holes will always remain positioned to the left side.

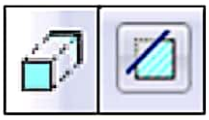




Solid Modeling Basics

1. Layer Cake method

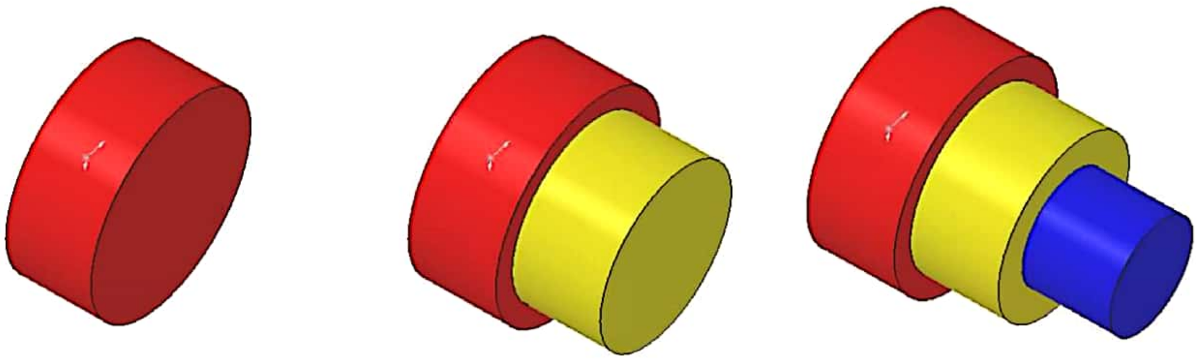
Extruded Boss/Base (Creates/Adds material)



Extruded Cut (Removes material)

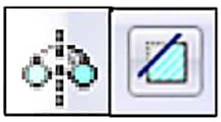
Ingredients:

- Profile



2. Revolve method

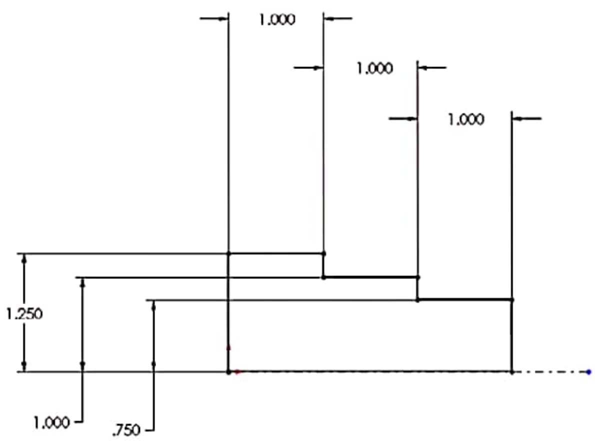
Revolve Boss/Base (Creates/Adds material)

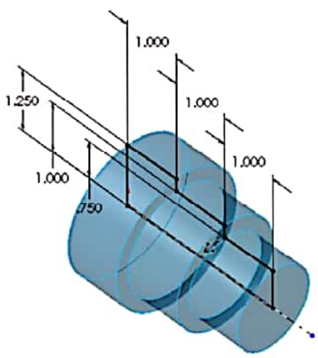


Revolve Cut (Removes material)

Ingredients:

- Profile
- Center Line *(Note: The profile cannot cross over the center line!)*

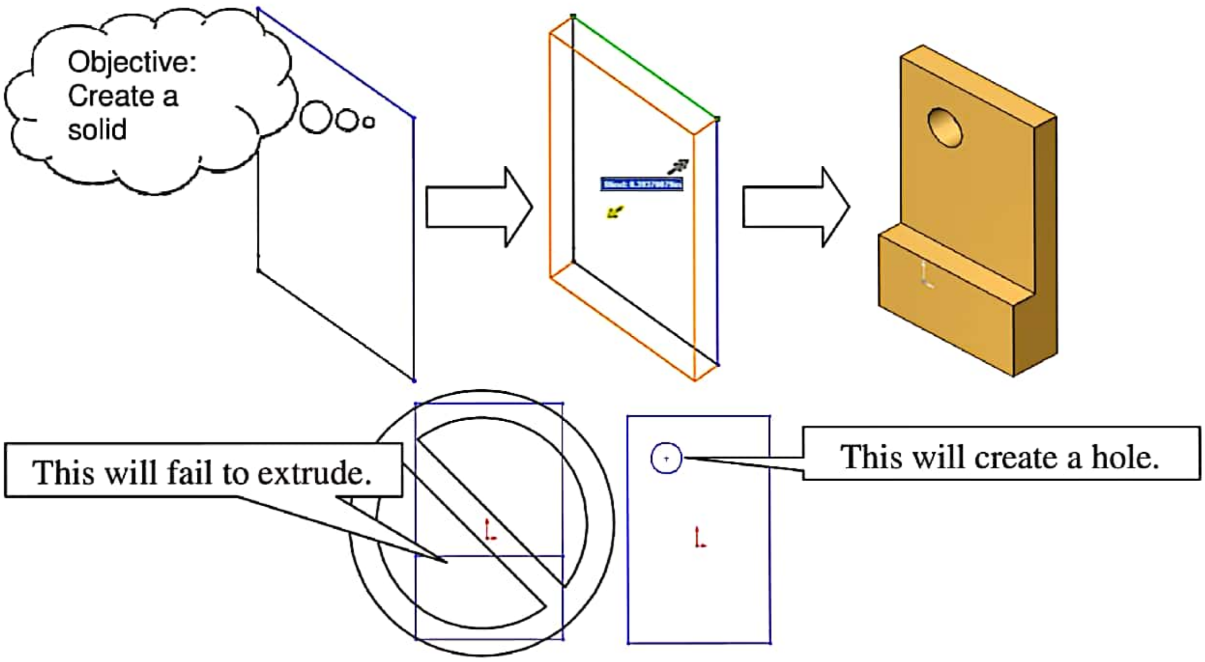


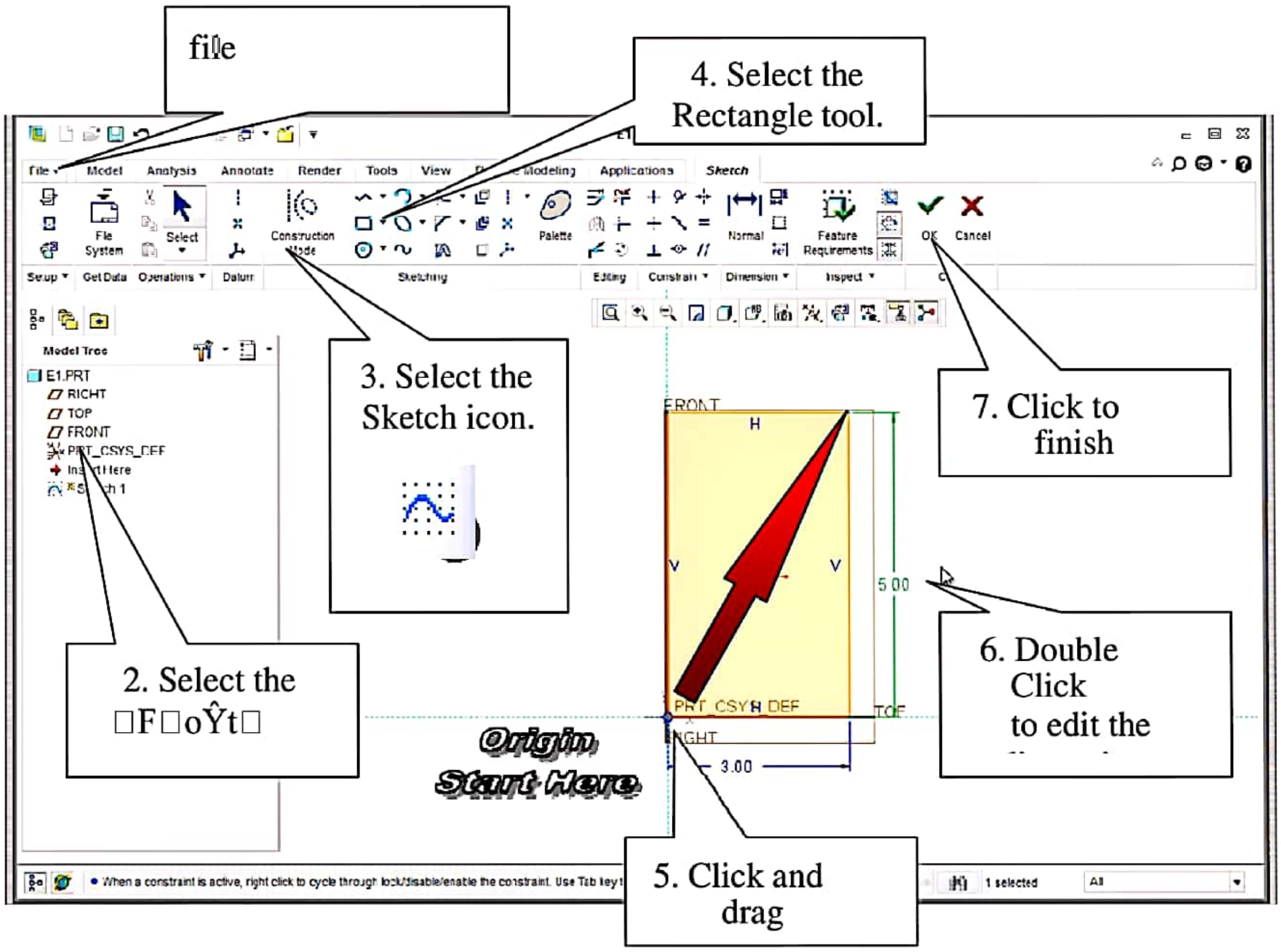


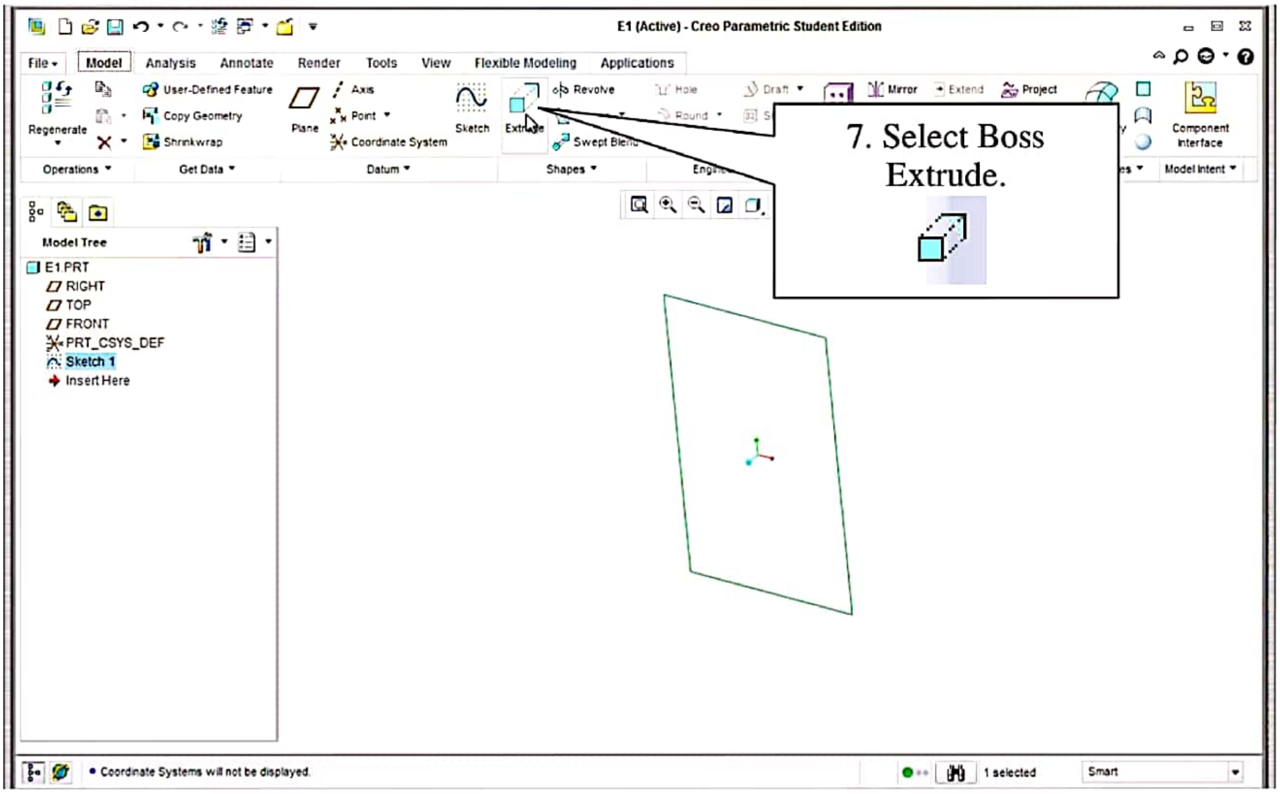
EXERCISE 1

Introduction to basic part modeling

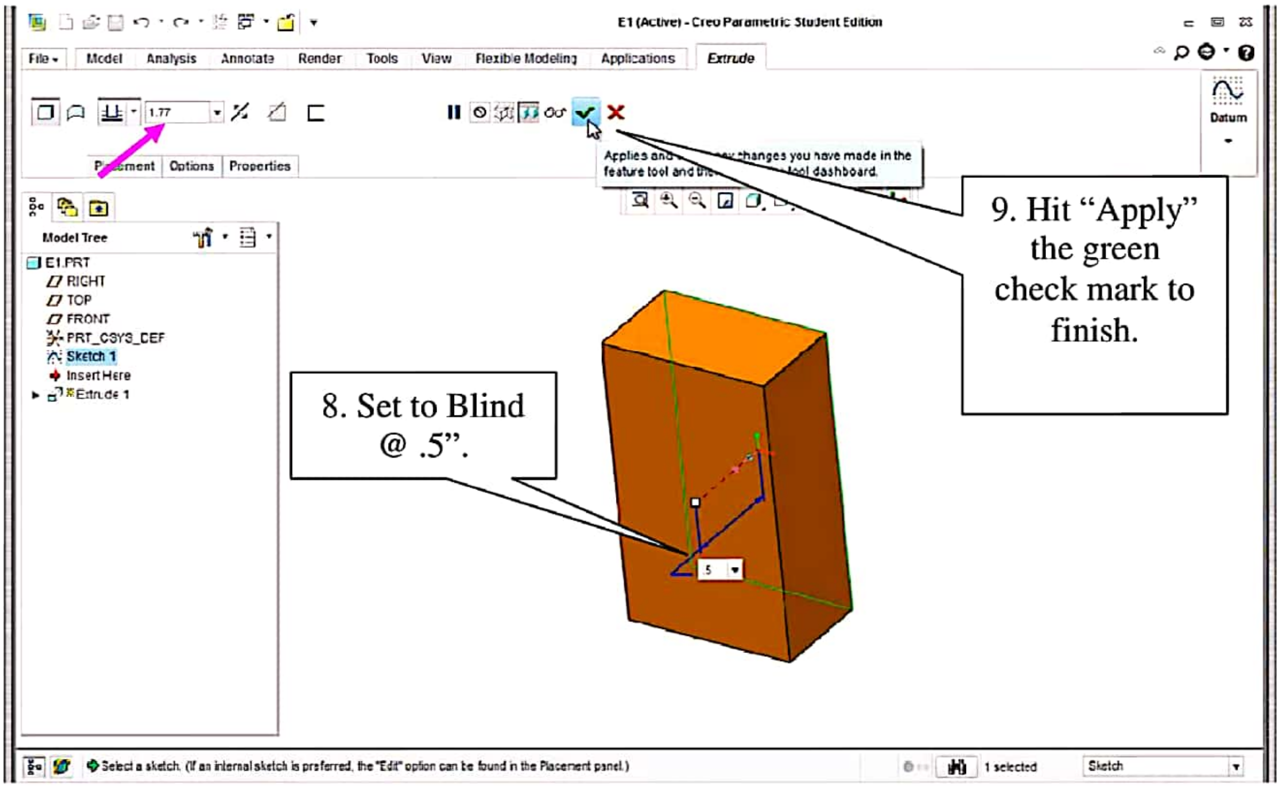
***Base Extrude Features** create a 3D solid representation by extruding a 2 dimensional profile of the entity.*

