

Kirstie Plantenberg

Engineering Graphics Essentials with AutoCAD® 2019 Instruction

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CHAPTER 2

DRAWING IN AUTOCAD®

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CHAPTER SUMMARY

In this chapter you will learn how to navigate through and create drawings within AutoCAD's 2-D drawing workspace. Entities within the AutoCAD® environment are created within a coordinate framework. Objects such as Lines, Circles, Polygons, etc... are created using a set of defining coordinates points. Although this may sound complex, AutoCAD® makes creating professional looking engineering drawings very simple. By the end of this chapter you will be able to create and edit complex geometries.

2.1) INTRODUCTION

AutoCAD® allows you to visualize, document and share a design idea in either a 2-D or 3-D environment. This chapter will focus on AutoCAD's 2-D drawing capabilities. AutoCAD® gives you the tools to create accurate and professional looking detailed and assembly drawings. Figure 2.1-1 shows a detailed drawing created in AutoCAD's 2-D drawing workspace.

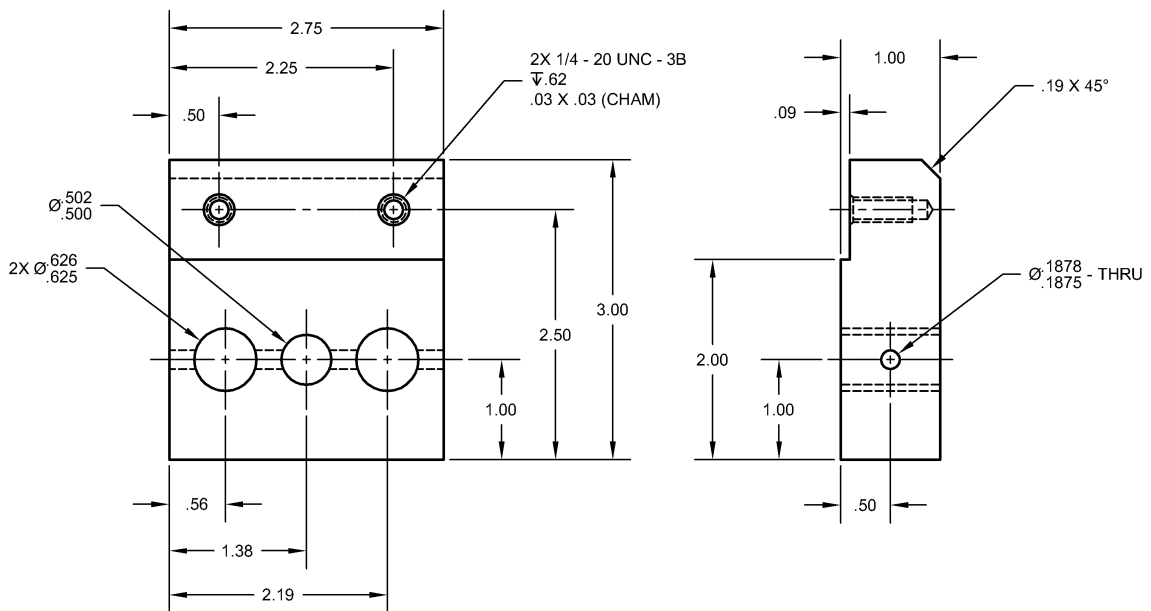


Figure 2.1-1: AutoCAD® drawing example

2.1.1) Navigating through the AutoCAD tutorials

The AutoCAD® tutorials presented in this book have specific objectives which are stated at the beginning. Each tutorial is designed to introduce the user to a particular feature of AutoCAD® or to a set of commands. As you will see, there are several ways to access AutoCAD's commands. For each command sequence, the tutorial will usually state one or two ways to access a command. This doesn't mean that you would not be able to access it in another way.

The format of each tutorial is consistent. For each step, a general statement of what is to be accomplished is given, followed by a command sequence that takes you step by step through the process. Within the tutorials the font, capitalization, and boldness of

the text are used to convey information. The text format scheme used in each tutorial is as follows:

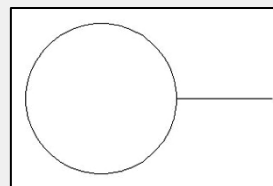
- **Underlined text:** Gives a location of where the command or options should be entered or selected.
- ***Italic text:*** Gives the name of a window, palette, ribbon, tab, or a field.
- **Bold text:** Text that should be typed in the *Command* window or into a field to access a command. Many AutoCAD commands may be entered in an abbreviated form. The “line” command may be accessed by typing the full word **LINE** or by just typing **L**. Therefore, when a command is written in full within a tutorial contained in this book, only the capitalized letters need to be typed. For example, if you see **ELLipse**, you may type the full word but you only need to type **EL**.
- **Bold italic text:** Represents the name of a command, toggle, button, or icon that should be selected.
- **Bold Courier font:** Indicates a key on the keyboard that needs to be pressed.
- **Normal text:** Gives instructions.
- **Courier font:** Represents AutoCAD prompts within the *Command* window.

An example of a command sequence from one of the tutorials is given in the following example. The command sequence illustrates the use of text format to convey information. The command sequence is used to create a line that is arrayed around an existing circle.

Example of an AutoCAD® tutorial command sequence

- 1) Create a total of **16** sun rays using a polar **ARray**. Each sun ray **Line** is **0.5** inches long.

a) **Command:** **L** or **line** or **Draw toolbar:**



b) Specify first point: **QUADrant** or

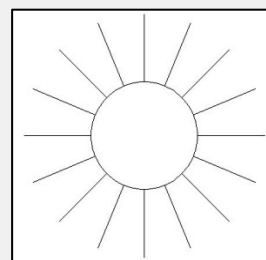


Object Snap toolbar:

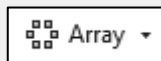
of Select the right side quadrant of the sun body circle.

c) Specify next point or [Undo]: **@.5<0**

d) Specify next point or [Undo]: **Enter**



e) **Command:** **ARray** or **Modify toolbar:**



i. Select objects: **Select the line.**

ii. Select objects: **Enter**

iii. Enter array type [Rectanglular/PATh/POLar] <Rectangular>: **PO**

iv. Specify center point of array or [Base point/Axis of rotation]: **cen**

v. of **Select the center of the circle.**

vi. Select grip to edit array or [ASsociative/Base point/Items/Angle between /Fill angle/ROWS/Levels/ROTate items/eXit] <eXit>: **F**

vii. Specify the angle to fill (+=ccw, -=cw) or [EXpression] <360>: **360**

viii. Select grip to edit array or [ASsociative/Base point/Items/Angle between /Fill angle/ROWS/Levels/ROTate items/eXit] <eXit>: **I**

ix. Enter number of items in array or [EXpression] <6>: **16**

x. Select grip to edit array or [ASsociative/Base point/Items/Angle between /Fill angle/ROWS/Levels/ROTate items/eXit] <eXit>: **Enter**

2.2) AUTOCAD'S WORKSPACES AND USER INTERFACE

AutoCAD® has three predefined workspaces: *Drafting & Annotation*, *3D Basics* and *3D Modeling*. Each workspace allows for easy access to operations that are relevant to the current workspace.

AutoCAD's user interface is workspace dependent. We will focus on the *Drafting & Annotation* workspace in this book. The *Drafting & Annotation* workspace user interfaces are shown in Figure 2.2-1. The important areas of the interfaces are identified and will be discussed in the sections indicated.

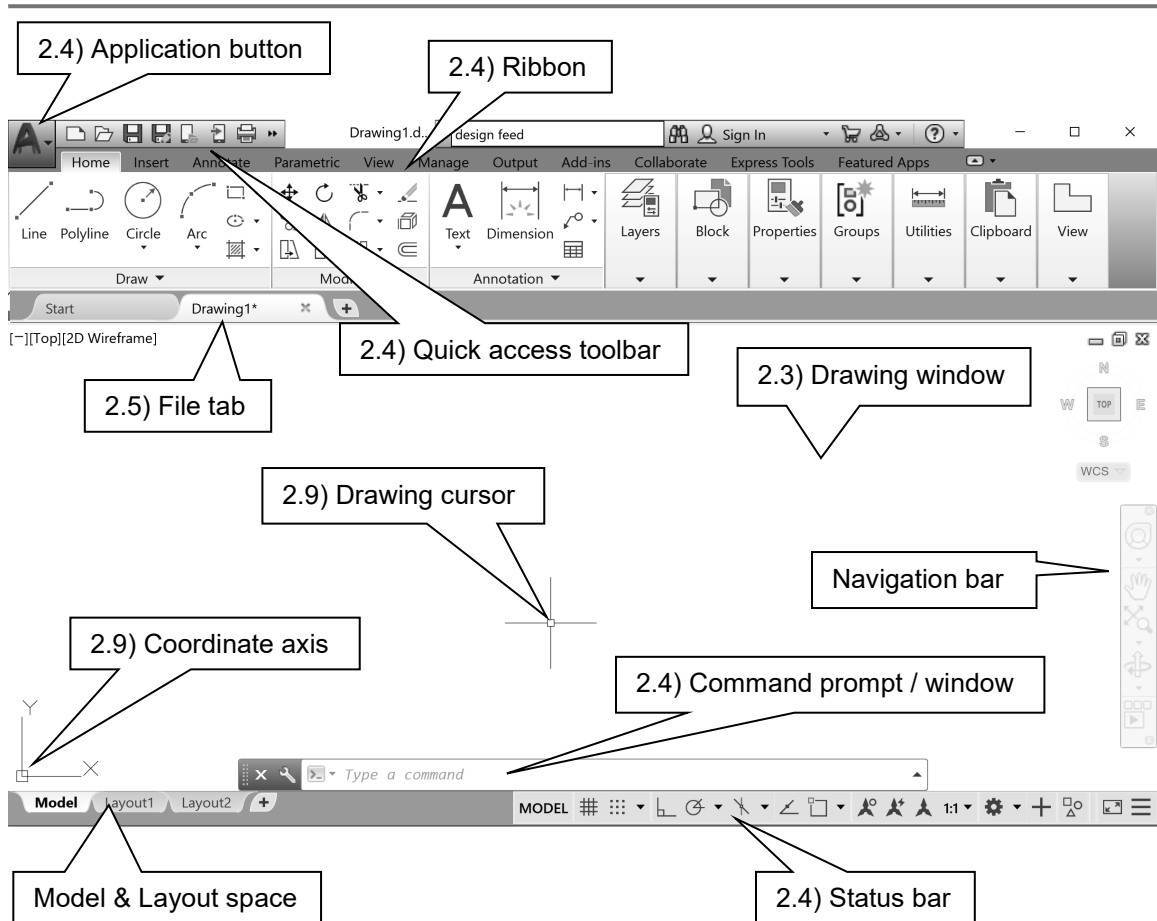


Figure 2.2-1: *Drafting & Annotation* workspace user interface

2.3) THE DRAWING AREA

The drawing area/window is the place where you create and view your drawing. The background color of the drawing window may be changed to suit the user's preference as shown in Figure 2.3-1.

The drawing area is as large as you need it to be. The usable drawing area does not just consist of the area that you can see. You can pan around the drawing area using the **Pan** command to reveal areas of your drawing that are out of view. You can also

Zoom in and out to reveal more or less of the drawing area. Both the **Pan** and **Zoom** commands are located in the *Utilities* ribbon panel in the *Home* tab or the *Standard* toolbar. You can also pan and zoom by manipulating the mouse wheel.

- **Pan:** Hold down the mouse wheel and move the mouse.
- **Zoom:** Rotate the mouse wheel.
- **Zoom All:** Double click the mouse wheel.

