CAD

LABORATORY MANUAL

B.TECH

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Prepared by:

Mr. Shailesh babu

Assistant Professor

Department of Aeronautical Engineering

MALLA REDDY COLLEGE OF ENGINEERING AND TECHNOLOGY

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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 drafting
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. Colors 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.   CREO 3.0

3.   CATIA V5

4.   ANSYS

5.   NSC NASTRAN

6.   IDEAS

7.   SOLID WORKS

8.   HYPERMESH

9.   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.

STARTING UP AUTOCAD 2016

1. Select the **AutoCAD 2016** option on the *Program* menu or select the **AutoCAD2016** icon on the *Desktop*. Click **Start Drawing** to start a new drawing.
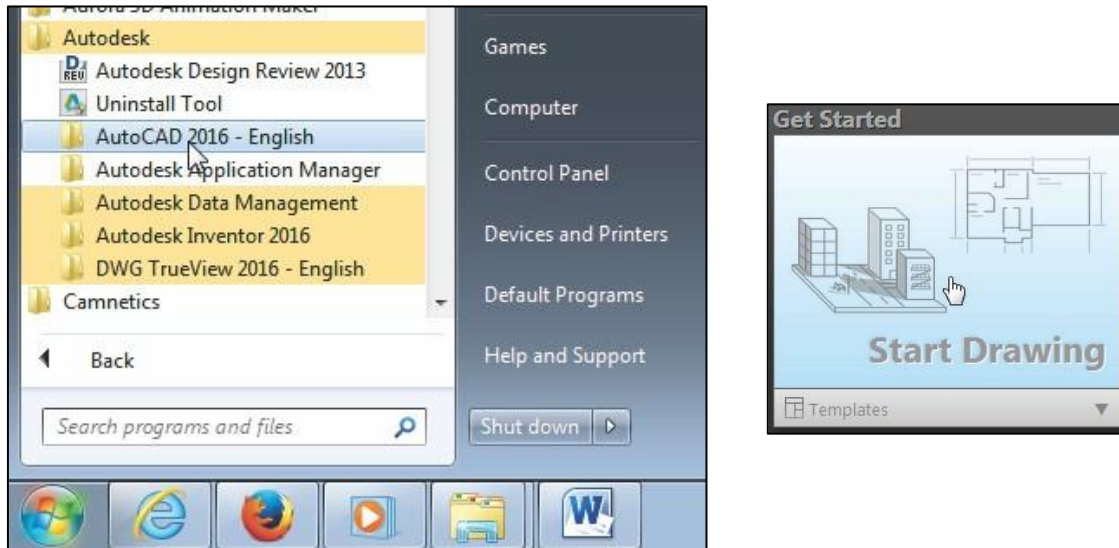
Once the program is loaded into memory, the **AutoCAD 2016** main drawing screen will appear on the screen.

Note that AutoCAD automatically assigns generic names, *Drawing X*, as new drawings are created. In our example, AutoCAD opened the graphics window using the default system units and assigned the drawing name *Drawing1*.
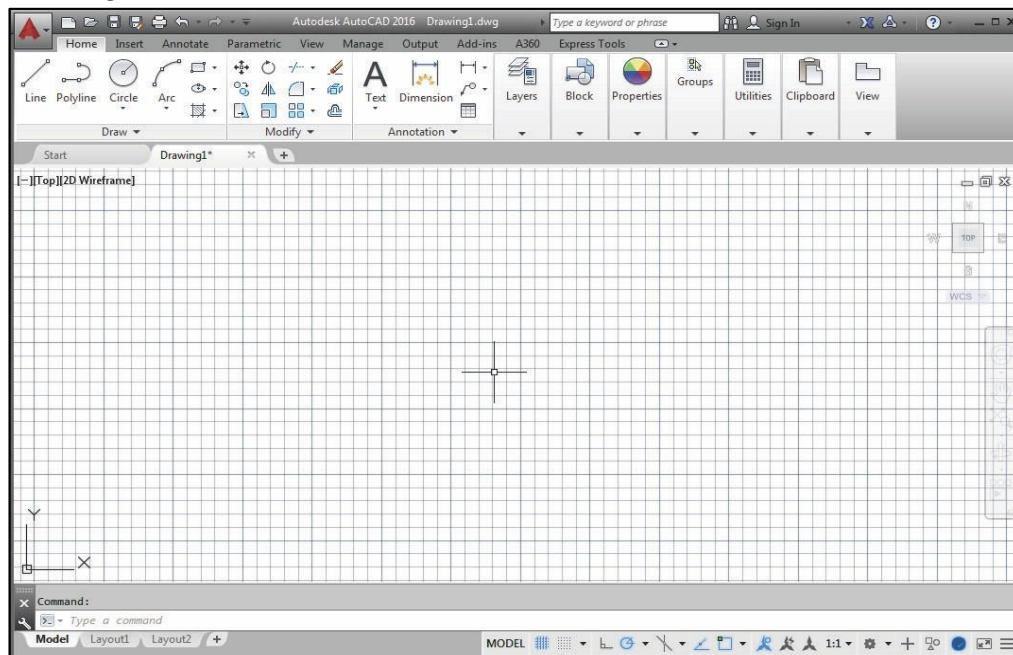
2. If necessary, click on the down-arrow in the *Quick Access* bar and select **Show Menu Bar** to display the **AutoCAD** *Menu Bar*. The *Menu Bar* provides access to all AutoCAD commands.
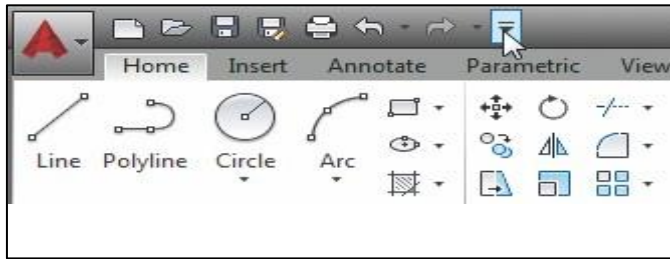
3. To switch on the **AutoCAD Coordinates Display**, use the *Customization option* at the bottom right corner.
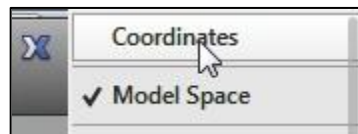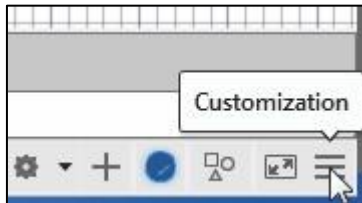
Drawing Units Setup

Every object we construct in a CAD system is measured in **units**. We should determine the system of units within the CAD system before creating the first geometric entities.

1. In the *Menu Bar* select:
[Format] [Units]
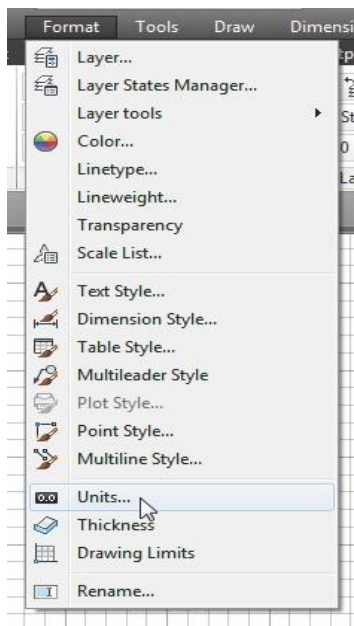
• The AutoCAD *Menu Bar* contains multiple pull-down menus where all of the AutoCAD commands can be accessed. Note that many of the menu items listed in the pull-down menus can also be accessed through the *Quick Access* toolbar and/or *Ribbon* panels.

• Click on the *Length Type* option to display the different types of length units available. Confirm the *Length Type* is set to **Decimal**.
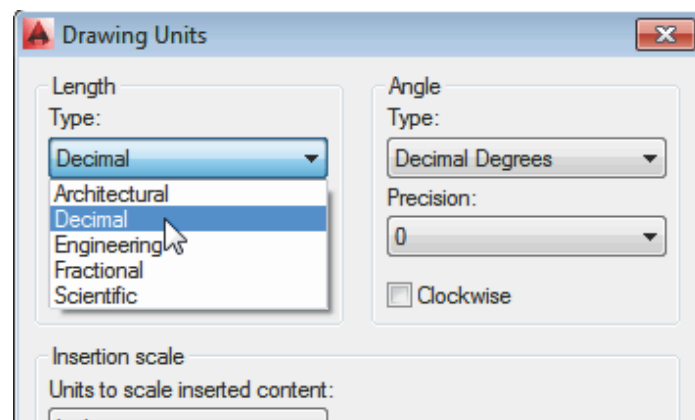
3. Angle Type also can be changed by clicking the pull down menu of *Angle Type*.
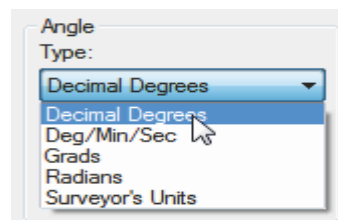
4. In the *Drawing Units* dialog box, set the *Length Type* to **Decimal**. This will set the measurement to the default *English* units, inches.
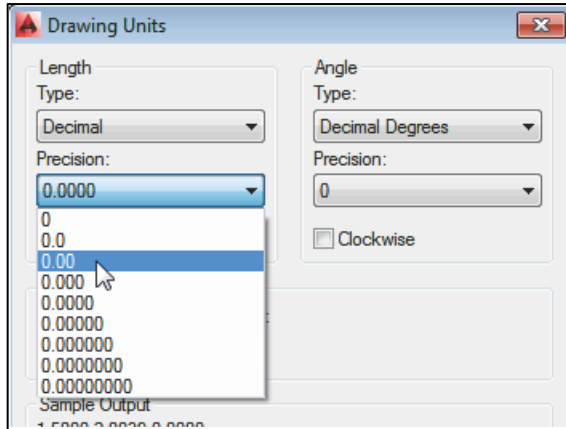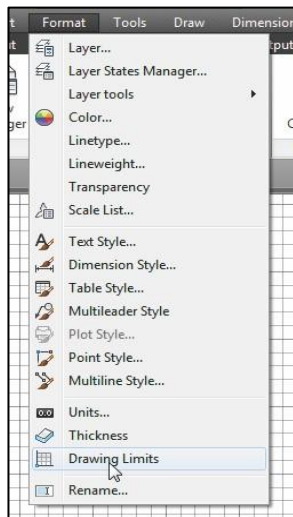

5. Set the *Precision* to **two digits** after the decimal point as shown in the above figure.

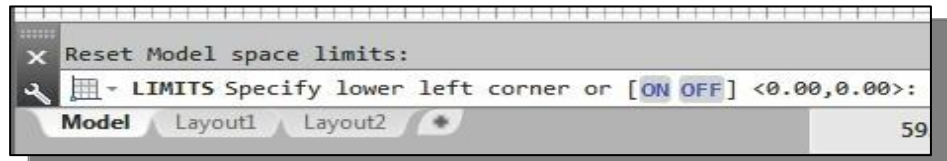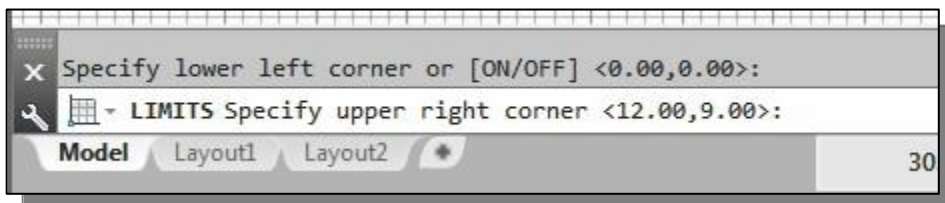6. Pick **OK** to exit the *Drawing Units* dialog box.
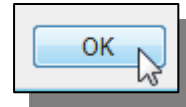
Drawing Area Setup

We will set up the **Drawing Limits** by entering a command in the command prompt area. Setting the Drawing Limits controls the extents of the display of the *grid*. It also serves as a visual reference that marks the working area. It can also be used to prevent construction outside the grid limits and as a plot option that defines an area to be plotted/printed. Note that this setting does not limit the region for geometry construction.

1. In the *Menu Bar* select:
   [Format] [Drawing Limits]

2. In the command prompt area, the message "*Reset Model Space Limits: Specify lower left corner or [On/Off] <0.00,0.00>:*" is displayed. Press the **ENTER** key once to accept the default coordinates <**0.00,0.00**>.


3. In the command prompt area, the message "*Specify upper right corner <12.00,9.00>:*" is displayed. Press the **ENTER** key again to accept the default coordinates <**12.00,9.00**>.

4. On your own, move the graphics cursor near the upper-right corner inside the drawing area and note that the drawing area is unchanged. (The Drawing Limits command is used to set the drawing area, but the display will not be adjusted until a display command is used.)

5. Inside the *Menu Bar* area select: **[View] [Zoom] [All]**
   The **Zoom All** command will adjust the display so that all objects in the drawing are displayed to be as large as possible. If no objects are constructed, the Drawing Limits are used to adjust the current viewport.

6. Move the graphics cursor near the upper-right corner inside the drawing area and note that the display area is updated.

7. Hit the function key [**F7**] once to turn **off** the display of the *Grid* lines.
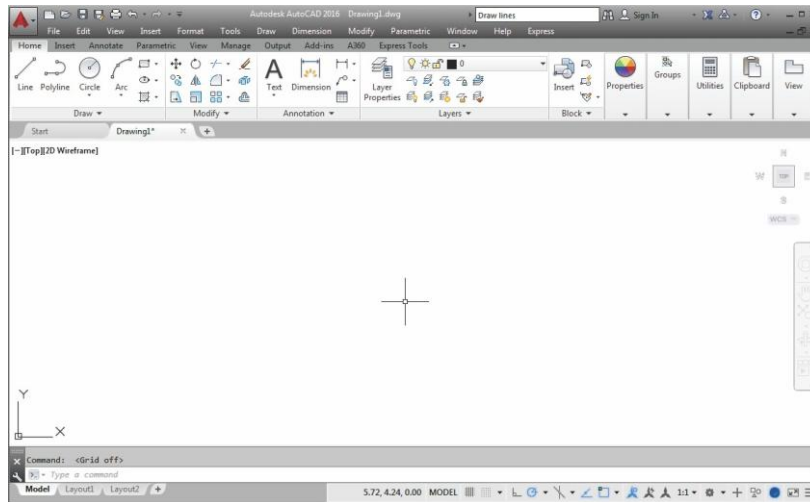
• Note that function key [**F7**] is a quick key, which can be used to quickly toggle on/off the grid display. Also, note the *command prompt area* can be positioned to dock below the drawing area or float inside the drawing area as shown.

Visual Reference

The method we just used to create the freehand sketch is known as the **interactive method**, where we use the cursor to specify locations on the screen. This method is perhaps the fastest way to specify locations on the screen. However, it is rather difficult to try to create a line of a specific length by watching the displayed coordinates. It would be helpful to know what one inch or one meter looks like on the screen while we are creating entities. **AutoCAD 2016** provides us with many tools to aid the construction of our designs. For example, the *GRID* and *SNAP MODE* options can be used to get a visual reference as to the size of objects and learn to restrict the movement of the cursor to a set increment on the screen.

The *GRID* and *SNAP MODE* options can be turned *ON* or *OFF* through the *Status Bar*. The *Status Bar* area is located at the bottom left of the AutoCAD drawing screen, next to the cursor coordinates.

The second button in the *Status Bar* is the *SNAP MODE* option and the third button is the *GRID DISPLAY* option. Note that the buttons in the *Status Bar* area serve two functions: (1) the status of the specific option, and (2) as toggle switches that can be used to turn these special options *ON* and *OFF*. When the corresponding button is *highlighted*, the specific option is turned *ON*. Using the buttons is a quick and easy way to make changes to these *drawing aid* options. The buttons in the *Status Bar* can also be switched on and off in the middle of another command.
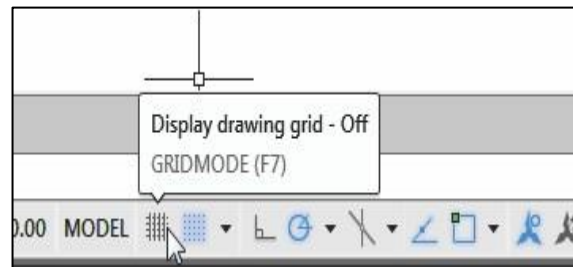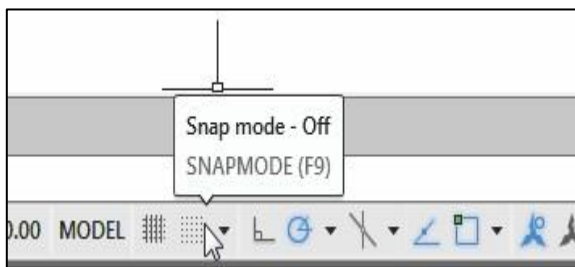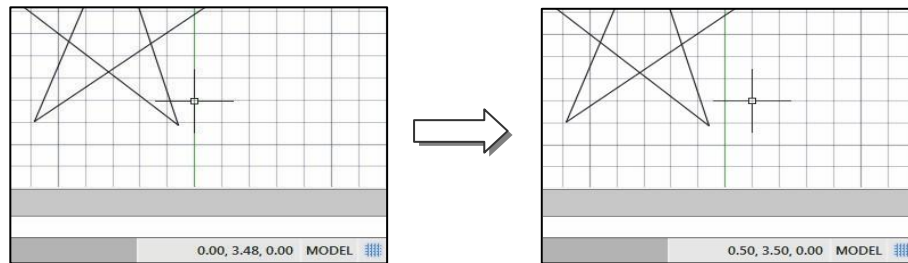

GRID ON

1. Left-click the **GRID** button in the *Status Bar* to turn **ON** the *GRID DISPLAY* option. (Notice in the command prompt area, the message *"<Grid on>"* is also displayed.)
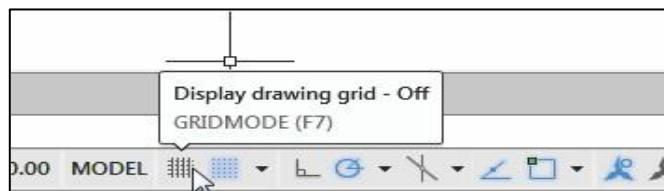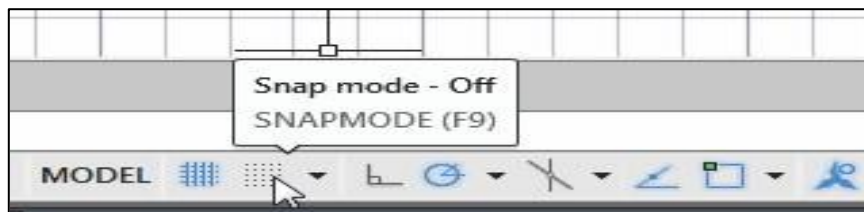
2. Move the cursor inside the graphics window, and estimate the distance in between the grid lines by watching the coordinates displayed at the bottom of the screen.

The *GRID* option creates a pattern of lines that extends over an area on the screen. Using the grid is similar to placing a sheet of grid paper under a drawing. The grid helps you align objects and visualize the distance between them. The grid is not displayed in the plotted drawing. The default grid spacing, which means the distance in between two lines on the screen, is 0.5 inches.

SNAP MODE ON

1. Left-click the *SNAP MODE* button in the *Status Bar* to turn *ON* the *SNAP* option.

2. Move the cursor inside the graphics window, and move the cursor diagonally on the screen. Observe the movement of the cursor and watch the *coordinates display* at the bottom of the screen.

The *SNAP* option controls an invisible rectangular grid that restricts cursor movement to specified intervals. When *SNAP* mode is on, the screen cursor and all input coordinates are snapped to the nearest point on the grid. The default snap interval is 0.5 inches and aligned to the grid points on the screen.