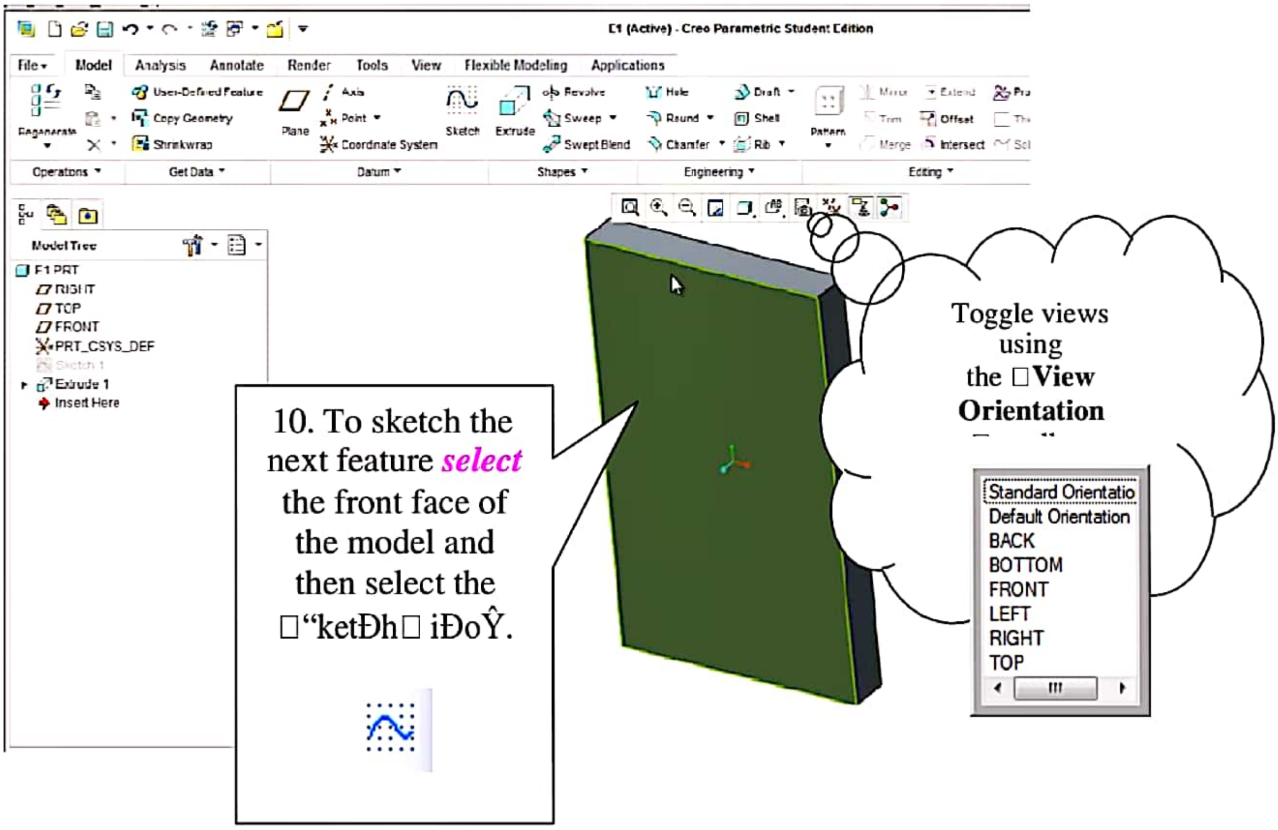




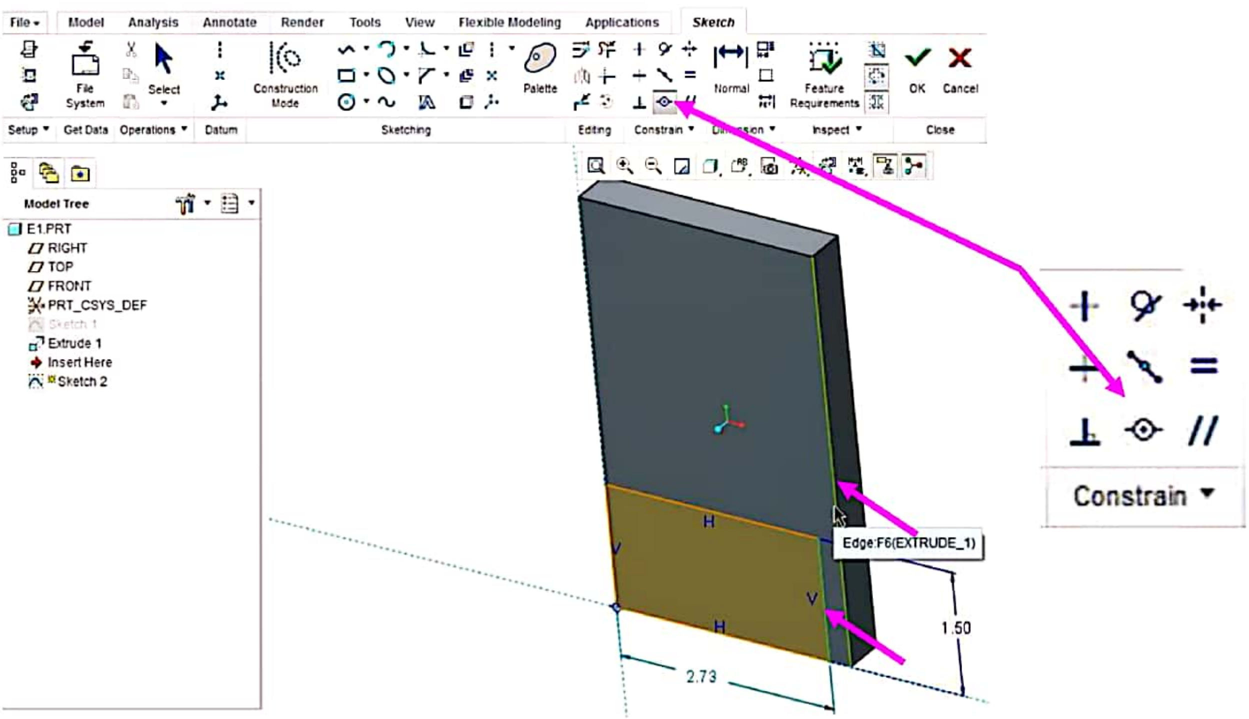


NOTE: When dimensioning use the dimension tool and make edge selections, mouse center button click to apply dimension.

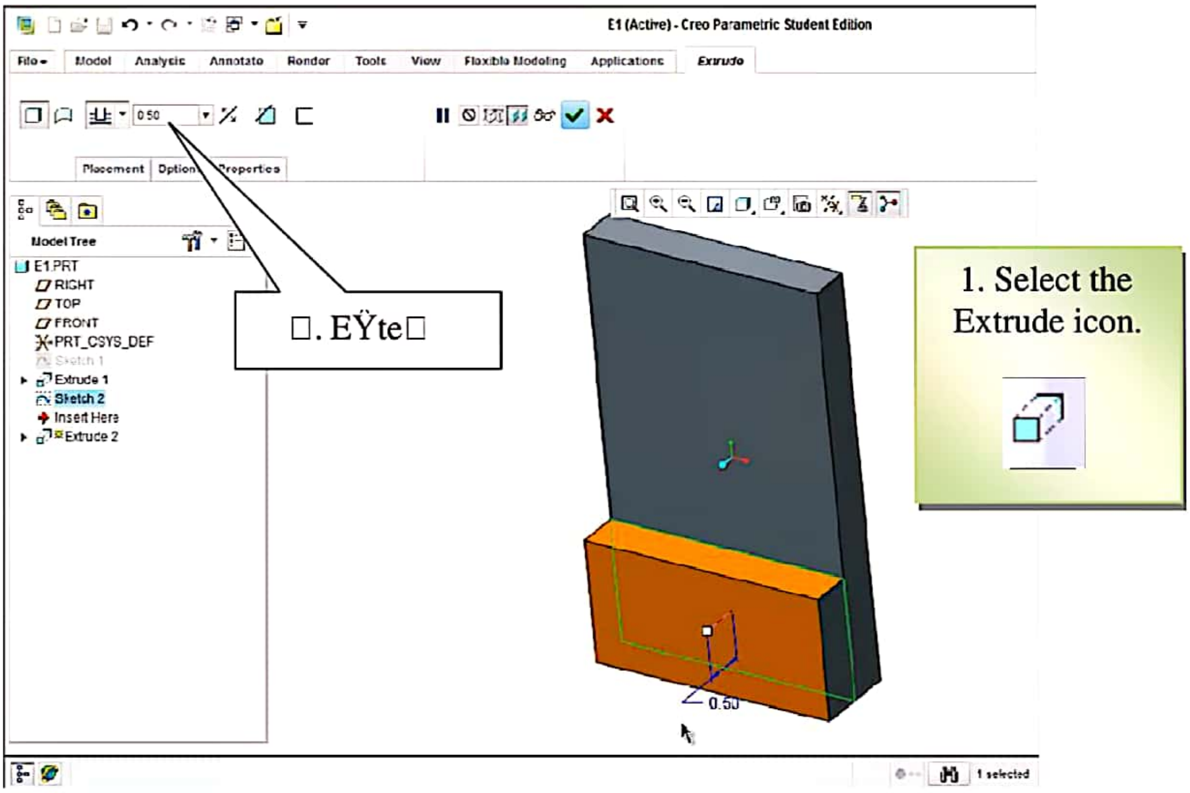




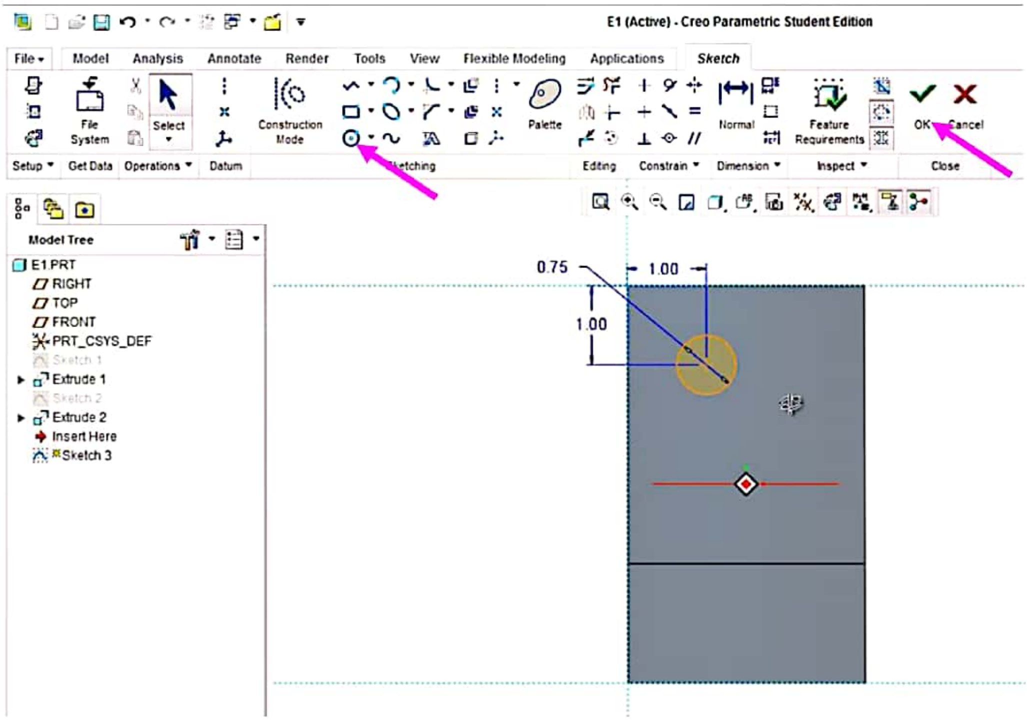
Adding a constraint – Ctrl Select both left edges of sketch and solid. Select Coincident



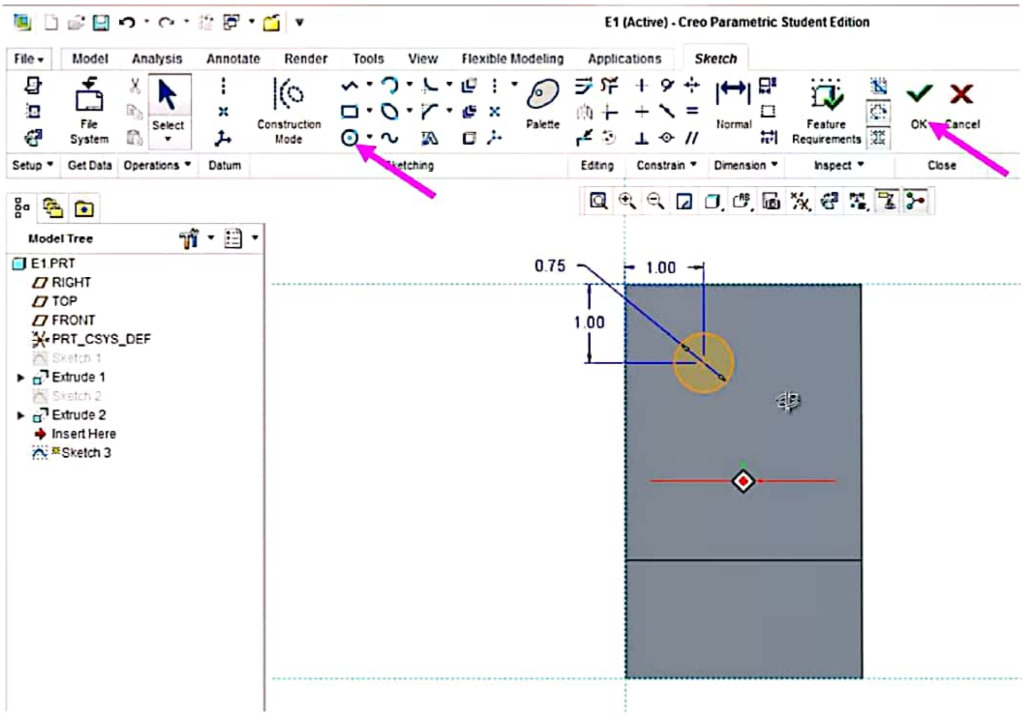
Extrude



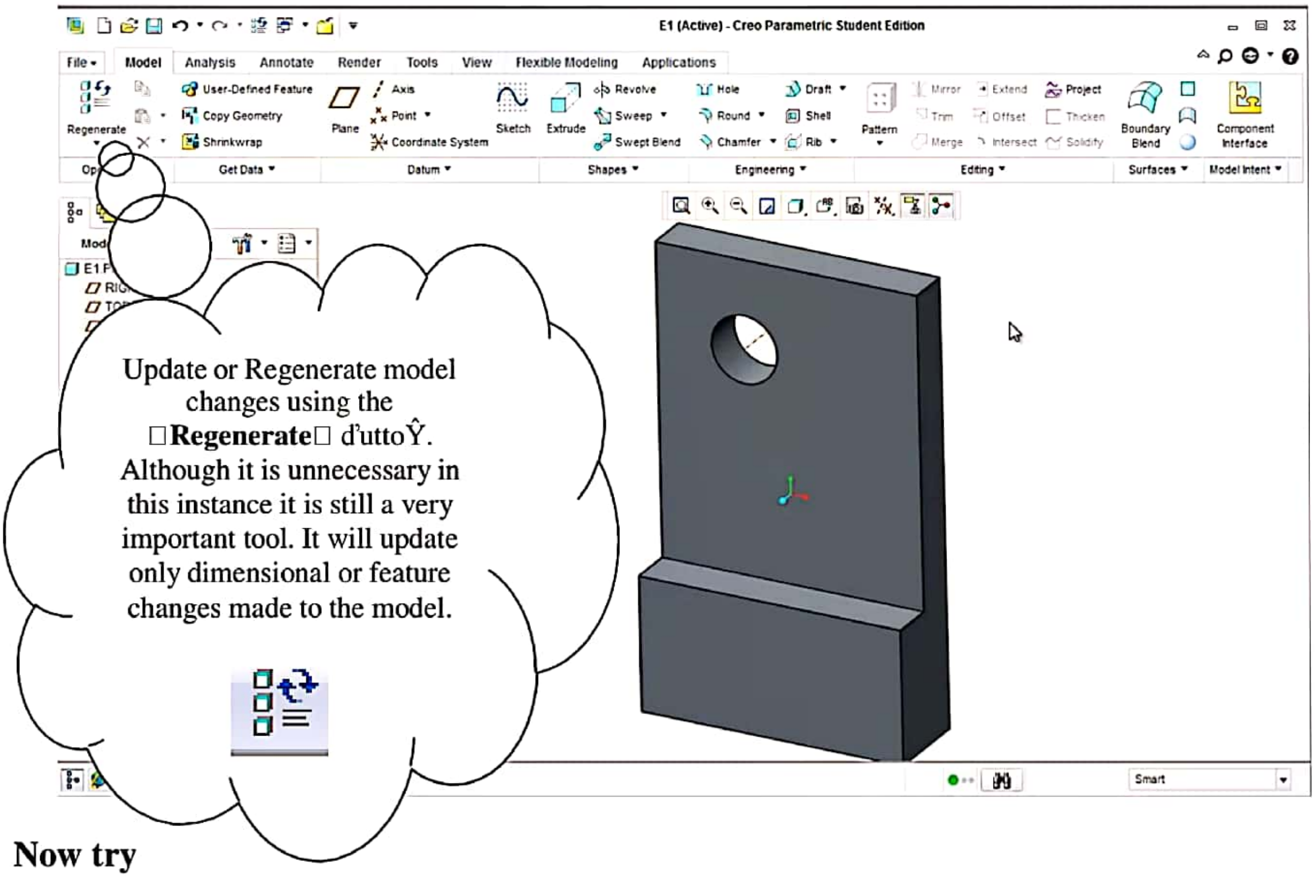
Select the face, select sketch icon and draw a circle on the face. Dimension, Hit □Ok□



Select the face, select sketch icon and draw a circle on the face. Dimension, Hit □Ok□

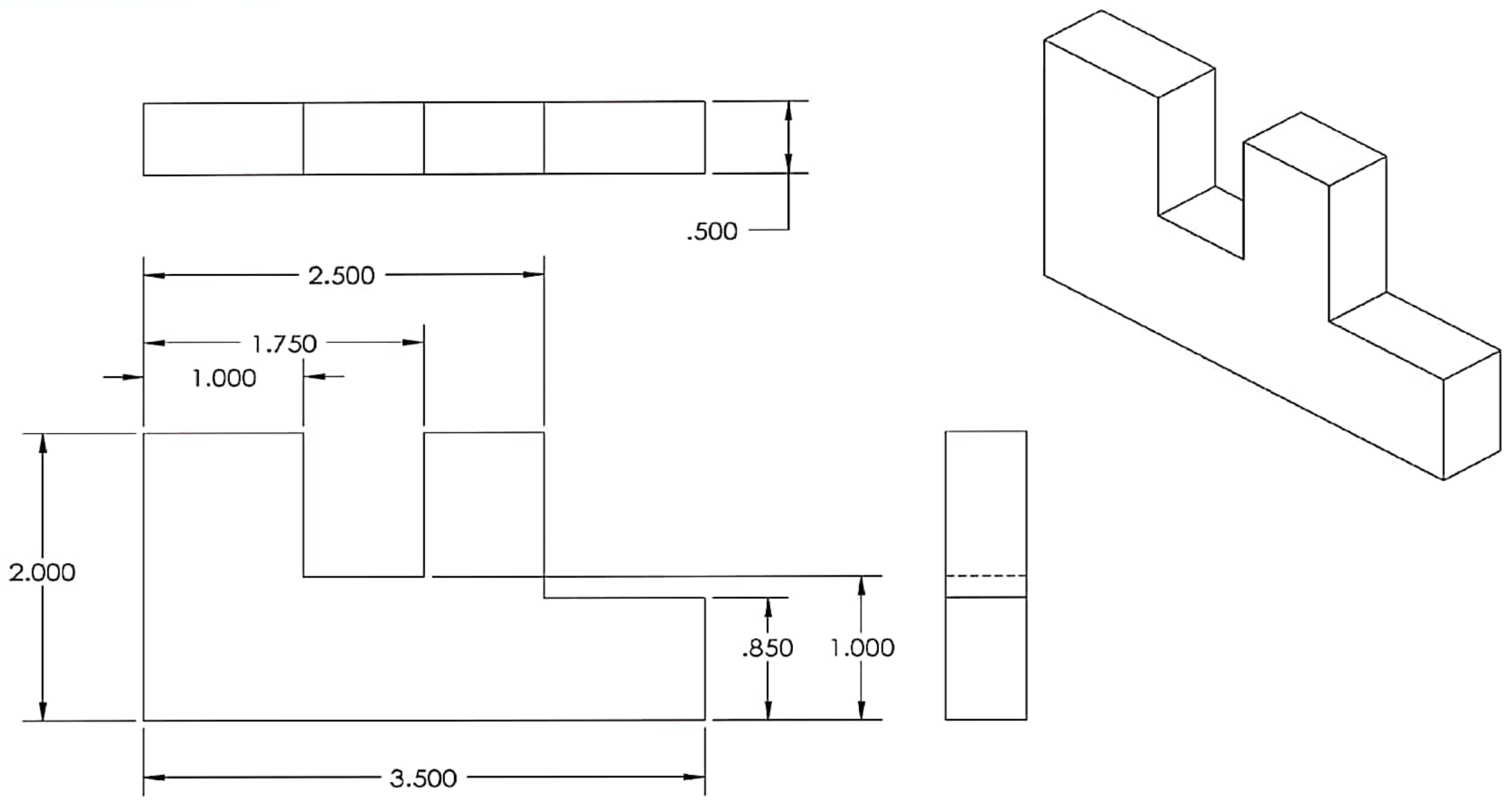


Go to file save



Now try

NOTE: Patterns/Arrays and Mirroring will be covered in the next three Ðhapters. Please try to ŵodel LAB 1 without usiŶg theŵ. It's good practice to just dimension and sketch all geometry when first starting out learning this software.