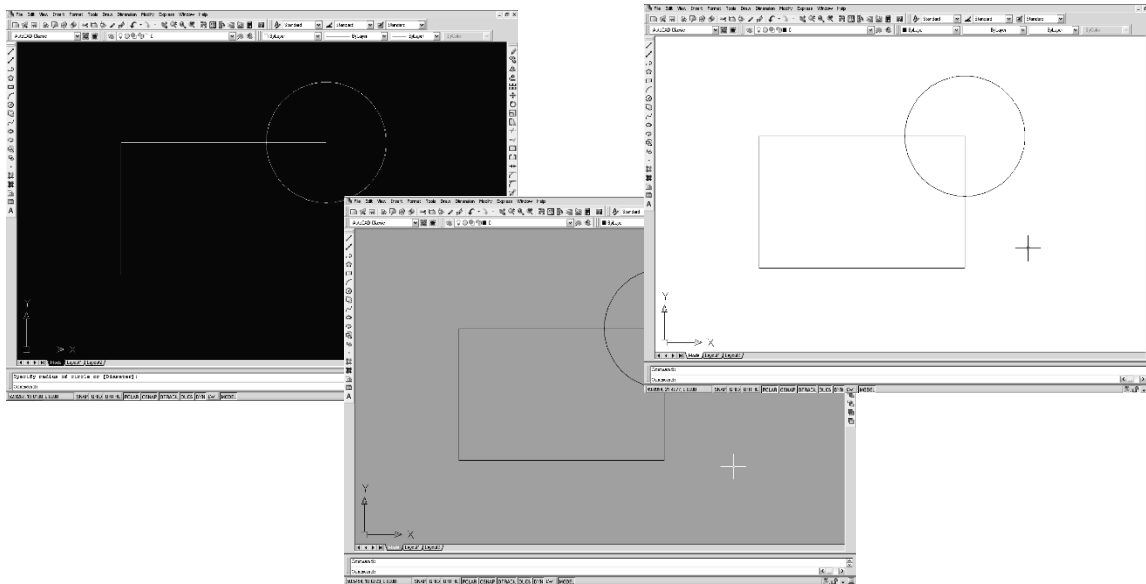




Figure 2.3-1: Drawing area background color

Changing the drawing area's background color

- 1) Application button : Select the **Options**  icon at the bottom of the menu.
- 2) Options window – Display tab: Select the **Colors...** button.
- 3) Drawing Window Colors window:
 - a) Context field: Select **2D model space**
 - b) Interface element field: Select **Uniform background**
 - c) Color field: Select your color preference.
 - d) Select the **Apply & Close** button.

Because the drawing area is so large, it is a good idea to indicate the region that you wish to use. This is your drawing size or limits. This is usually the area that will be printed. You can change your drawing size using the **LIMITS** command.

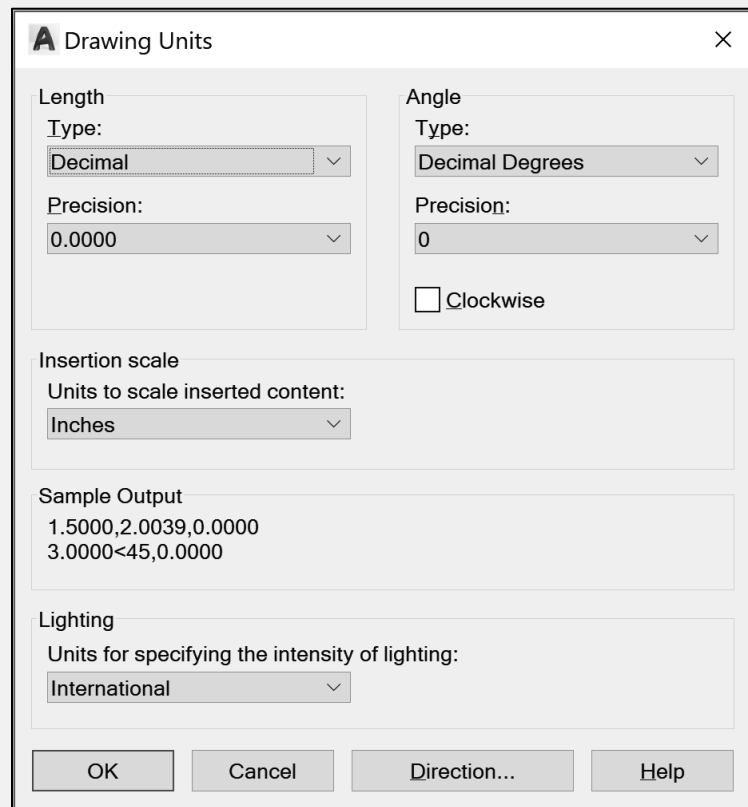
Setting your drawing size

- 1) Command: **limits**
- 2) Specify lower left corner or [ON/OFF] <0.0000,0.0000>: **Enter** (The lower left corner of your limits should always remain 0,0.)
- 3) Specify upper right corner <420.0000,297.0000>: **280,216** (This changes a Metric drawing area to the equivalent of an 11 x 8.5 sheet of paper.)

The units (i.e., inches, millimeters, feet) used to draw objects in the drawing area can be selected using the **UNITS** command.

Setting your drawing units, precision and angle directions

- 1) **Command: UNITS**
- 2) **Drawing Units window:** Use this window to set your units and precision.



2.4) ACCESSING AUTOCAD COMMANDS

AutoCAD® allows its users to access commands in many different ways, one of which will no doubt become your favorite. The most common ways of accessing AutoCAD's commands within the *Drafting & Annotation* workspace are through the application button, ribbon, command window, shortcut menus, and status bar.

2.4.1) Application button

Menus are available through the *Application button* in the upper left corner of the drawing window. These menus contain the commands used to create, save, print and manage your drawing. Figure 2.4-1 shows the *Application button*. The *Search menu* (near the top) allows you to perform a keyword command search. The search even works for commands that are not directly accessible through the *Application button*. Note the location of the **Options** button at the bottom. This button will be referenced several times in the upcoming tutorials.

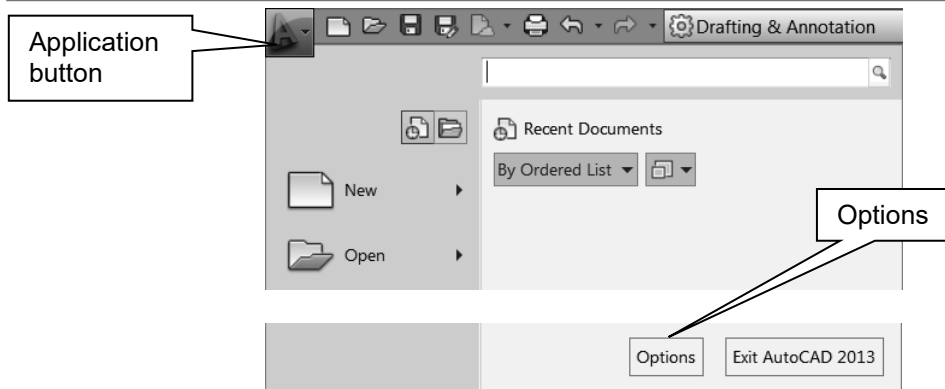


Figure 2.4-1: Application button

2.4.2) The ribbon

The *Ribbon* provides, in a single location, commands that are relevant to the current workspace. It is organized into a series of tabs. Several ribbon panels are located in each tab area. Each panel contains related commands. Figure 2.4-2 shows the *Drafting & Annotation* workspace *Ribbon*. Individual ribbon panels may be expanded by clicking on the black arrow located in the bottom right corner of each panel. Once the mouse moves off of an expanded panel, the panel will collapse unless it is pinned. The ribbon may be docked horizontally or vertically. When it is docked vertically, the text tabs are replaced with icons. The ribbon as a whole or individual panels may float freely anywhere in the drawing window. Panels may also be added to or removed from a specific ribbon tab. This requires accessing the CUI (Customize User Interface) manager.

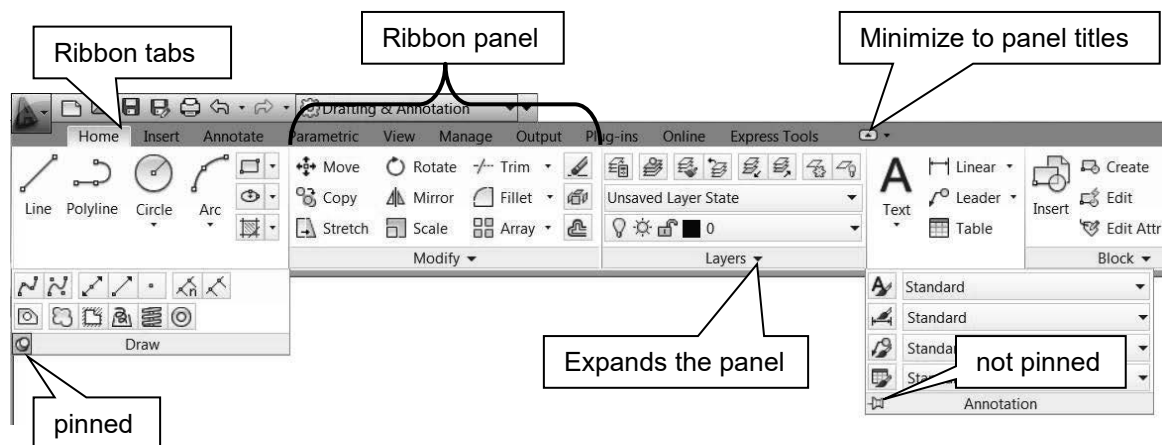
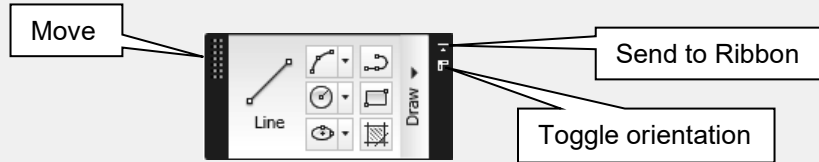


Figure 2.4-2: The Ribbon

Floating and docking ribbon panels

- 1) **Floating:** Click on a ribbon panel title and drag it to the drawing window.
- 2) **Docking:** Once the panel is floating, it may be moved, sent back to the ribbon, or the orientation may be changed.



Tooltips appear when your mouse hovers over an icon. For the first few seconds an abbreviated tooltip appears. If the mouse remains over the icon, an expanded tooltip will appear as shown in Figure 2.4-3. The tooltips help you navigate through the command steps if you are uncertain how to proceed. Tooltip appearance may be modified by accessing in the **Display** tab of the **Options** window (*Application button – Options – Display tab*).

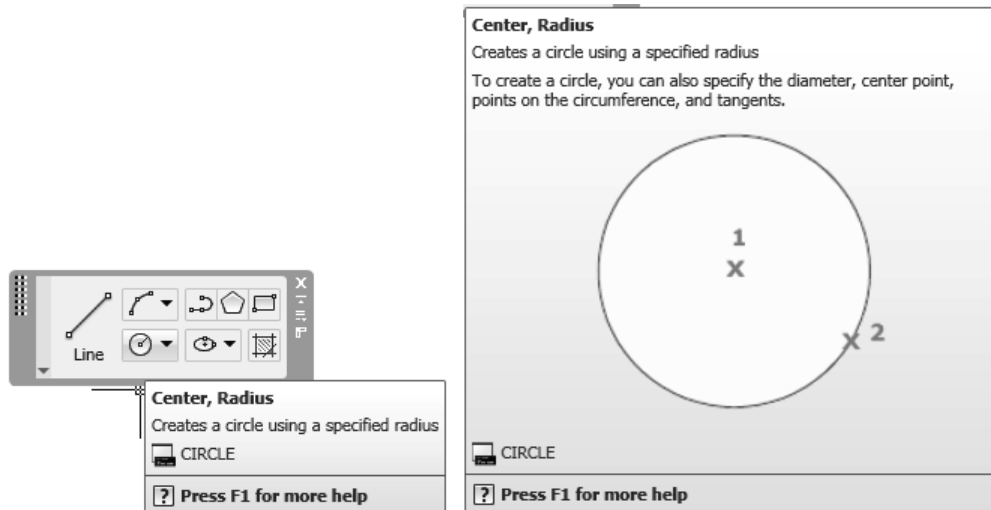


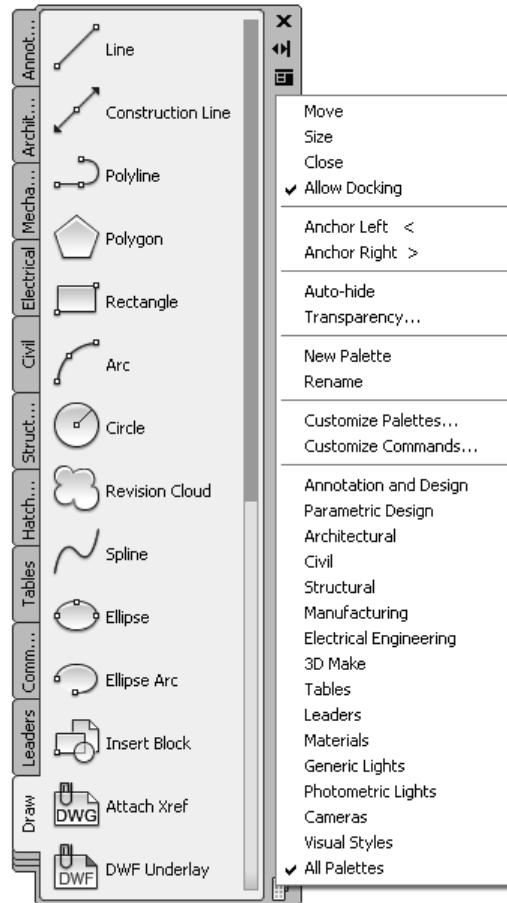
Figure 2.4-3: Tooltips

2.4.3) Tool palettes

Tool palettes are tabbed areas within the *Tool Palettes* window that provide an efficient method for organizing, sharing, and repeatedly using blocks, hatches, and your most frequently used commands. The Tool Palettes may be activated within the **Palettes** panel in the **View** tab.