Using the *ERASE* Command

One of the advantages of using a CAD system is the ability to remove entities without leaving any marks. We will erase two of the lines using the Erase command.
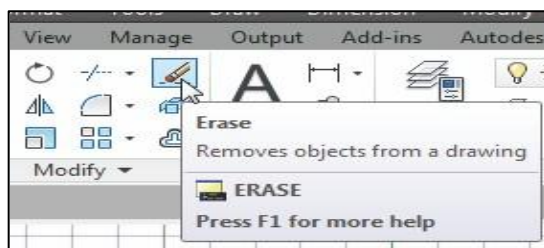
1. Pick **Erase** in the *Modify* toolbar. (The icon is a picture of an eraser at the end of a pencil.) The message "*Select objects*" is displayed in the command prompt area and AutoCAD awaits us to select the objects to erase.

2. Select any two lines on the screen; the selected lines are highlighted as shown in the figure below.

3. To **deselect** an object from the selection set, hold down the [**SHIFT**] key and select the object again.

4. **Right-mouse-click** once to accept the selections. The selected two lines are erased.


Repeat the Last Command
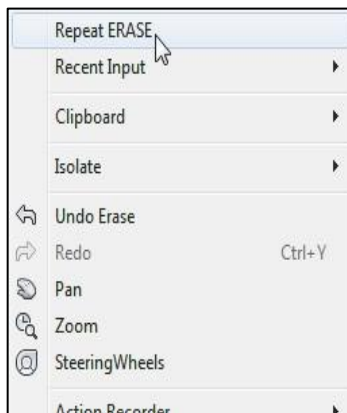
1. Inside the graphics window, click once with the **right- mouse-button** to bring up the popup option menu.

2. Pick **Repeat Erase**, with the left-mouse-button, in the popup menu to repeat the last command. Notice the other options available in the popup menu.

   AutoCAD 2016 offers many options to accomplish the same task. Throughout this text, we will emphasize the use of the **AutoCAD Heads-up Design**^{TM} interface, which means we focus on the screen, not on the keyboard.

3. Move the cursor to a location that is above and toward the left side of the entities on the screen. Click Left-mouse once to start a corner of a rubber-band window.

First corner

Second corner
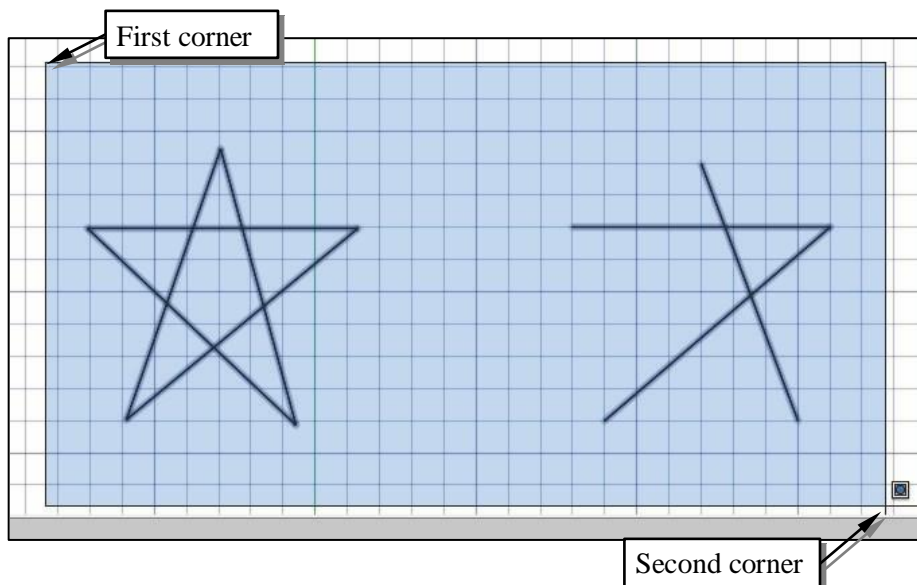
4. Move the cursor toward the right and below the entities, and then left-mouse-click to enclose all the entities inside the **selection window**. Notice all entities that are inside the window are selected. (Note the *enclosed window selection* direction is from **top left** to **bottom right**.)

5. Inside the graphics window, **right-mouse-click** once to proceed with erasing the selected entities.
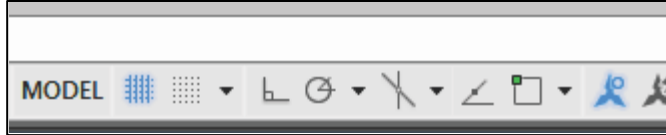
On your own, create a free-hand sketch of your choice using the **Line** command.
Experiment with using the different commands we have discussed so far. Reset the status buttons so that only the *GRID DISPLAY* option is turned *ON* as show
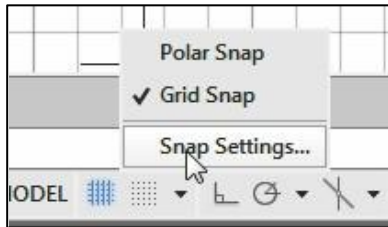
1. In the *Status Bar* area, **right-mouse-click** on *SnapMode* and choose **[Snap settings]**.

2. In the *Drafting Settings* dialog box, select the **Snap and Grid** tab if it is not the page on top.

GRID Style Setup

3. Change *Grid Style* to **Display dotted grid in 2D model Space** as shown in the below figure.
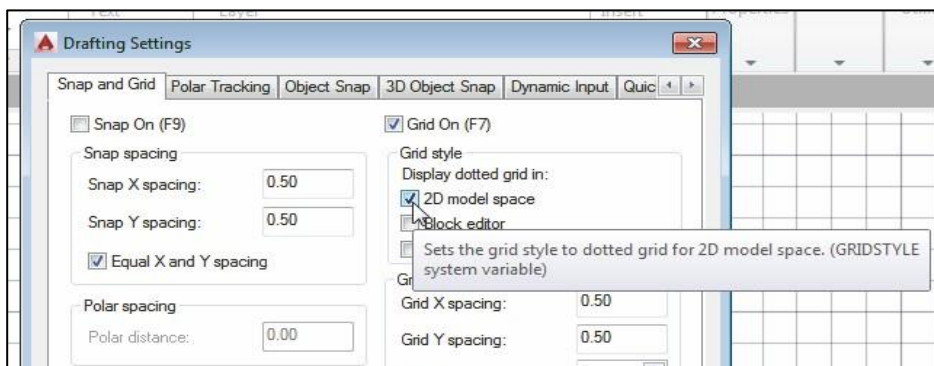
4. Pick **OK** to exit the *Drafting Settings* dialog box.

The GuidePlate

We will next create a mechanical design using the different coordinate entry methods.
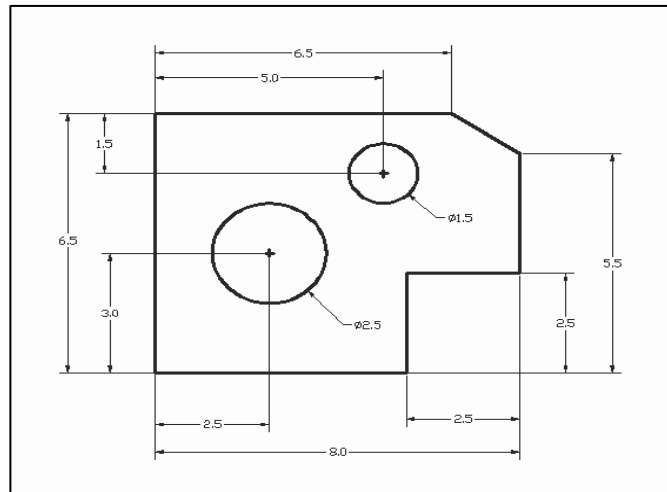
The rule for creating CAD designs and drawings is that they should be created at **full size** using real-world units. The CAD database contains all the definitions of the geometric entities and the design is considered as a virtual, full-sized object. Only when a printer or plotter transfers the CAD design to paper is the design scaled to fit on a sheet. The tedious task of determining a scale factor so that the design will fit on a sheet of paper is taken care of by the CAD system. This allows the designers and CAD operators to concentrate their attention on the more important issues – the design.
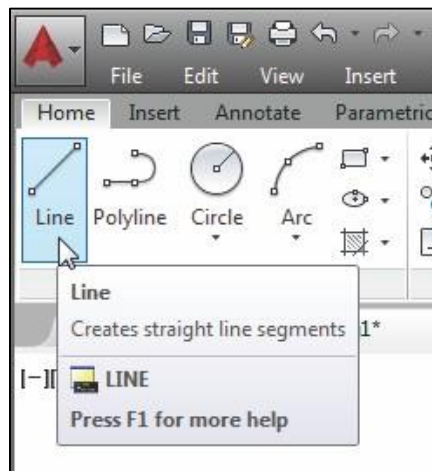
1. Select the **Line** command icon in the *Draw* toolbar. In the command prompt area, near the bottom of the AutoCAD graphics window, the message "*_line Specify first point:*" is displayed. AutoCAD expects us to identify the starting location of a straight line.

2. We will locate the starting point of our design at the origin of the *world coordinate system*.

Command: _line Specify first point: **0,0**
(Type **0,0** and press the [**ENTER**] key once.)

3. We will create a horizontal line by entering the absolute coordinates of the second point.
    Specify next point or [Undo]: **5.5,0** [**ENTER**]

4. We will create a horizontal line by entering the absolute coordinates of the second point.
    Specify next point or [Undo]: **5.5,0** [**ENTER**]

5. Move the cursor, which appears as a hand inside the graphics window, near the center of the drawing window, then push down the **left-mouse-button** and drag the display toward the right and top side until we can see the sketched line. (Notice the scroll bars can also be used to adjust viewing of the display.)

6. Press the **[Esc]** key to exit the *Pan-Realtime* command. Notice that AutoCAD goes back to the Line command.

7. We will create a vertical line by using the *relative rectangular coordinates entry method*, relative to the last point we specified:
*Specify next point or [Close/Undo]:* **@0,2.5** **[ENTER]**
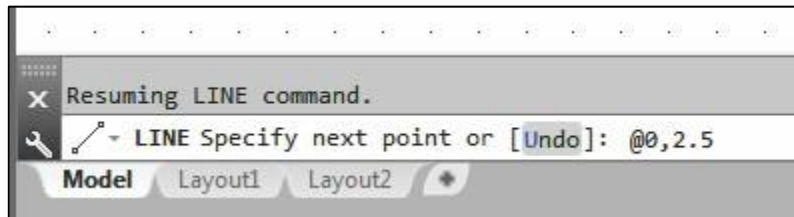
8. We can mix any of the entry methods in positioning the locations of the endpoints. Move the cursor to the *Status Bar* area, and turn **ON** the *SNAP MODE* option.
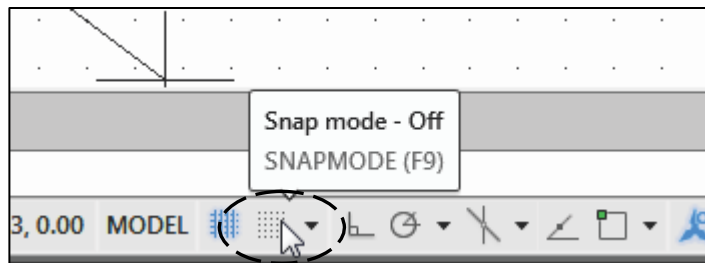

Note that the Line command is resumed as the settings are adjusted.

9. Left-click once on the coordinates display area to switch to a different coordinate display option. Note the coordinates display area has changed to show the length of the new line and its angle. Each click will change the display format of the cursor coordinates.
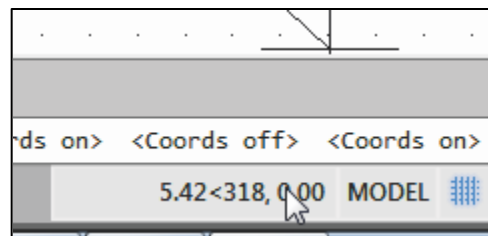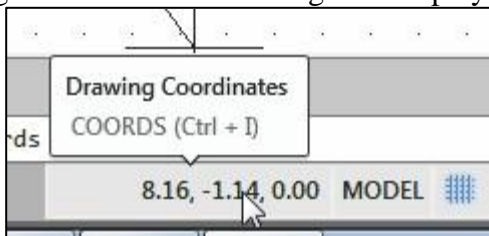
10. On your own, left-click on the coordinates display area to observe the switching of the coordinate display; set the display back to using the world coordinate system.

11. Create the next line by picking the location, world coordinates (*8,2.5*), on the screen.
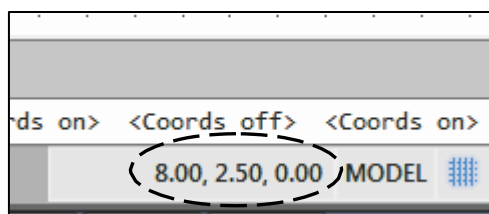
12. We will next use the *relative polar coordinates entry method*, relative to the last point we specified:

> *Specify next point or [Close/Undo]:* **@3<90** [**ENTER**]
> (Distance is **3** inches with an angle of **90** degrees.)
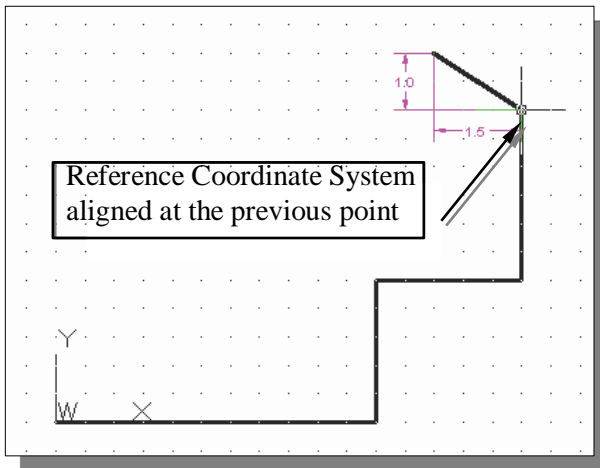
13. Using the *relative rectangular coordinates entry method* to create the next line, we can imagine a *reference coordinate system* aligned at the previous point. Coordinates are measured along the two reference axes.

Reference Coordinate System aligned at the previous point

> *Specify next point or [Close/Undo]:* **@-1.5,1** [**ENTER**]

(-**1.5** and **1** inches are measured relative to the reference point.)

14. Move the cursor directly to the left of the last point and use the *direct distance entry technique* by entering **6.5** [**ENTER**].
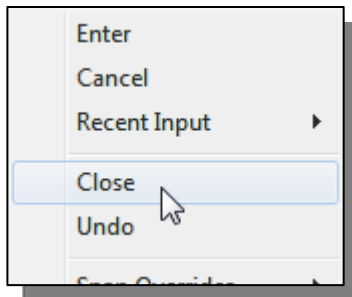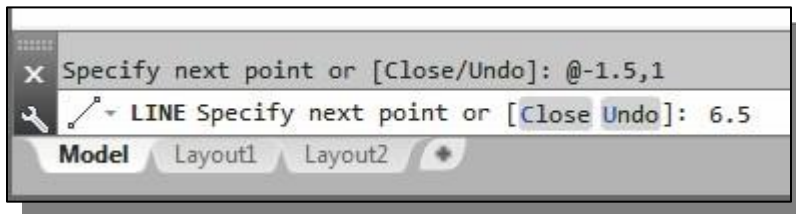
15. For the last segment of the sketch, we can use the **Close** option to connect back to the starting point. Inside the graphics window, **right-mouse-click** and a *popup menu* appears on the screen.

16. Select **Close** with the left-mouse-button to connect back to the starting point and end the Line command.
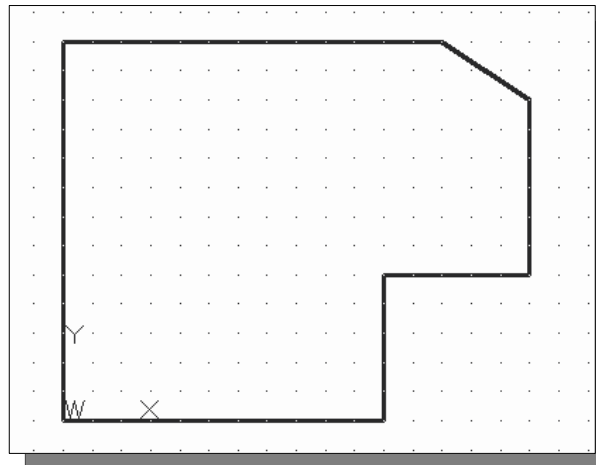
Creating *Circles*

The menus and toolbars in **AutoCAD 2016** are designed to allow the CAD operator to quickly activate the desired commands.

1. In the *Draw* toolbar, click on the little triangle below the circle icon. Note that the little triangle indicates additional options are available.

2. In the option list, select: **[Center, Diameter]**
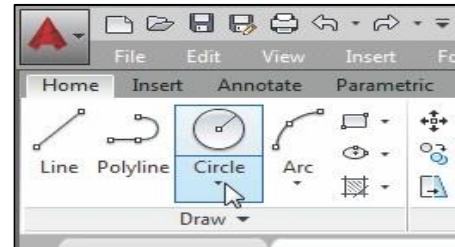
Notice the different options available under the circle submenu:

- **Center, Radius**: Draws a circle based on a center point and a radius.

- **Center, Diameter**: Draws a circle based on a center point and a diameter.

- **2 Points**: Draws a circle based on two endpoints of the diameter.

- **3 Points**: Draws a circle based on three points on the circumference.

- **TTR–Tangent, Tangent**, **Radius**: Draws a circle with a specified radius tangent to two objects.

- **T T T –Tangent, Tangent, Tangent**: Draws a circle tangent to three objects.

3. In the command prompt area, the message "*Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:*" is displayed. AutoCAD expects us to identify the location of a point or enter an option. We can use any of the four coordinate entry methods to identify the desired location. We will enter the **world coordinates (*2.5,3*)** as the center point for the first circle.

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **2.5,3** [**ENTER**]
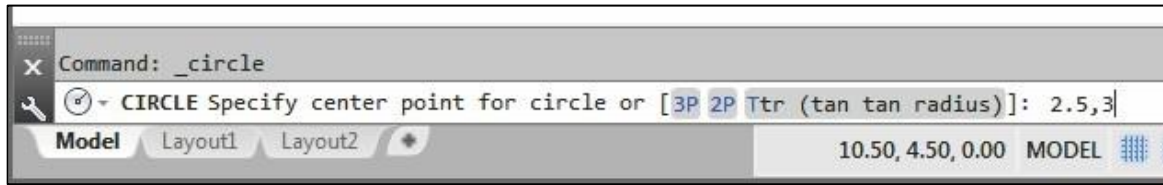
4. In the command prompt area, the message "*Specify diameter of circle:*" is displayed.
   Specify diameter of circle: **2.5** [**ENTER**]
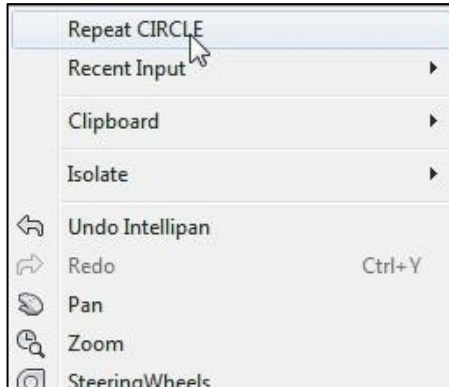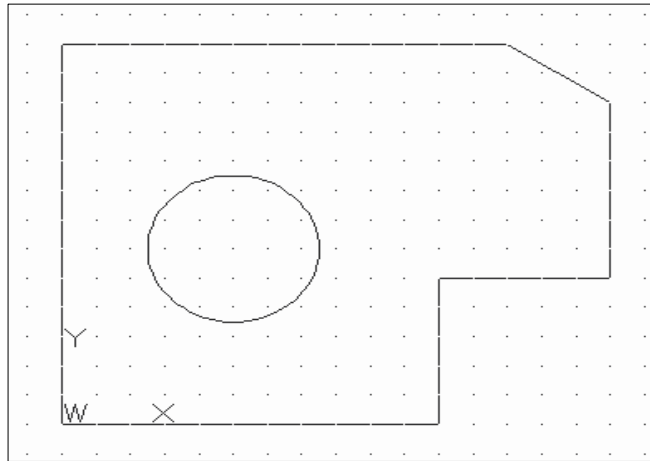
5. Inside the graphics window, right-mouse-click to bring up the popup option menu.