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APPLICATION

UNLESS OTHERWISE SPECIFIED:

DIMENSIONS ARE IN INCHES
TOLERANCES:
FRACTIONAL ±
ANGULAR: MACH ± BEND ±
TWO PLACE DECIMAL ±
THREE PLACE DECIMAL ±

INTERPRET GEOMETRIC TOLERANCING PER:

MATERIAL

FINISH

DO NOT SCALE DRAWING

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MFG APPR.		
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SCALE: 1:1 WEIGHT: SHEET 1 OF 1

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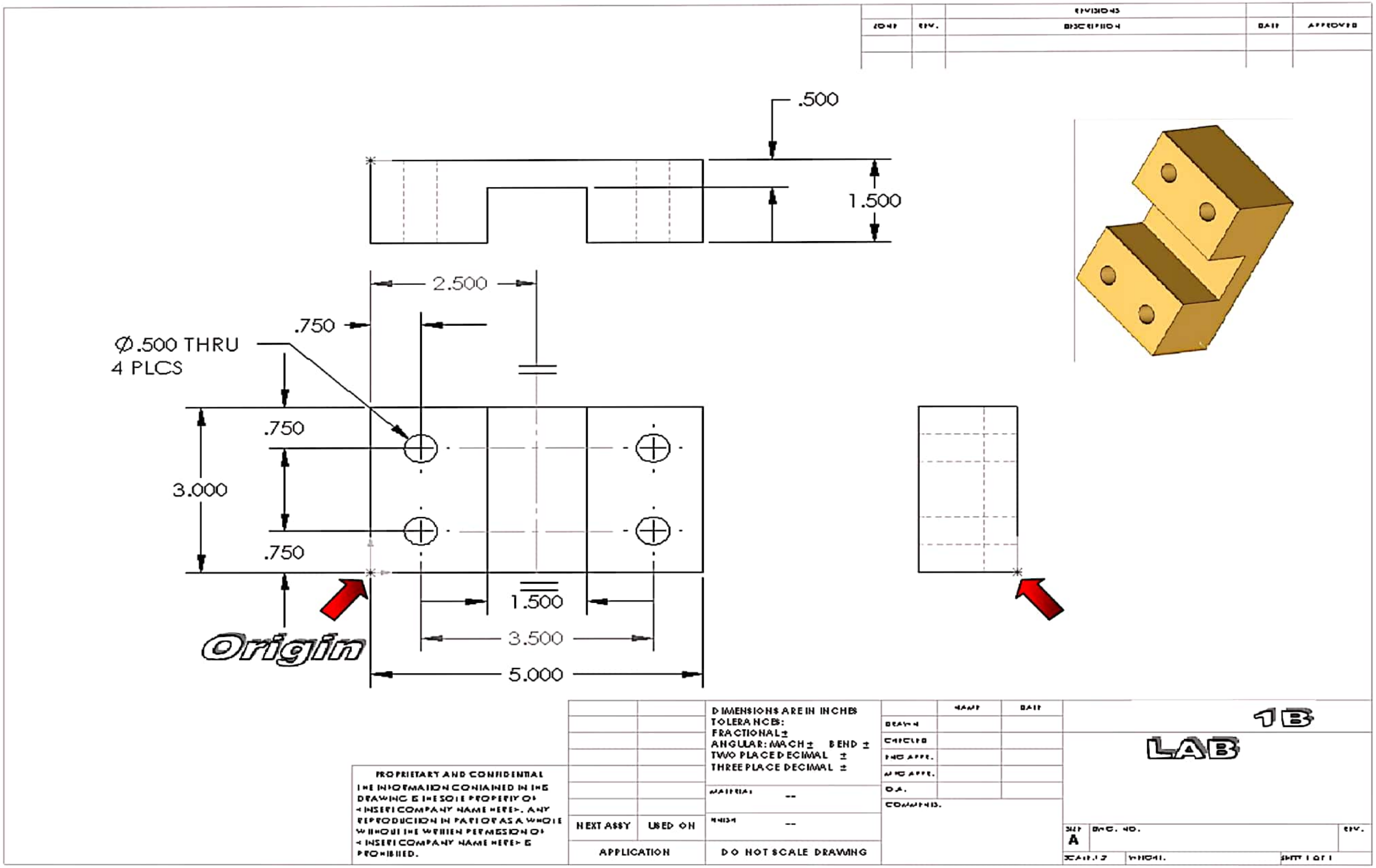
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ANGULAR: MACH ± BEND ±
TWO PLACE DECIMAL ±
THREE PLACE DECIMAL ±
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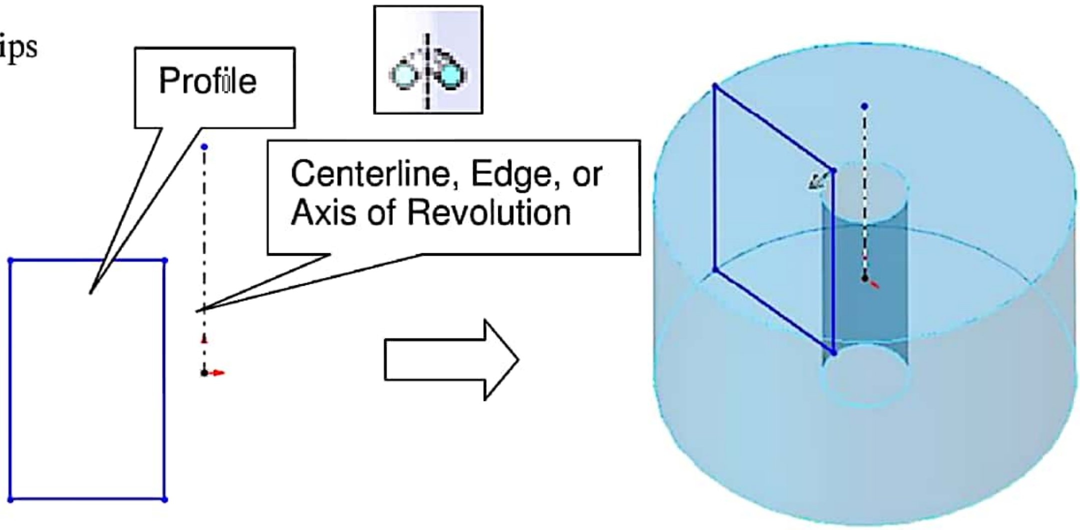
MFG APPR.
Q.A.
COMMENTS:
1B
LAB
SIZE A
DWG. NO.
REV.
SCALE: 1:2
WEIGHT:
SHEET 1 OF 1

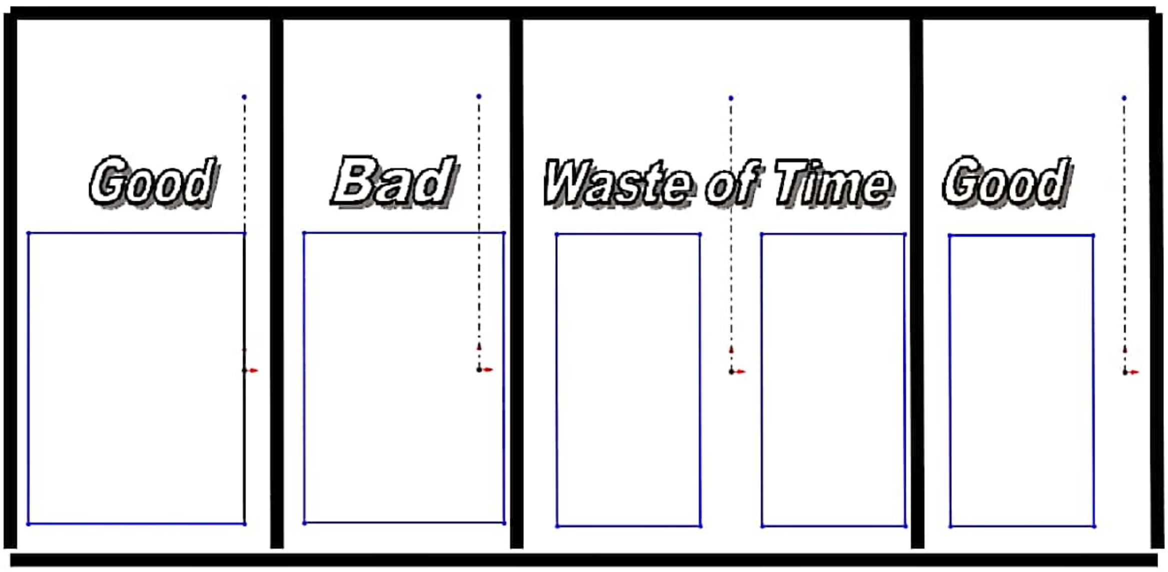
EXERCISE 2

Revolved Features

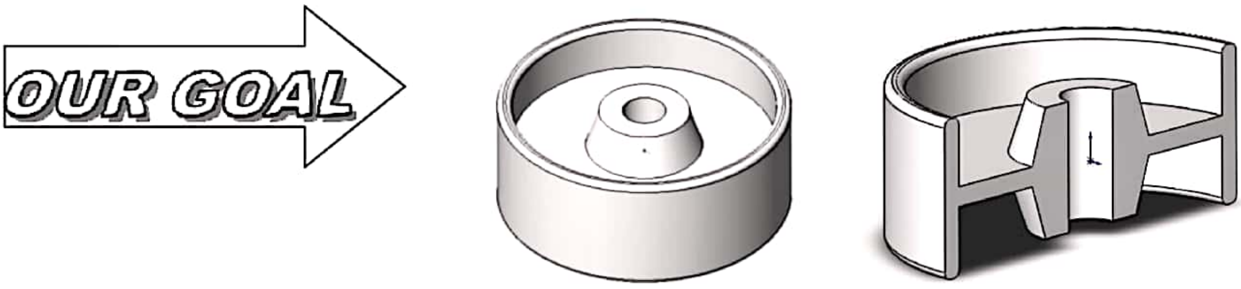
Revolved Feature - creates features that add or remove material by revolving one or more profiles around a centerline. The feature can be a solid, a thin feature, or a surface.

Tips

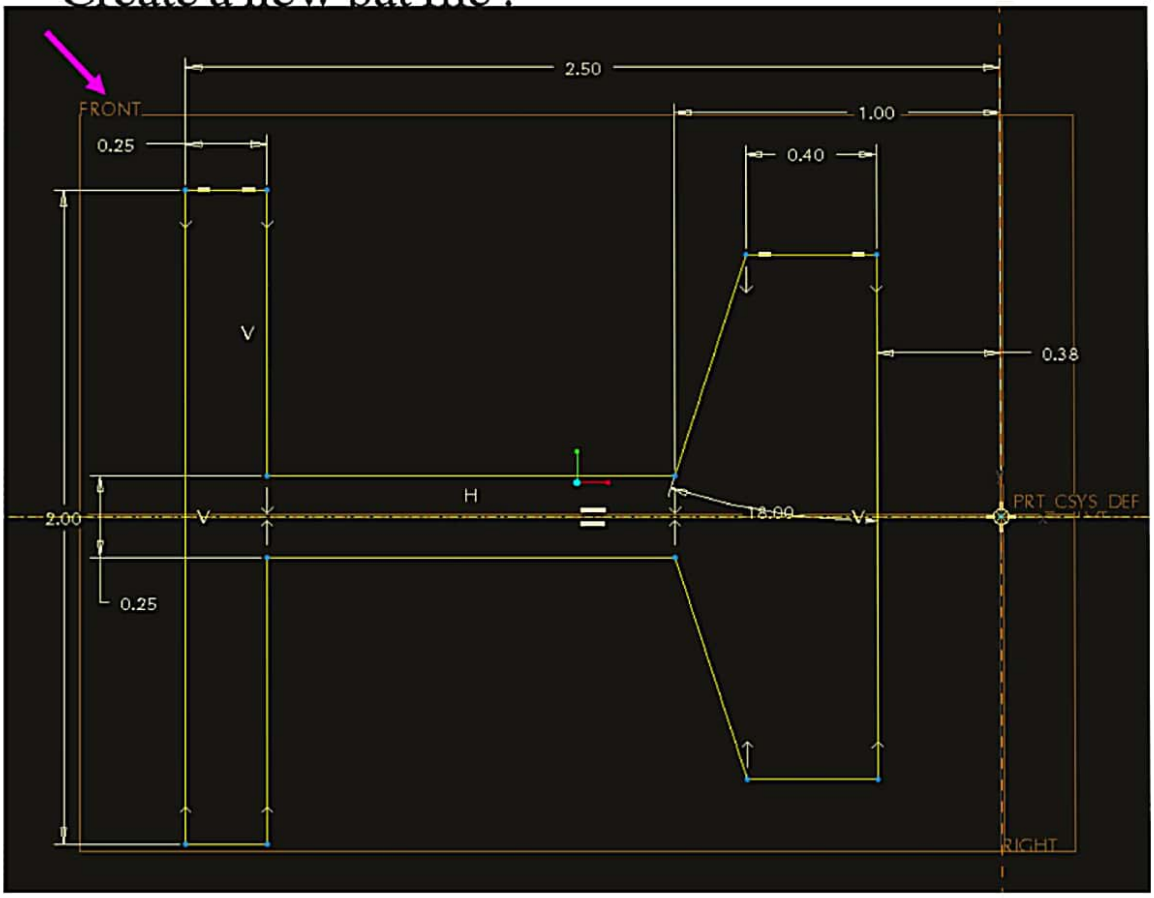




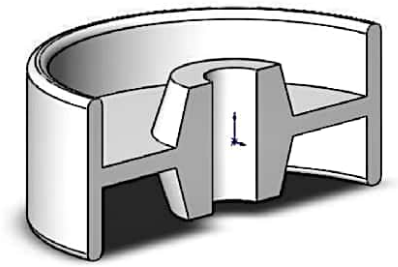
The profile should never cross over the centerline, nor should there be profiles on both sides of the centerline.

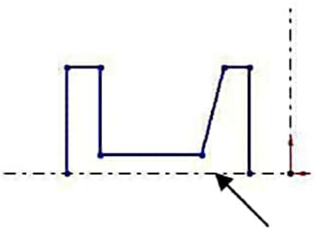


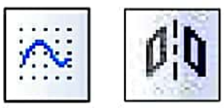
1. Create a new pat file .

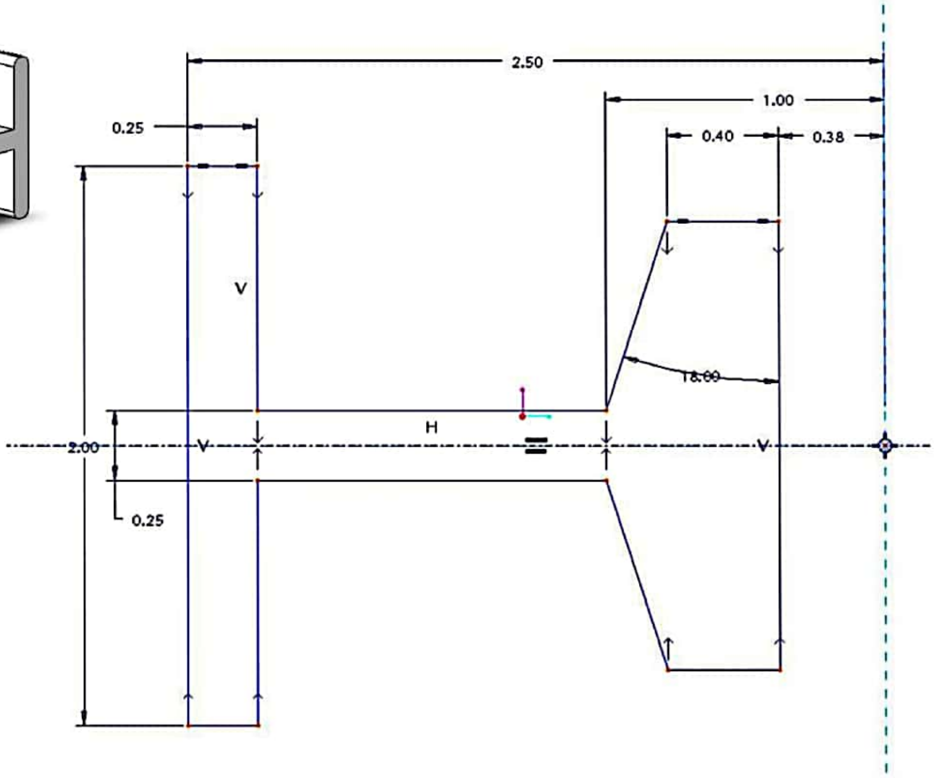


2. using the dimension tool to create a ¼ of the geometry and then sweep it to the other side. Make sure you finish adding the dimensions.