The options and settings for the *Tool Palettes* are accessible from shortcut menus that are displayed when you right-click in different areas of the *Tool Palettes* window. The bottom portion of the shortcut menu, starting with *Dynamic Blocks*, are subsets of all the palettes. Other settings include:

- **Allow Docking:** Toggles the ability to dock or anchor palette windows. If this option is selected, a window can be docked when you drag it over a docking area at the side of a drawing. Selecting this option also makes *Anchor Right* and *Anchor Left* available. Press the **Ctrl** key if you want to prevent docking as you move the *Tool Palettes* window.
- **Auto-hide:** Controls the display of the palette when it is floating. When this option is selected, only the tool palette title bar is displayed when the cursor moves outside the tool palette. When this option is cleared, the palette stays open continuously.
- **Transparency:** Sets the transparency of the Tool Palettes window so it does not obscure objects behind it.



The items within a tool palette are called tools. You can create your own tools by dragging objects such as dimensions, multiline text, polylines, blocks, hatches, etc... from your drawing area onto a tool palette. You can then use the new tool to create objects with the same properties as the object you dragged to the tool palette. Once you add a command to a tool palette, you can click on the tool to execute the command.

Adding a tool to a tool palette

- 1) Place your cursor over the *Tool Palettes* window and click your right mouse button. A shortcut menu will appear.
- 2) **Shortcut menu:** Select **New Palette**.
- 3) Name the new palette **My Palette**.
- 4) **Drawing area:** Select the object you wish to add to your palette.
- 5) Place your cursor over your object, right click your mouse button and drag the object to the *My Palette* area.
- 6) To create another object like the one used to create the tool, just click on the icon in the *My Palette* area.

Move
Size
Close
 Allow Docking

Anchor Left <
Anchor Right >

Auto-hide
Transparency...

New Palette
Rename

2.4.4) Command prompt / window

The *Command* window (Figure 2.4-4) is the place where you can type in AutoCAD commands, enter coordinates, and select what type of information you wish to enter. This is also the place where AutoCAD prompts you and asks you questions. It is important to read the *Command* prompt when working with an unfamiliar command. The *Command* window, by default, is located at the bottom of the drawing area. However, it can be moved, resized, and closed to suit the user's need. If the *Command* window has been closed, you can type **ctrl + 9** to reopen it.

To enter a command using the keyboard, type the command name on the *Command* line and press **Enter** or the **Spacebar**. Commands entered at the *Command* line are not case sensitive and you do not have to move the mouse over the *Command* window before you type. When the *Dynamic Input* is switched on and is set to display dynamic prompts, the entered commands will appear near the cursor.

Most commands may be abbreviated, for example: Line = L, Circle = C, Move = M, and Copy = CO. Abbreviated command names are called command aliases and are defined in the acad.pgp file.

When you type a command, you will see a set of suggestions. To select one of these suggested commands, click on it with your mouse to activate the command. The system variable **Inputsearchdelay** controls the number of milliseconds to delay before the command line suggestion list is displayed. A finer level of control may be accessed through the *Input Search Options* window. This is opened by typing the command **Inputsearchoptions**.

The *Command* window shown in Figure 2.4-4 shows the process used to draw a circle. First, **C** is entered to activate the CIRCLE command. Second, AutoCAD prompts us to enter a center point or choose another option. A center point was chosen with the mouse. Third, AutoCAD prompts you for a radius or the option of a diameter. **D** was entered to indicate that a diameter is to be entered. Lastly, AutoCAD prompts you for the circle's diameter. The diameter is then entered.

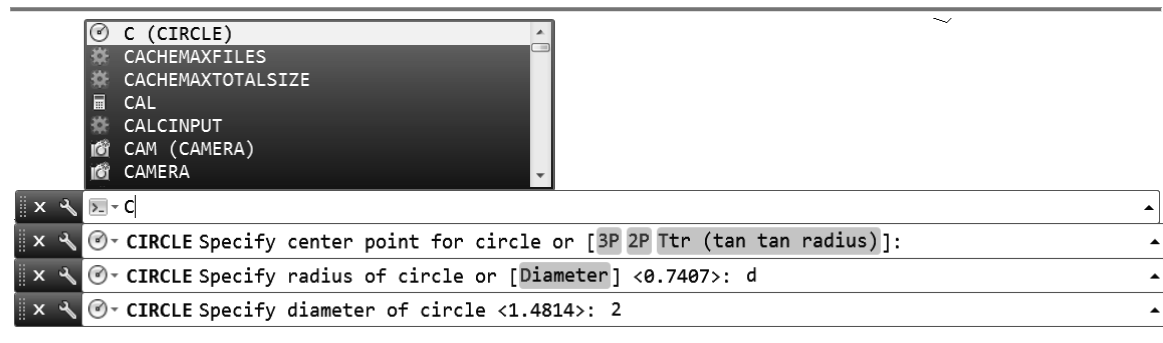
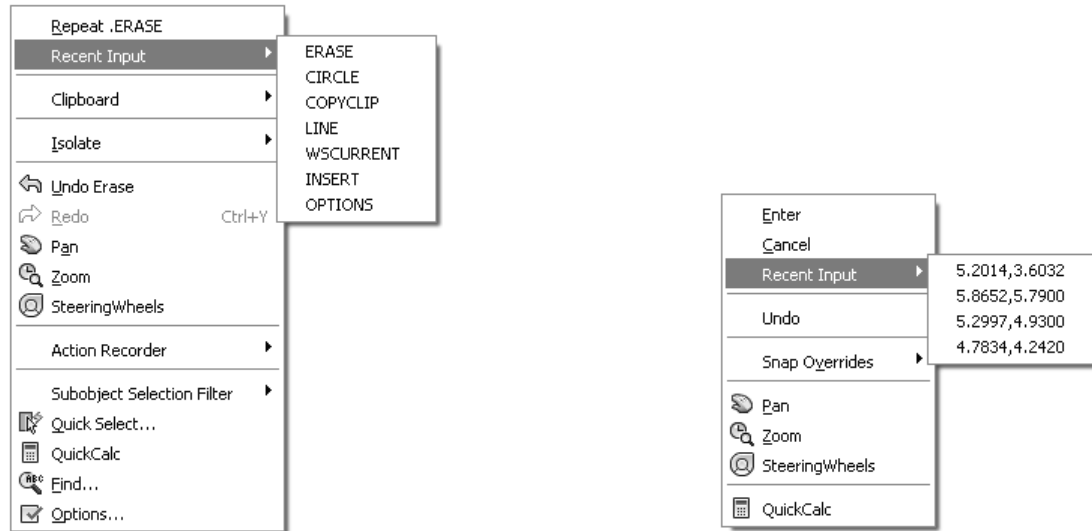


Figure 2.4-4: The *Command* window

A command may be repeated by pressing **Enter** or the **Spacebar**. A history of inputs may be shown by clicking on the black arrow on the right side of the command window. To find a command, you can type a letter on the command line and a list of commands starting with that letter will be shown (see Figure 2.4-4). To restart a recently used command, right click on the command line to access a shortcut menu.

2.4.5) Shortcut menus

Shortcut menus allow quick access to commands that are relevant to your current activity. You can display different shortcut menus when you right-click different areas of the screen and during a command. The shortcut menus shown in Figure 2.4-5 appear when you right click in the drawing window and during the LINE command.



Drawing window shortcut menu

LINE command shortcut menu

Figure 2.4-5: Shortcut menu examples

2.4.6) Application status bar

The *Application Status bar* displays the coordinate values of your cursor, drawing tools, navigation tools, and tools for quick view and annotation scaling. It is located at the bottom of the drawing area. Figure 2.4-6 shows the *Status Bar*. The buttons in the status bar are active or on when they are highlighted or light blue (they look gray when they are off). The status bar may contain the coordinate readout. This reads the location of your cursor in the drawing area. Next to the coordinate reading are the drawing tools. These commands are frequently turned on and off throughout the drawing process. The *Application Status bar* options may be accessed by clicking on the *Customization* icon.

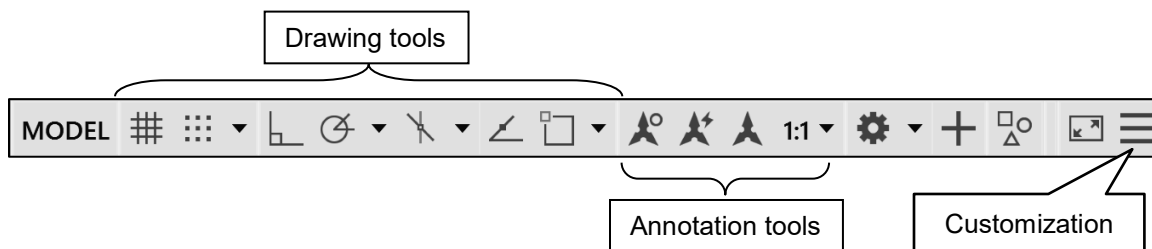


Figure 2.4-6: The *Status Bar*

2.4.7) Quick access toolbar

The *Quick Access* toolbar, displayed in the *Drafting & Annotation* workspace, is located at the very top of the drawing window next to the *Application* button. The *Quick Access* toolbar may be customized by adding or removing commands. This is done by right clicking on the toolbar and selecting **Customize Quick Access toolbar...**, or selecting the arrow at the end of the toolbar. The *Quick Access* toolbar is shown in Figure 2.4-7.



Figure 2.4-7: The *Quick Access* toolbar

The *Quick Access* toolbar contains the following commands (reading left to right):


- **QNew:** Opens a new drawing.
- **Open...:** Opens an existing drawing. (Ctrl+O)
- **Save:** Saves the current drawing. (Ctrl+S)
- **Save as:** Allows you to save the current drawing under a different name. (Ctrl+Shift+S)
- **Open from Web & Mobile:** Allows you to open a file from the online cloud.
- **Save to Web & Mobile:** Allows you to save a file to the online cloud.
- **Plot...:** Plots or prints the current drawing. (Ctrl+P)
- **Undo:** Used to undo previous command or actions.
- **Redo:** Used to redo commands that have been undone.

2.4.8) Touch screen

For those using a touch-enabled screen or interface, the system variable **TOUCHMODE** controls the display of the *Touch* panel on the ribbon. If **TOUCHMODE** is set to 0, then the *Touch* panel is not displayed on the ribbon. If it is set to 1, then the *Touch* panel is displayed on the ribbon. The system variable **MAXTOUCHES** indicates whether a digitizer or touch pad is in use.

2.5) STARTING, SAVING, AND OPENING DRAWINGS

2.5.1) Starting a new drawing

When starting a new drawing (QNEW ) , you have a choice of either starting from the *Create New Drawing* window or the *Select Template* window. The *Create New Drawing* window allows you to set up a drawing to your preferences. You may set parameters such as the units (Imperial or Metric), the size of the drawing, and the degree of precision. The *Select Template* window allows you to choose from predefined templates. Figure 2.5-1 shows both startup windows. The **STARTUP** variable is used to choose what is displayed when the application is started, or which window will appear when you start a new drawing. It has 4 values that may be set (i.e., 0, 1, 2, and 3). However, for starting a new drawing, only 0 and 1 are of interest. If **STARTUP** = 0, then

the *Select Template* window will appear. If STARTUP = 1, then the *Create New Drawing* window will appear.

Template drawings store all the settings for a drawing and may also include predefined layers, dimension styles, and views. Template drawings are distinguished from other drawing files by the .dwt file extension. Several template drawings are included in AutoCAD®. You can make additional template drawings by changing the extensions of drawing file names to .dwt.