6. Pick **Repeat CIRCLE** with the left-mouse- button in the popup menu to repeat the last command.

7. Using the *relative rectangular coordinates entry method*, relative to the center-point coordinates of the first circle, we specify the location as (**2.5,2**).

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **@2.5,2** [**ENTER**]

8. In the command prompt area, the message "*Specify Radius of circle: <2.50>*" is displayed. The default option for the Circle command in AutoCAD is to specify the *radius* and the last radius used is also displayed in brackets.
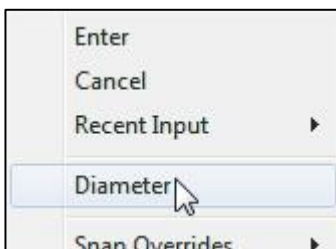
9. Inside the graphics window, **right-mouse-click** to bring up the popup option menu and select **Diameter** as shown.

10. In the command prompt area, enter **1.5** as the diameter.

17

Specify Diameter of circle<2.50>: **1.5** [**ENTER**]
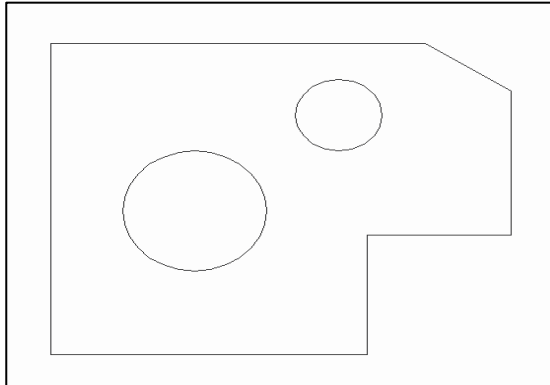
Saving the CAD Design

1. In the *Application Menu*, select:

    [Application] [Save]

    Note the command can also be activated with the quick-key combination of [**Ctrl**]+[**S**].

2. In the *Save Drawing As* dialog box, select the folder in which you want to store the CAD file and enter **GuidePlate** in the *File name* box.
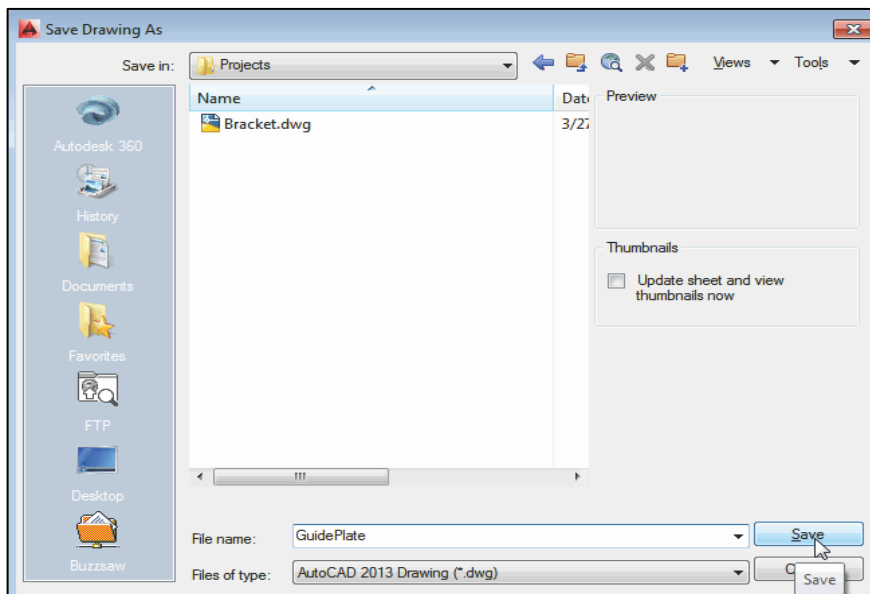
3. Click **Save** in the *Save Drawing As* dialog box to accept the selections and save the file. Note the default file type is DWG, which is the standard AutoCAD drawing format.

Close the Current Drawing

Several options are available to close the current drawing:
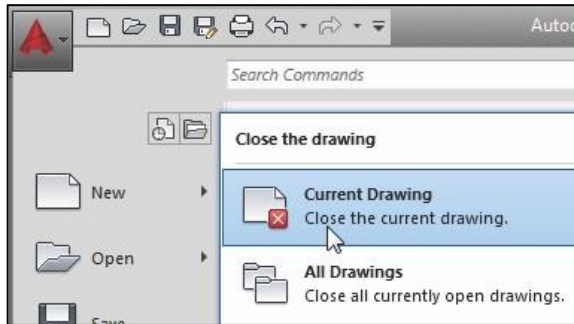
Select **[Close] [Current Drawing]** in the *Application Menu Bar* as shown.

Enter **Close** at the command prompt.
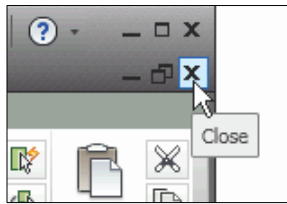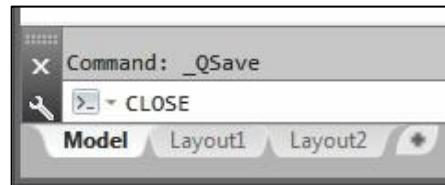
The third option is to click on the **[Close]** icon located at the upper-right-hand corner of the drawing window.

Drawing Units Setup

Every object we construct in a CAD system is measured in **units**. We should determine the system of units within the CAD system before creating the first geometric entities.
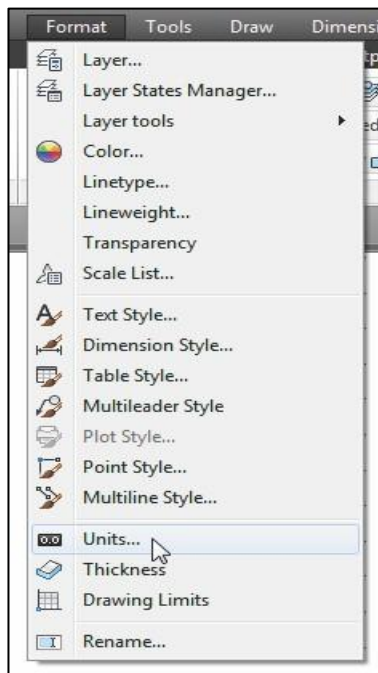
1. In the *Menu Bar* select:
   [Format] [Units]

• The AutoCAD *Menu Bar* contains multiple pull-down menus where all of the AutoCAD commands can be accessed. Note that many of the menu items listed in the pull-down menus can also be accessed through the *Quick Access* toolbar and/or *Ribbon* panels.

2. Click on the *Length Type* option to display the different types of length units available. Confirm the *Length Type* is set to **Decimal**.
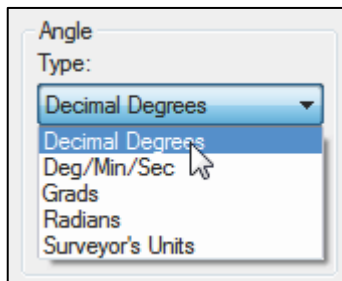
3. On your own, examine the other settings that are available.

4. In the *Drawing Units* dialog box, set the *Length Type* to **Decimal**. This will set the measurement to the default *English* units, inches.

5. Set the *Precision* to **two digits** after the decimal point as shown in the above figure.

6. Pick **OK** to exit the *Drawing Units* dialog box.
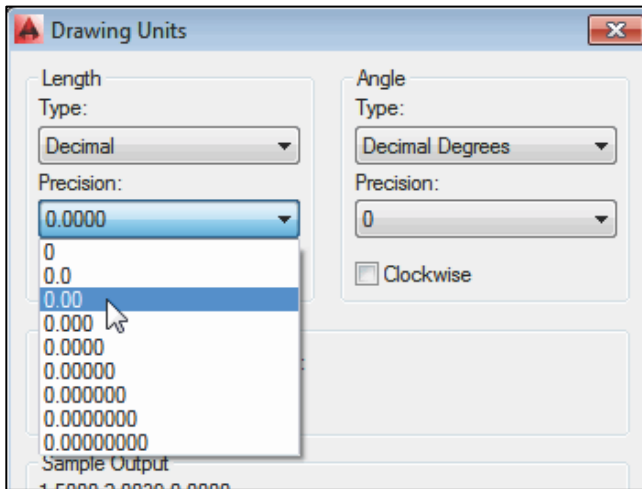
Drawing Area Setup

We can set up the **Drawing Limits** by entering a command in the command prompt area. Setting the Drawing Limits controls the extents of the display of the *grid*. It also serves as a visual reference that marks the working area. It can also be used to prevent construction outside the grid limits and as a plot option that defines an area to be plotted/printed. Note that this setting does not limit the region for geometry construction.

1. In the *Menu Bar* select:
   [Format] [Drawing Limits]

2. In the command prompt area, the message "*Reset Model Space Limits: Specify lower left corner or [On/Off] <0.00,0.00>:*" is displayed. Press the **ENTER** key once to accept the default coordinates <**0.00,0.00**>.

3. In the command prompt area, the message "*Specify upper right corner <12.00,9.00>:*" is displayed. Press the **ENTER** key again to accept the default coordinates <**12.00,9.00**>.

4. On your own, move the graphics cursor near the upper-right corner inside the drawing area and note that the drawing area is unchanged. (The Drawing Limits command is used to set the drawing area, but the display will not be adjusted until a display command is used.)

5. Inside the *Menu Bar* area select:
    [View] [Zoom] [All]

    The **Zoom All** command will adjust the display so that all objects in the drawing are displayed to be as large as possible. If no objects are constructed, the Drawing Limits are used to adjust the current viewport.

6. Move the graphics cursor near the upper-right corner inside the drawing area, and note that the display area is updated.

7. In the *Status Bar* area, **right-mouse-click** on *SnapMode* and choose **[Snap Settings]**.

8. In the *Drafting Settings dialog box*, switch **on** the **Snap** and **Grid** options as shown.
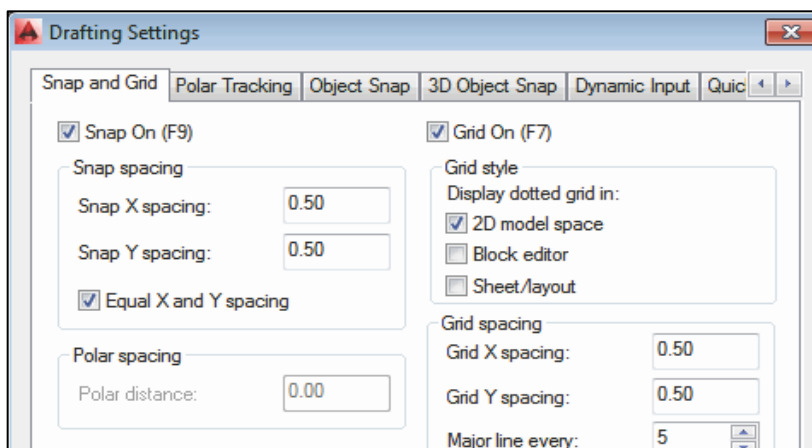
On your own, exit the *Drafting Settings dialog box* and reset the status buttons so that only *GRID DISPLAY* and *SNAP MODE* are turned *ON* as shown.
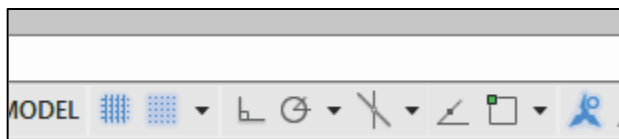
Using the *ERASE* Command

The vertical line on the right was created as a construction line to aid the construction of the rest of the lines for the design. We will use the Erase command to remove it.
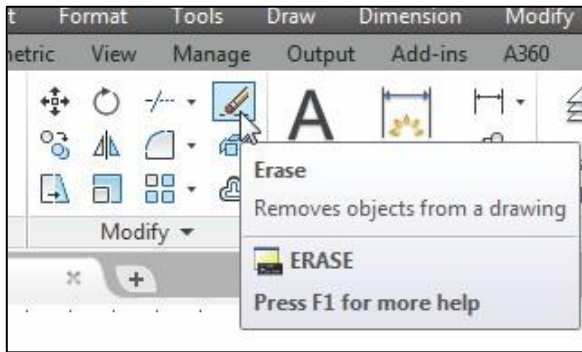
1. Pick **Erase** in the *Modify* toolbar. The message "*Select objects*" is displayed in the command prompt area and AutoCAD awaits us to select the objects to erase.
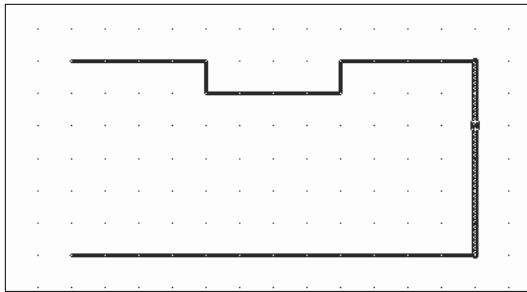
2. Select the vertical line as shown.

3. Click once with the **right-mouse-button** to accept the selection and delete the line.

Using the Arc Command
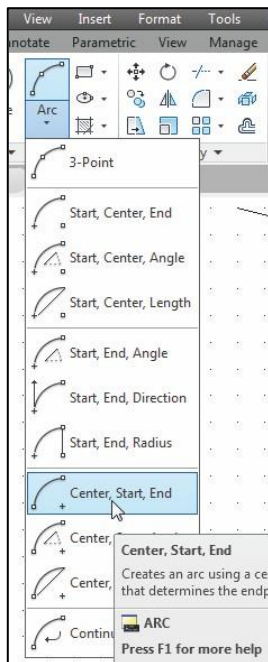
1. Click the down-arrow icon of the **Arc** command in the *Draw* toolbar to display the different Arc construction options.

   AutoCAD provides eleven different ways to create arcs. Note that the different options are used based on the geometry conditions of the design. The more commonly used options are the **3-Points** option and the **Center-Start-End** option.

2. Select the **Center-Start-End** option as shown. This option requires the selection of the center point, start point and end point location, in that order, of the arc.

23
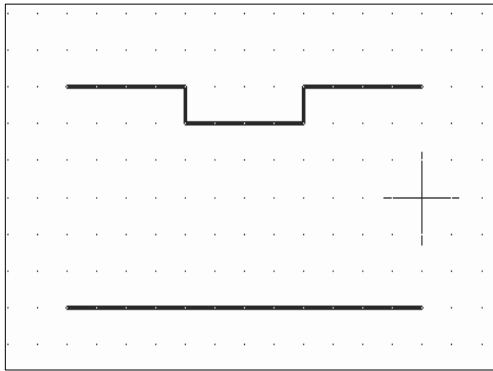
3. Move the cursor to the middle of the two horizontal lines and align the cursor to the two endpoints as shown. Click once with the **right-mouse-button** to select the location as the center point of the new arc.
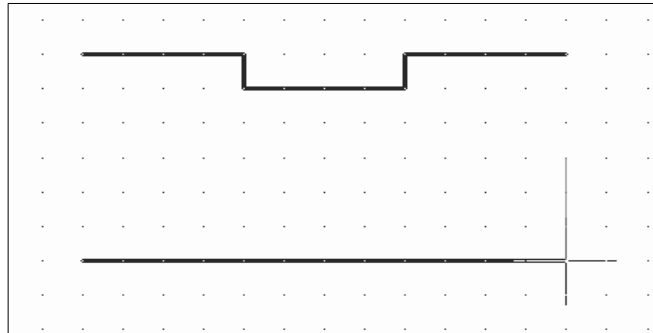
4. Move the cursor downward and select the right endpoint of the bottom horizontal line as the start point of the arc.
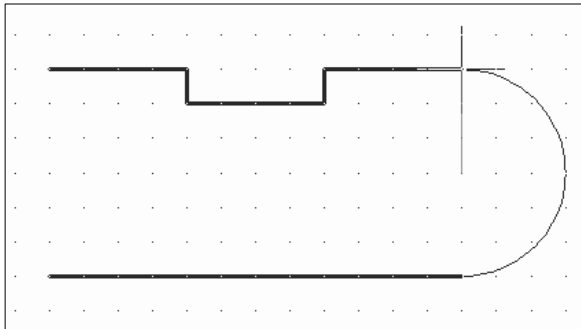
5. Move the cursor to the right endpoint of the top horizontal line as shown. Pick this point as the endpoint of the new arc.

6. On your own, repeat the above steps and create the other arc as shown. Note that in most CAD packages, positive angles are defined as going counterclockwise; therefore, the starting point of the second arc should be at the endpoint on top.
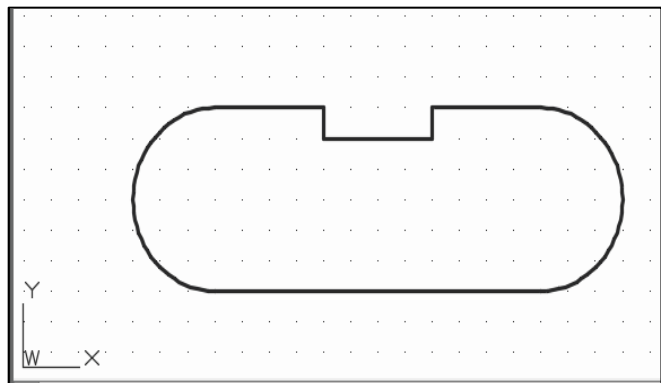
Review Questions:

1. What are the advantages and disadvantages of using CAD systems to create engineering drawings?

2. What is the default AutoCAD filename extension?

3. How do the *GRID* and *SNAP* options assist us in sketching?

4. List and describe the different **coordinate entry methods** available in AutoCAD.

5. When using the Line command, which option allows us to quickly create a line- segment connecting back to the starting point?

6. List and describe the two types of coordinate systems commonly used for planar geometry.

7. Which key do you use to quickly cancel a command?

8. When you use the Pan command, do the coordinates of objects get changed?

3. 2 - D Figures Using ACAD

(All dimensions are in inches.)