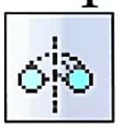


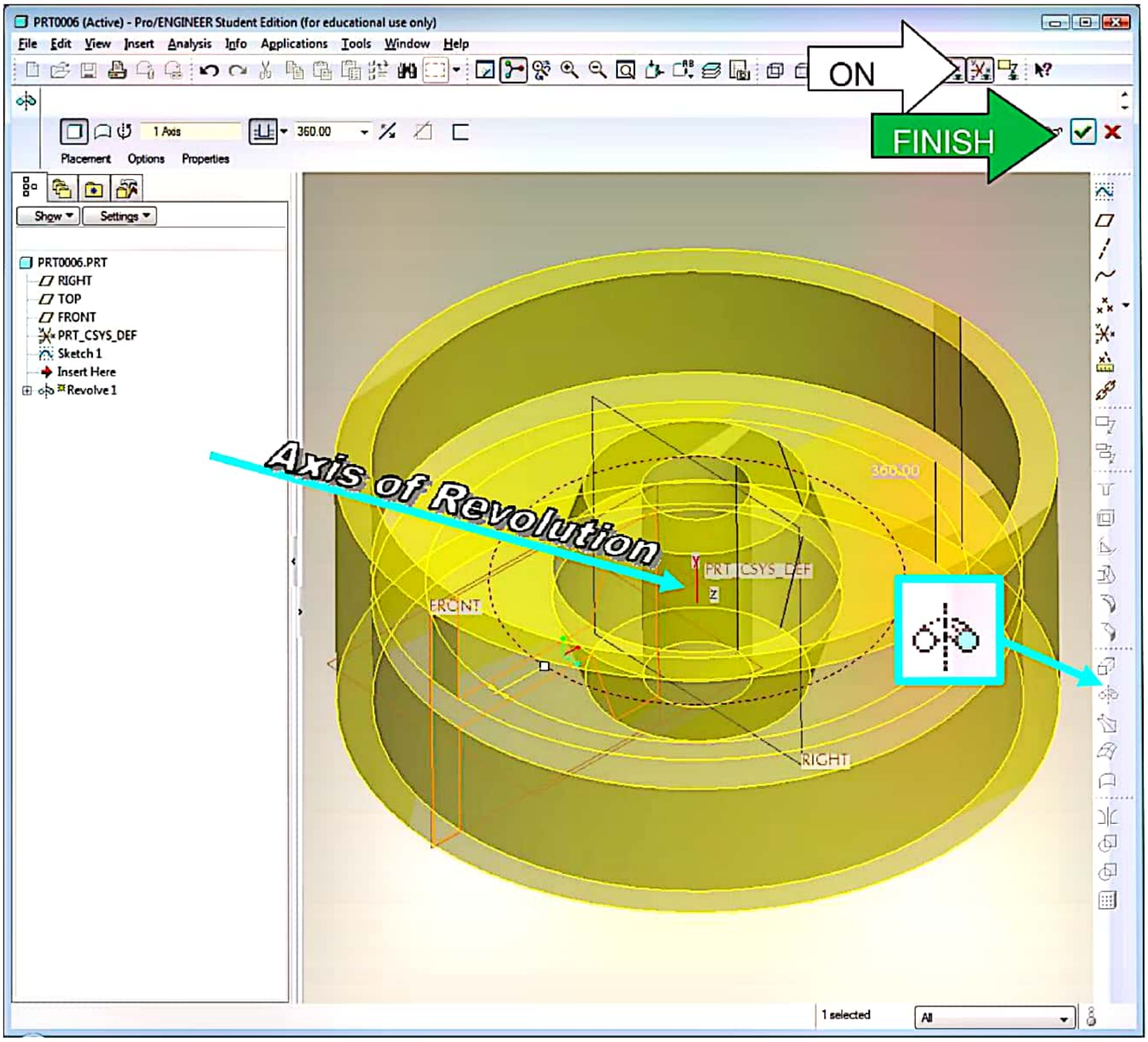






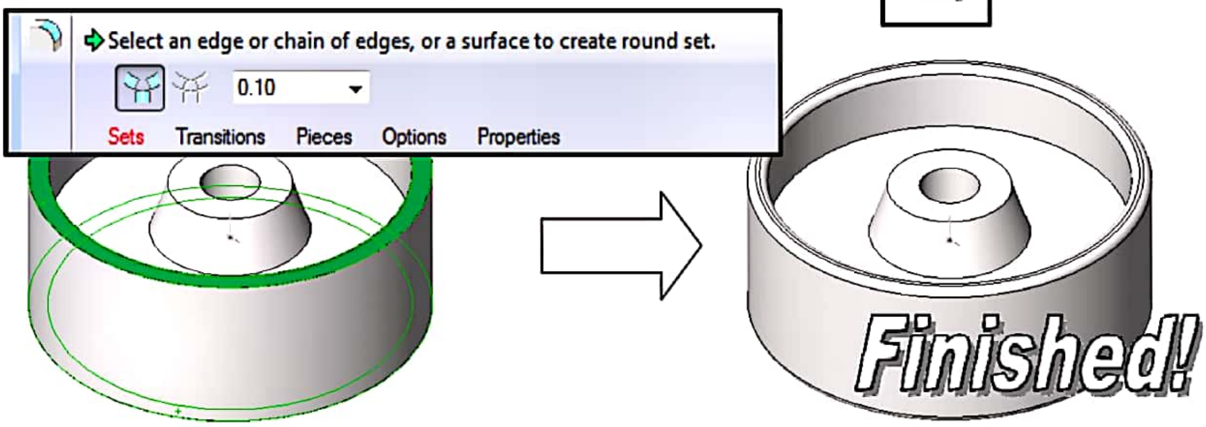
1. Select the **Revolve** feature icon. Ten select the axis/centerline.



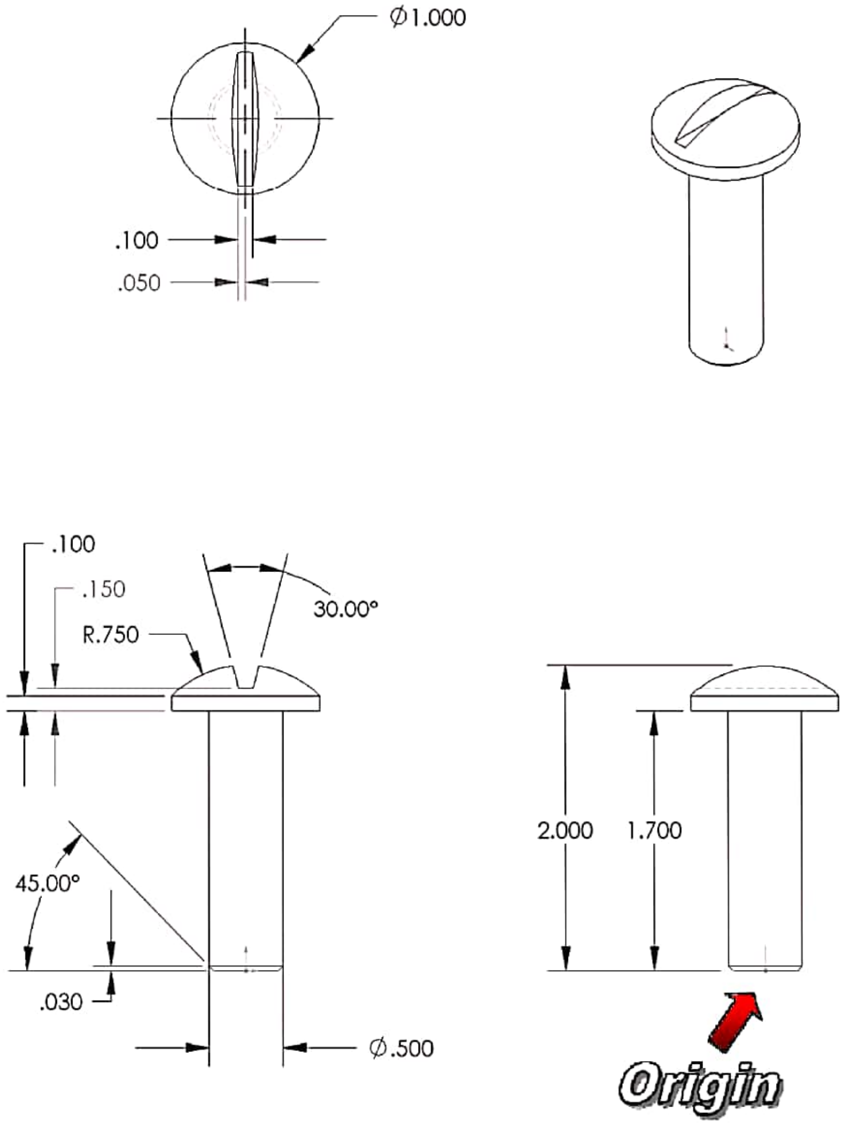


Rounds

4. Select the top and bottom edges and add a  ds/fillets.



REVISIONS				
ZONE	REV.	DESCRIPTION	DATE	APPROVED



		DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL ± ANGULAR: MACH ± BEND ± TWO PLACE DECIMAL ± THREE PLACE DECIMAL ±		NAME	DATE	LAB 2
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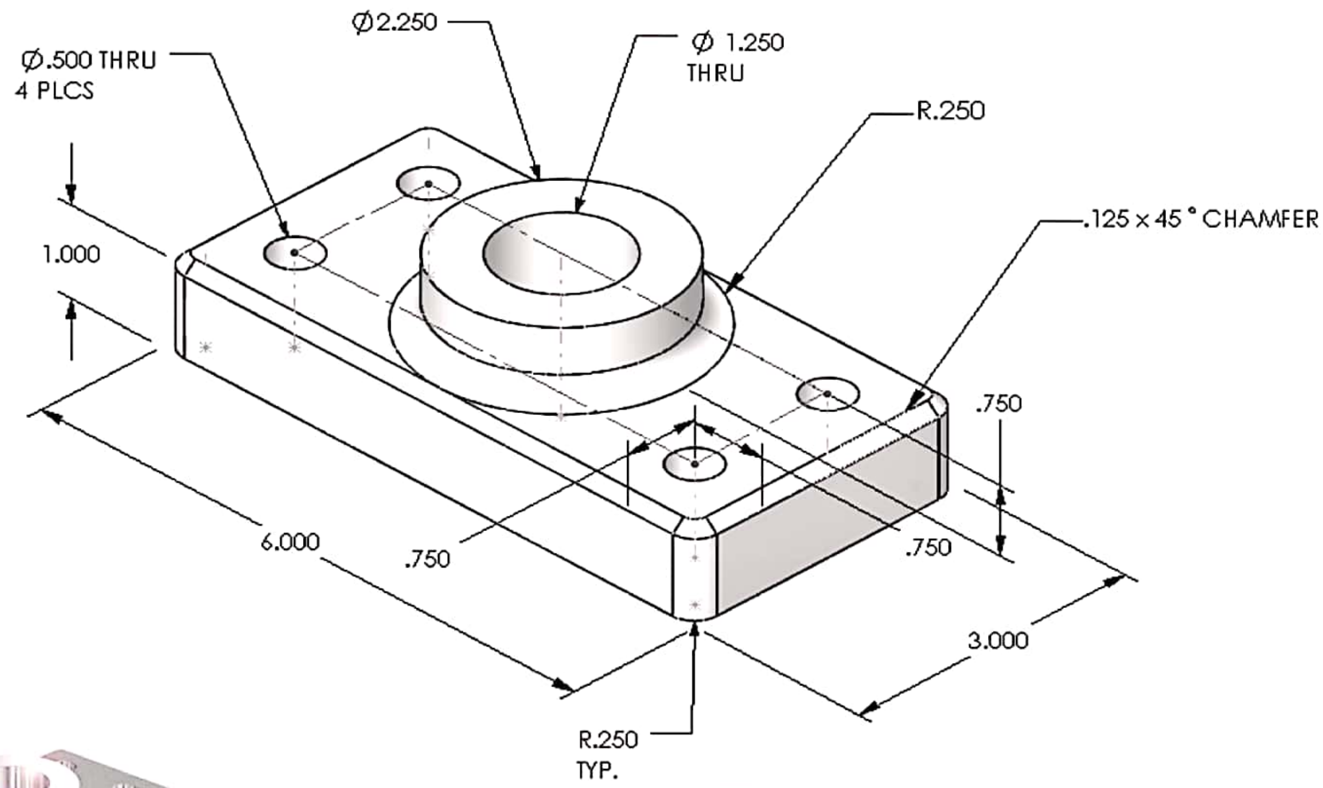
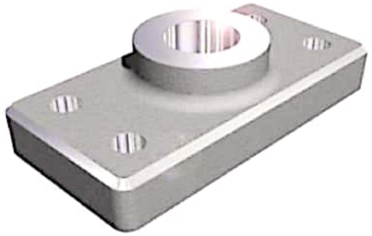
REVISIONS

ZONE	REV.	DESCRIPTION	DATE	APPROVED





QUIZ 1

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TWO PLACE DECIMAL ±
THREE PLACE DECIMAL ±

MATERIAL --

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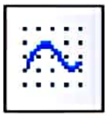
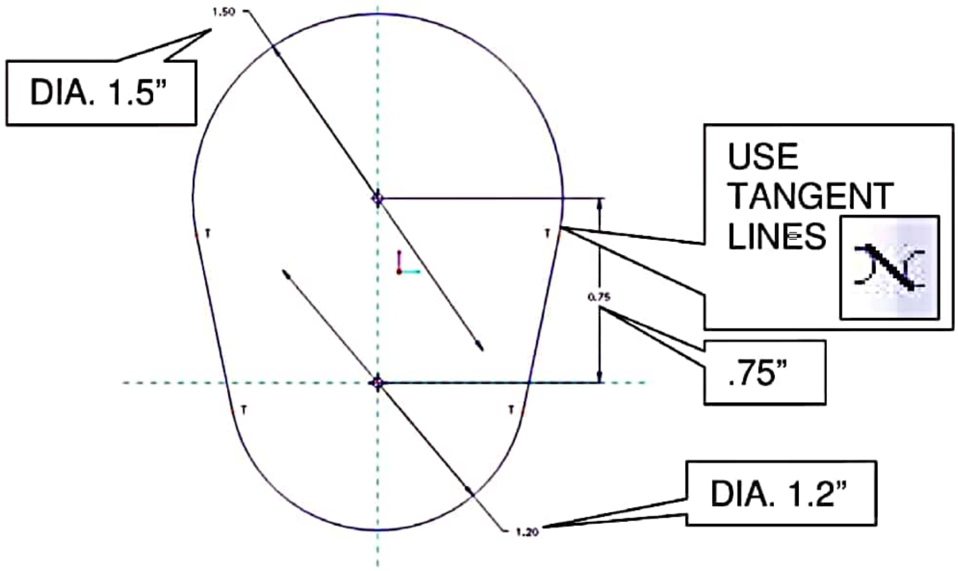
SCALE: 1:1 WEIGHT: SHEET 1 OF 1