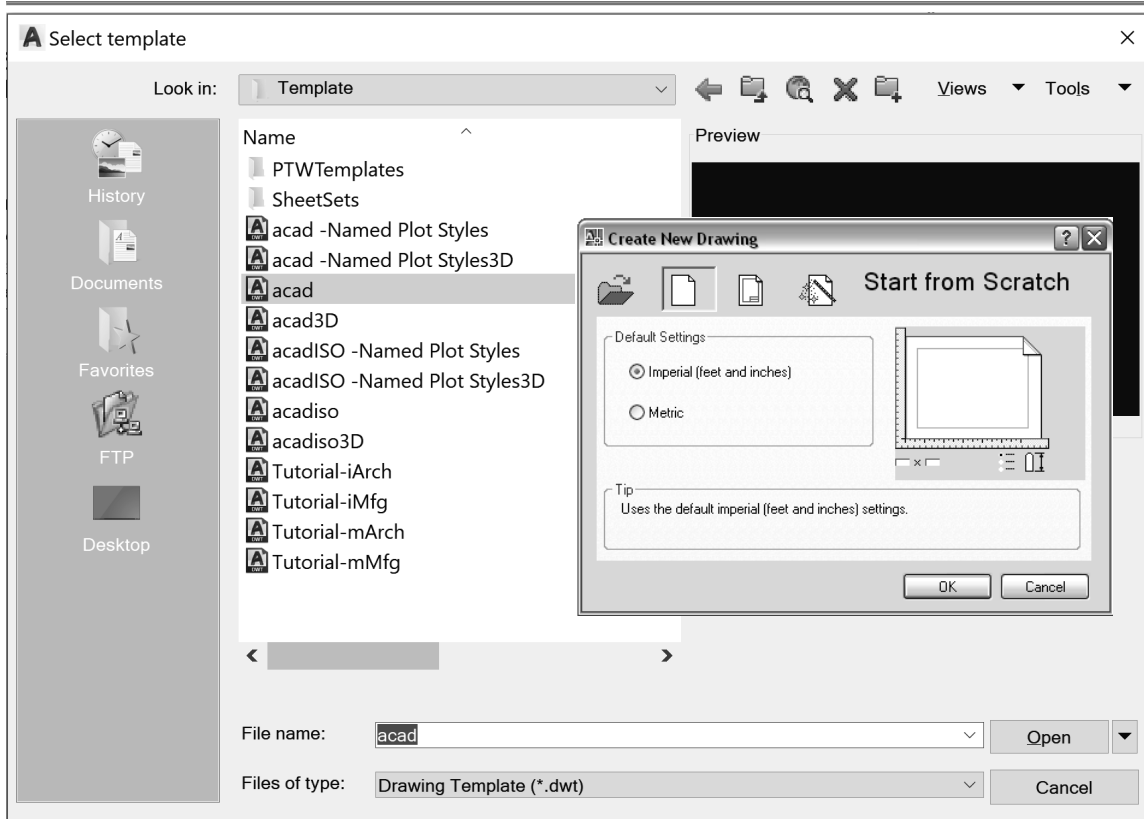

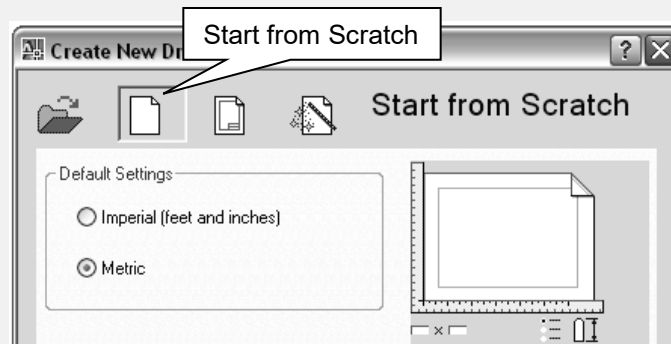


Figure 2.5-1: Startup window

Starting a new drawing using the *Create New Drawing* window

- 1) Command: **startup**
- 2) Enter new value for STARTUP <0>: **1**
- 3) Quick Access toolbar:  or Application button: **File – New...** (Ctrl+N).
The *Create New Drawing* window will appear.
- 4) Create New Drawing window: Activate the **Start from Scratch** button, activate either **Imperial** or **Metric** toggle, and then select **OK**.



- 5) Quick Access toolbar: 
- 6) Create New Drawing window:
 - a) Activate the **Use a Wizard** button.
 - b) Select a Wizard field: Select **Advanced Setup** and then **OK**.



- 7) The wizard will take you through a setup which will allow you to choose your drawings units, angle, angle measure, angle direction and drawing area.

2.5.2) Saving and opening a drawing

When saving (or open) a drawing (**Application** button - **Save** or **Saveas** or **Open**), you have the option of saving (or opening) the following file types.

- **DWG** (DraWinG) is a binary file format used for storing two and three dimensional design data and metadata. Most of what you draw will be saved in this format.
- **DWT** is a template file. These files are used as a starting point when starting a new drawing. They may contain drawing preferences, settings, and title blocks that you do not want to create over and over again for every new drawing.
- **DXF** (Drawing Interchange Format, or Drawing Exchange Format) is a CAD data file format developed by Autodesk® for enabling data interoperability between AutoCAD® and other programs.
- **DWS** is a standards file. To set standards, you create a file that defines properties for layers, dimension styles, linetypes, and text styles, and you save it as a standards file with the .dws file name extension.

2.5.3) File tab

File tabs are displayed at the top of the drawing area as shown in Figure 2.5-2. They help provide an easy way for you to access all the open drawings in the application. The file tab usually displays the full name of the file. A preview of the file and all its spaces appears when you hover your mouse over the tab. The plus button on the right end of the file tabs displays the *Select Template* dialog box to create a new drawing. The following are the system variables that relate to the file tabs.

- **FILETAB** opens the file tabs.
- **FILETABCLOSE** closes the file tabs.
- **FILETABPREVIEW** controls the type of preview (1 = thumbnail, 0 = list).

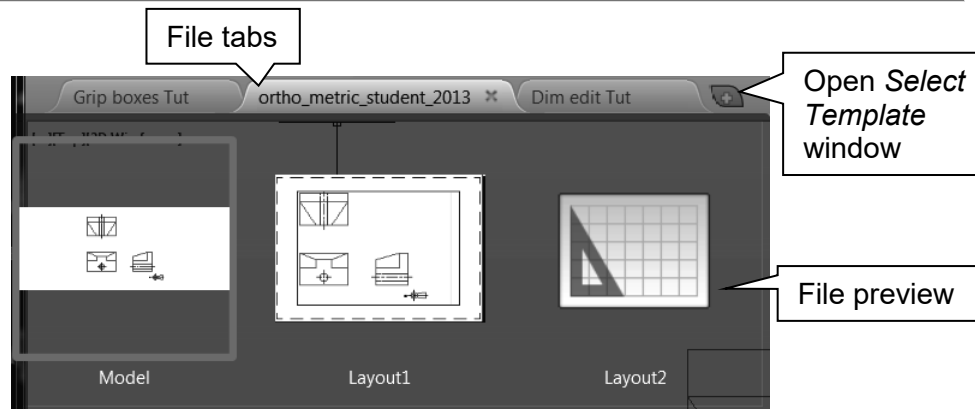


Figure 2.5-2: File tab preview

2.5.4) Saving and opening from the web or mobile device.

The **Save to web & mobile**  and **Open from web & mobile**  allow you to save and open drawings online from any remote location and from any device that has internet access. The first time you use either of these commands, AutoCAD will prompt you to install the **Save to AutoCAD web & mobile** plugin. If you don't already have an account, the program will ask you to create one.

2.6) CUSTOMIZE USER INTERFACE (CUI)

The CUI allows you to adjust the user interface and drawing area to match the way you work. Many of these settings are available from shortcut menus and the *Options* window. Some workspace elements, such as the presence and location of ribbon panels, toolbars and palettes can be specified and saved using the *Customize User Interface* manager. However, you should not make major changes to the CUI if you are using a public access computer.

2.7) USER INTERFACE AND STARTUP TUTORIAL

The objective of this tutorial is to set up AutoCAD's user interface and start a new drawing. AutoCAD® allows you to set up an interface that suits the user. As you become more familiar with using AutoCAD®, you will develop an interface configuration that best suits you.

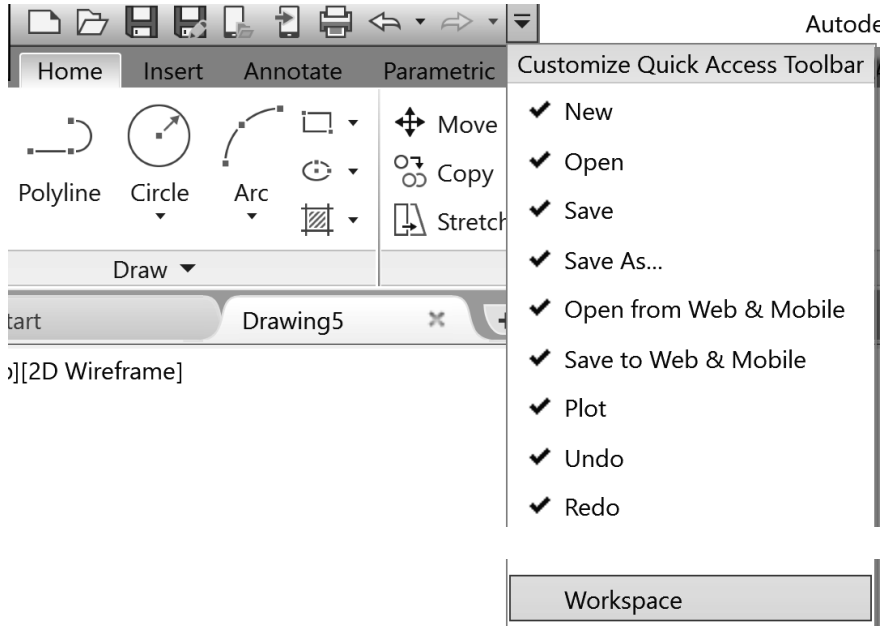
2.7.1) Setting up the user interface

- 1) View the *User Interface* video and read sections 2.1) to 2.6).
- 2) Start AutoCAD 2019®.
- 3) When you first launch AutoCAD® you may or may not enter a window that looks like what is shown in the following illustration. This will depend on whether you are using a personal copy or a school computer where the settings have already been adjusted. If you **don't see** this window perform the following steps.
 - a) **Command: startup**
 - b) Enter new value for startup <1>: **3**
 - c) Close down AutoCAD® and restart the program.

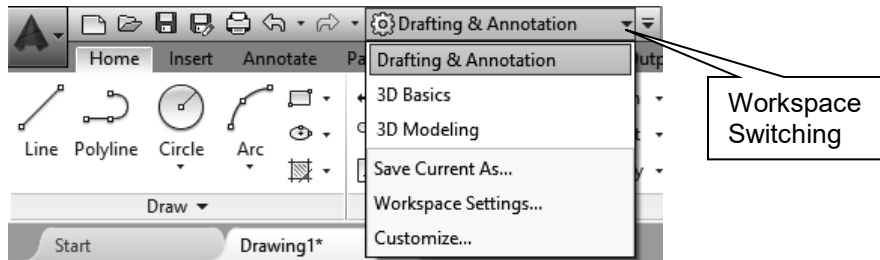


- 4) The above window is where you can access instructional videos and create an account which allows you to save, share and edit drawings online. Within the *Connect* area, click on **Sign in...**. Create an account and enter the necessary information.
- 5) Return to the AutoCAD start window and select **Start Drawing**.

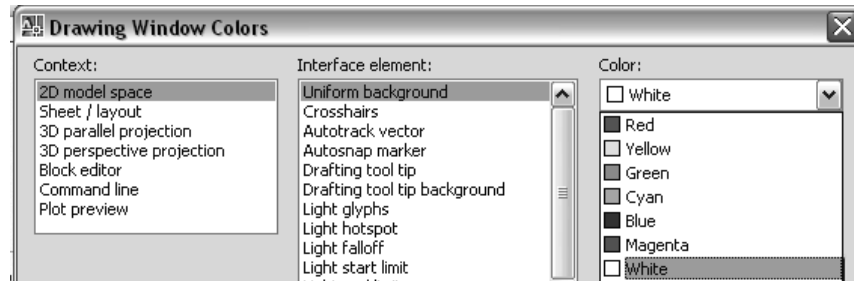
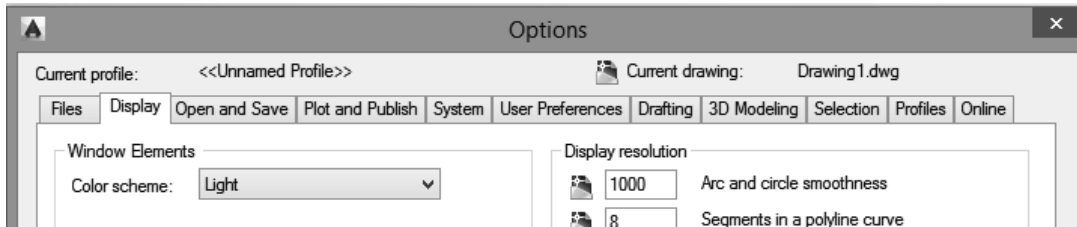
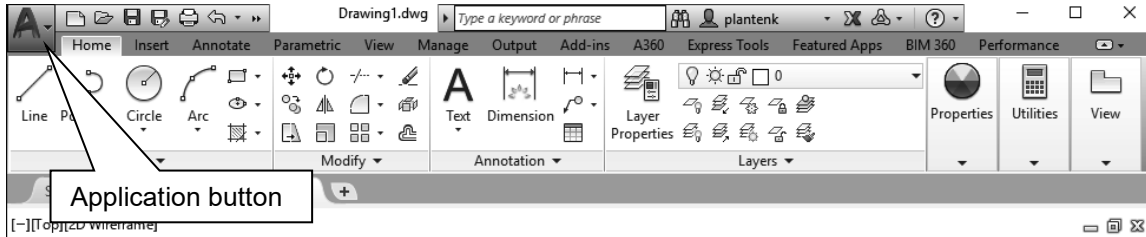
- 6) Activate the workspace selection menu.
 - a) Locate the *Quick access* menu up at the top left of the program window.
 - b) Click on the *Customize* arrow and select **Workspace**. The *Workspace* pulldown menu will be added to the *Quick access* menu.