1. Angle Spacer

2. Channel Plate

94 Dia

54 Dia

74

6 holes 12 Dia

100 R

10R

12R

9R

23 R

36

30°   30°   Quadrant

22   12

6 holes 14 Dia

57

104

78

58

20R

22 Dia

2 holes 18 Dia   36 Dia

90

R8
Ø62
R4
R20
R18
65
R100
R15
R10
110

40R
40
94R
130
40R
246
78R
26R
36R
24
6R
135
60
24
52
100
168
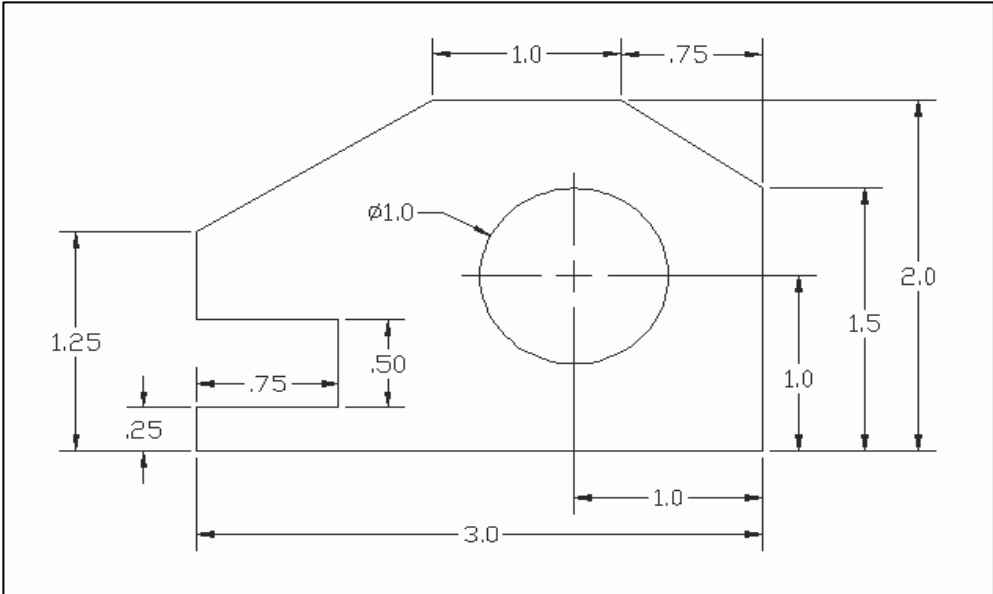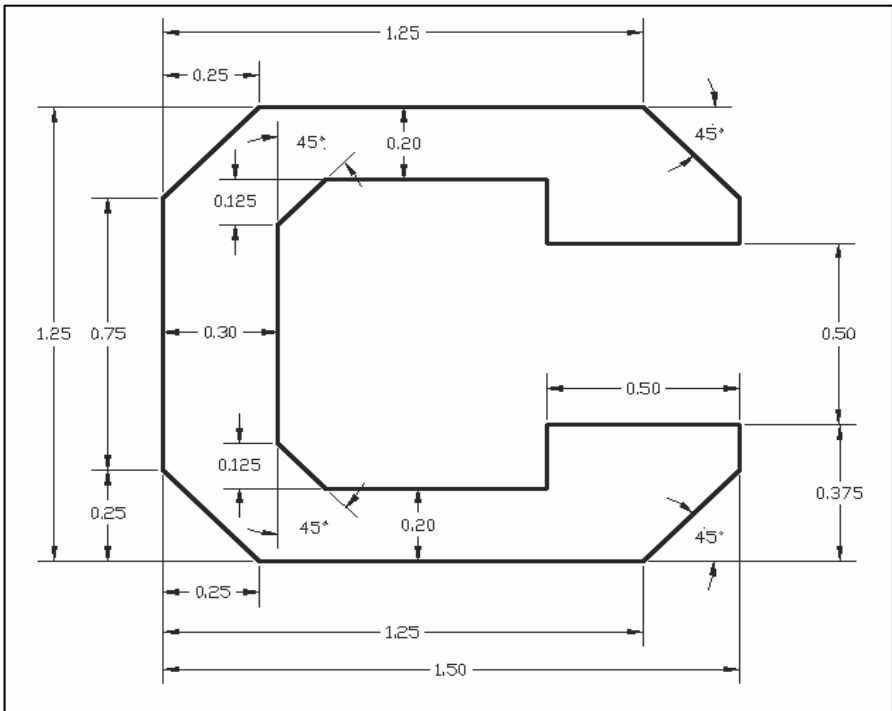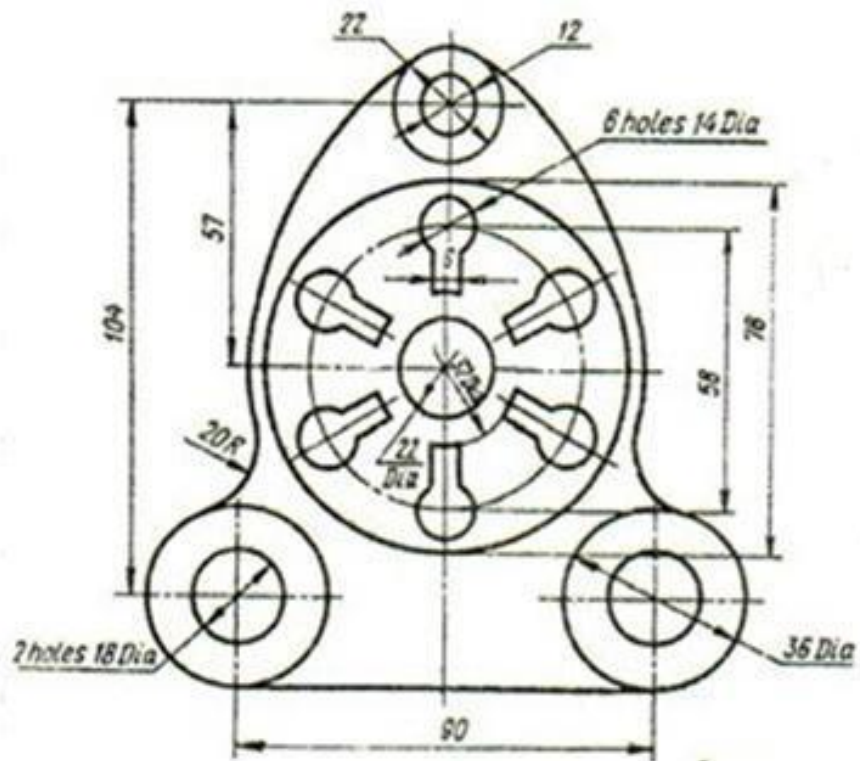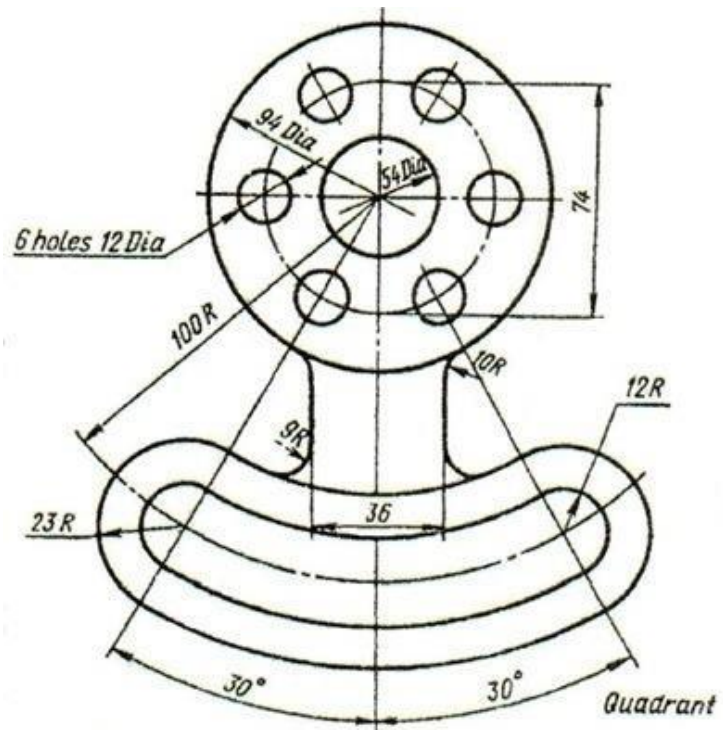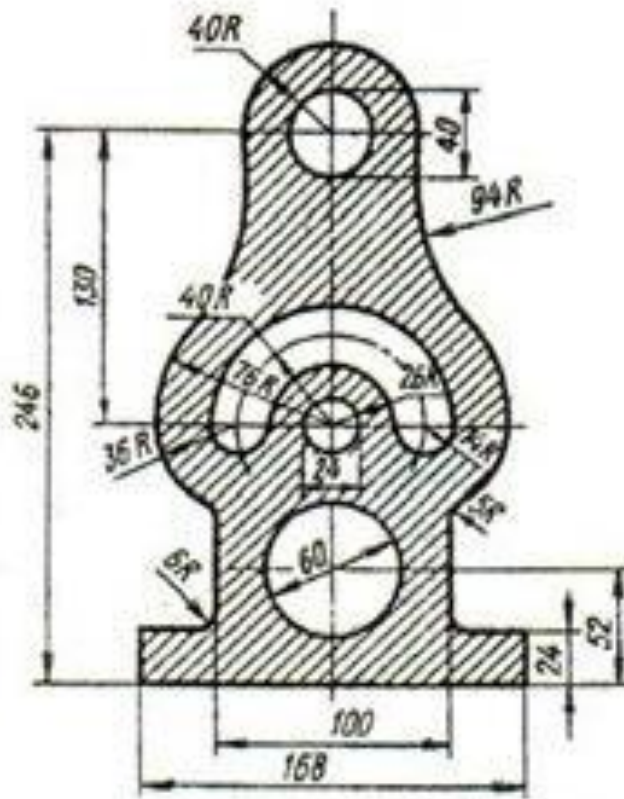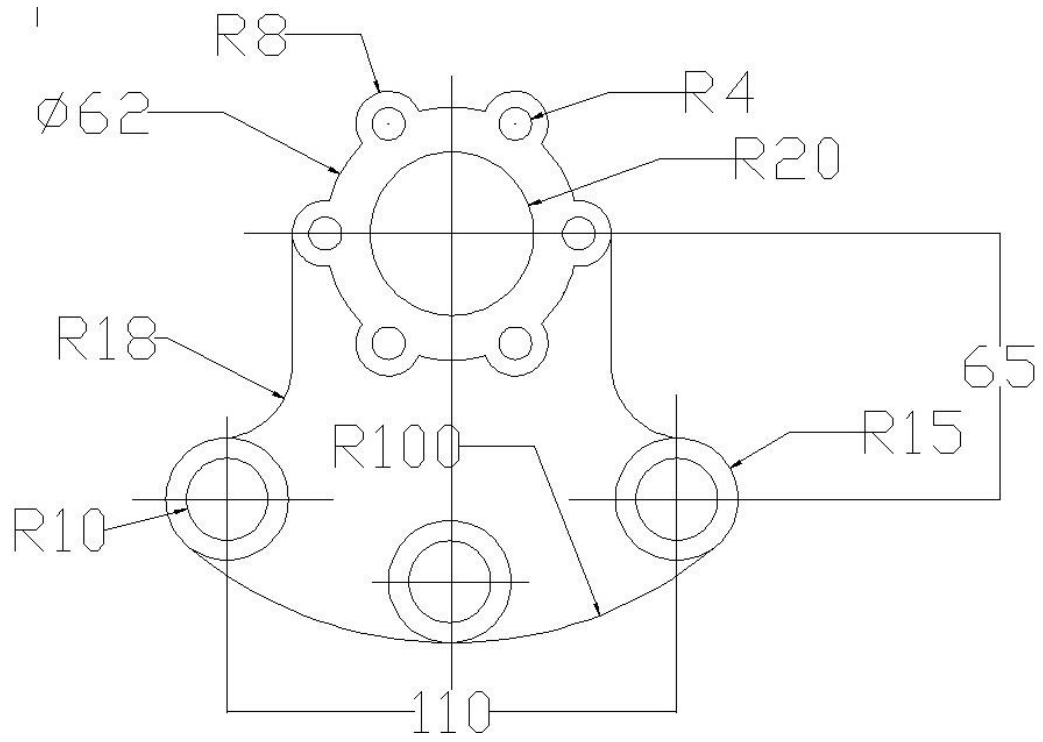
Isometric Drawings Using ACAD 2016

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
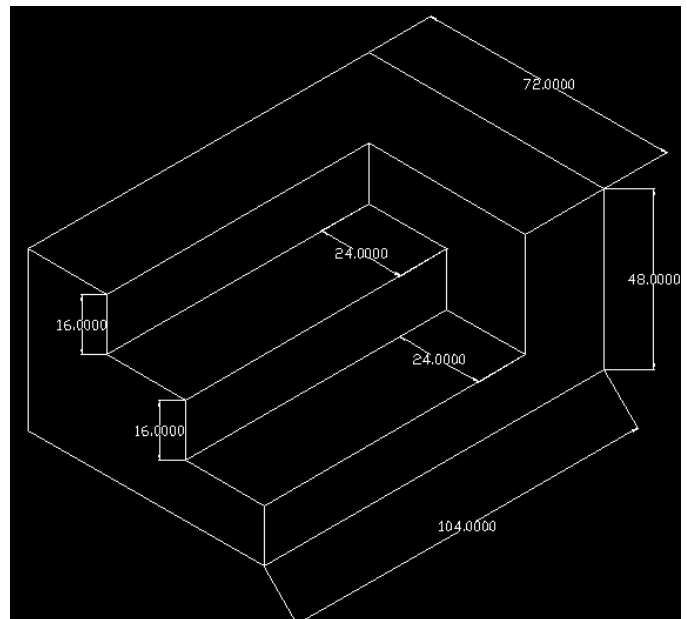
FIGURE 2
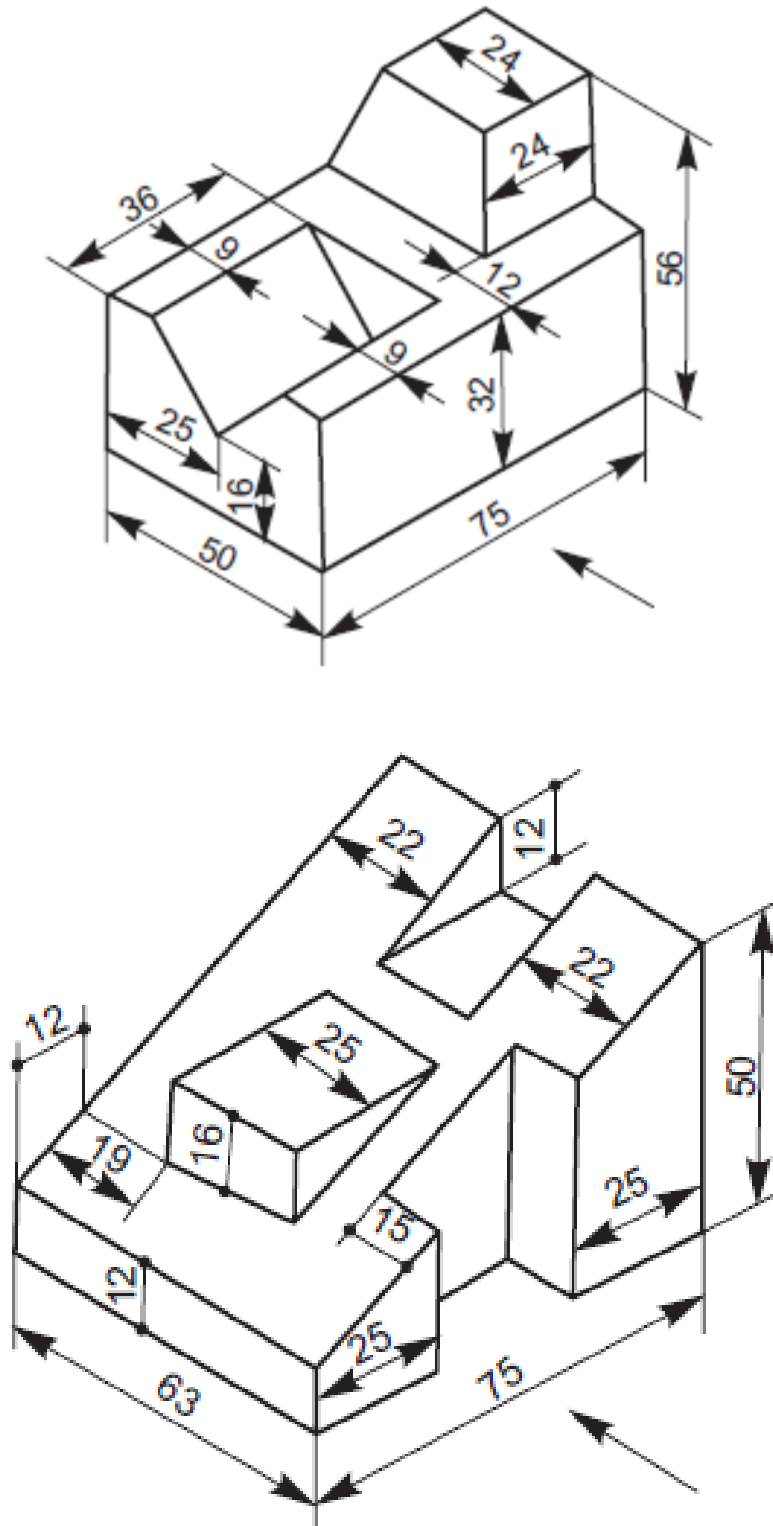
INTRODUCTION TO CREO 3.0

Creo Parametric 3.0 Interface
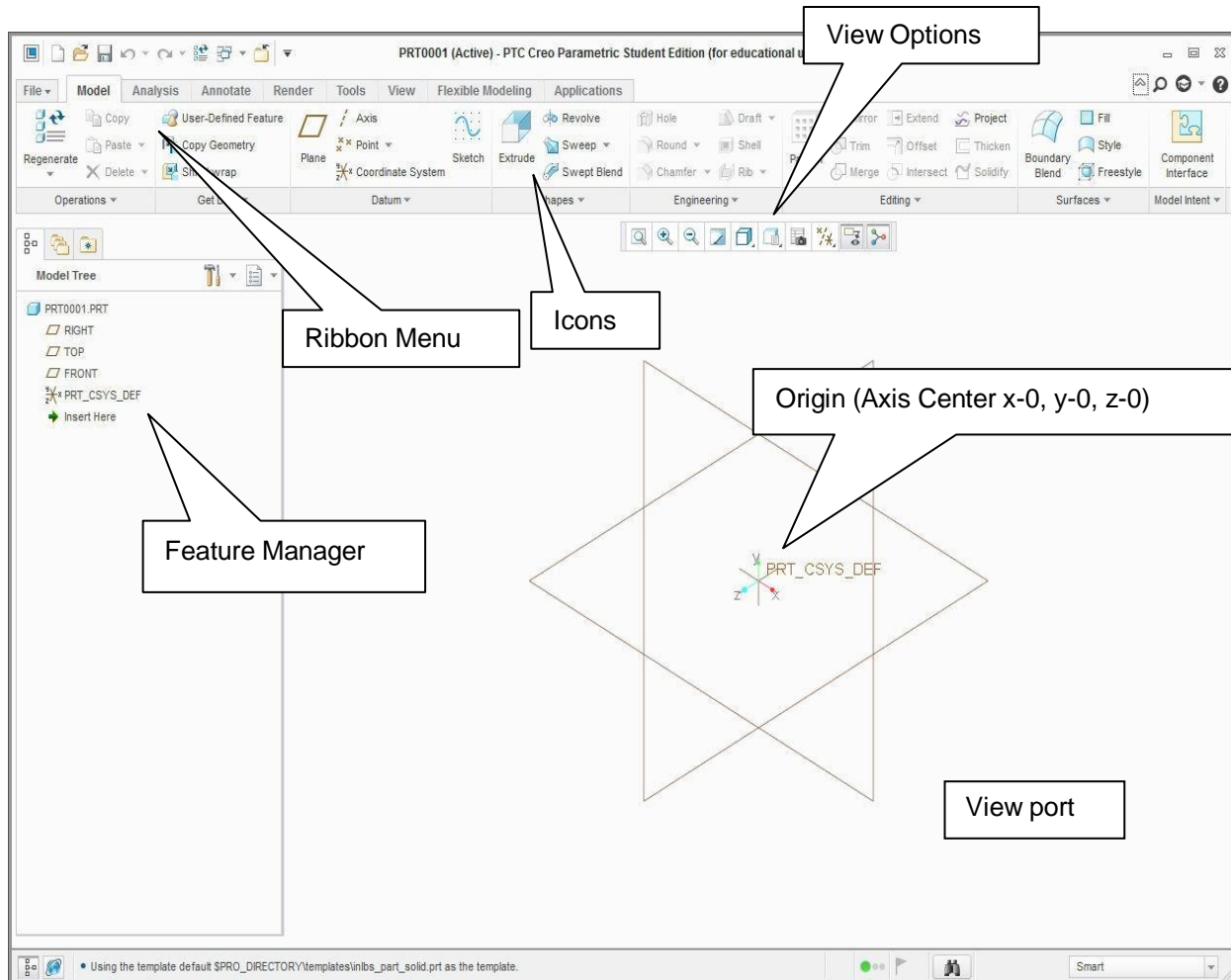
Mouse Buttons