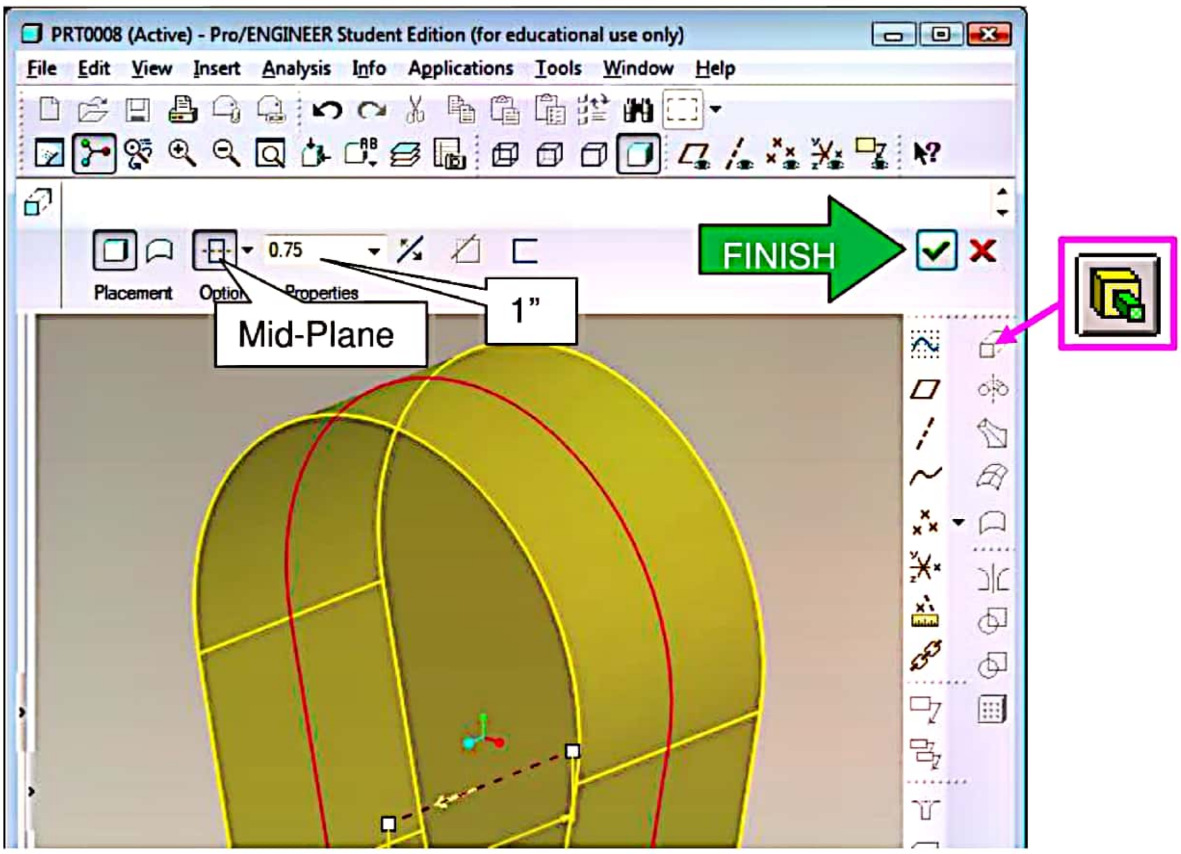
EXERCISE 3
Secondary Feature Modeling

1. Sketch the geometry as shown below on the front plane.

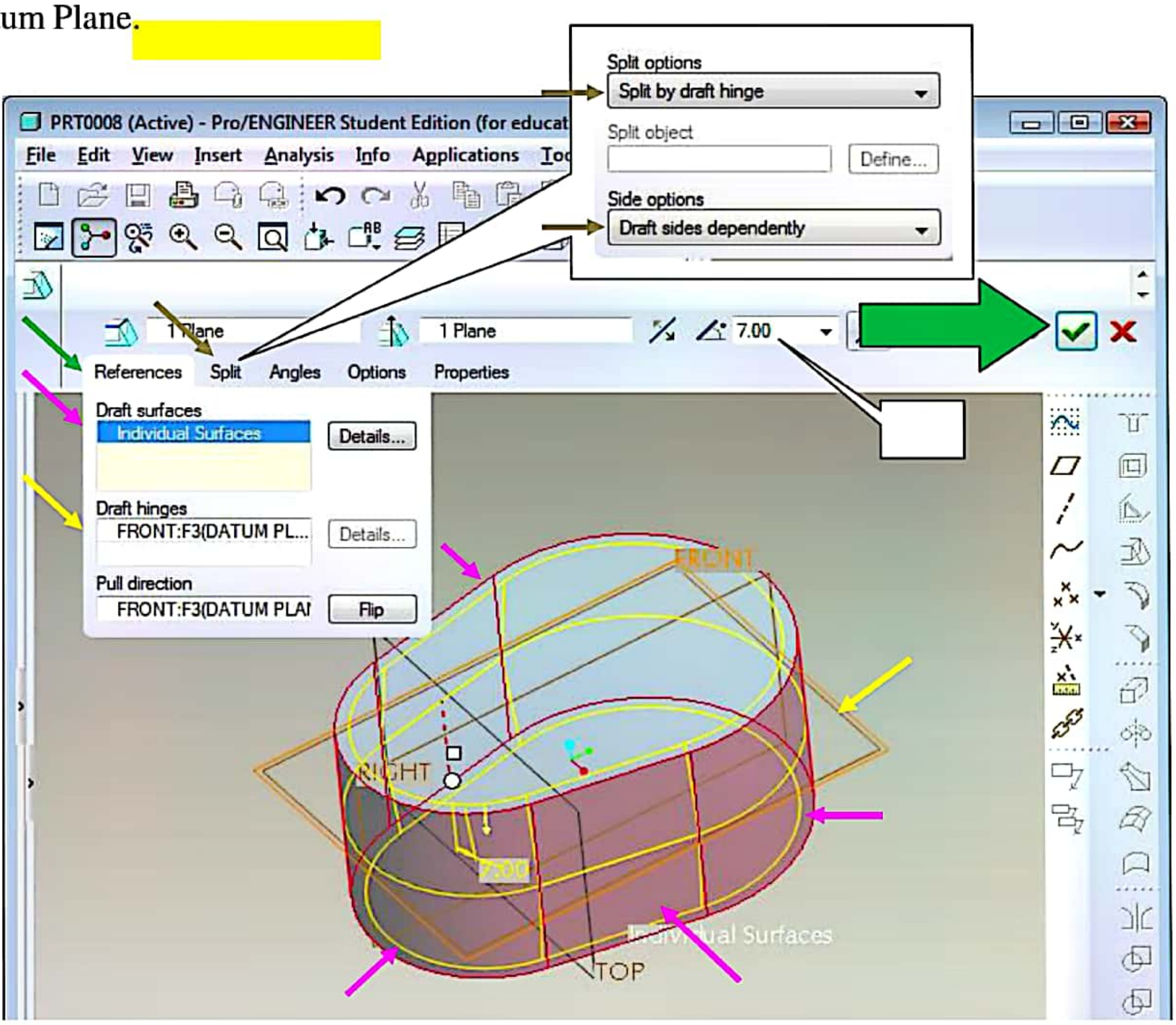




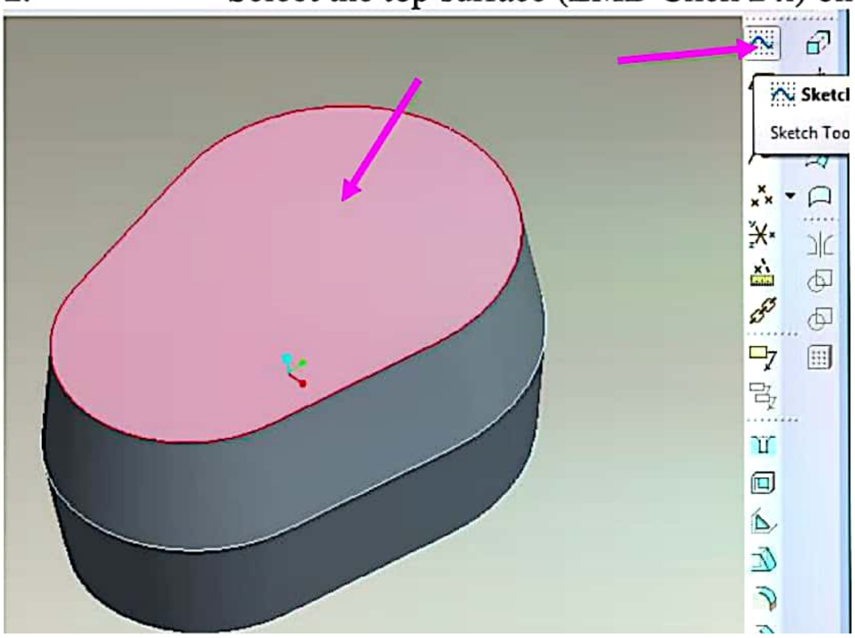
2. **Extrude**. Select Mid-Plane,

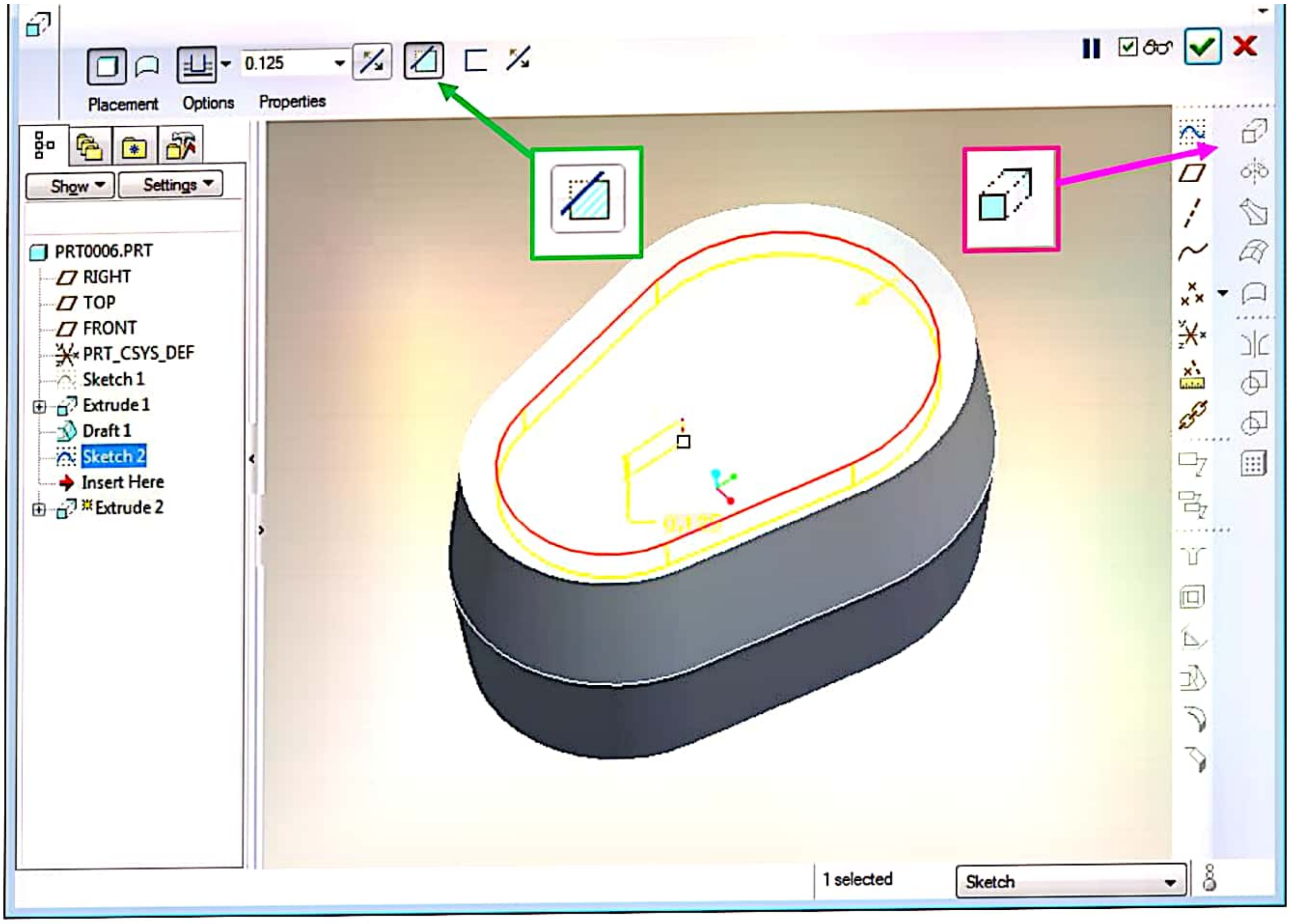


1. **DRAFT**: Select the Draft tool, and then References, Ctrl select all side faces of the model. Then Click on the draft hinges dialog box, and select the Front Datum Plane.

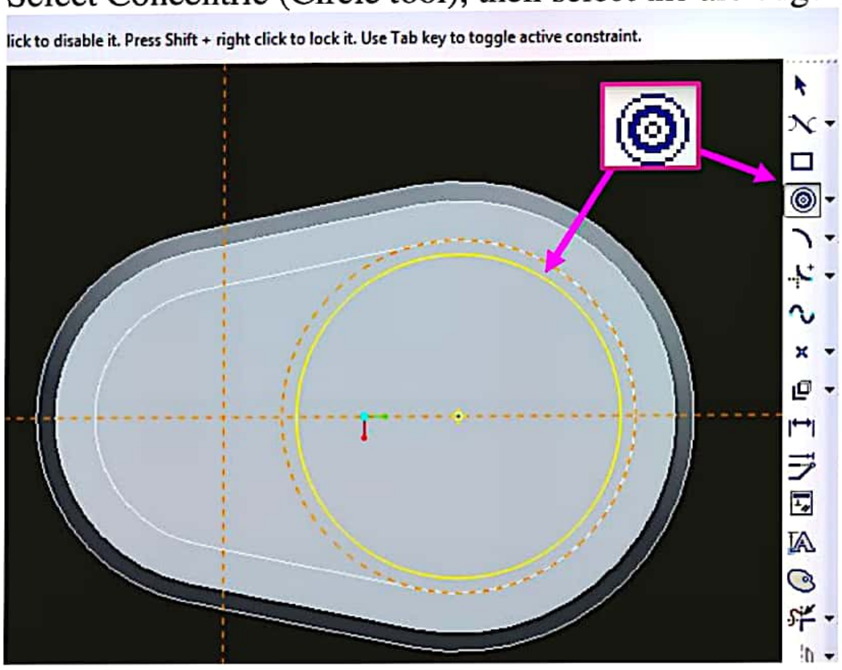


2. Select the top surface (LMB Click 2 x) on the model and start a sketch on it.

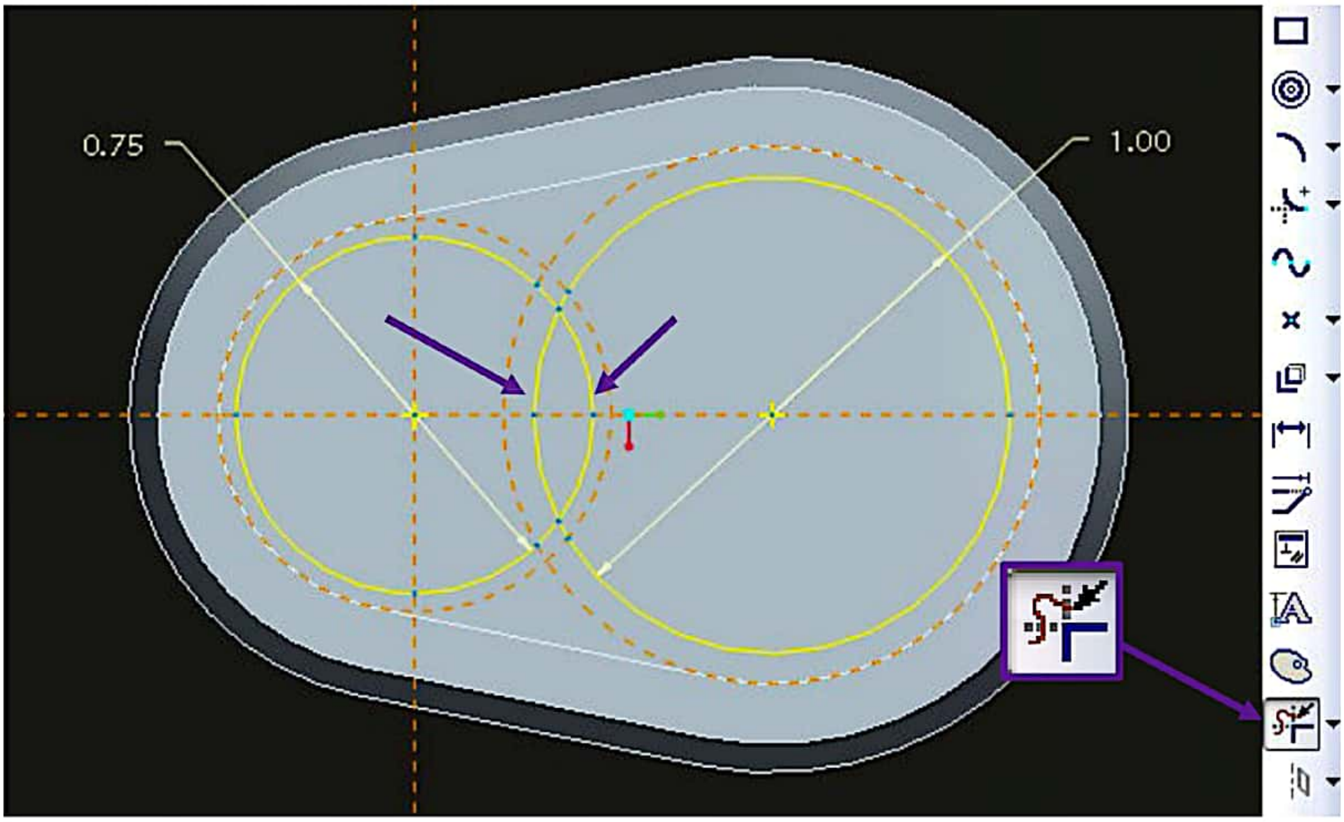




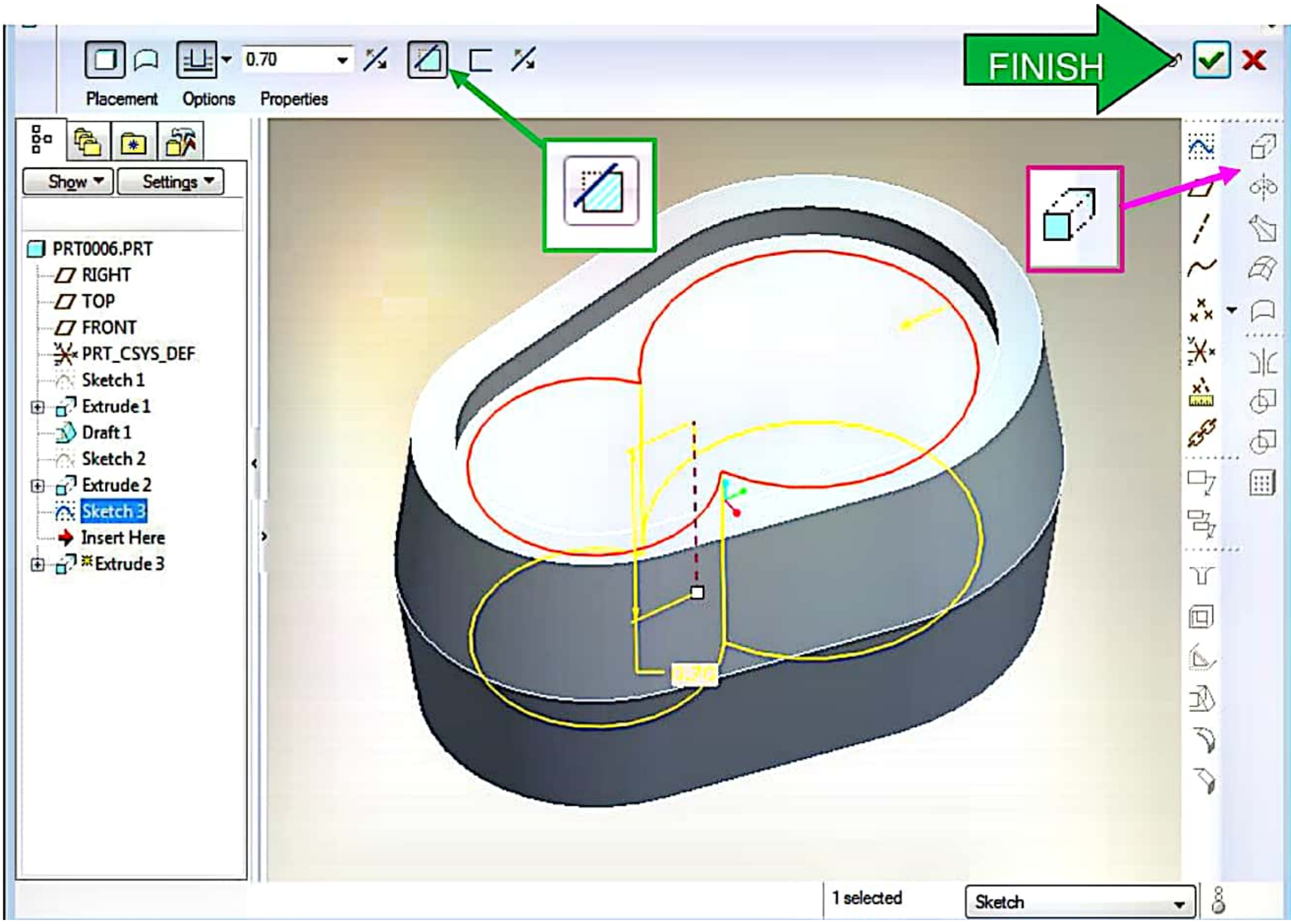
Select Concentric (Circle tool), then select the arc edge of the part.

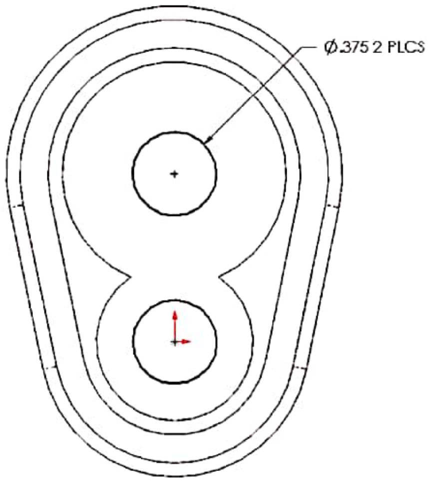


9. **Trim** the intersection.

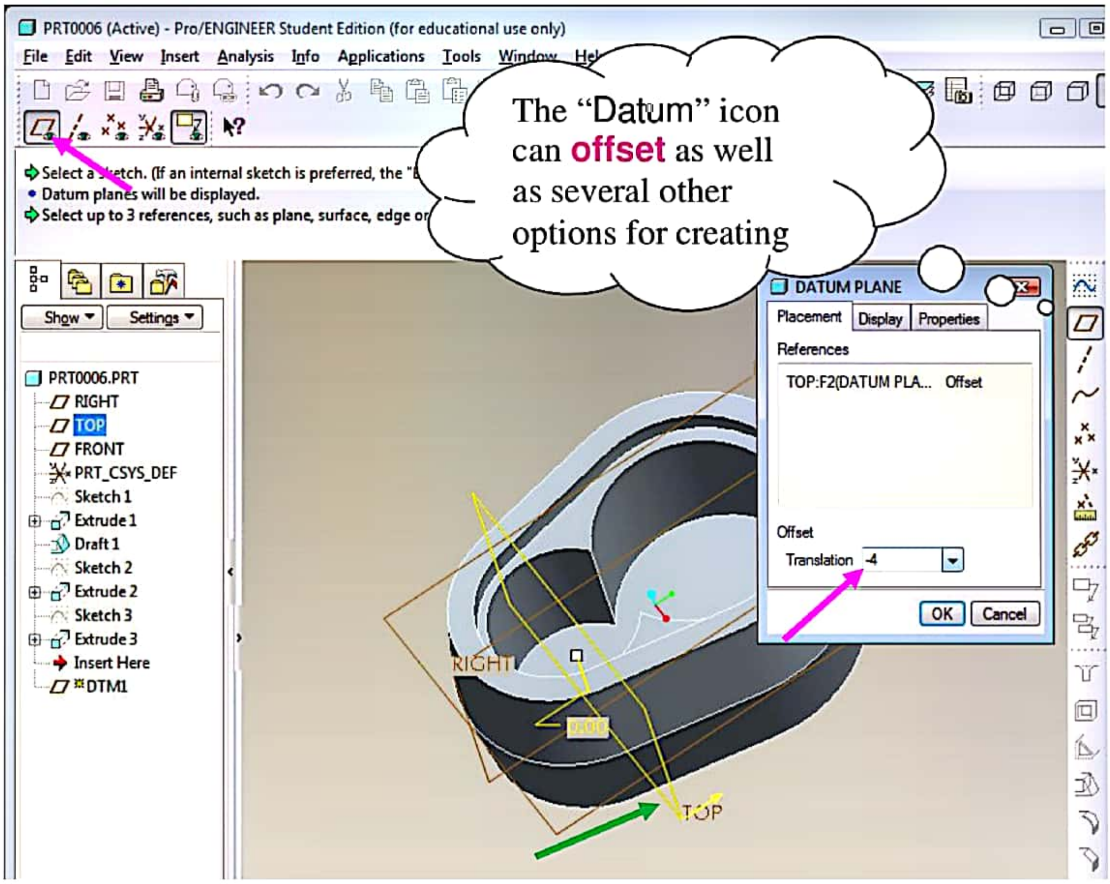


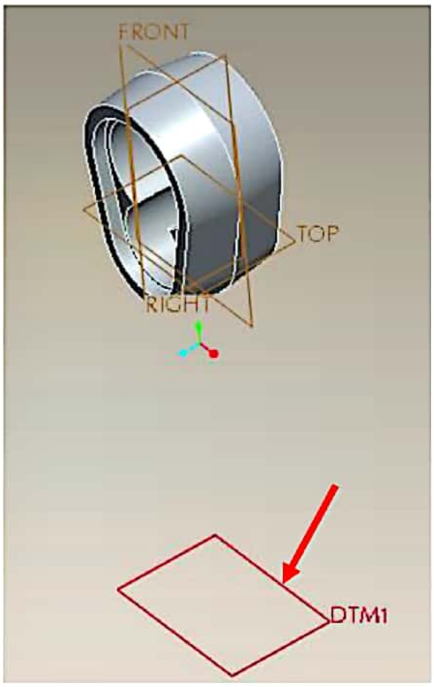
9. Select the extrude icon, and cut depth.