- 7) Select the **Drafting & Annotation** workspace (if not already selected).




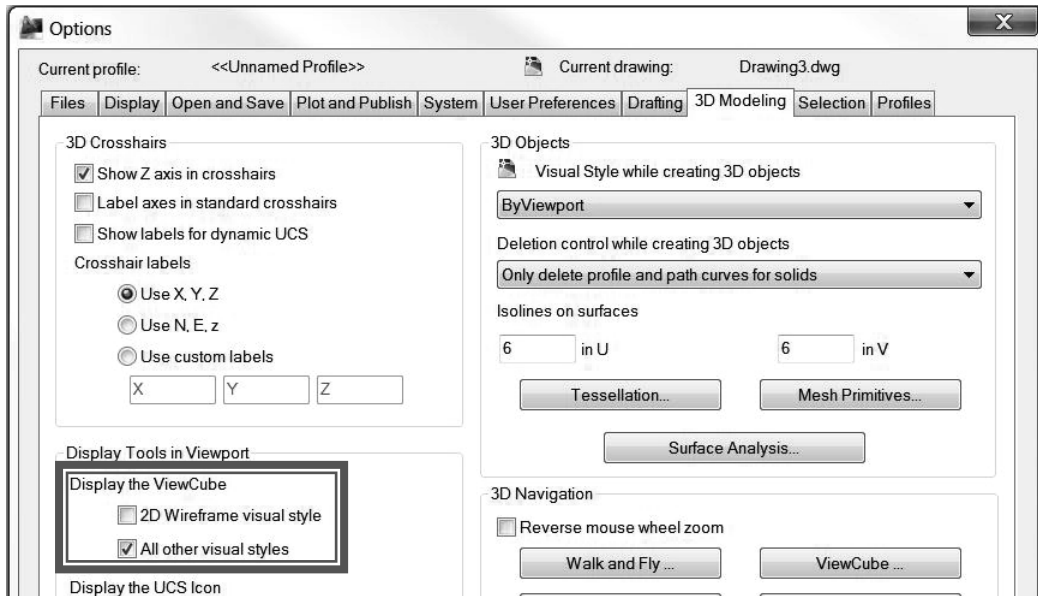
- 8) Change the color scheme of your workspace to white.
 - a) Application button: Select the **Options** icon at the bottom of the menu.
 - b) Options window – Display tab – Color scheme area: Select the **Light** option.
 - c) Options window – Display tab: Select the **Colors...** button.
 - d) Drawing Window Colors window:
 - i. Context: **2D model space**
 - ii. Interface element: **Uniform background**
 - iii. Color: Select **White**.
 - e) Drawing Window Colors window: Select the **Apply & Close** button.
 - f) Options window: Select **OK**.





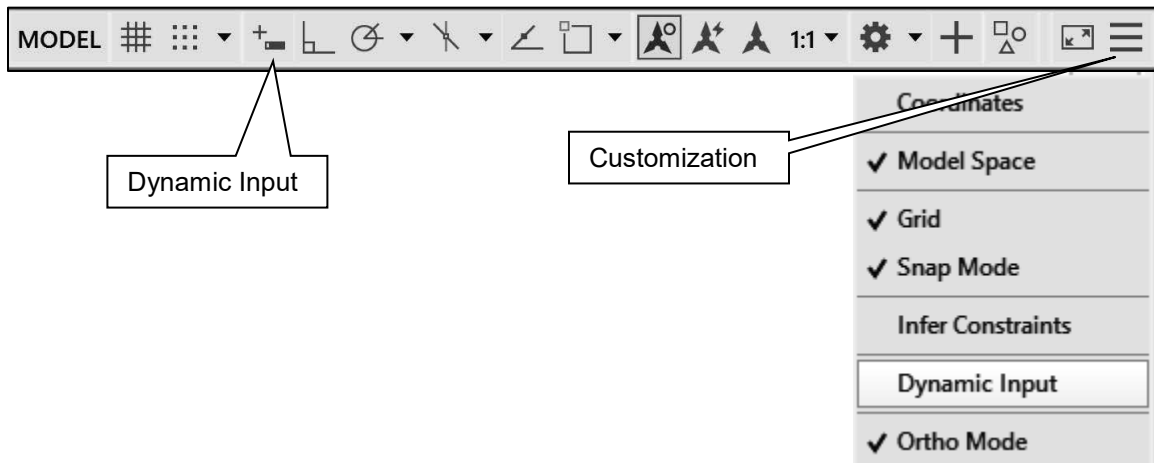
9) If you see the *View Cube* in the upper right of your drawing area, you may want to hide it. The *View Cube* is a navigation tool that allows you to switch between viewing directions. While this is very useful in 3D space, it is not very useful in 2D space.



- a) Application button : Select the **Options** icon at the bottom of the menu.
- b) Options window – 3D Modeling tab – Display Tools in Viewport area: Deactivate the **2D Wireframe visual style** check box.
- c) Options window: Select **OK**.



10) In the *Drawing Status bar*, deactivate the **Grid**  and **Dynamic Input** . The *Drawing Status bar* is located at the bottom/right of the drawing area. The command icons should look gray and not highlighted blue after they are deactivated. If the *Dynamic Input* command is not shown, click on the **Customization** icon and add it to the *Status bar*.



2.7.2) Starting a new drawing

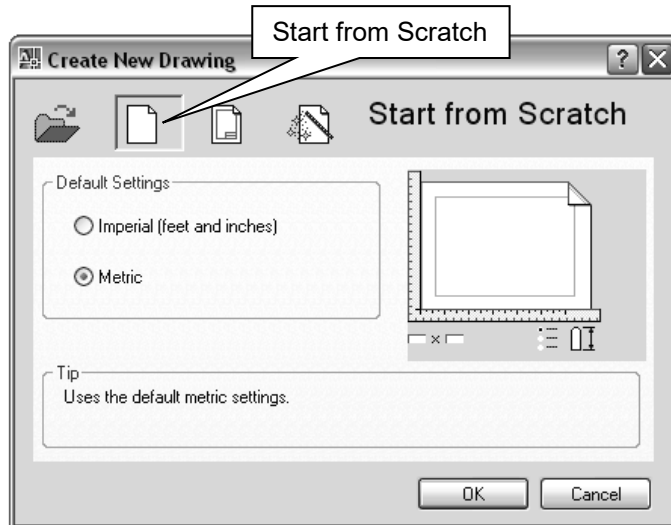
- 1) Enable the *Create New Drawing* window.
 - a) Command: **startup**
 - b) Enter new value for `STARTUP <0>`: **1**

- 2) Set the units for your drawing.
 - a) Quick Access toolbar: **New**



A *Create New Drawing* window will appear.

- b) Create New Drawing window: Activate the **Start from Scratch** button, select the **Metric** toggle and then select **OK**.



- 3) Enter the *Advanced Setup* wizard.
 - a) Quick Access toolbar: **New**



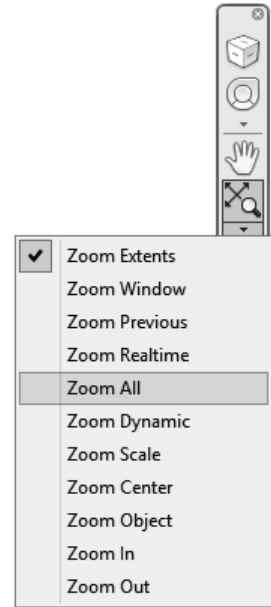
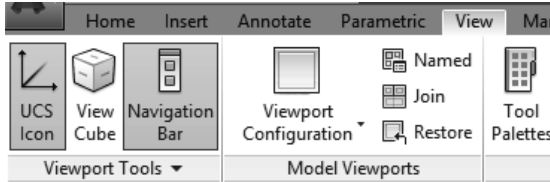
- b) Create New Drawing window:
 - i. Activate the **Use a Wizard** button.
 - ii. Select a Wizard field: Select **Advanced Setup**.
 - iii. **OK**.






- 4) Step through the *Advanced Setup* wizard and set the following parameters.
 - a) Set your *Units* to **Decimal** with a precision of **0**.
 - b) Set your *Angle* to **Decimal Degrees** with a precision of **0**.
 - c) Set your *Angle Measurement* to **East**.
 - d) Set your *Angle Direction* to **Counter-Clockwise**.
 - e) Set your *Area* to a width of **297 mm** and a length (height) of **210 mm**.
 - f) When you are done setting your parameters, select **Finish**.

Problem? If your paper size is 12 x 9, you need to re-enter the *Create New Drawing* window and select *Metric* units. (Repeat steps 2 & 3)

- 5) Visualize your drawing area.
- a) Locate the *Navigation Bar* icon in the *View Ribbon* tab. Activate and deactivate the *Navigation bar*. The *Navigation bar* is located on the right side of the drawing area. Watch it appear and disappear as you click on the *Navigation bar* icon. End with the ***Navigation Bar* activated or showing**.



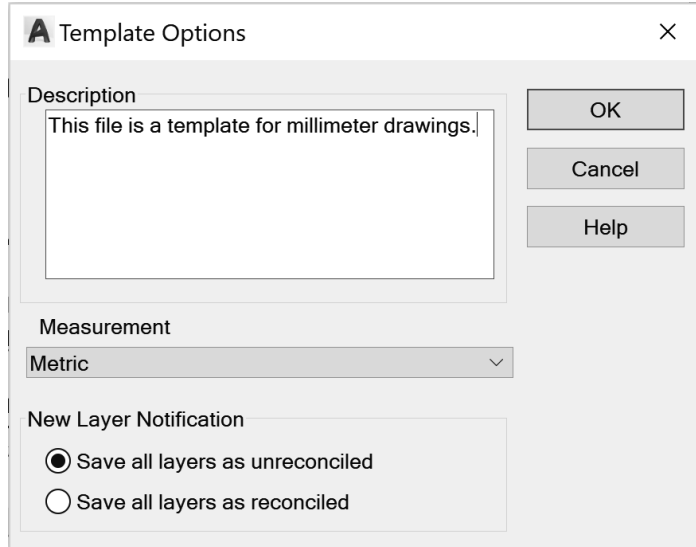
- b) Navigation bar: Zoom All. (You may also *Zoom Extents* by double clicking your mouse wheel.)
 - c) Command: grid
 - d) Specify grid spacing(X) or ON/OFF/Snap/Major/aDaptive/Limits/Follow/Aspect] <10>: **5**
 - e) Notice that your *Grid* icon  in the *Drawing Status Bar* has become active.
 - f) Navigation bar: Zoom Realtime
 - i. Move your mouse to the center of the drawing area. Click and hold your left mouse button and move the mouse down until you can see the edges of your grid and then release your left mouse button.
 - ii. **Esc** to exit the command.
 - g) Zoom using the mouse.
 - i. If your mouse has a wheel, take the cursor to the center of the drawing area and roll the wheel to zoom in and out.
 - h) Navigation bar:  Pan
 - i. Move your mouse to the center of the drawing area. Click and hold your left mouse button and move the mouse around the drawing area. When you are done, release your left mouse button.
 - ii. **Esc**
 - i) Pan using the mouse.
 - i. If your mouse has a wheel, take the cursor to the center of the drawing area, press and hold the wheel and move your mouse around to pan.
 - j) **Zoom All**
- 6) Turn off your **GRID** .

2.7.3) Saving your drawing as a template

- 1) Save your drawing as an AutoCAD® drawing template that you will be able to use again.



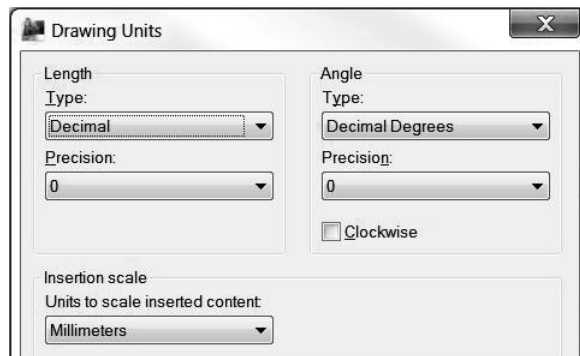
- a) Application button: **Save As** (Ctrl+Shift+S)
- b) Save Drawing As window: Select a file type of **AutoCAD Drawing Template (*.dwt)**.
- c) Save Drawing As window: Select the location where you want to save the file.
- d) Save Drawing As window: Name your file **set-mm.dwt**, and select **Save. A Template Description** window will appear.
- e) Template Options window: Describe your template file in the **Description** field, select your **Measurement** units and then select **OK**.