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.**
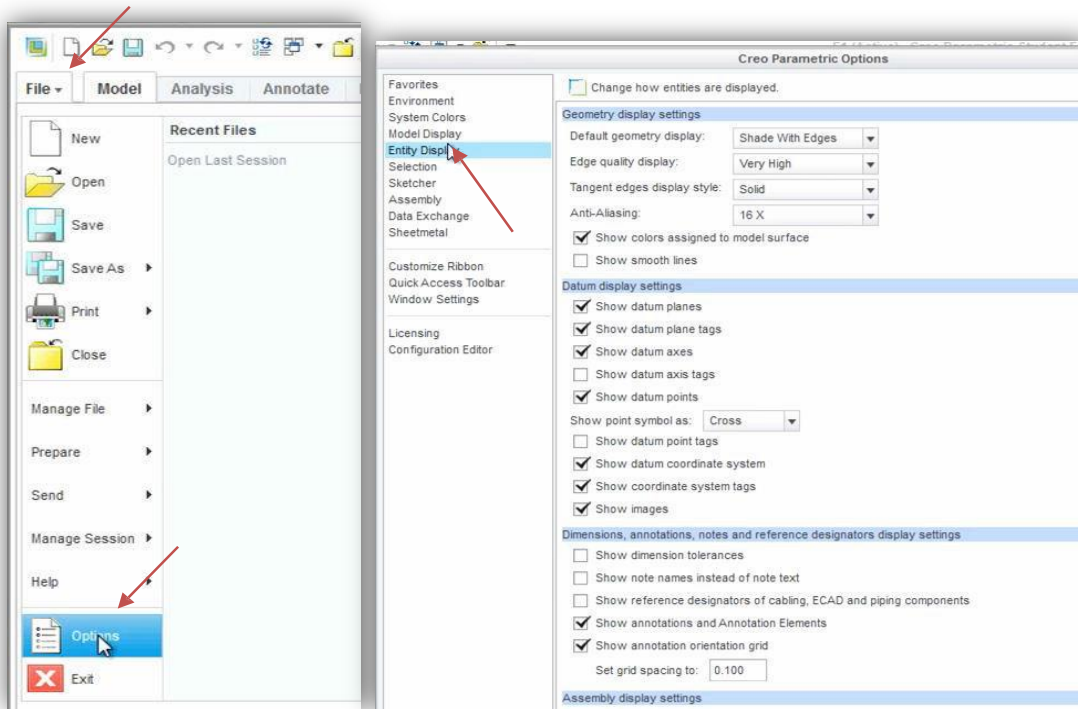
Model Properties

View options

Zoom In

Repaint

Objects

Spin Center

Refit

Zoom Out

Shaded

Standard Orientatio
Default Orientation
BACK
BOTTOM
FRONT
LEFT
RIGHT
TOP

Create Cross-sections
for a drawing

View Manager

Simp Rep | Xsec | Orient | All

New | Edit ▼ | Display ▼

Names

➔ Master Rep

Symbolic Rep

Geometry Rep

Graphics Rep

Properties >> | Close

Sketching

Rectangle

Spline

Constraint/
Relations

Trim

Apply/Finish

Circle

Fillet

Point

Mirror

Dim

Line

Arcs
• 3 Point
• Tangent
• Concentric

Text

Convert
Entities &
Offset

Cancel

Start Sketch

Datum Plane

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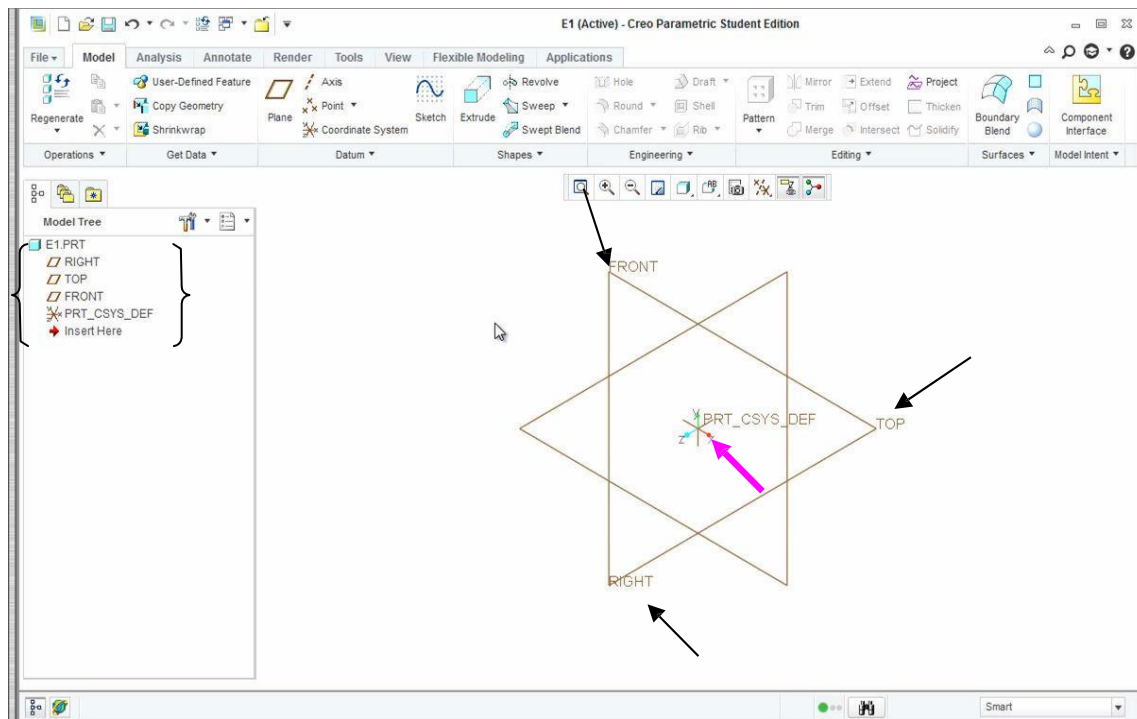
To start a sketch Pre-select the plane or face you desire to sketch on and then select the Sketch Icon.
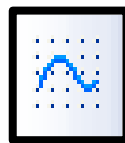
Sketch Options
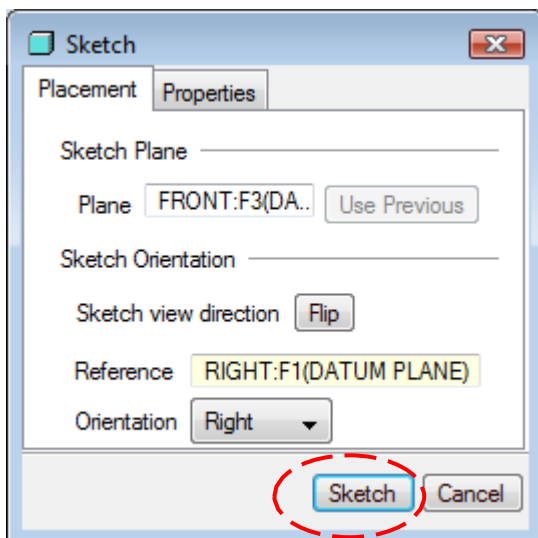
Controlling your geometry

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.

Here is an example of adding a relationship between two geometric entities.
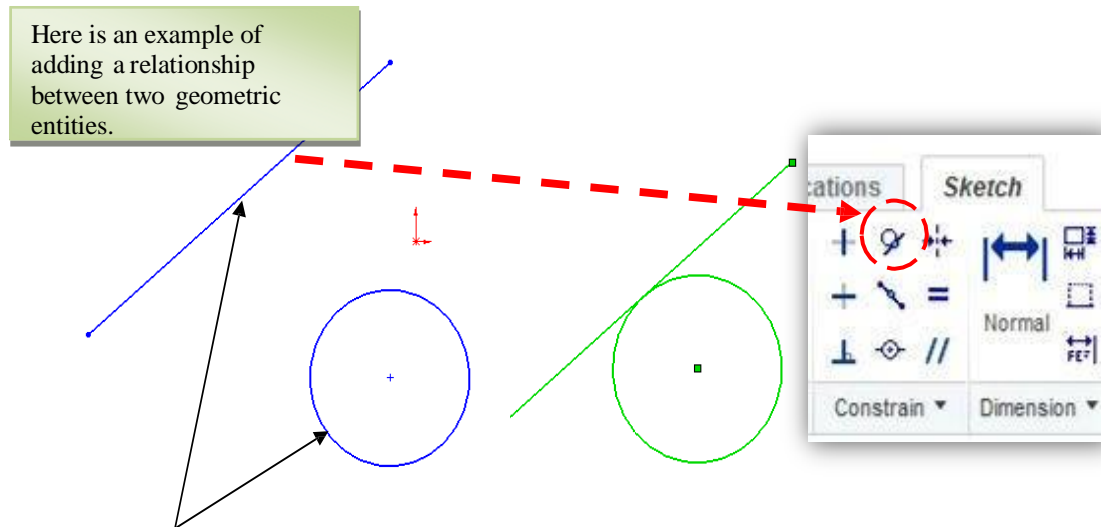
Cautious sketching can save time

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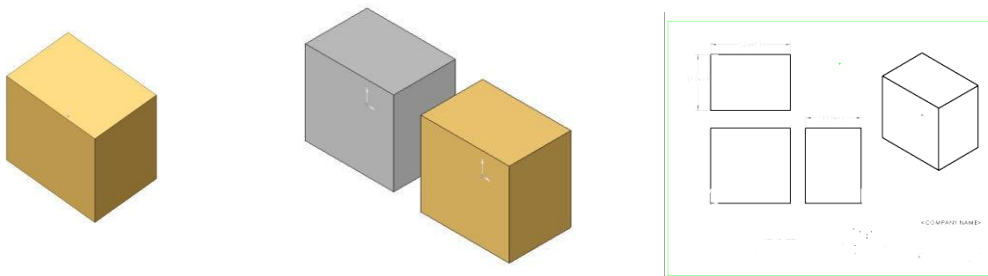
Switching between documents (Activating a document)
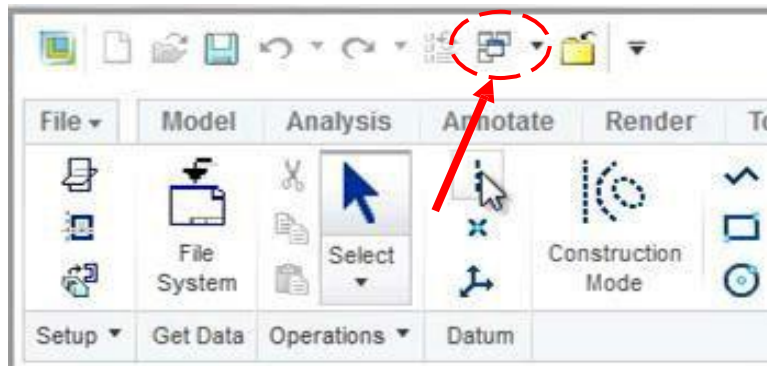
Select the Window pull-down menu and you will see the available documents. Click on the document you wish to work on from the list to update it.

Sketch Constraints (Relations)
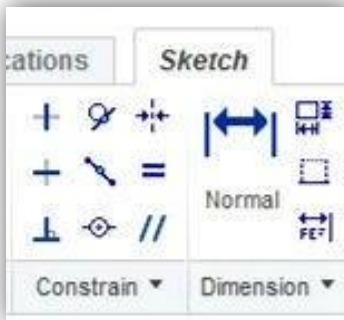
Constraint	Geometric entities to select	Resulting Constraint
Horizontal or Vertical	One or more lines or two or more points.	The lines become horizontal or vertical (as defined by the current sketch space). Points are aligned horizontally or vertically.
Collinear	Two or more lines.	The items lie on the 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...
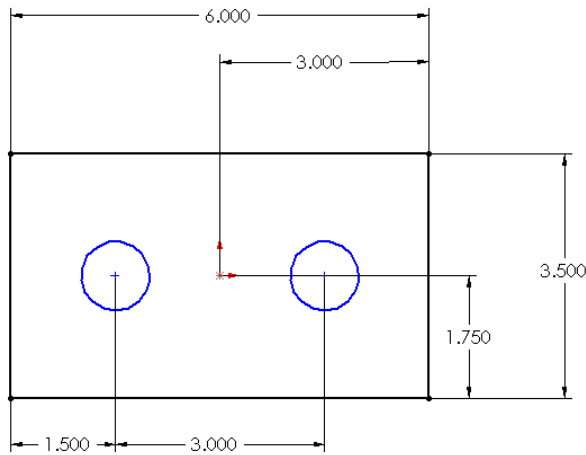
Strong versus Weak Dimensions - Double click and change to make them Strong!

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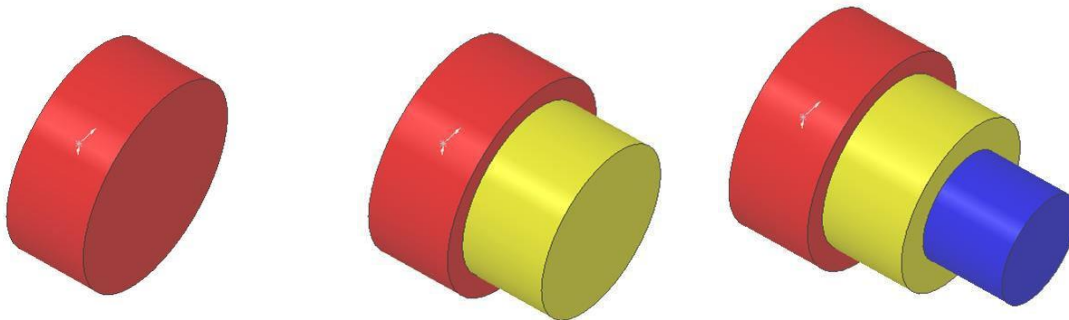
Dimensioning this way will enable the length of the bracket to change but the holes will always remain positioned to the left side.

Sketch

Normal

Constrain ▼   Dimension ▼
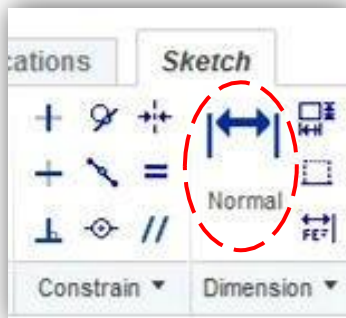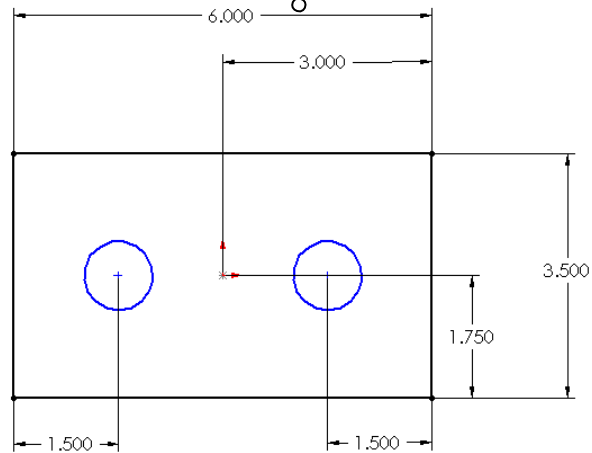
Solid Modeling Basics

1.   *Layer Cake method*

Extruded Boss/Base (Creates/Adds material)
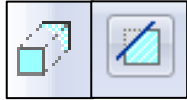
Extruded Cut (Removes material)

Ingredients:
- Profile

2.   *Revolve method*
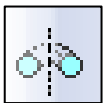
Revolve Boss/Base (Creates/Adds material)
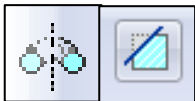
Revolve Cut (Removes material)
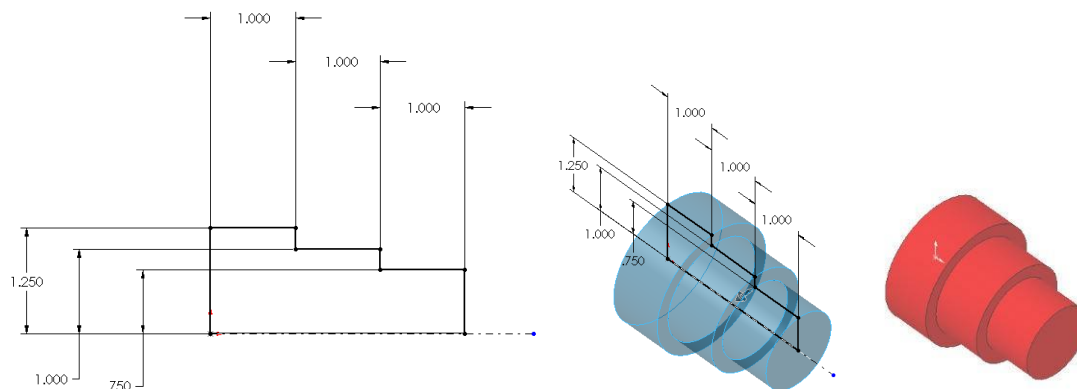
EXERCISE 1

Introduction to basic part modeling

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

Objective: Create a solid

Blind: 0.39370078in

This will fail to extrude.

This will create a hole.

file

4. Select the Rectangle tool.

7. Click to finish sketch.

3. Select the Sketch icon.

6. Double Click to edit the dimensions.

2. Select the □F□oŶt□

5. Click and drag across.

Origin Start Here

Model Tree
E1.PRT
  RIGHT
  TOP
  FRONT
  PRT_CSYS_DEF
  Insert Here
  Sketch 1

FRONT
H
V   V
5.00
PRT_CSYS_DEF   TOP
RIGHT
3.00

When a constraint is active, right click to cycle through lock/disable/enable the constraint. Use Tab key

1 selected   All

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

10. To sketch the next feature *select* the front face of the model and then select the "ketÐh iÐoŶ.

Toggle views using the **View Orientation**

Standard Orientatio
Default Orientation
BACK
BOTTOM
FRONT
LEFT
RIGHT
TOP
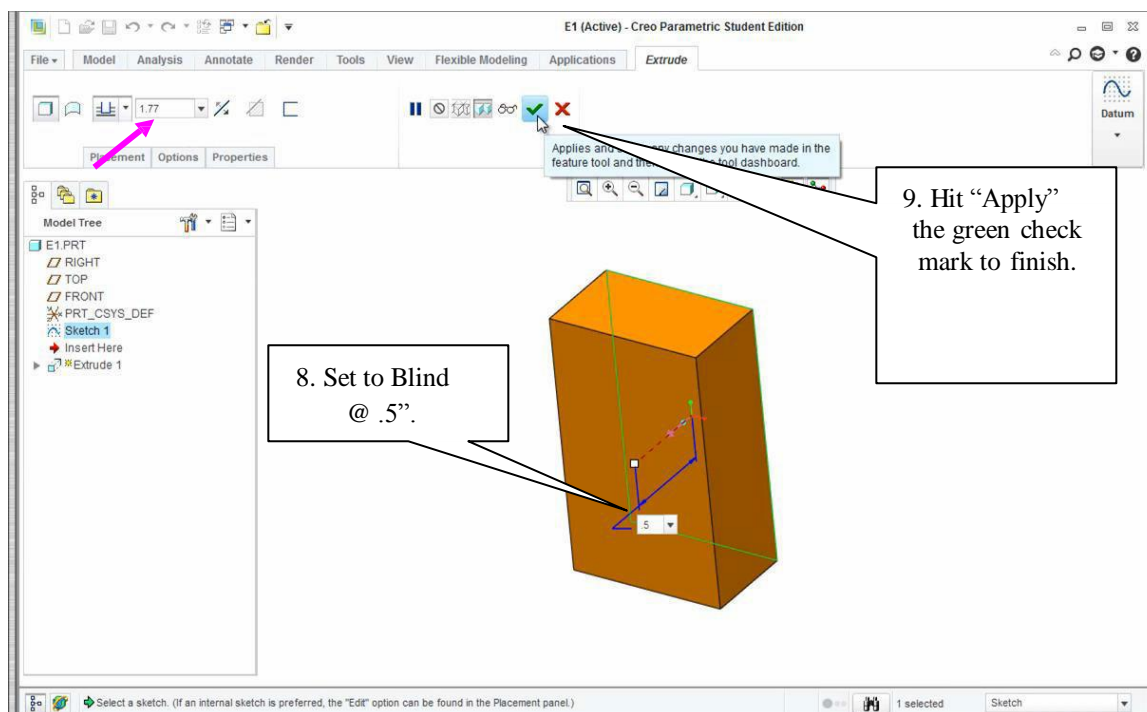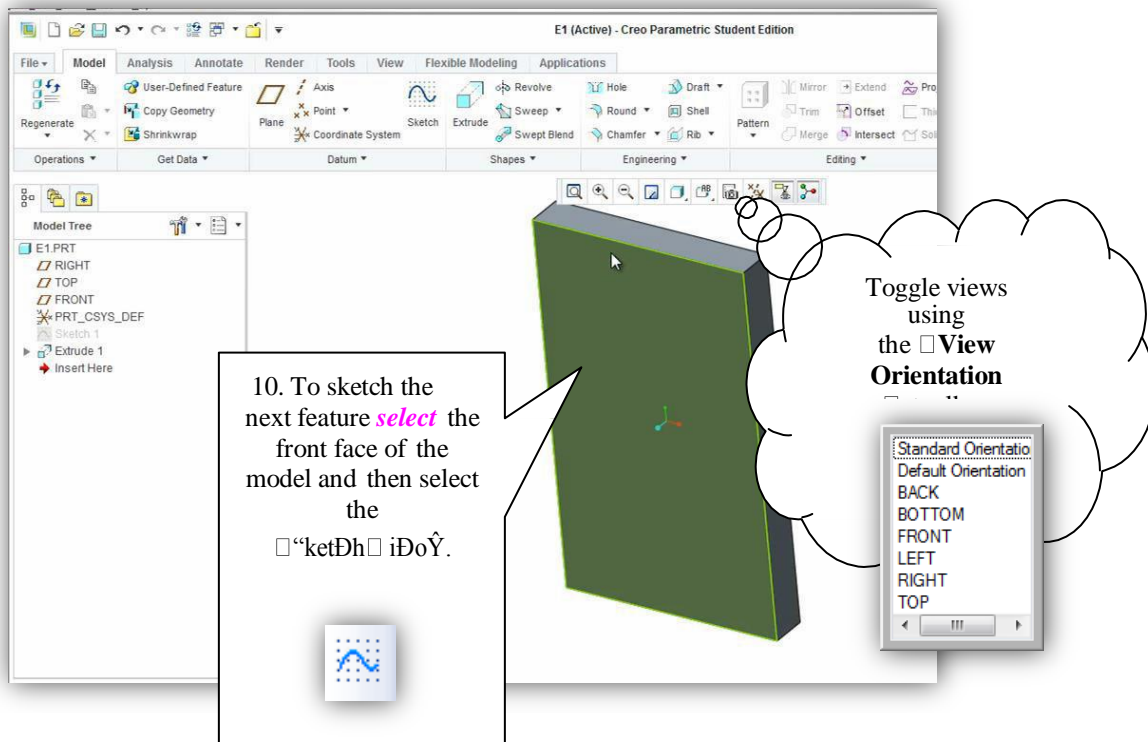
Adding a constraint – Ctrl Select both left edges of sketch and solid. Select Coincident