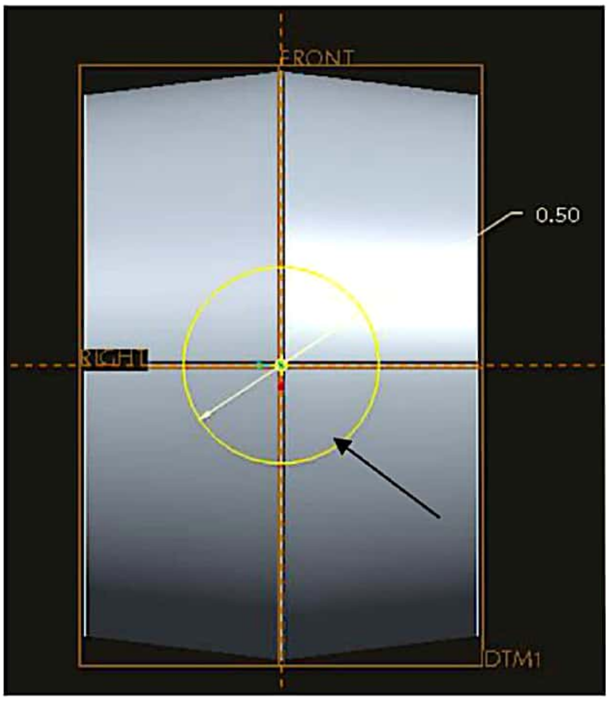


11. **DATUM PLANE OFFSET:** Select the Top datum plane, then select the Datum icon. Set to **-4 offset**.

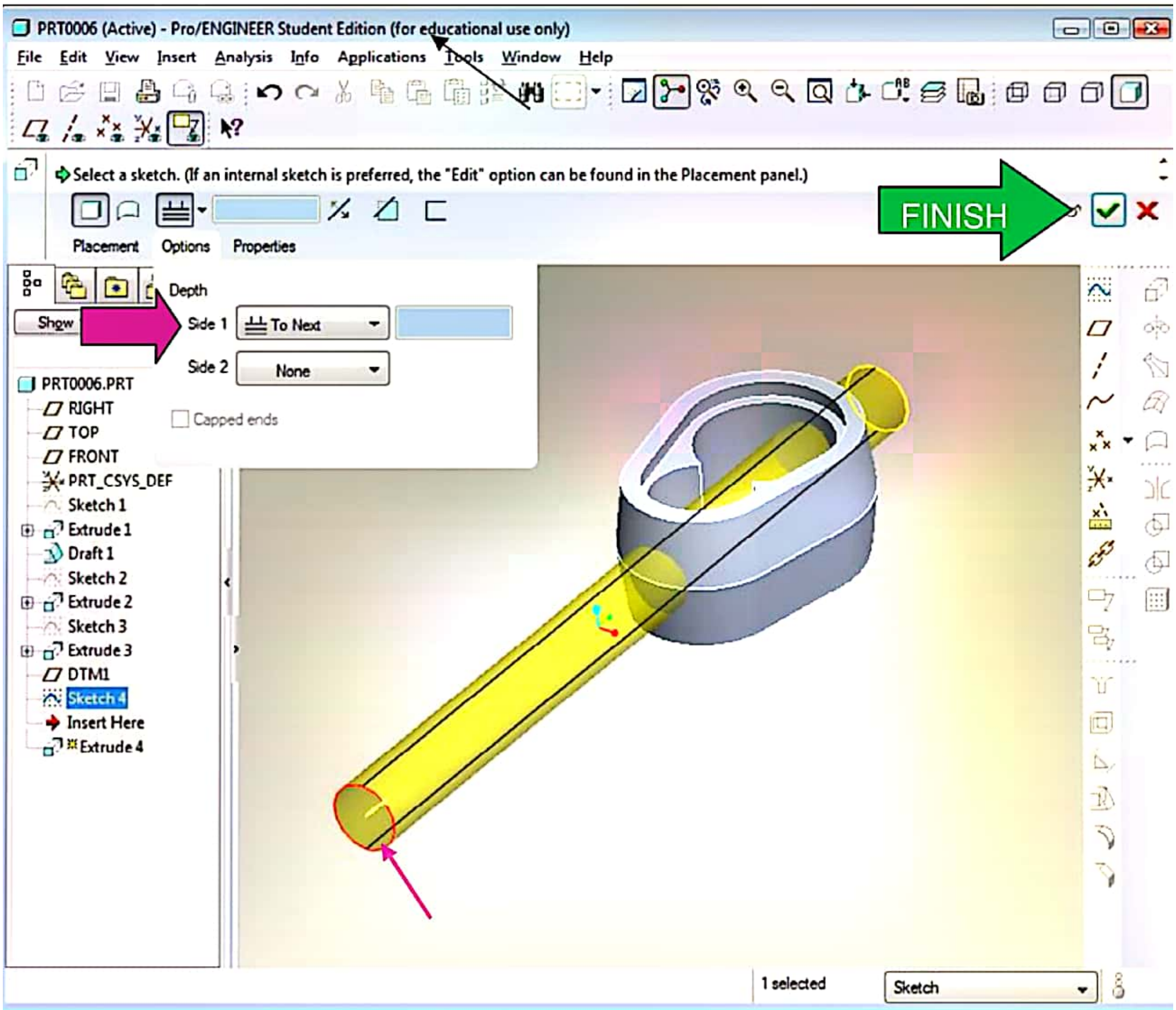


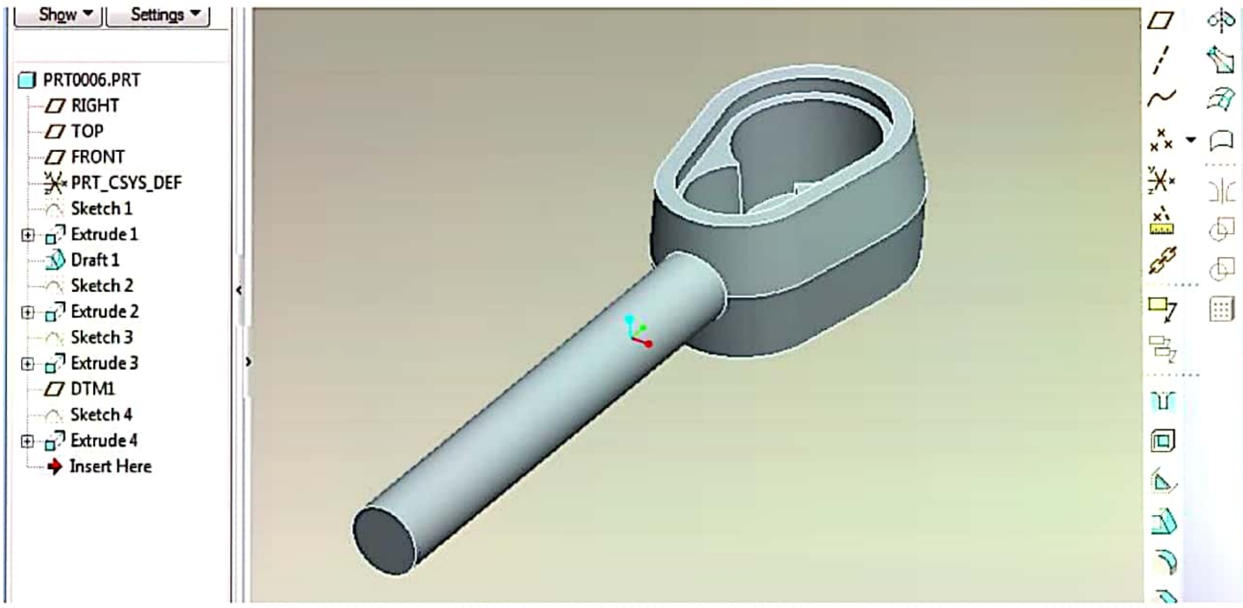


12. Extrude

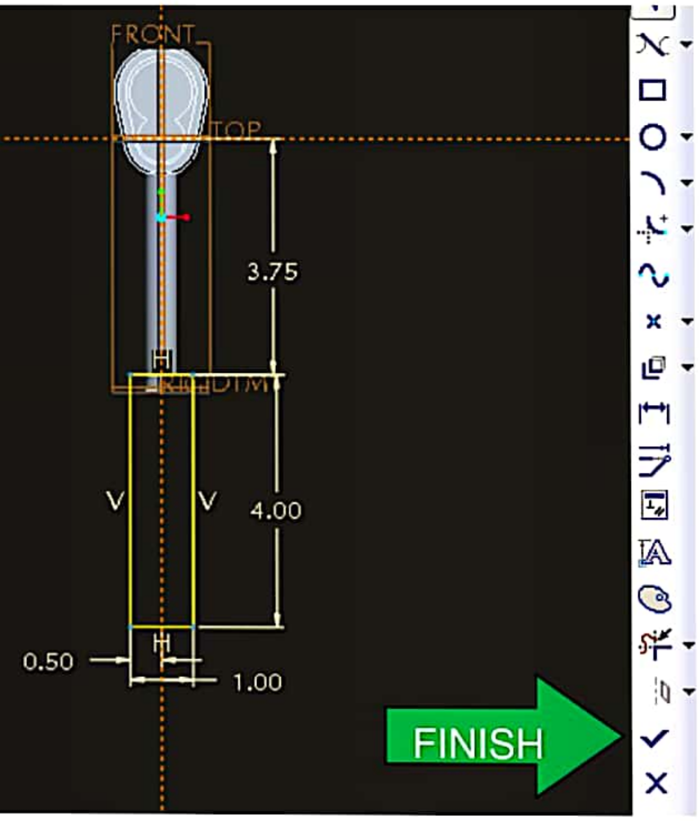


15.Select the circle and use the setting as shown in the illu stration below.

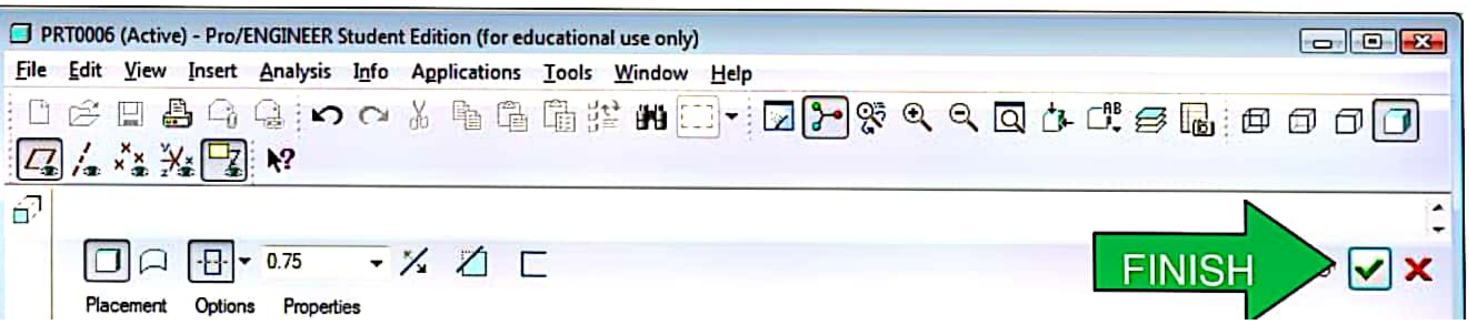


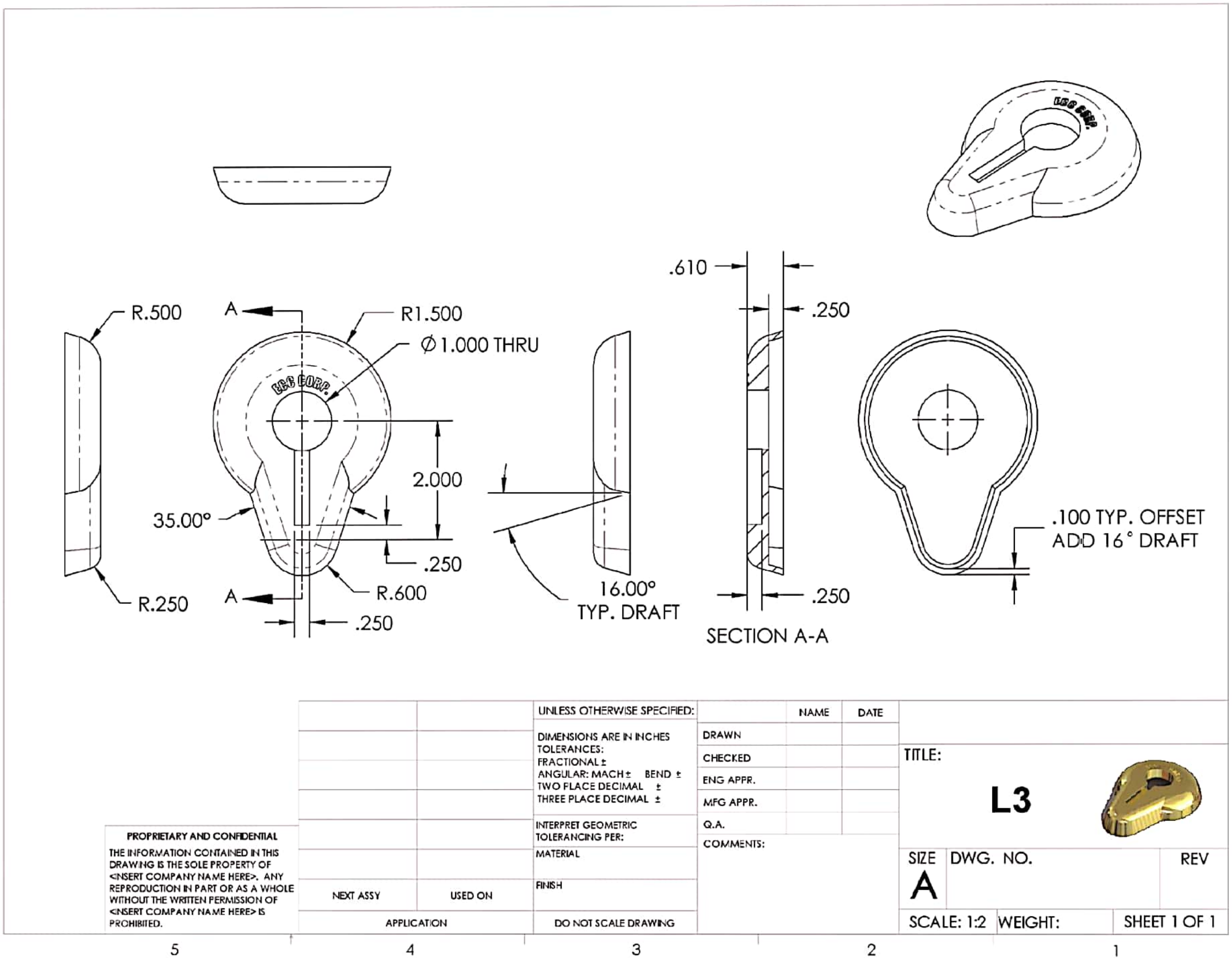


7. Start a sketch on the front datum plane and draw a rectangle vith the following dimensions.



18. Extrude boss using the mid-plane option and .750 thick.




R.500
R.250
A
A
R1.500
Ø1.000 THRU
ECC CORP.
2.000
35.00°
.250
R.600
.250
16.00°
TYP. DRAFT
.610
.250
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SECTION A-A
.100 TYP. OFFSET
ADD 16° DRAFT
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DIMENSIONS ARE IN INCHES
TOLERANCES:
FRACTIONAL ±
ANGULAR: MACH ± BEND ±
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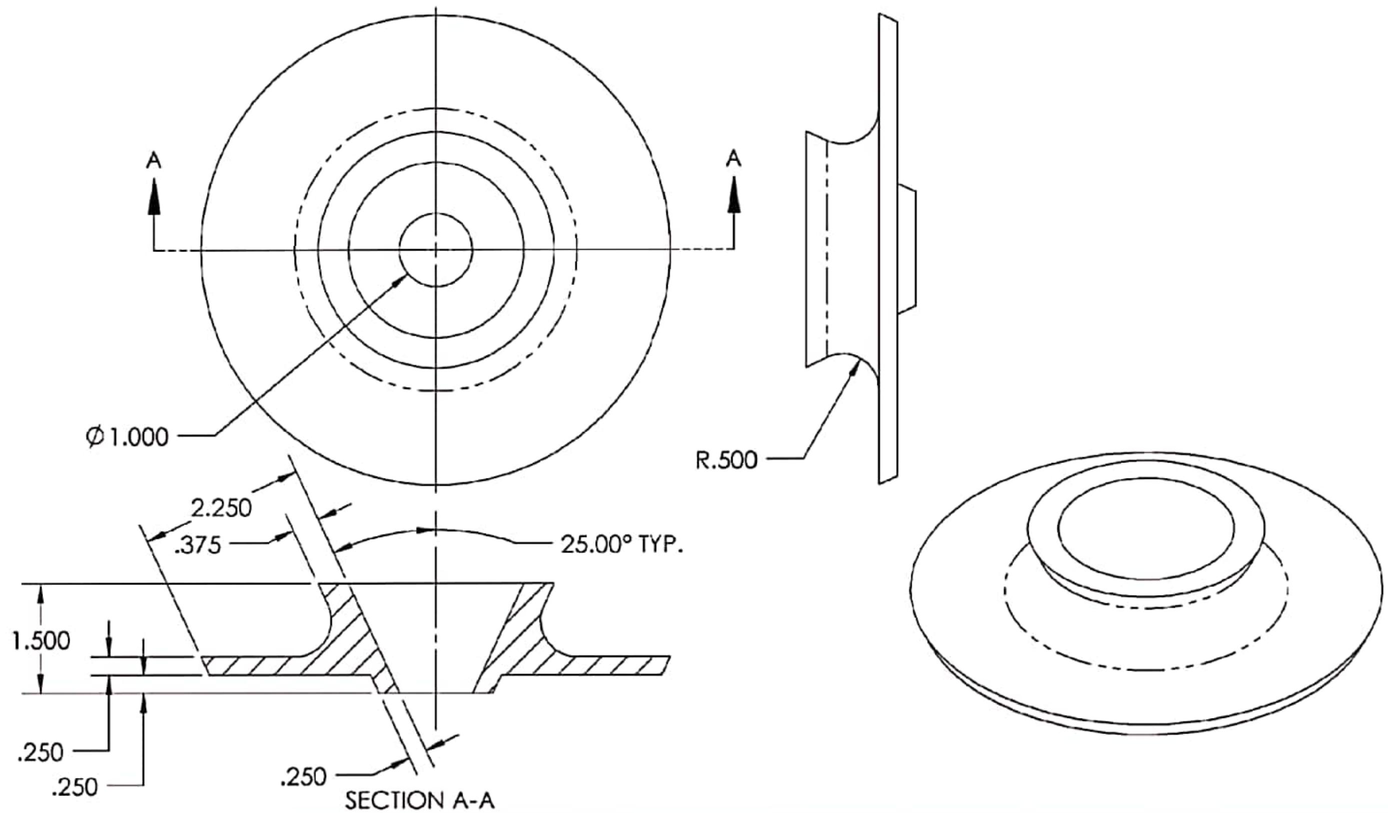

UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN INCHES
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ANGULAR: MACH ± BEND ±
TWO PLACE DECIMAL ±
THREE PLACE DECIMAL ±
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APPLICATION

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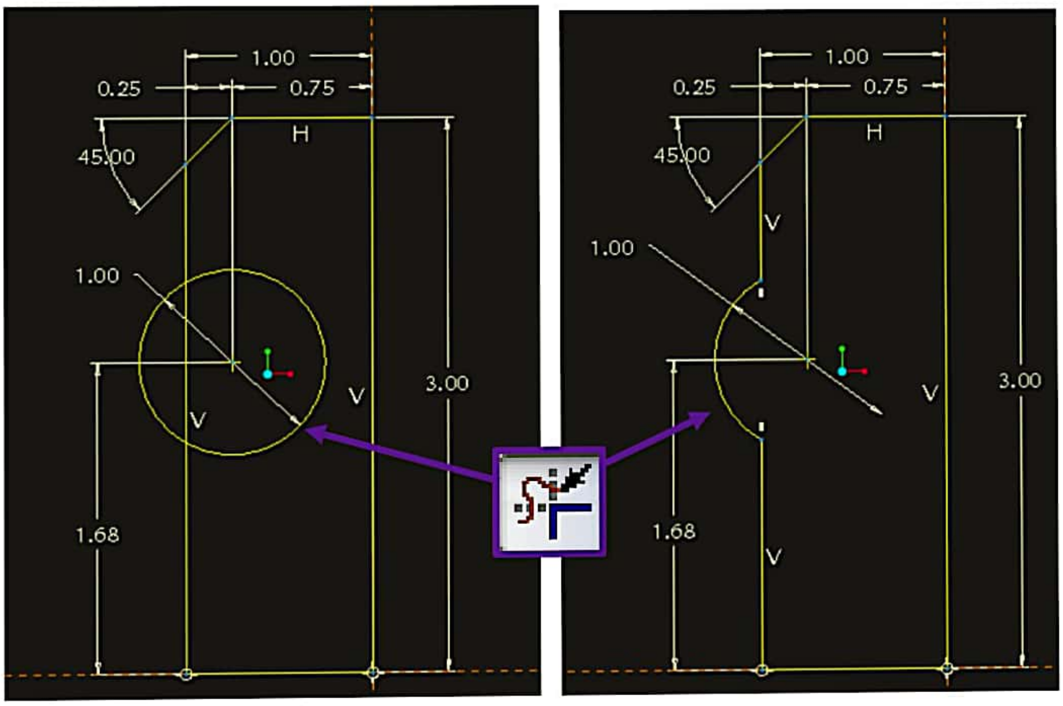
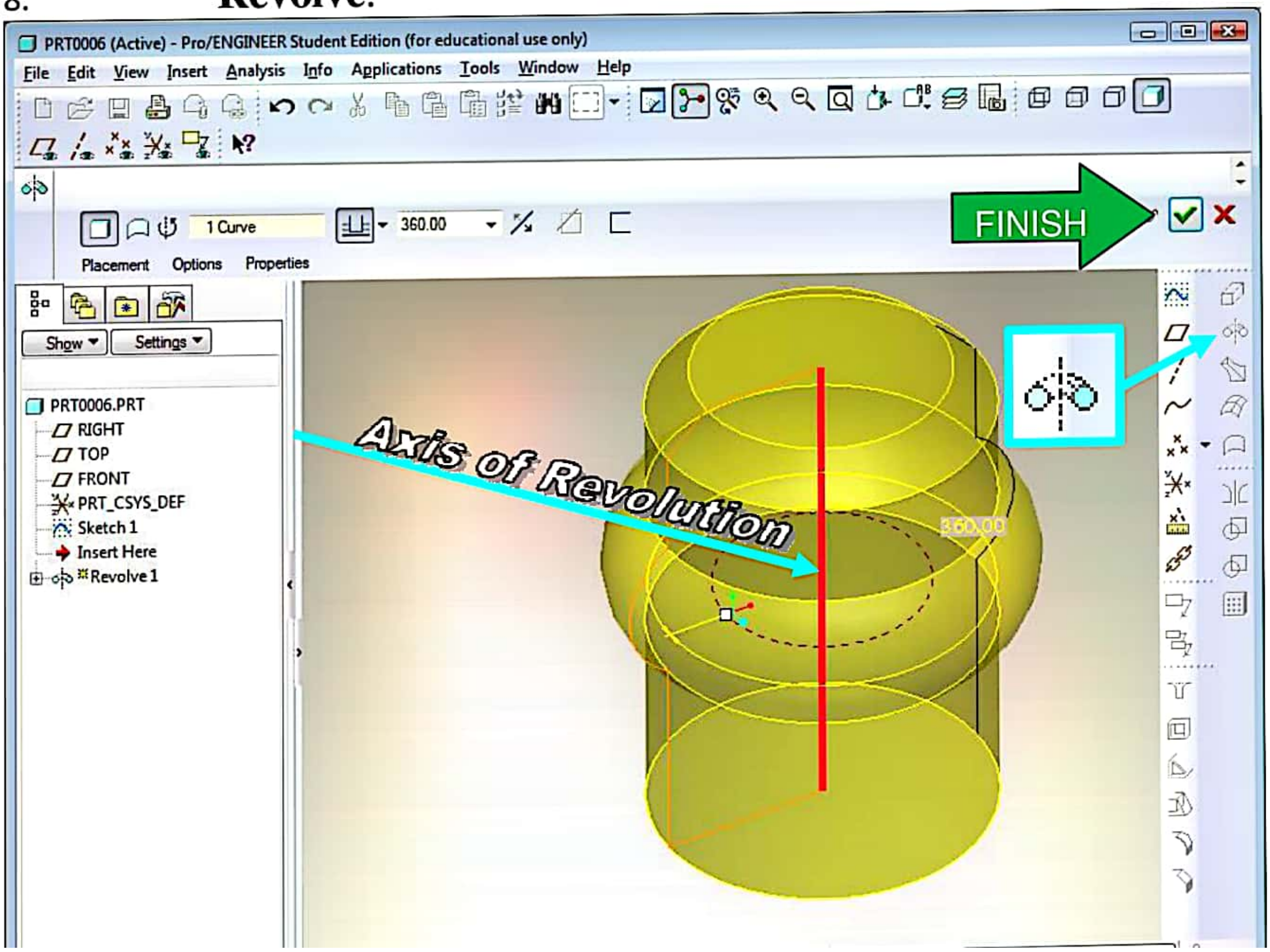
EXERCISE 4
Secondary Feature Modeling

1. sketch the geometry as shown below

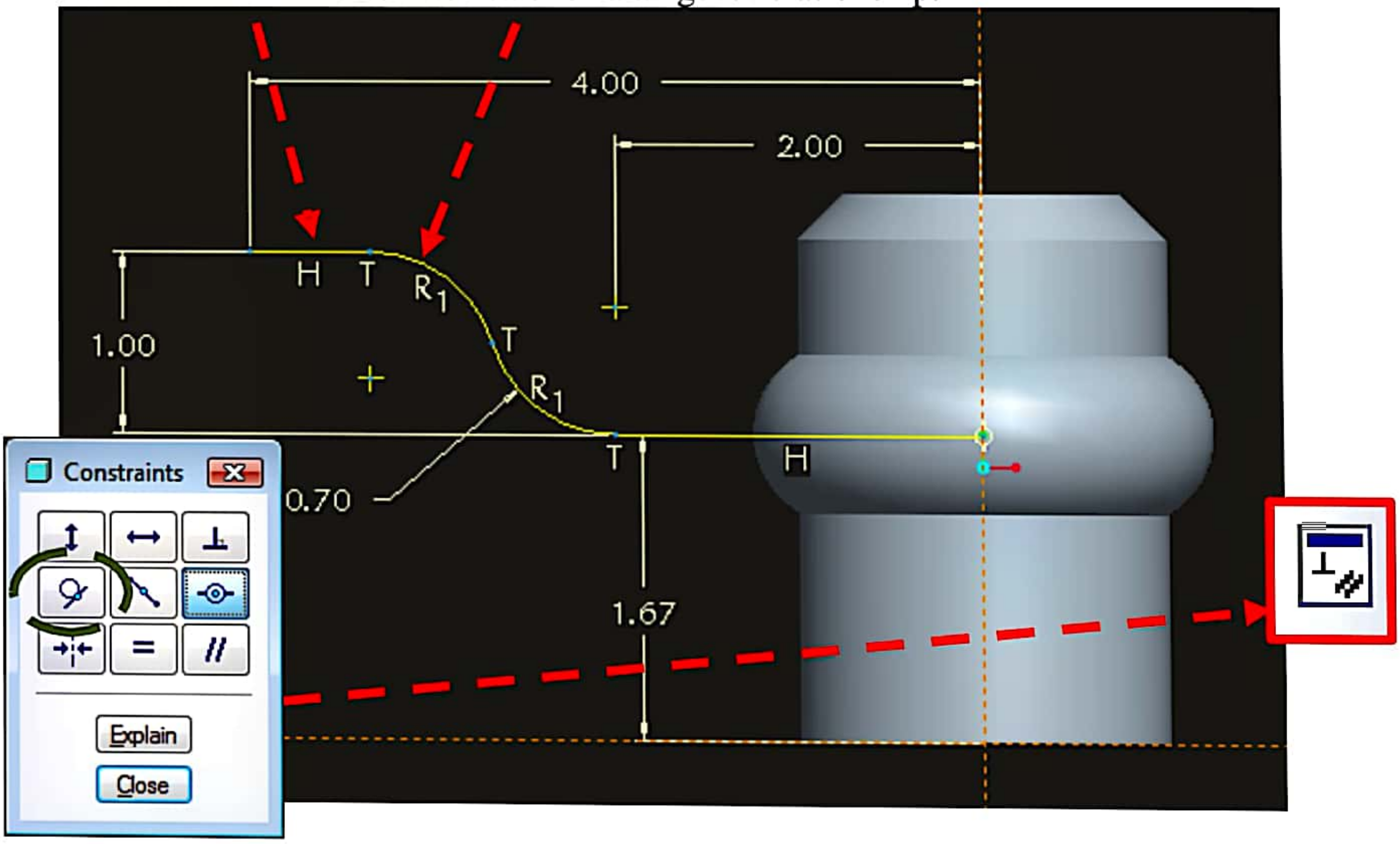
Then **Trim.**



8. **Revolve.**



line and the arc attached to it to establish a tangent relationship.

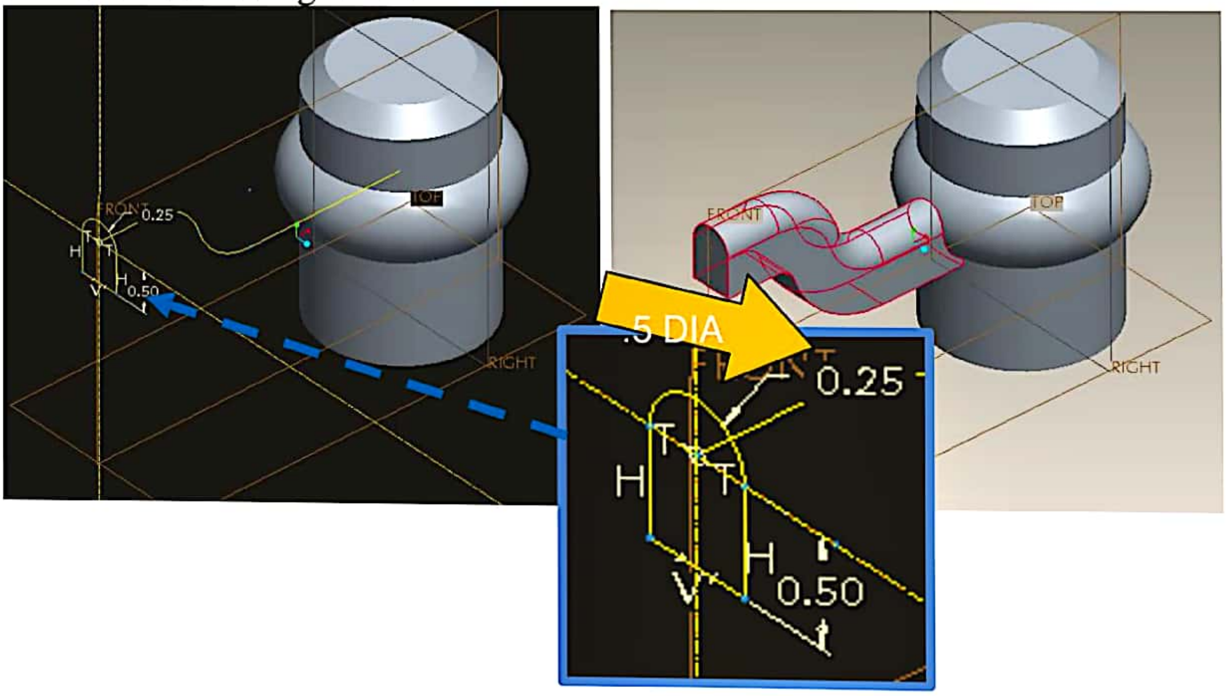


9. **Sweeps**: left side of the curve we just created to create a new sketch datum at the end.

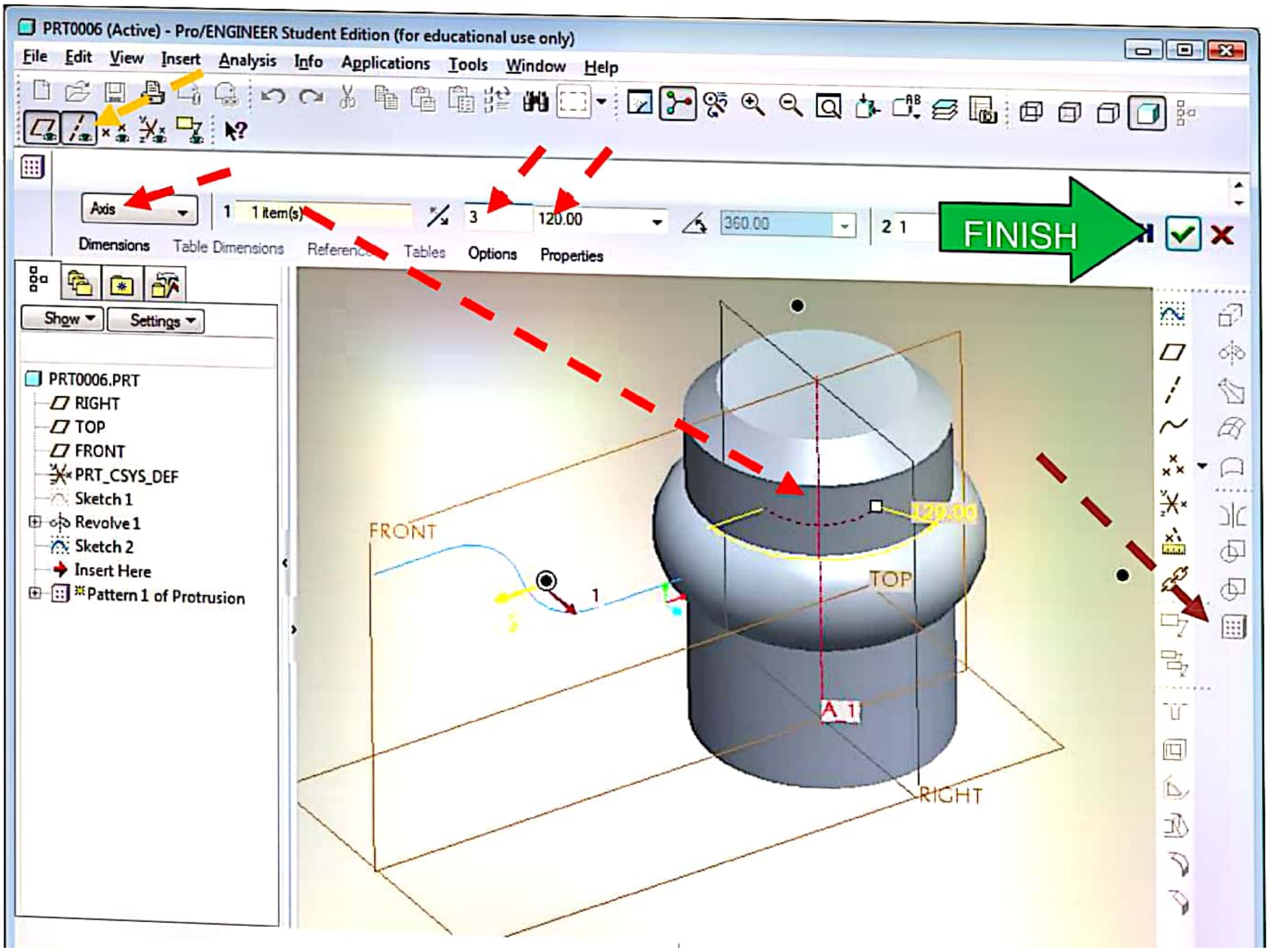
10. Also select: □SelectTraj/Curve Chain/Select All/Done/Done□

Select an edge from chain, pick 'Done' when finished.