Note: You may save your drawings and templates on your hard drive, portable memory device, or to the web using the *Save to Web & Mobile* command.

2.7.4) Checking your drawing parameters

- 1) Check your drawing units.
 - a) Command: **un** or **units**
 - b) Drawing Units window: Make sure that your units and precision are correct and then select **OK**.



- 2) Check the size of your drawing.
 - a) Command: **limits**
 - b) Specify lower left corner or [ON/OFF] <0,0>: **Enter**
 - c) Specify upper right corner <280,216>: **Enter** (If needed, you can change your drawing size by entering larger or smaller numbers in this area.)

- 3) Quick Access toolbar: **Save**

Exercise 2.7-1: User interface and startup

Create and save a **set-inch.dwt** drawing template with the following drawing parameters.

- Start a new drawing and set your units to **Inches**.
- Select **OK** in the *Create New Drawing* window.
- Start a new drawing. Inch units should already be selected.
- Set up your drawing parameters.
 - Set your precision to **0.00**.
 - Set your drawing size to **11** by **8.5**.
- Use the command **UNITS** to check your settings.

2.8) COORDINATES

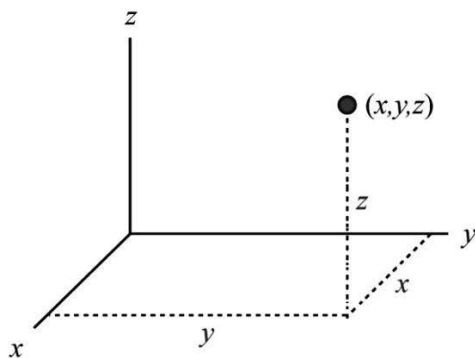
2.8.1) Cartesian and polar coordinates

Objects created in AutoCAD such as lines and circles are defined by coordinate points. The user may specifically state the defining coordinate points using the *Command* window or use the geometry of other objects to define the coordinate points. The two main coordinate systems used by AutoCAD are the Cartesian coordinate system (x,y) and the polar coordinate system (r,θ). Let's review both.

2.8.2) Cartesian coordinate system

The Cartesian coordinate system consists of three mutually perpendicular axes. The axes are labeled the x -axis, the y -axis and the z -axis as shown in Figure 2.8-1. The point where all the axes meet is called the origin. The origin is defined to be the zero location (0,0,0). The location of any point in space can be identified by an x position, a y position and a z position relative to the origin. Since this chapter only deals with AutoCAD's 2-D capabilities, a point in space will be defined by an x and y value. The z value will always remain zero.

3-D Cartesian coordinate system



2-D Cartesian coordinate system

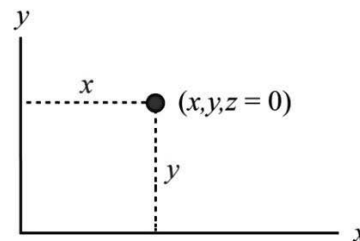


Figure 2.8-1: The Cartesian coordinate system

To illustrate the use of Cartesian coordinates in AutoCAD, let's look at the sequence of commands used to draw the line shown in Figure 2.8-2. As shown, the line lies in the x - y plane. It starts at the point $x = 2$ and $y = 1$. It ends at the point $x = 6$ and $y = 3$. This is the command sequence used to create this line in AutoCAD:

- **Command:** `l` or `line`
 - `LINE` Specify first point: **2,1**
 - Specify next point or [Undo]: **6,3**
 - Specify next point or [Undo]: Press **Enter** to end the `LINE` command.
-

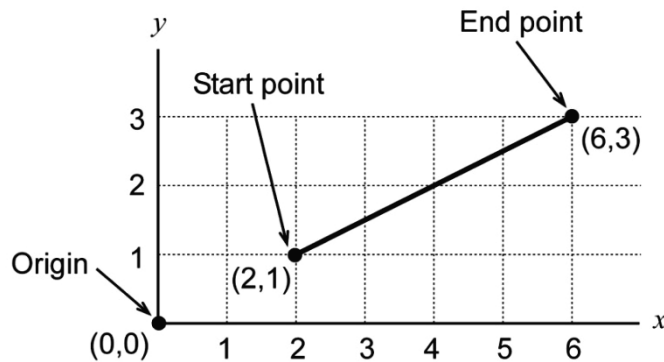


Figure 2.8-2: A line defined by Cartesian coordinate points

2.8.3) Polar Coordinates

The 2-D polar coordinate system consists of two mutually perpendicular axes. The axes are labeled the x -axis and the y -axis as shown in Figure 2.8-3. The point where the axes meet is called the origin. The origin is defined to be the zero location (0,0). The location of any point in space can be identified by the radial coordinate r and the angular coordinate θ . The radial coordinate is the shortest measured distance between the origin and the point under consideration, and the angular coordinate θ is the angle between the radial coordinate line and the x -axis.

The angular coordinate θ is measured positive counterclockwise starting at the positive x -axis. Therefore, if a point lies on the positive x -axis, its angular coordinate is zero. If a point lies on the y -axis, its angular coordinate is 90 degrees. The angular coordinate directions are illustrated in Figure 2.8-3.

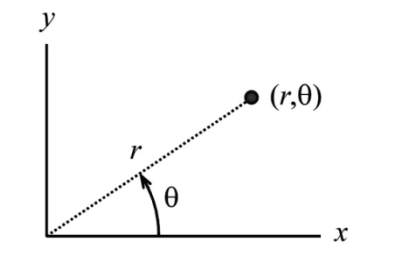
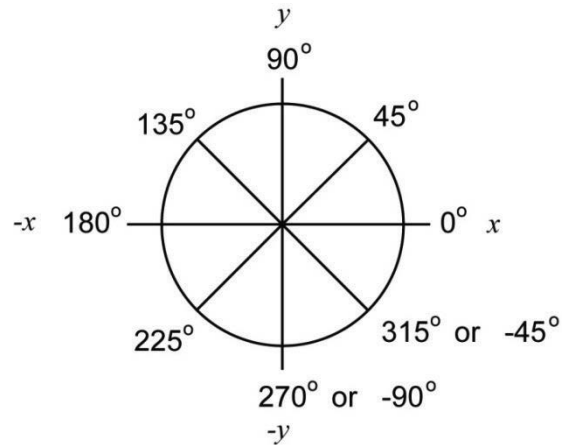
2-D Polar Coordinate SystemPolar Coordinate Directions

Figure 2.8-3: The polar coordinate system.

To illustrate the use of polar coordinates in AutoCAD®, let's look at the sequence of commands used to draw the line shown in Figure 2.8-4. As shown, the line lies in the x - y plane. It starts at the point $r = 2.5$ and $\theta = 60$ degrees. It ends at the point $r = 7$ and $\theta = 10$ degrees. This is the command sequence used to create this line in AutoCAD®:

- Command: **l** or **line**
- **LINE** Specify first point: **2.5<60**
- Specify next point or [Undo]: **7<10**
- Specify next point or [Undo]: Press **Enter** to end the **LINE** command.

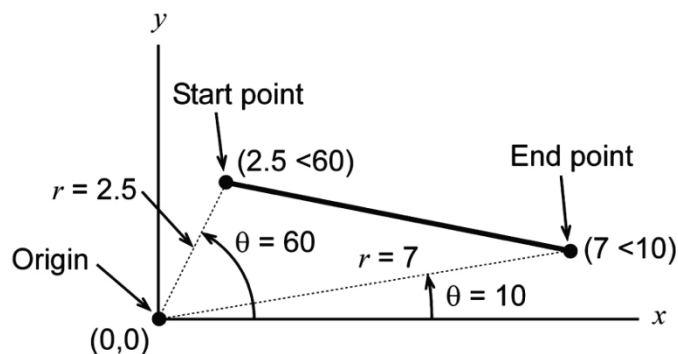


Figure 2.8-4: A line defined by polar coordinate points

2.8.4) Relative coordinates

Many times the start point of a line is unknown or the length and angle of the line is known but not the coordinate for the end point. Therefore, AutoCAD® allows you to enter coordinate points that are relative to the last point entered and not relative to the

origin. It is like making the last point entered a temporary origin. The symbol @ is placed before the coordinate point if it is to be relative to the last point entered.

To illustrate the use of relative coordinates, let's look at the sequence of commands used to draw the two lines shown in Figure 2.8-5. This is the command sequence used to create these lines in AutoCAD®:

Line 1

- **Command:** `l` or `line`
- `LINE` Specify first point: Using the mouse, select a point anywhere in the drawing area.
- Specify next point or [Undo]: `@4,2`
- Specify next point or [Undo]: Press **Enter** to end the `LINE` command.

Line 2

- **Command:** `l` or `line`
- `LINE` Specify first point: `2,1`
- Specify next point or [Undo]: `@5<25`
- Specify next point or [Undo]: Press **Enter** to end the `LINE` command.

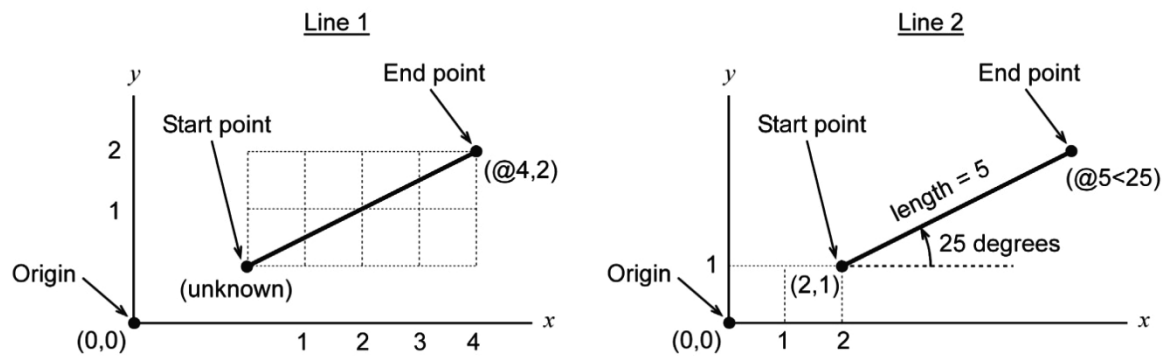


Figure 2.8-5: Lines defined by relative coordinates

2.8.5) World coordinate system (WCS) and user coordinate system (UCS)

AutoCAD® has two different coordinate systems: the *world coordinate system* (WCS), which is fixed and cannot be moved, and the *user coordinate system* (UCS), which is movable. The WCS origin is always located in the same place. When opening a new drawing, it is located in the bottom left corner of the drawing window. The UCS origin may be translated and its axes rotated using the commands found in the *UCS* panel. The UCS is very useful when drawing objects that are relative to a point on an existing object. Figure 2.8-6 shows the coordinate axes for the WCS and a translated and rotated UCS. Notice that the WCS coordinate axes have a box at the origin and the UCS does not. Visually this is how you can tell them apart. The properties of the coordinate axes may be changed to suit the user's preferences.