Edge:F6(EXTRUDE_1)

1.50

2.73

Constrain ▼
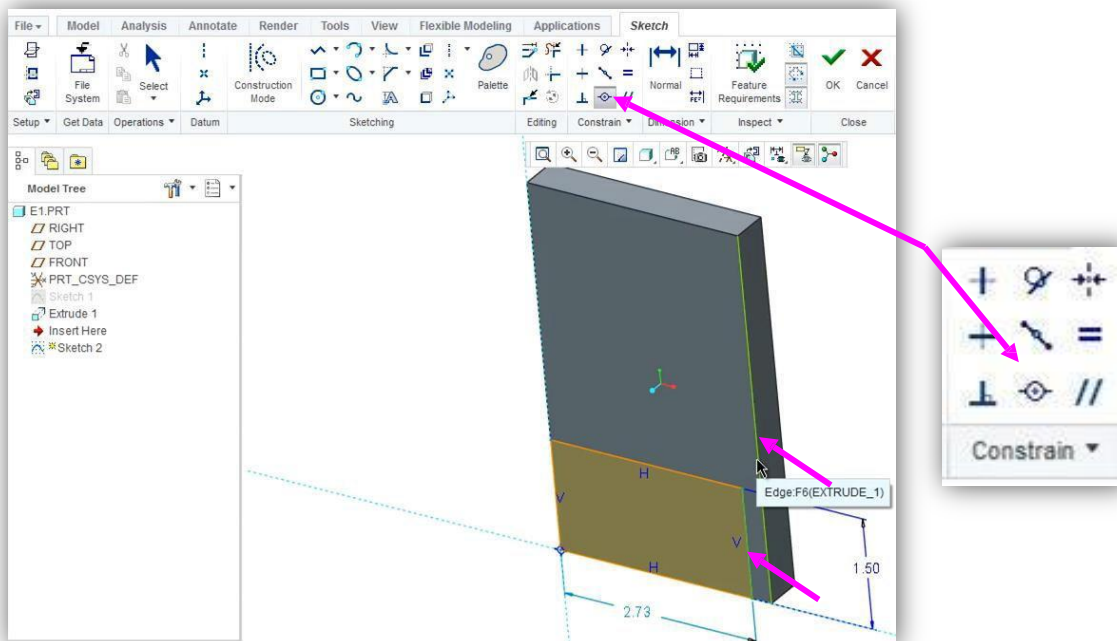
Extrude
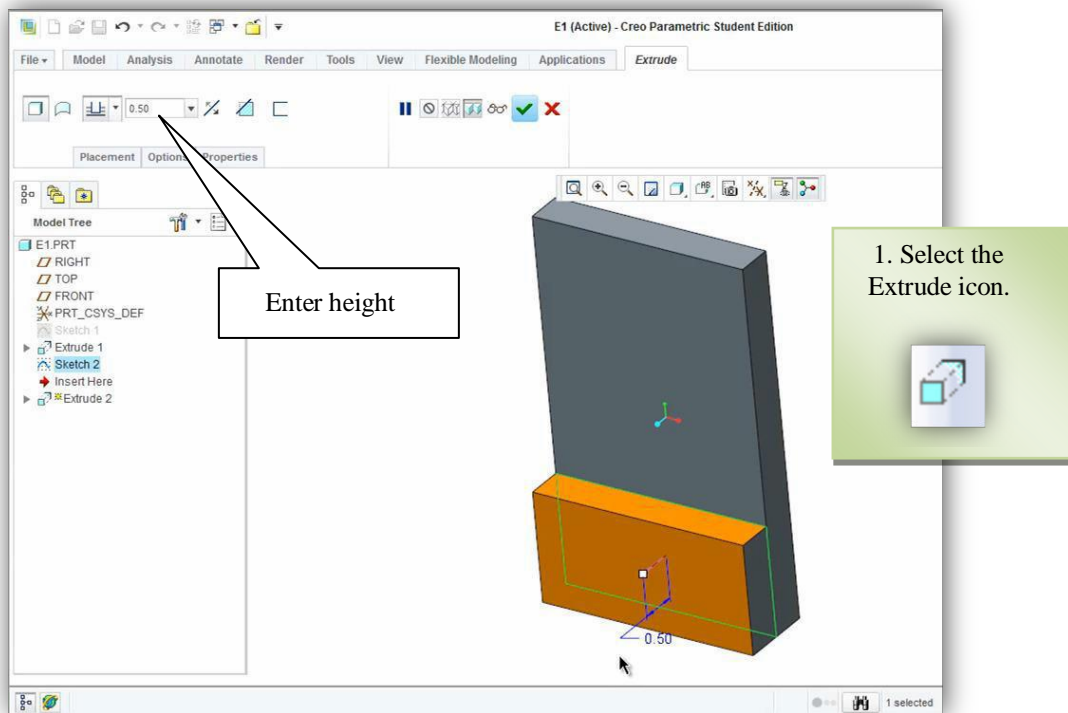
Select the face, select sketch icon and draw a circle on the face. Dimension, Hit ☐Ok☐
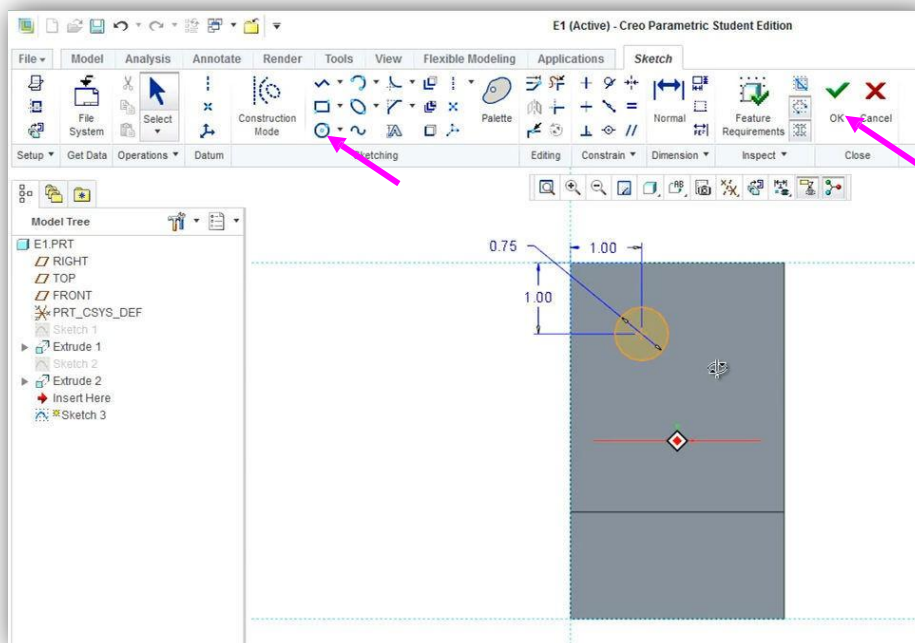
Select the face, select sketch icon and draw a circle on the face. Dimension, Hit ☐Ok☐

Go to file save

Update or Regenerate model changes using the ☐**Regenerate**☐ button. Although it is unnecessary in this instance it is still a very important tool. It will update only dimensional or feature changes made to the model.
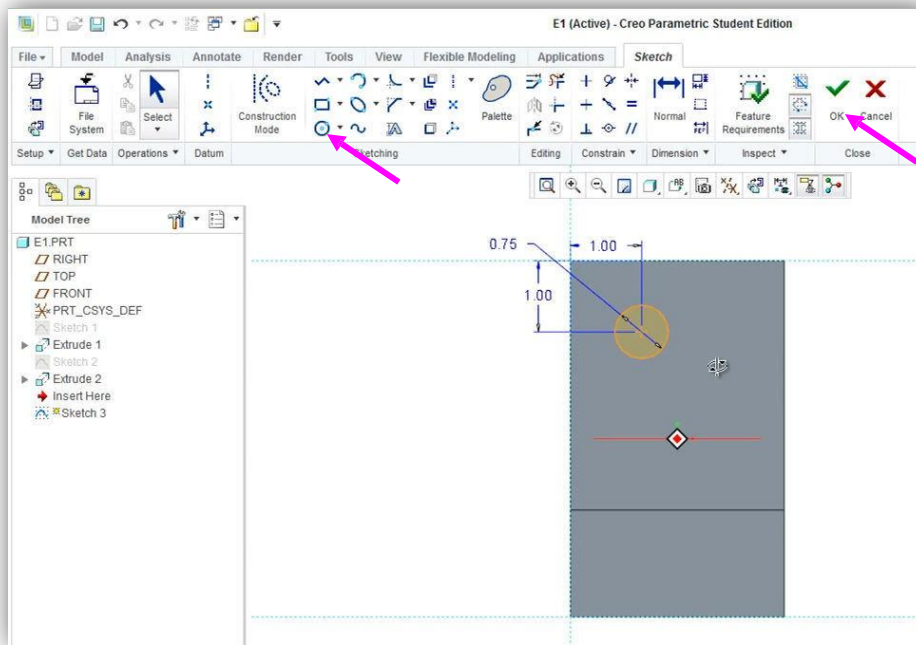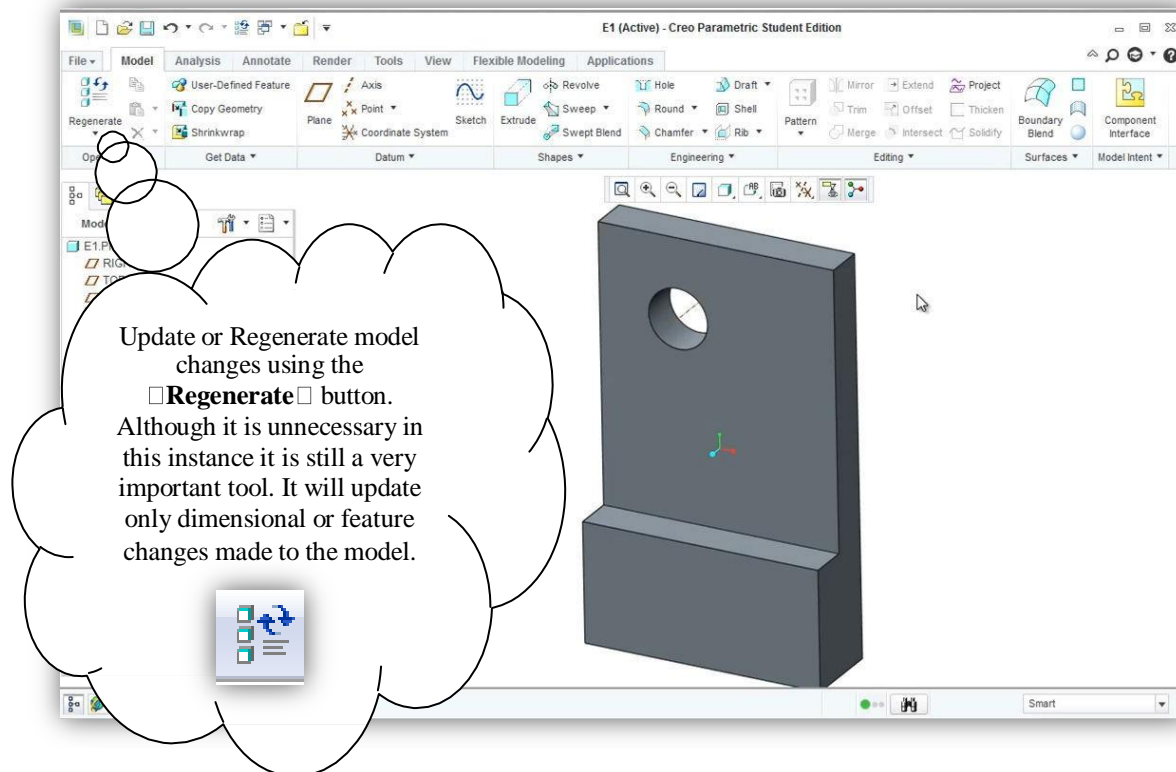
EXERCISE 2

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…

Profile

Centerline, Edge, or Axis of Revolution
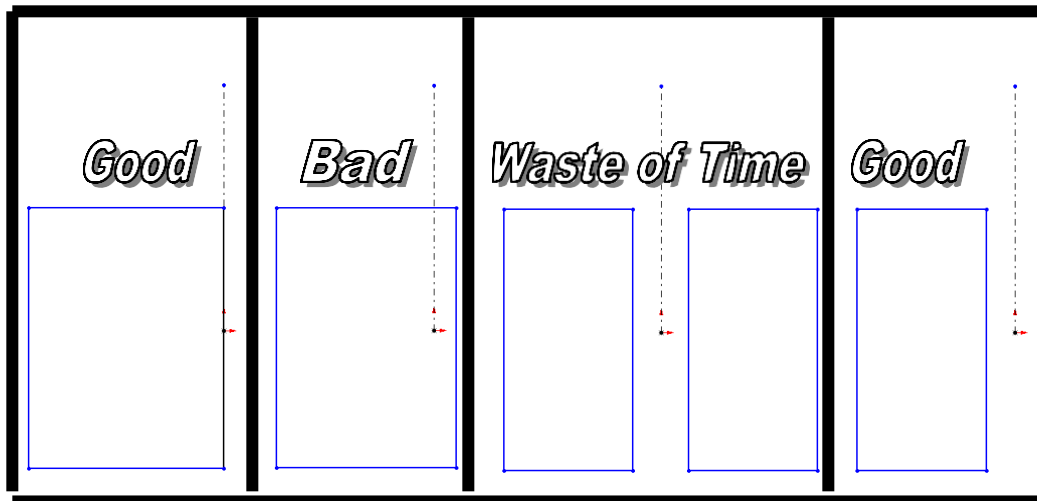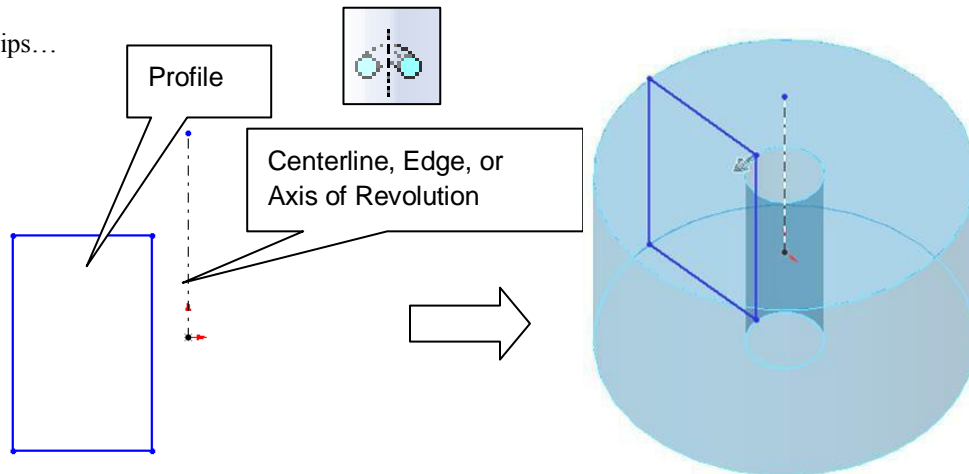
The profile should never cross over the centerline, nor should there be profiles on both sides of the centerline.