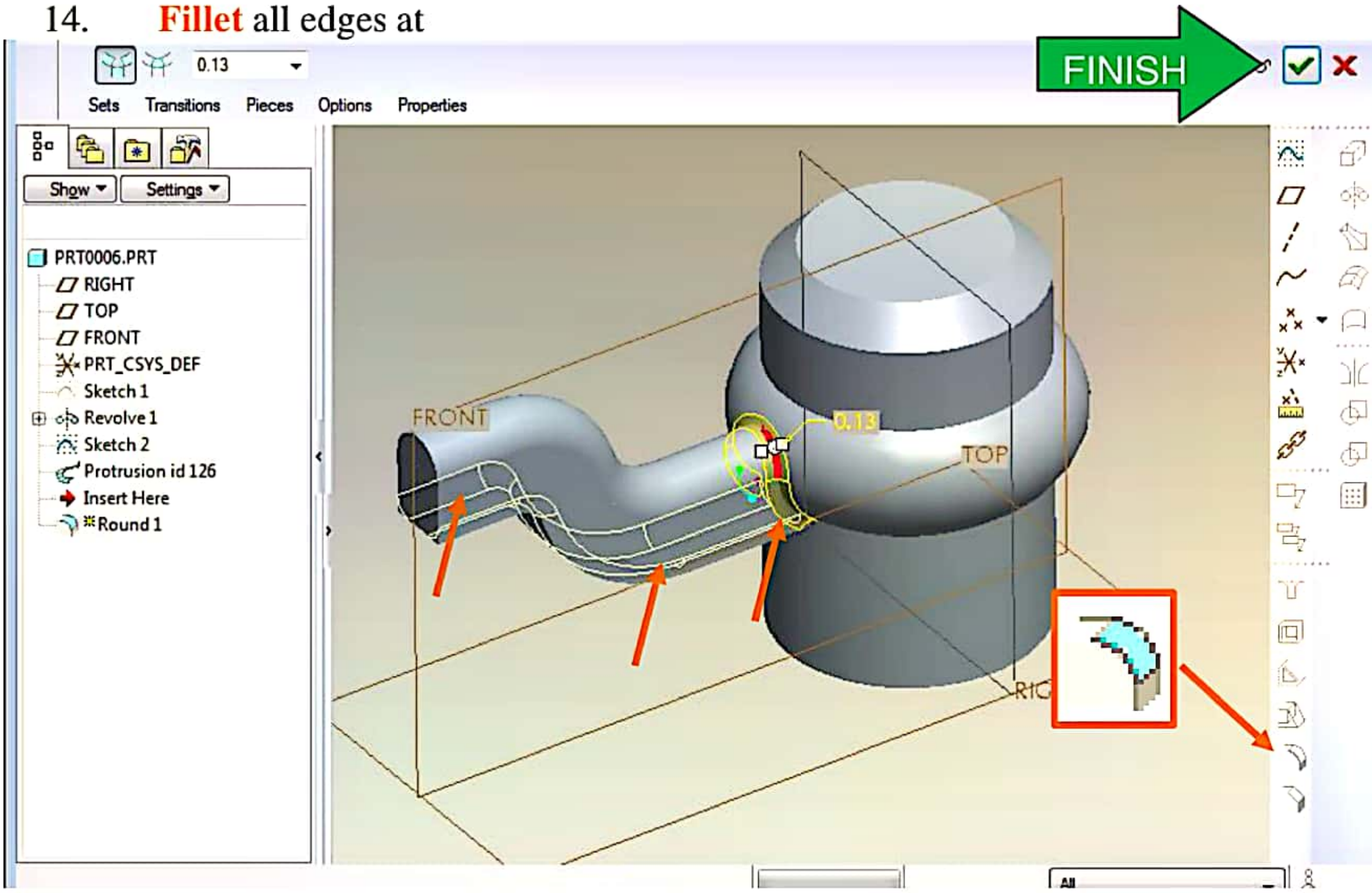
8.Draw the following sketch

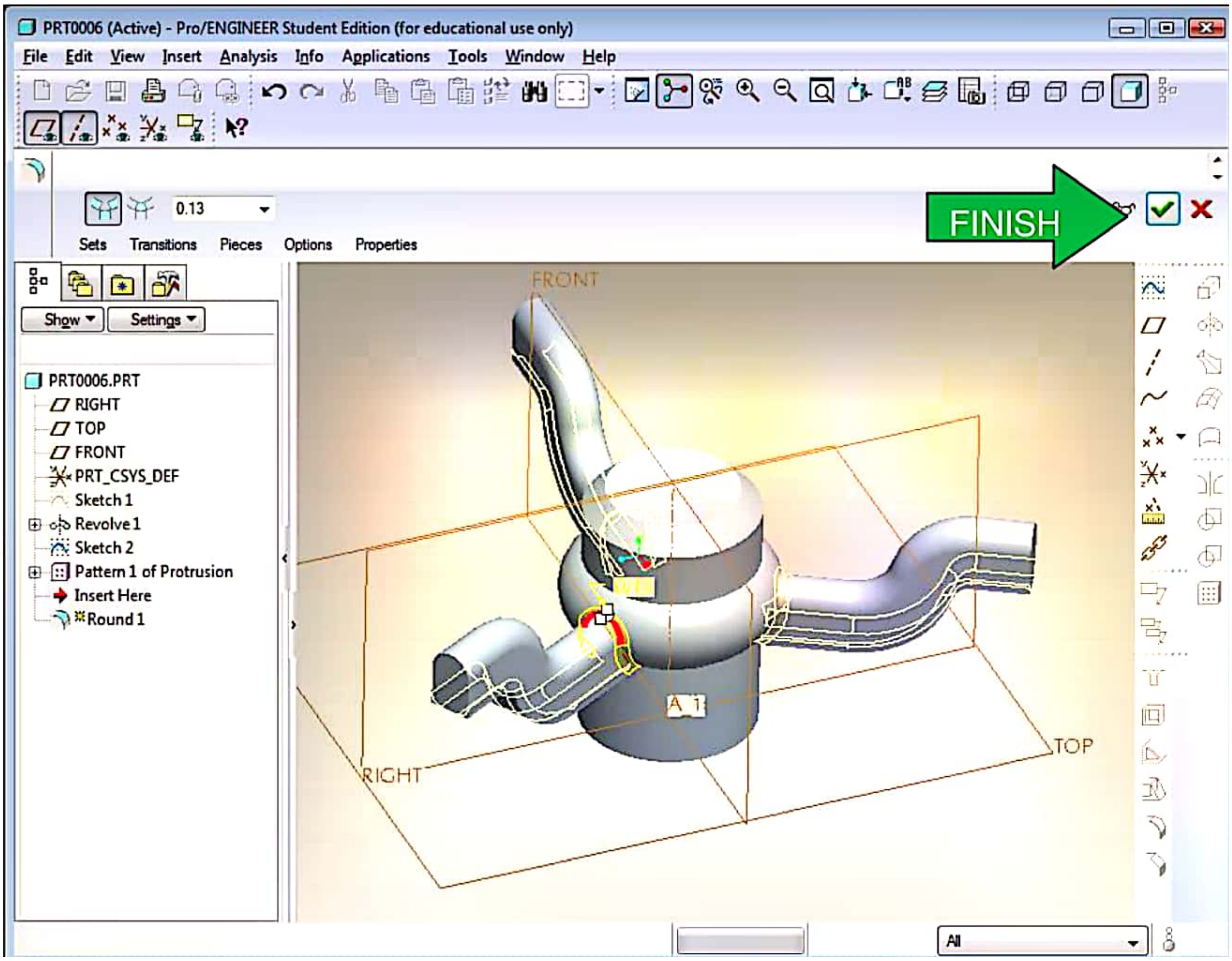


8. **Pattern** *Circular Pattern*: 360°/3 = 120° (*NOTE: First select the spoke to activate the icon.*) "SelectAxis□ also select the □view axis□

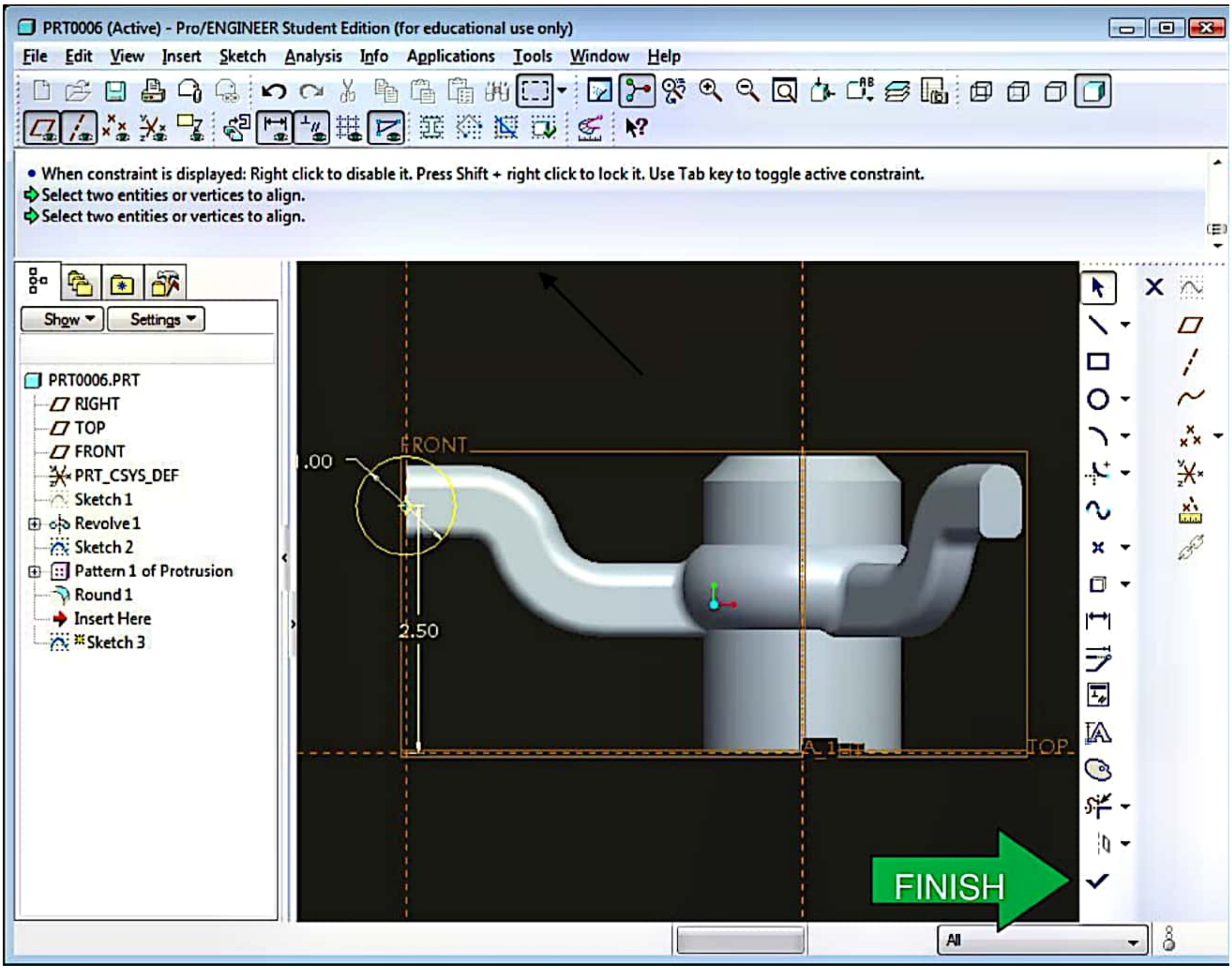


14. **Fillet** all edges at

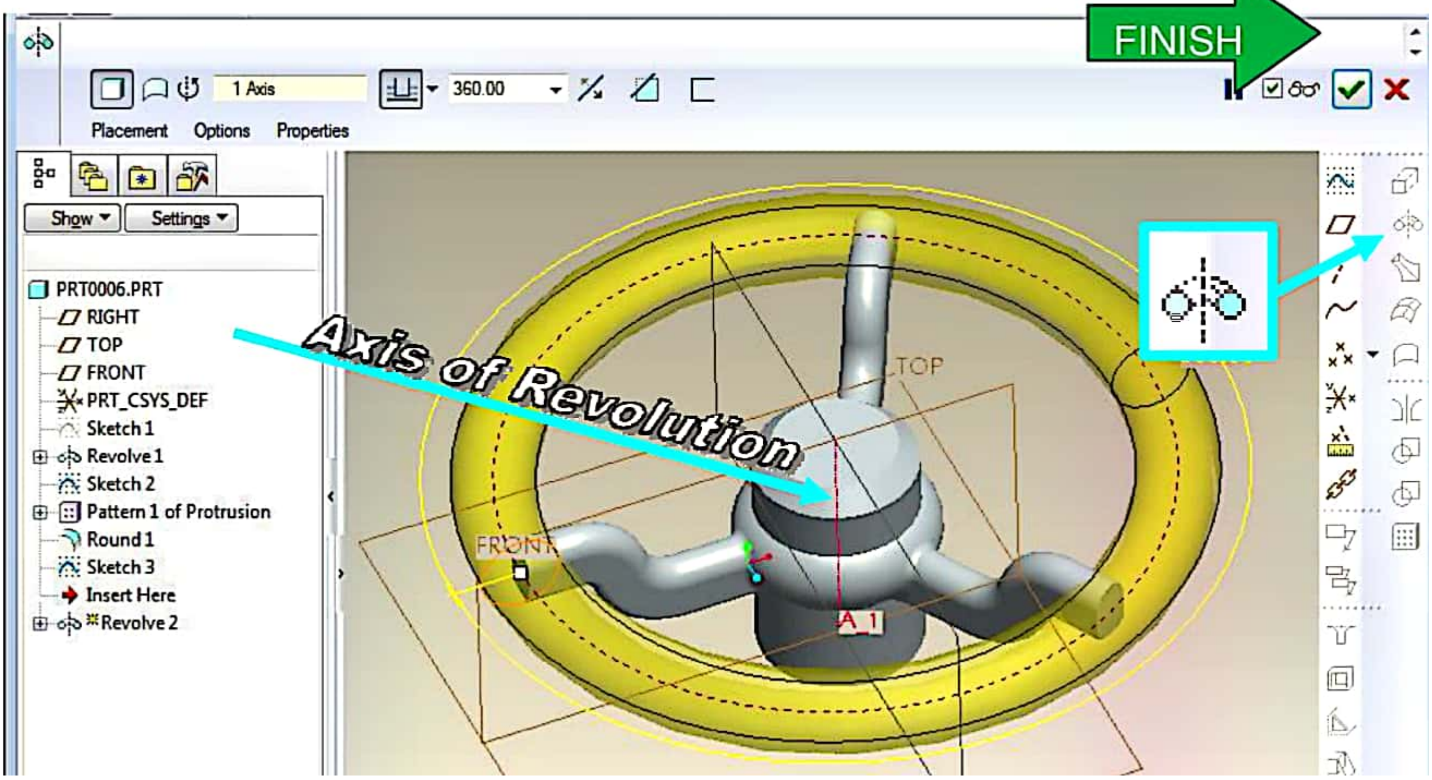


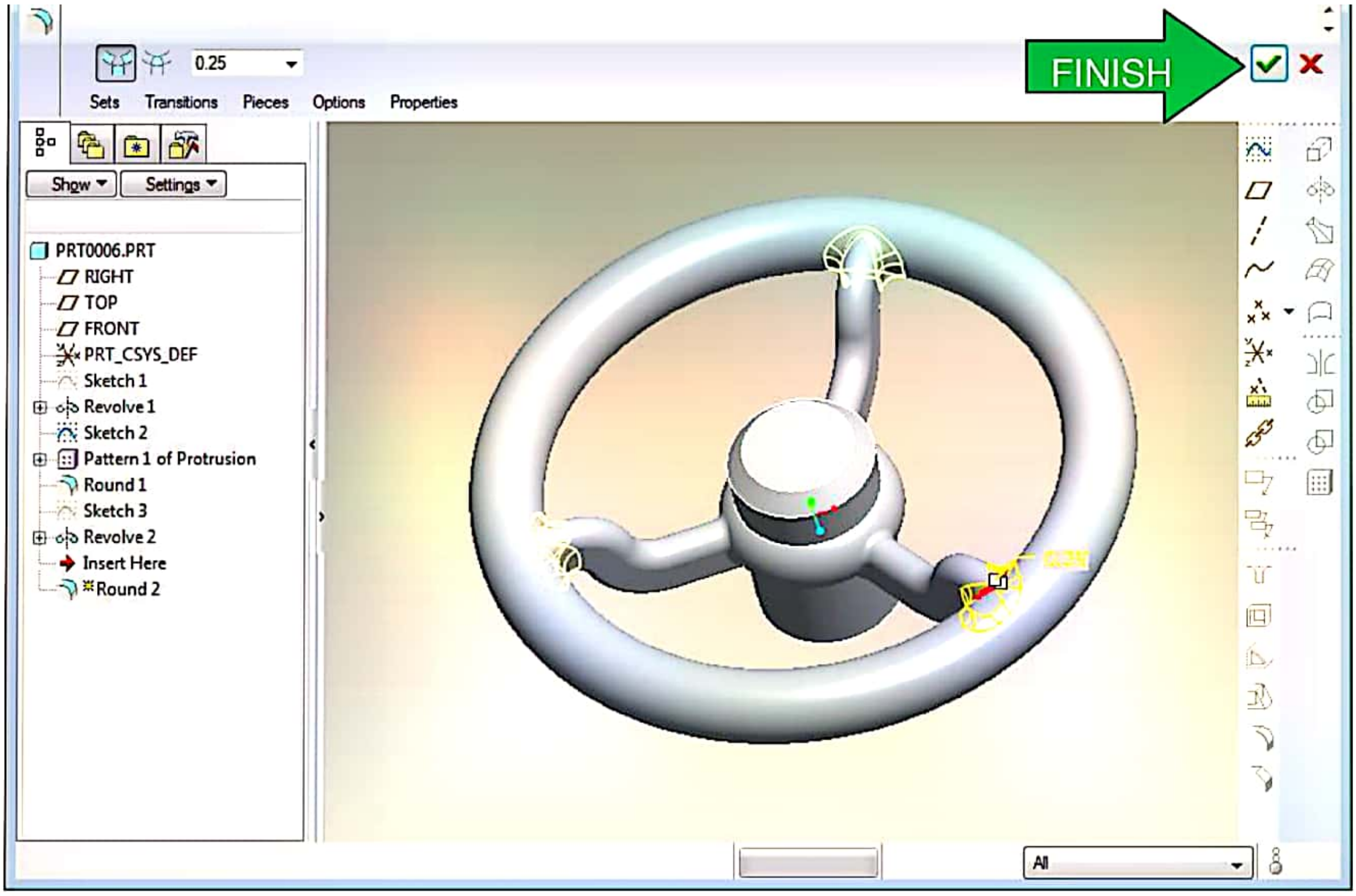


15. Rebuild after completion.



16. REVOLVE

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FINISH
Sets
Transitions
Pieces
Options
Properties
Show
Settings
PRT0006.PRT
RIGHT
TOP
FRONT
PRT_CSYS_DEF
Sketch 1
Revolve 1
Sketch 2
Pattern 1 of Protrusion
Round 1
Sketch 3
Revolve 2
Insert Here
Round 2
All

FINISHED


Finished