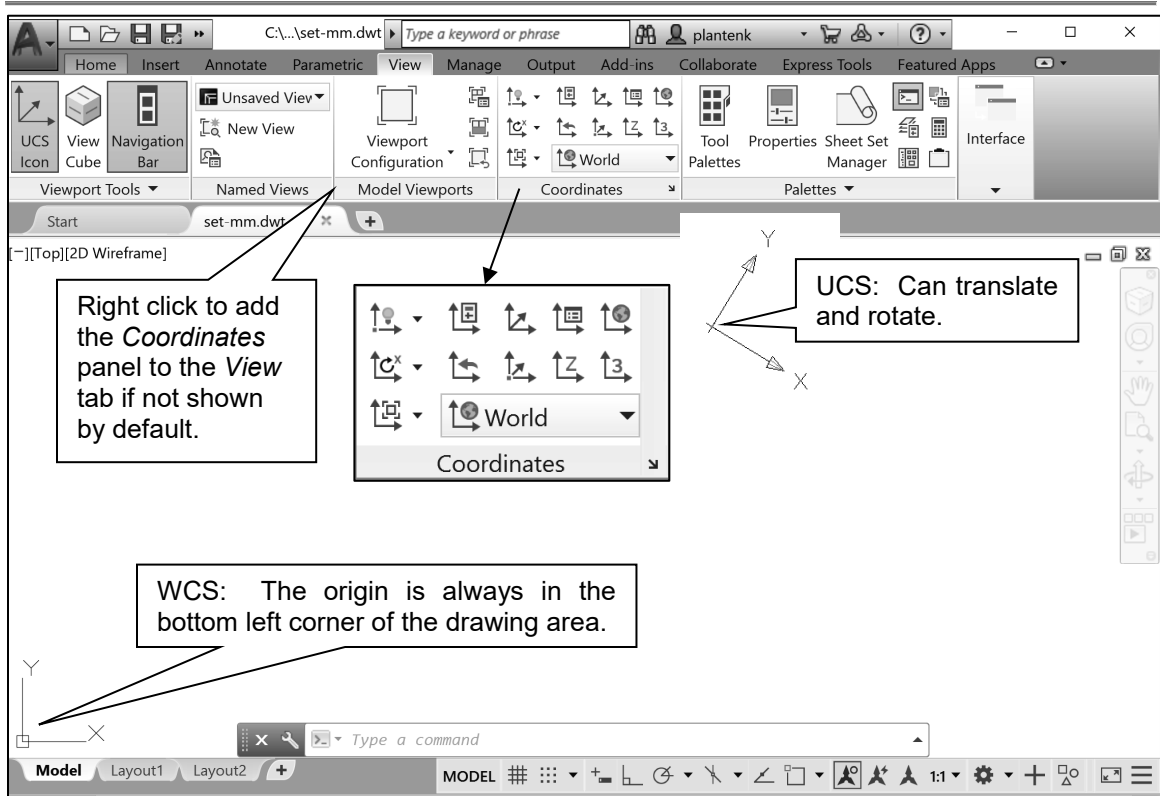

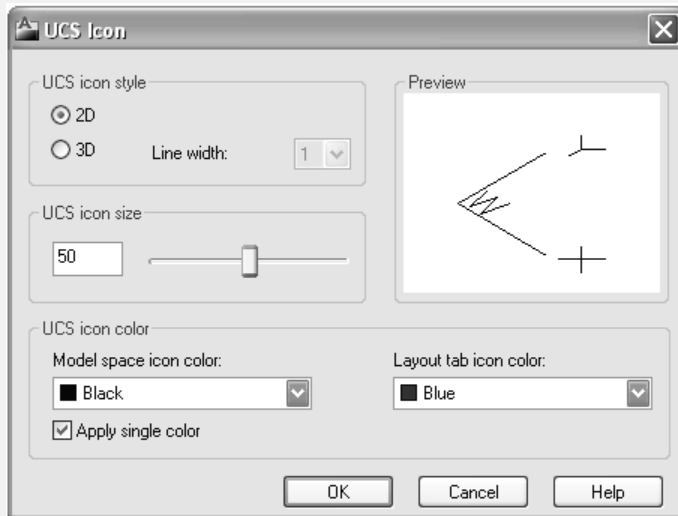


Figure 2.8-6: AutoCAD's coordinate axes

Changing properties of the UCS icon

- 1) View tab - Coordinates panel:  (UCS icon, Properties...)
- 2) Within the UCS Icon window, you can change the UCS icon style, its size and its color.



2.8.6) The coordinates panel

The *Coordinates* panel, shown in Figure 2.8-7, is located in the *View* tab and contains commands that allow you to switch back and forth between the WCS and the UCS. There are also several commands that manipulate the UCS.

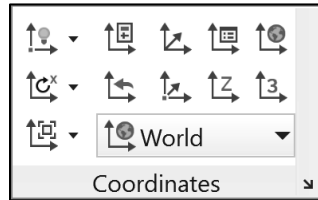





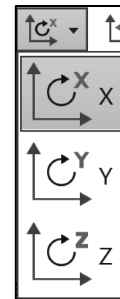



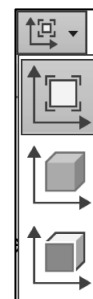
Figure 2.8-7: The *Coordinates* panel


The *Coordinates* panel contains the following commands:





- **UCS:**  Manages the user coordinate system.
- **Named:**  Manages the defined user coordinate systems.
- **World:**  Activates the WCS.
- **Previous:**  Restores the previous UCS.
- **Origin:**  Allows you to change the origin of your UCS.
- **Rotate pull-down:** This command rotates the UCS coordinate system a specified number of degrees about the *x*, *y* or *z*-axis.



- **Z Axis Vector:**  Defines a UCS with a specified positive *z*-axis.
- **Align pull-down:** Allows you to align the UCS.
 - **View:** Aligns the *xy*-plane of the UCS with the screen.
 - **Object:** Aligns the UCS with a selected object.
 - **Face:** Aligns the UCS with the face of a 3D object.



- **3 Point:**  Defines a UCS using 3 points: an origin, a point on the *x*-axis and a point on the *y*-axis.

- Show UCS Icon at Origin:  Displays the UCS icon at the origin only.
- Show UCS icon:  Allows you to show the coordinate axis.
- Hide UCS icon:  Allows you to hide the coordinate axis.
- UCS Icon Properties:  Allows you to change the look of the coordinate axis.

2.8.7) Coordinate position

The coordinate position of your cursor is displayed at the left of the *Status Bar*. It is not visible by default. You need to select **Coordinates** in the *Status Bar Customization*. It dynamically states the x, y and z position of your cursor. To turn the coordinate display on and off, just click on the area where the coordinate numbers are displayed. The command **COORDS** controls whether the coordinate display is on or off and whether it displays absolute x,y,z coordinates or relative r,θ,z coordinates during a command. Figure 2.8-8 shows the coordinate display for the cursor position in both the x,y,z coordinates and the r,θ,z coordinates.

- Coords = 0: The coordinate display is off.
- Coords = 1: The coordinate display is on and reads absolute x,y,z coordinates at all times.
- Coords = 2: The coordinate display is on and reads absolute x,y,z coordinates before a command and relative r,θ,z coordinates during the command.

x,y,z coordinate display



r,θ,z coordinate display



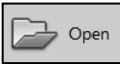
Figure: 2.8-8: Coordinate display

2.9) WCS/UCS COORDINATE SYSTEMS TUTORIAL

The objective of this tutorial is to familiarize the user with AutoCAD's coordinate systems. AutoCAD® has two different coordinate systems: the world coordinate system (WCS), which is fixed and cannot be moved, and the user coordinate system (UCS), which may be translated and rotated.


2.9.1) Coordinates

- 1) View the *UCS* video and read section 2.8).
- 2) Open your **set-inch.dwt** drawing template (created in Exercise 2.7-1).

- a) Application button:  (Ctrl+O)
- b) Select File window: Select the file type of **AutoCAD Drawing Template (*.dwt)** and then the location of the file (e.g., on your hard drive, online, etc.) and select the file **set-inch.dwt**. Select **Open** after you have selected the file.

- 3) Save your template file as a drawing file named **WCS-UCS Coord Tut.dwg**.


- a) Application button: **Save As...**
- b) Save Drawing As window:
 - i. Select a file type of **AutoCAD 2018 Drawing (*.dwg)**.
 - ii. Name your file **WCS-UCS Coord Tut.dwg**.
 - iii. Select a location for your file.
 - iv. Select **Save**.


- 4)  **(Zoom All)** - (**Z** Enter **a** Enter)

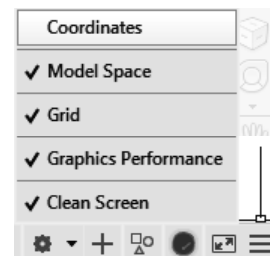
- 5) Show your coordinate axis at the origin.

- a) View ribbon tab – Coordinate panel: 

Problem? If your *Coordinate* panel is not shown, right click on one of the panel names, select **Show panel** and then select **Coordinate**.

- 6) **Pan**  your drawing a little to the right and a little up and **Esc** to exit the command.

- 7) Activate your coordinate readout. Select **Customize**  in the *Status Bar* and then **Coordinates**. A coordinate readout will appear next to the *Status Bar*.




- 8) Move your cursor around the graphics screen and watch the coordinate display change as your cursor changes locations.



- 9) Find the origin. It should be located at the intersection of the coordinate axes.
 10) Turn your coordinate readings off by clicking on the coordinate display reading.
 11) Turn your coordinate display back on.

2.9.2) Change to the user coordinate system (UCS)

- 12) Change the location of your **UCS** (User Coordinate System) origin to the middle of your drawing area.



- a) View ribbon tab - Coordinate panel:  (UCS - ORIGIN)
 b) UCS Specify new origin point <0, 0, 0>: Click the left mouse button somewhere in the middle of the drawing window. (Notice that the coordinate axis moves to the new origin point and the **w** or box at the origin disappears.)

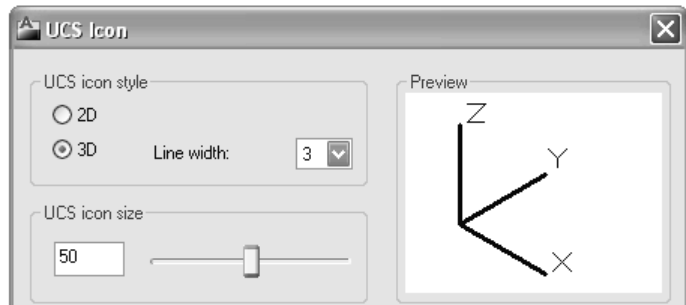
- 13) Find the origin.

- 14) Change back to the WCS (World Coordinate System).

- a) View ribbon tab - Coordinate panel: 

- 15) Change the appearance of the UCS.

- a) View ribbon tab - Coordinate panel:  (UCS icon, Properties...)
 b) UCS Icon window: Select the *UCS icon style* to be **3D** and change the coordinate axis line width to **3**.
 c) UCS Icon window: **OK**
 d) View ribbon tab - Coordinate panel:  (UCS icon, Properties...)
 e) UCS Icon window: Select the *UCS icon style* to be **2D**.
 f) UCS Icon window: **OK**



2.9.3) Coordinate display and drawing

- 1) Change the value of your **COORDS** system variable so that the coordinate display will read absolute x, y, z coordinates.
 - a) **Command:** **coords**
 - b) Enter new value for COORDS <2>: **1**

- 2) Draw a **Line**, but only enter the first point.
 - a) **Command:** **l** or **line**
 - b) Specify first point: Move your cursor to the center of your drawing area and click your left mouse button.
 - c) Specify next point or [Undo]: Move your cursor around the drawing area and watch the coordinate display change. Notice that the coordinate position of your cursor is relative to the origin of the WCS. Press **Esc** when done.

- 3) Change the value of your **COORDS** system variable to **2** so that the coordinate display will read relative polar coordinates (r, θ).

- 4) Draw a **Line**, but only enter the first point.
 - a) **Command:** **l** or **line**
 - b) Specify first point: Move your cursor to the center of your drawing area and click your left mouse button.
 - c) Specify next point or [Undo]: Move your cursor around the drawing area and watch the coordinate display change. Notice that the coordinate position of your cursor is relative to the first point of the line. Press **Esc** when done.


2.9.4) Drawing using the UCS coordinate system


- 1) Draw a **REctangle**, near the center of the graphics screen, which is 4 inches long and 3 inches high.
 - a) **Command:** **rec** or **rectangle**
 - b) Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: Select a point near the center of your drawing area.
 - c) Specify other corner point or [Area/Dimensions/Rotation]: **d**
 - d) Specify length for rectangles <10.00>: **4**
 - e) Specify width for rectangles <10.00>: **3**
 - f) Specify other corner point or [Area/Dimensions/Rotation]: Move your mouse around the drawing screen. Notice that the rectangle flips from one position to the next. Click your left mouse button to select a position.

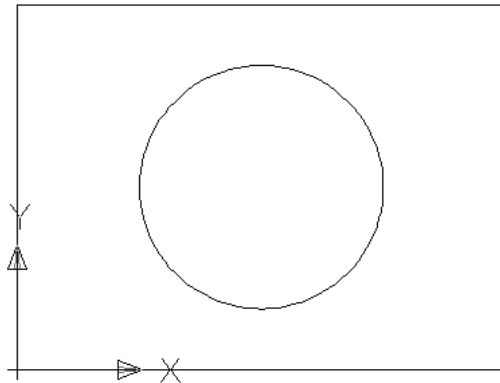
Problem?

- If your rectangle is too small, you are probably in a metric drawing.
- If your rectangle is off the screen, use **Pan** to bring it into view.

- 2) Turn your *Object Snap* on.

- a) Activate the **OBJECT SNAP** icon  in the *Drawing Status* bar. It should look highlighted.