OUR GOAL
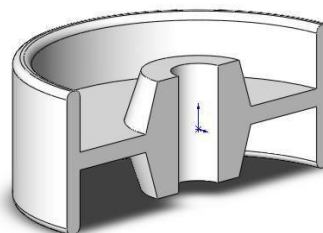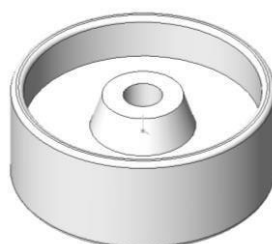
Create a new pat file .
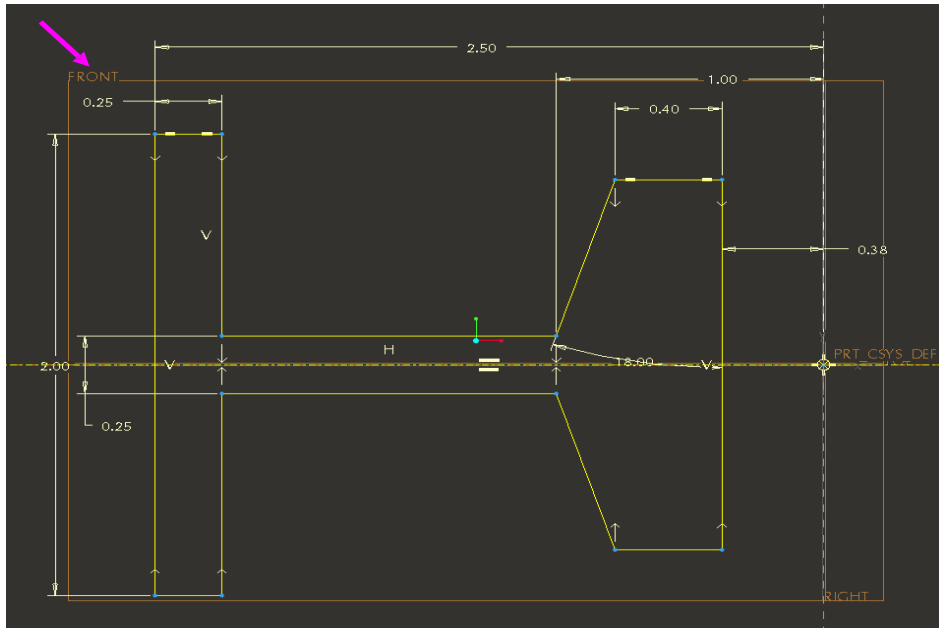
1.   Using the dimension tool to create a ¼ of the geometry and then sweep it to the other side. Make sure you finish adding the dimensions.
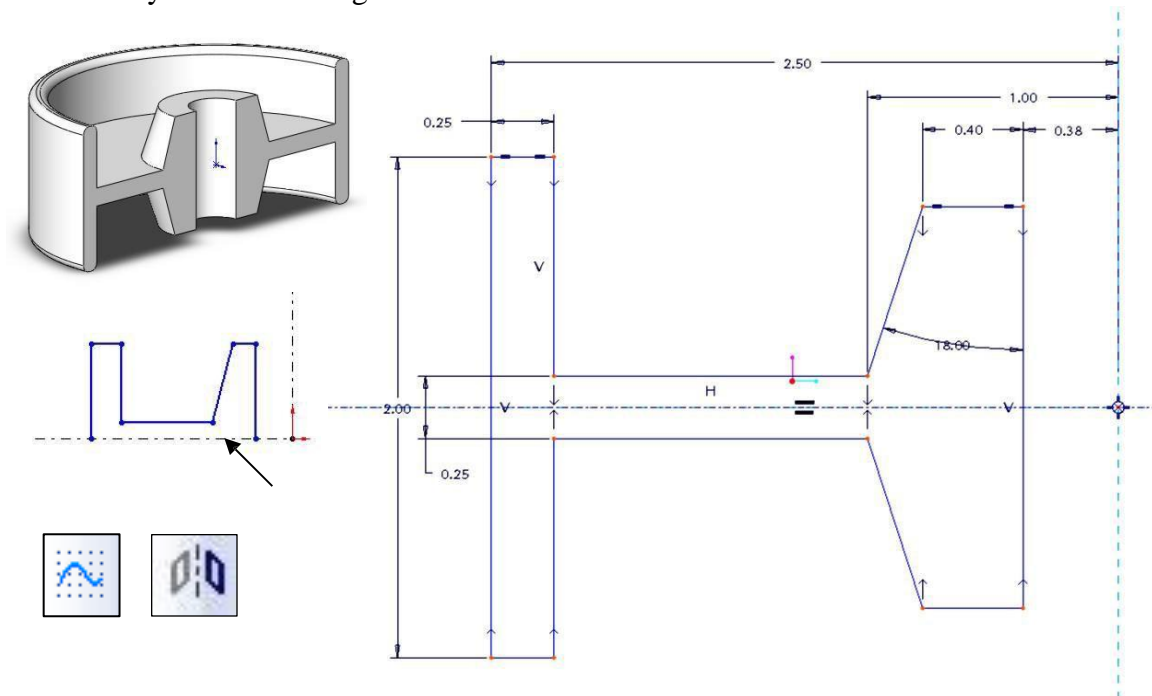
1.   Select the **Revolve** feature icon. Then select the axis/centerline.
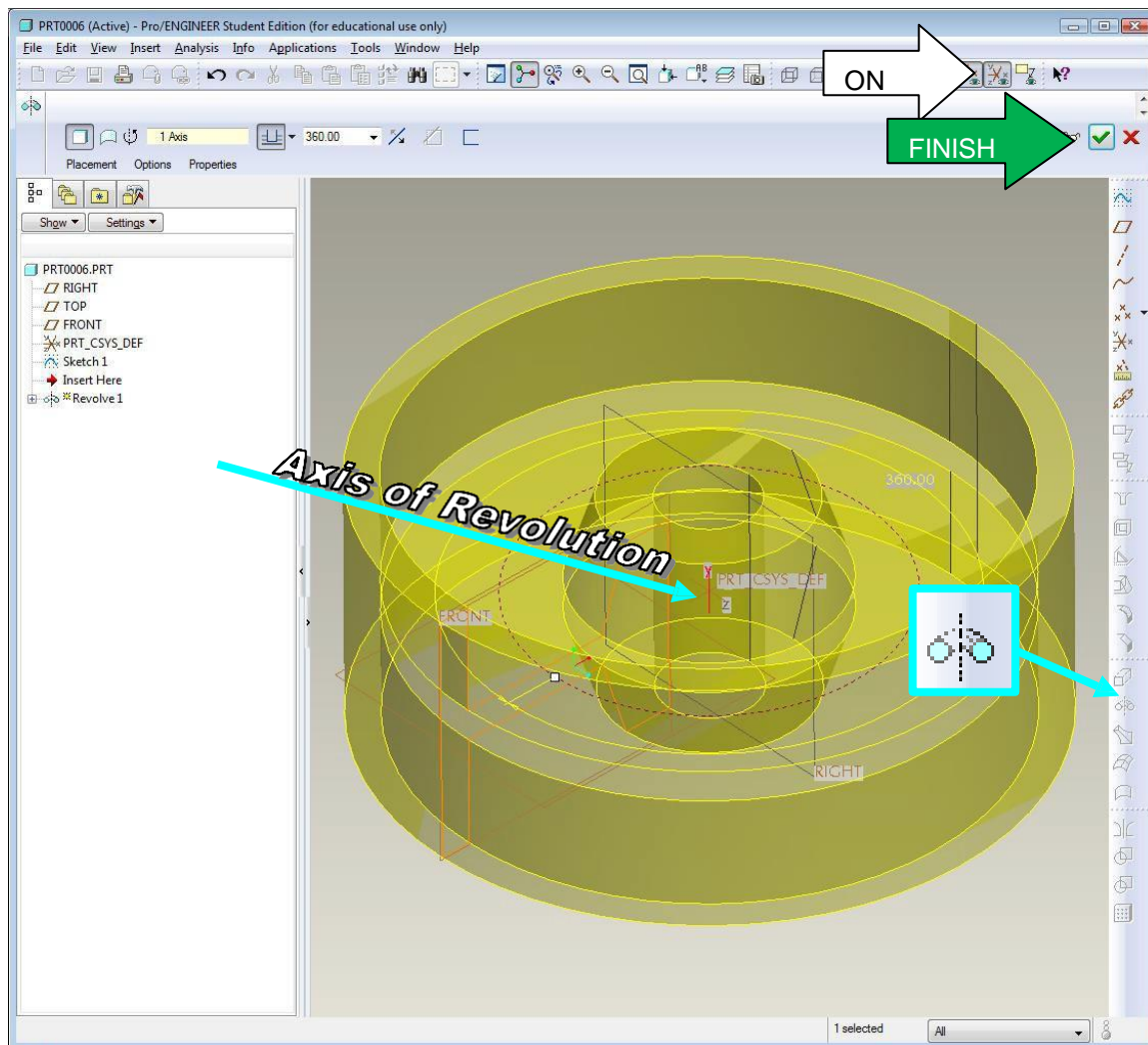
Rounds

4. Select the top and bottom edges and add a R.6 бб □ ds/fillets.
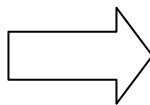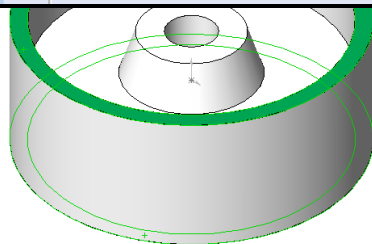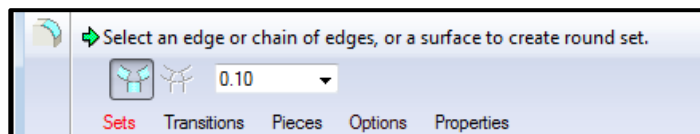
EXERCISE 4

Secondary Feature Modeling

1.   sketch the geometry as shown below
 Then **Trim.**
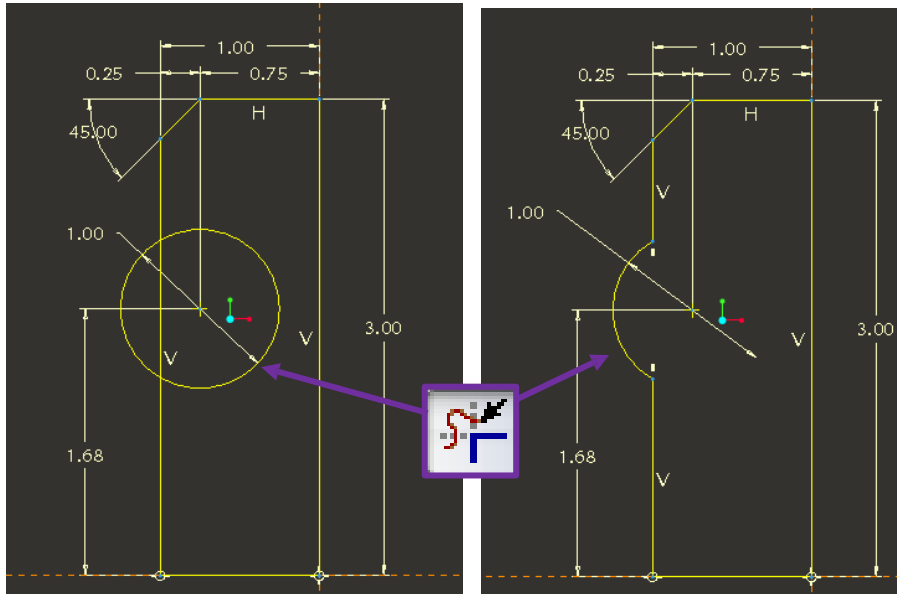
8.   **Revolve**.
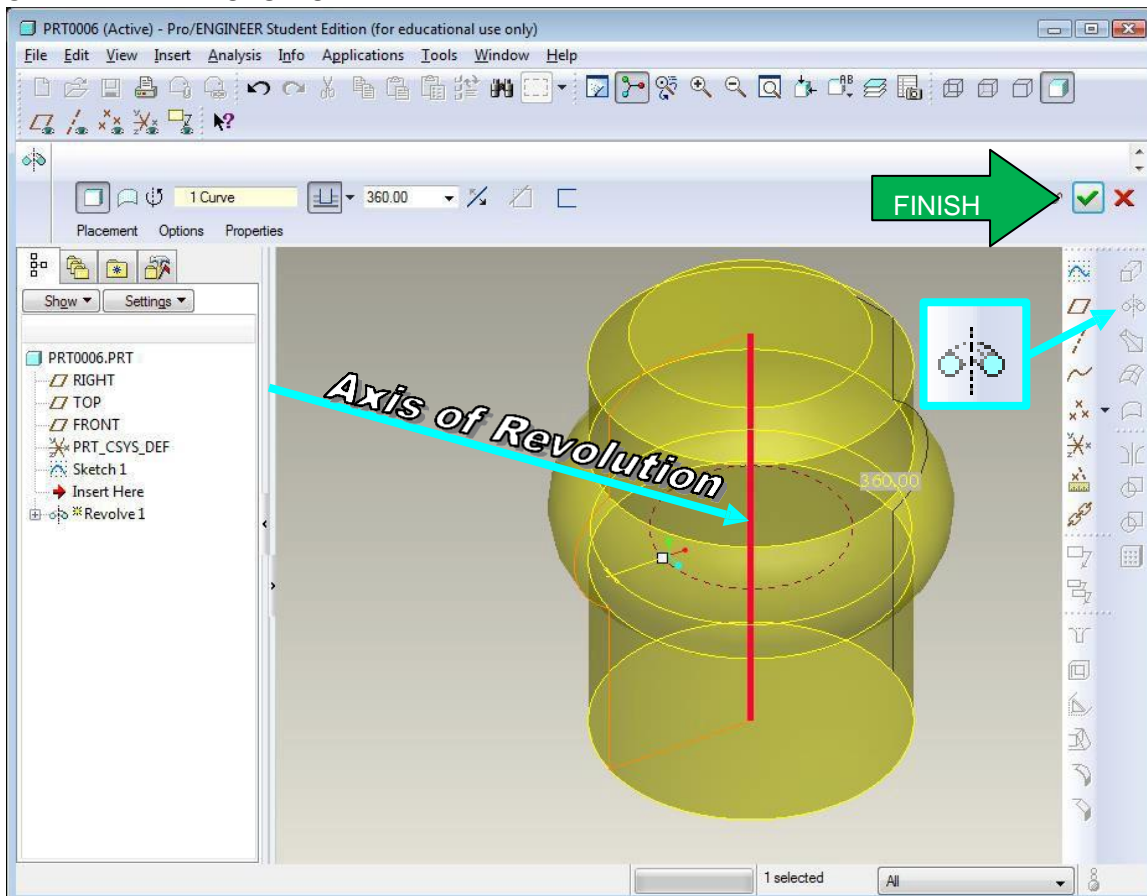
8.   **Constraints:** Select the Front datum plane and sketch the following. Use the Constraint tool and select the **Tangent** option. Then select the left most horizontal line and the arc attached to it to establish a tangent relationship.
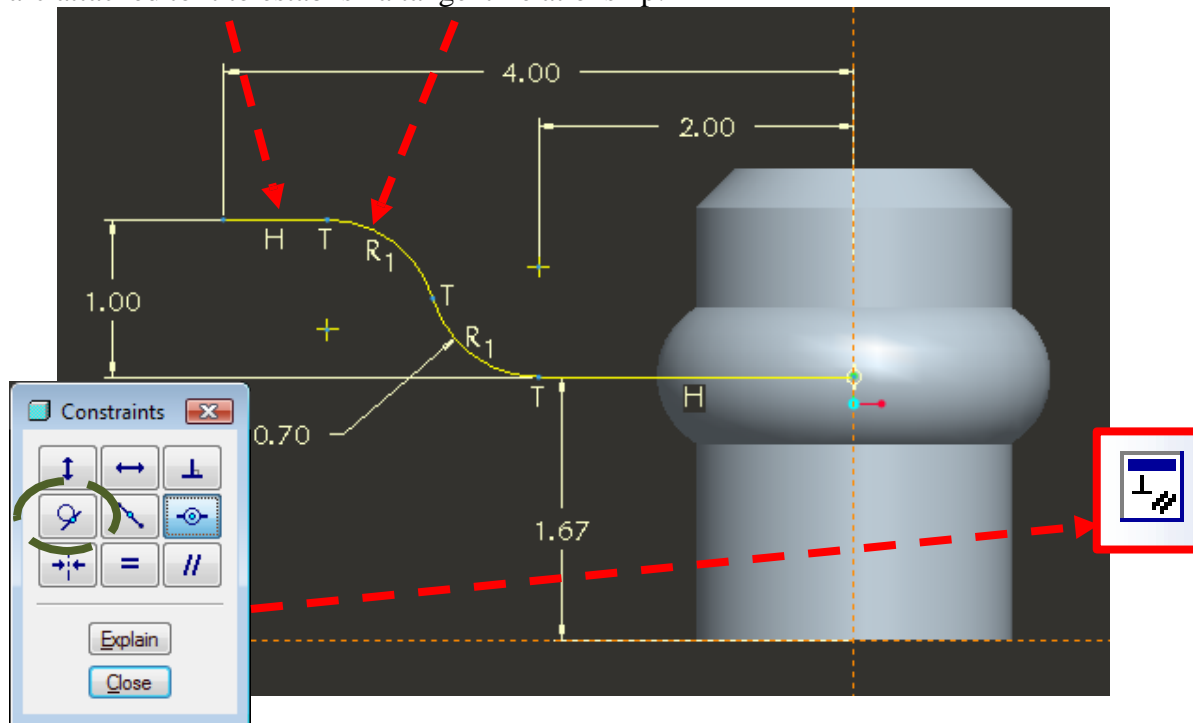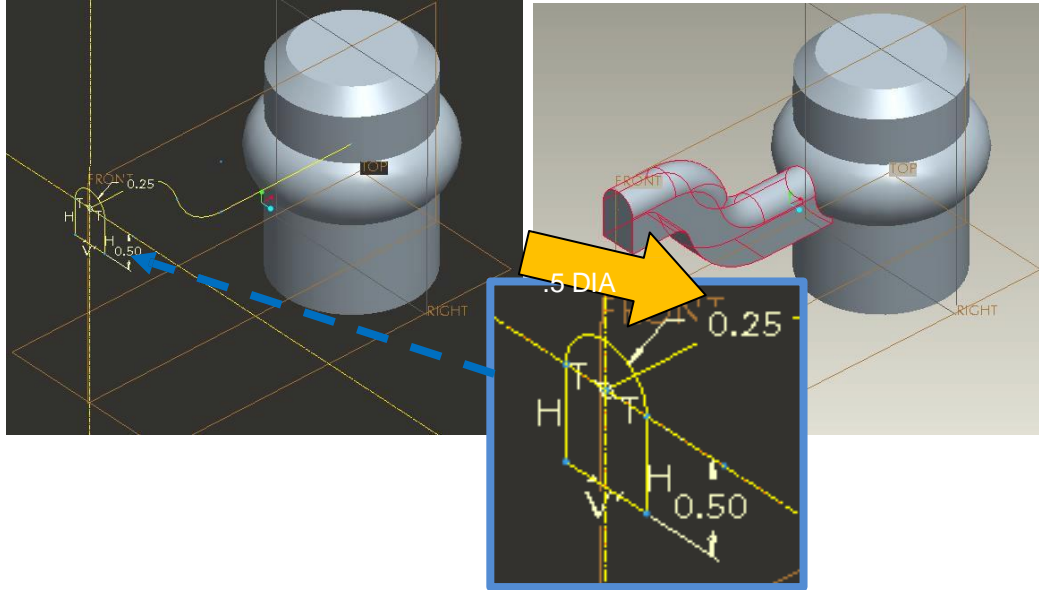

9.   **Sweeps**: left side of the curve we just created to create a new sketch datum at the end.

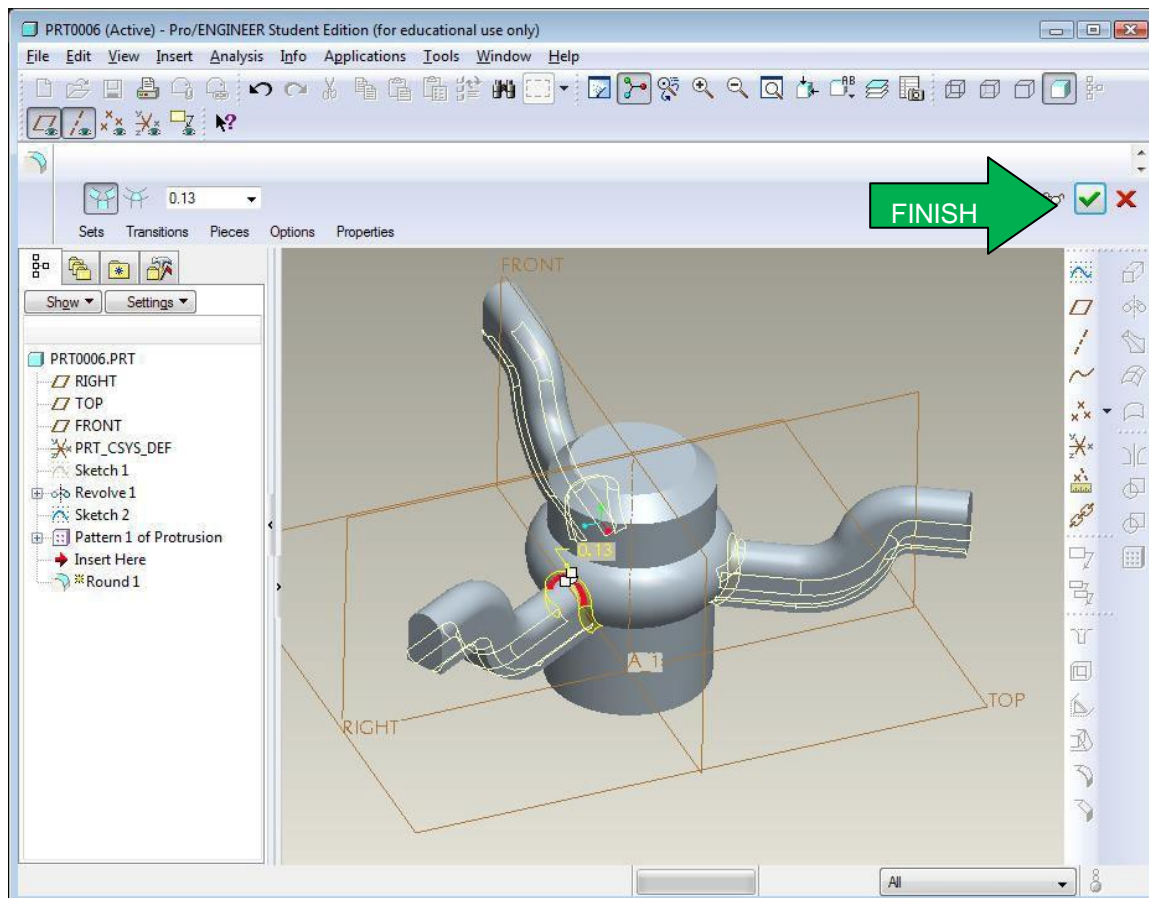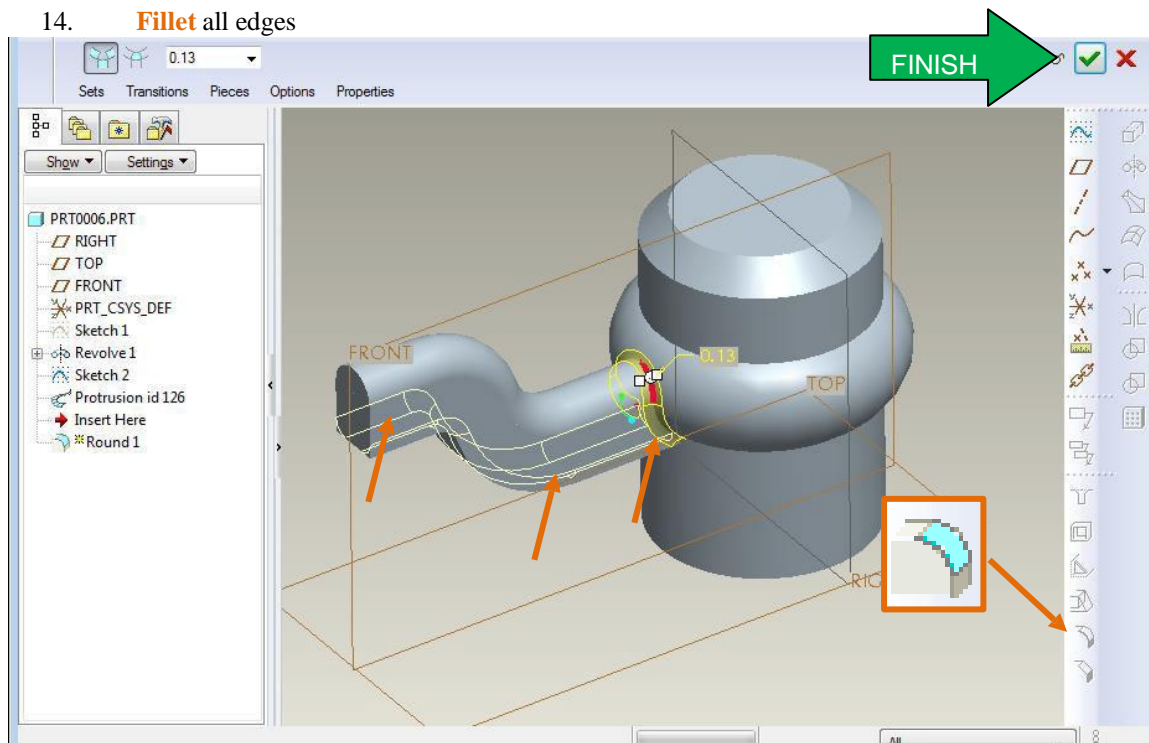10.   Also select: ☐SelectTraj/Curve Chain/Select All/Done/Done☐
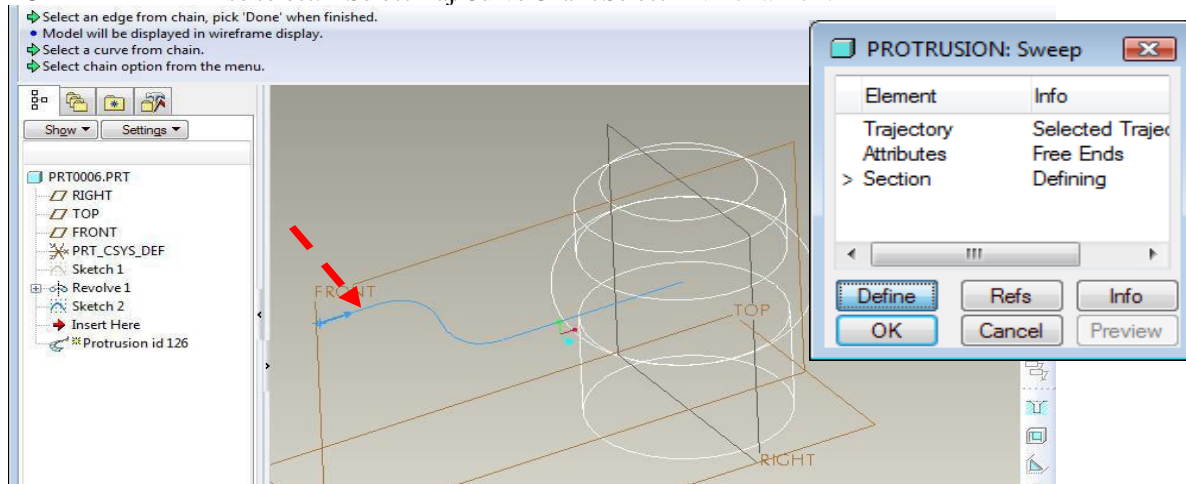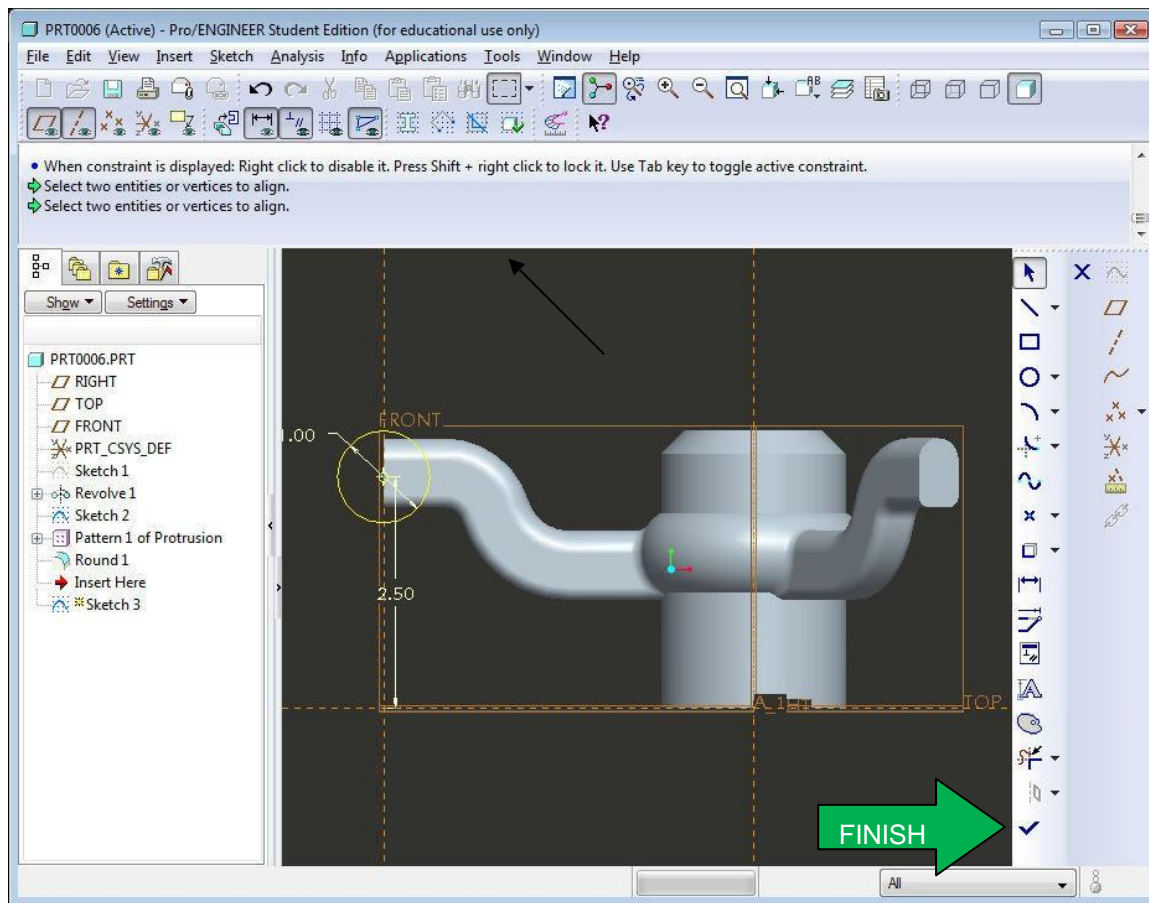
8.Draw the following sketch

8.   **Pattern** *Circular Pattern*: 360°/3 = 120° (*NOTE: First select the spoke to activate the icon.*) "SelectAxis☐ also select the ☐view axis☐
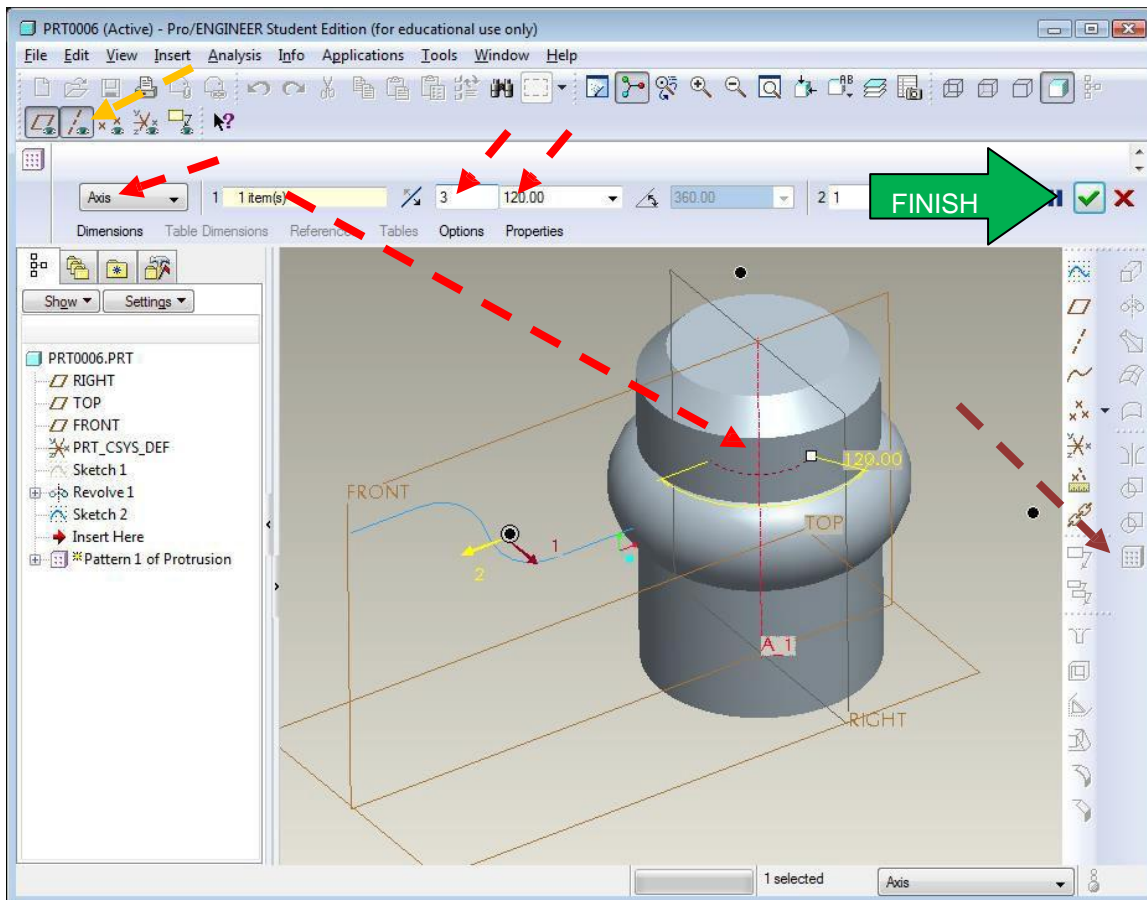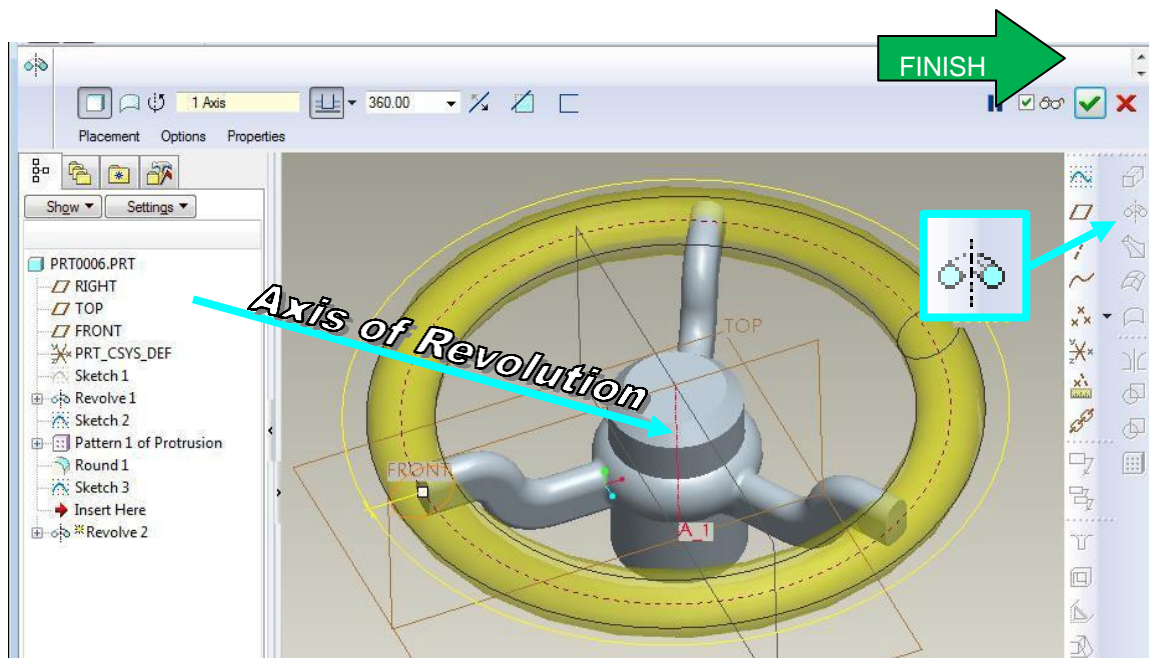
14.   **Fillet** all edges
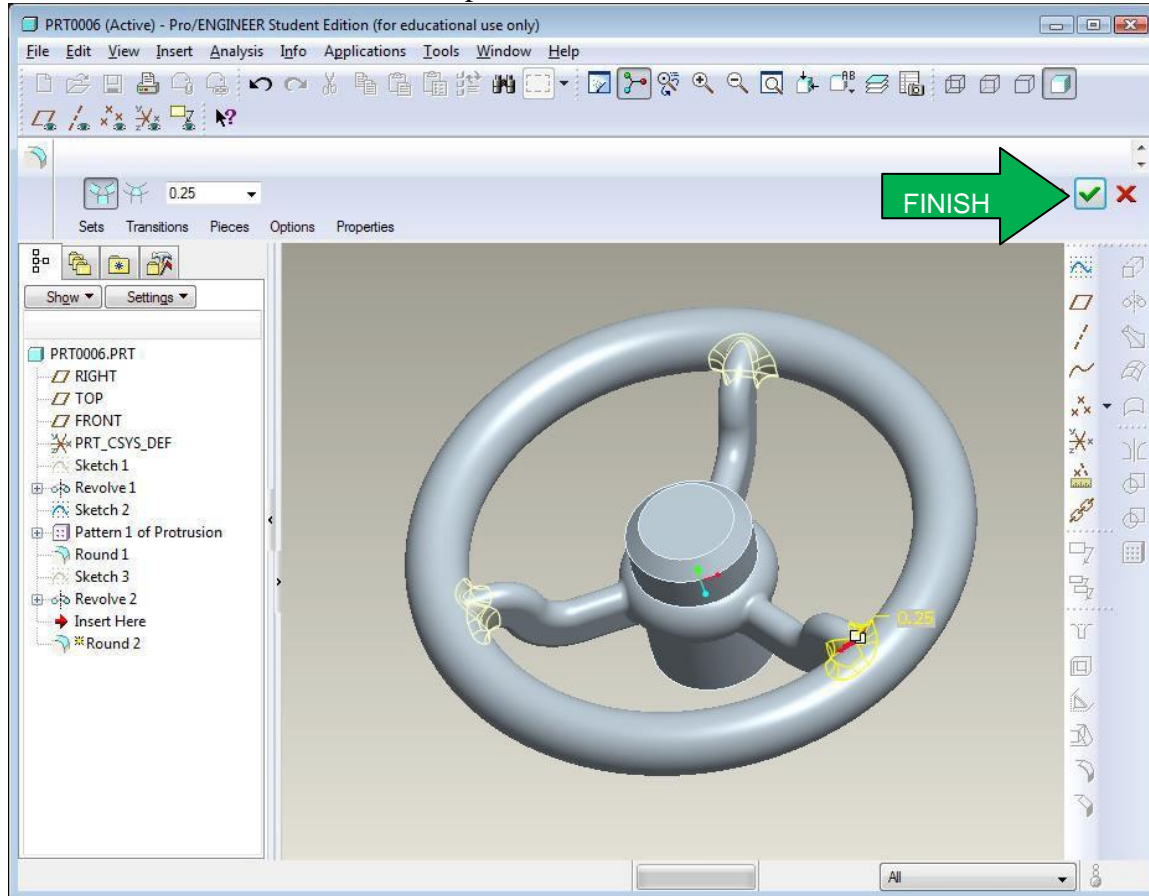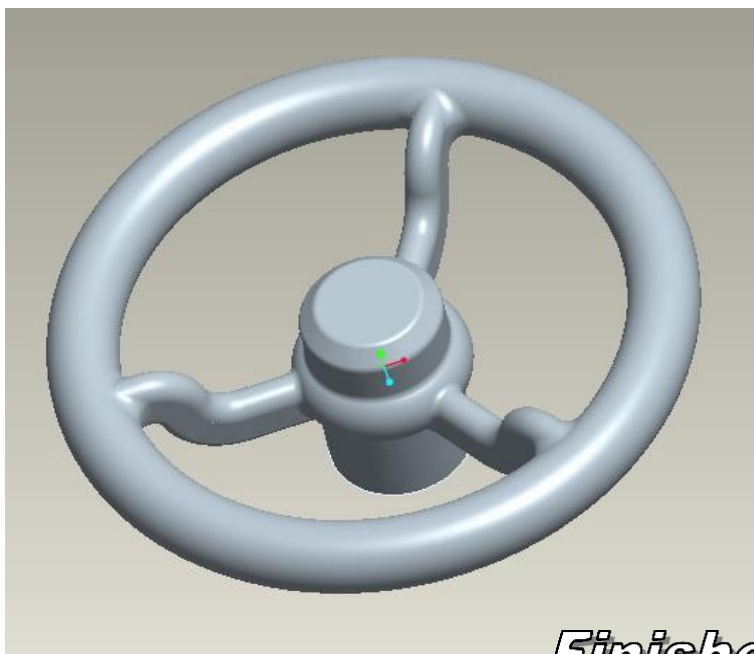
15.   Rebuild after completion.

16.   **REVOLVE**

15.   Add **Rounds** to the spoke – handle sections.

FINISHED