- 3) Draw a **Circle** of radius 1 whose center is located at the geometric center of the rectangle.
 - a) Change the origin of your **UCS** to the bottom left corner of the rectangle.
 - b) View ribbon tab - Coordinate panel:  (UCS – ORIGIN)
 - c) UCS Specify new origin point <0, 0, 0>: Move your cursor near the bottom left corner of the rectangle. As soon as a box appears locating the corner, click your left mouse button.
 - d) Command: **c** or **circle**
 - e) CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **2,1.5**
 - f) Specify radius of circle or [Diameter]: **1**



IMPORTANT! The UCS is used to reference existing geometry. The UCS allows us to position an object relative to a particular feature on an existing object.

- 4) **Save** your drawing.

2.10) DRAWING USING COORDINATES TUTORIAL

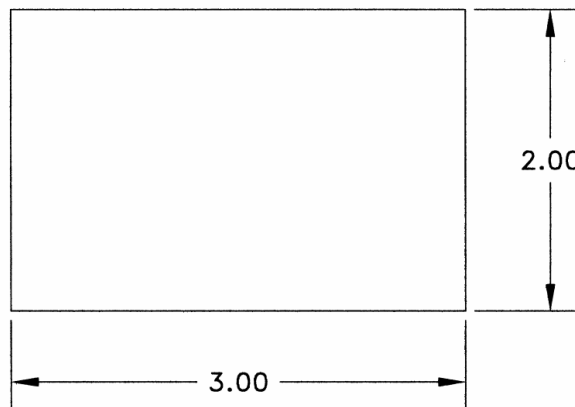
The objective of this tutorial is to familiarize the user with using both absolute and relative coordinates. You can enter the location and/or size of an object in AutoCAD by using either Cartesian (x,y) or polar coordinates ($r < \theta$). Both coordinate systems can be entered on an absolute basis (relative to the UCS or WCS origin) or a relative basis (relative to the last point entered). Relative coordinates are preceded by the symbol @.

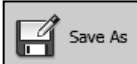
IMPORTANT! If you make a mistake when drawing, type **u** or **undo**. This will undo the last point or command that you entered. The UNDO command may be used repeatedly to undo several mistakes.

2.10.1) Drawing a rectangle using Cartesian coordinates

We will be drawing the rectangle shown using lines defined by absolute and relative Cartesian coordinates.

- 1) View the *Coordinates* video.
- 2) Open your **set-inch.dwt** drawing template.



- 3) Application button:  **Coordinate Tut – a.dwg**

- 4) Turn everything off in your *Drawing Status Bar* at the bottom of the drawing area. They should look gray.



- 5) Turn your **GRID** on and set the grid spacing to be 1 inch by 1 inch.

How?

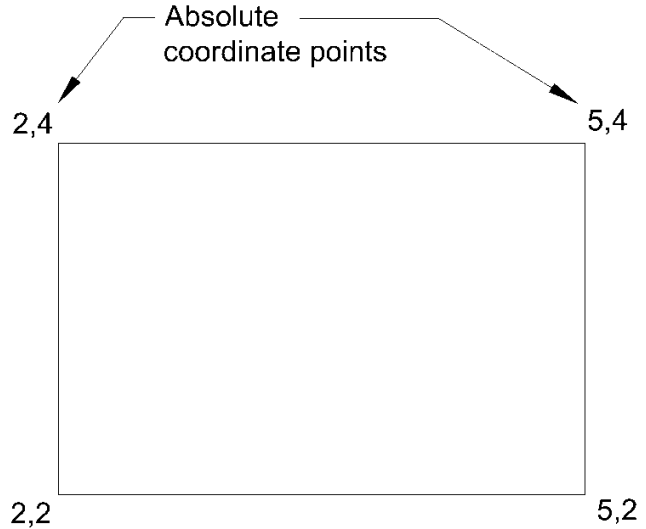
- a) Command: **grid**
- b) Specify grid spacing(X) or [ON/OFF/Snap/Major/aDaptive/Limits/Follow/ Aspect] <0.25>: **1**

Problem?

- If your grid is too dense, you are probably in a metric drawing.

- 6) **Zoom All.**

- 7) Draw a 3x2 rectangle using absolute coordinates.
- Command: I or line**
 - LINE Specify first point: **2,2**
 - Specify next point or [Undo]: **5,2**
 - Follow the figure and enter the remaining points. **Enter** when done.



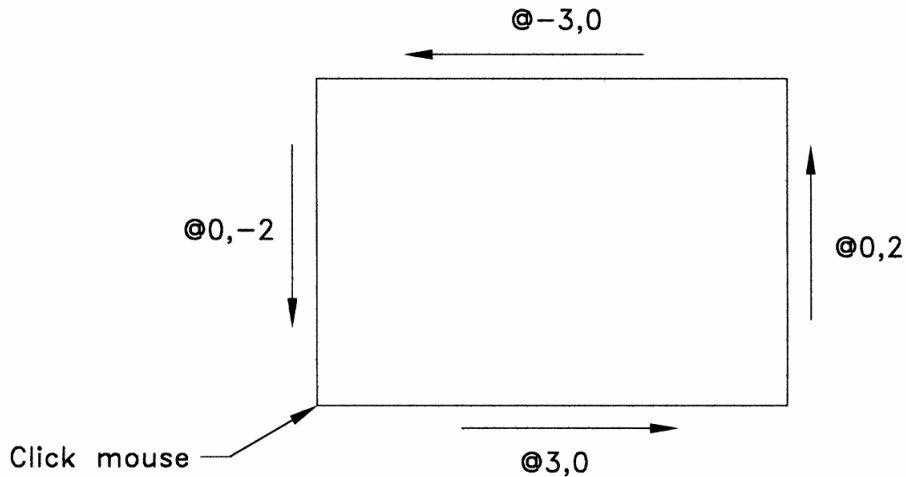
Note: An **absolute coordinate** is measured from the WCS origin. The figure shows the actual coordinate points used to create the rectangle.

Problem?

- If your rectangle is too small, you are probably in a metric drawing.

- Use the grid to confirm that the rectangle is 3 inches wide and 2 inches high.
- Draw the same rectangle using relative coordinates.

Note: A **relative coordinate** is measured from the last point drawn. The advantage of using relative coordinates is that you do not have to know where the original coordinate point is. We use the symbol **@** to tell AutoCAD that we are entering a relative coordinate.



- Command: I or line**
- LINE Specify first point: Select a point to the right of the first rectangle.
- Specify next point or [Undo]: **@3,0**
- Follow the figure and enter the remaining points. **Enter** when done.

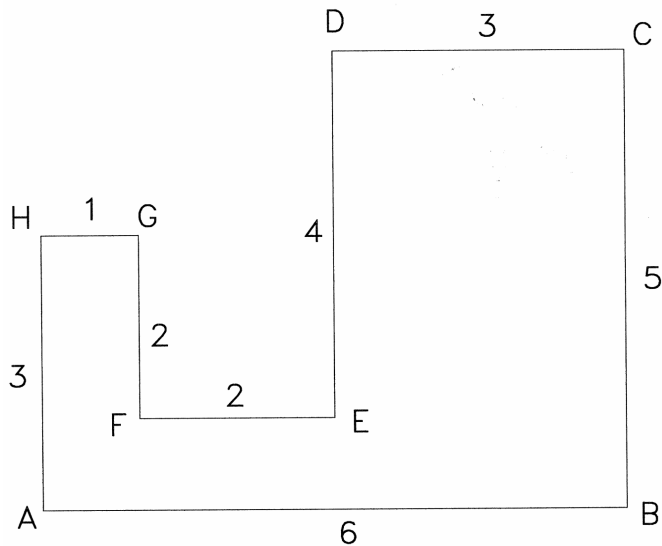


2.10.2) Drawing using relative polar coordinates

- 1) Review section 2.8.3).
- 2) **Erase** all the previously drawn objects.
 - d) Command: **e** or **erase**
 - e) Select object: **all**
 - f) Select object: **Enter**

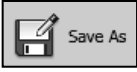


- 4) Draw the object shown below using relative polar coordinates.
 - a) Command: **l** or **line**
 - b) Specify first point: **3,2** (at "A") or choose any point near the bottom left corner of your drawing area.
 - c) Specify next point or [Undo]: **@6<0** (to "B")
 - d) Specify next point or [Undo]: **@5<90** (to "C")
 - e) Specify next point or [Close/Undo]: **@3<180** (to "D")
 - f) Specify next point or [Close/Undo]: **@4<270** (to "E")
 - g) Follow the figure and enter the remaining points. **Enter** when done.



2.10.3) Drawing using both Cartesian and polar coordinates

1) **Erase** all previously drawn objects.

2) Application button:  **Coordinate Tut – c.dwg**

3) Draw the object shown below using a combination of polar and Cartesian coordinates.

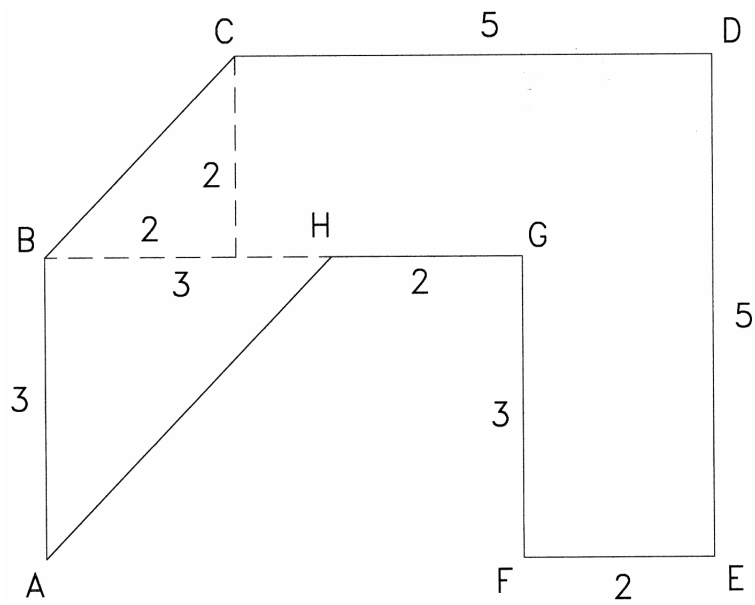
a) Command: **l** or **line**

b) Specify first point: **2,2** (at "A") or choose any point near the bottom left corner of your drawing area.

c) Specify next point or [Undo]: **@3<90** (to "B")

d) Specify next point or [Undo]: **@2,2** (to "C")

e) Follow the figure and enter the remaining points. **Enter** when done.



4) 

2.11) PRINTING

When you print or plot a drawing, you need to specify what region you would like to print. From within *Paper* space you usually print the *Layout*. However, in *Model* space, there are four different ways of selecting a region to print. The *Model* and *Paper* space will be discussed in “*Drawing Orthographic Projections in AutoCAD®*”.

1. Display: Prints everything that you can see, at the moment, in the drawing area.
2. Extents: Prints the minimum area which will include everything that is drawn.
3. Limits: Prints the area that you have defined as your drawing size.
4. Window: Prints the area that you select using a window.

Figure 2.11-1 and 2.11-2 show a drawing and the printing results using the first three region selection methods. The line around the drawing’s title block indicates the limits/drawing size.

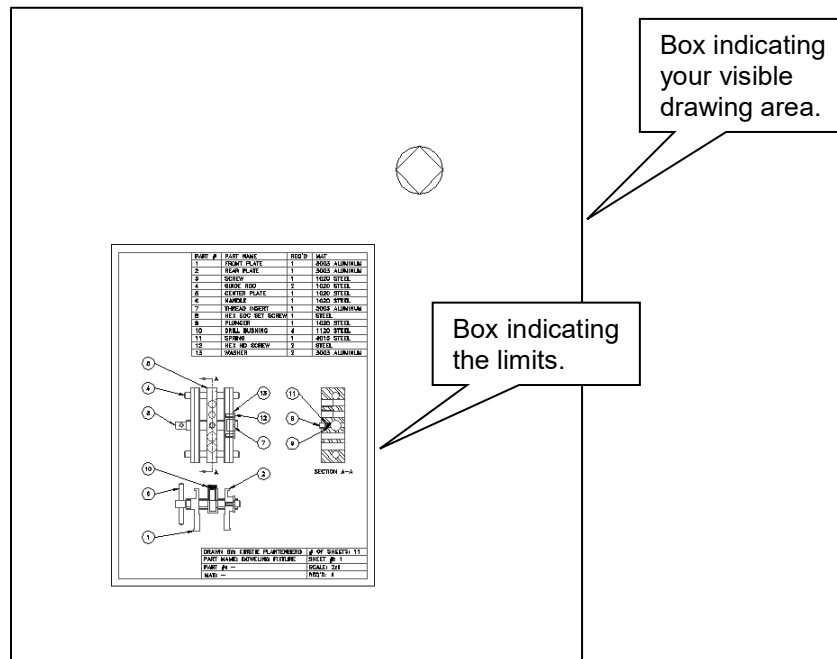
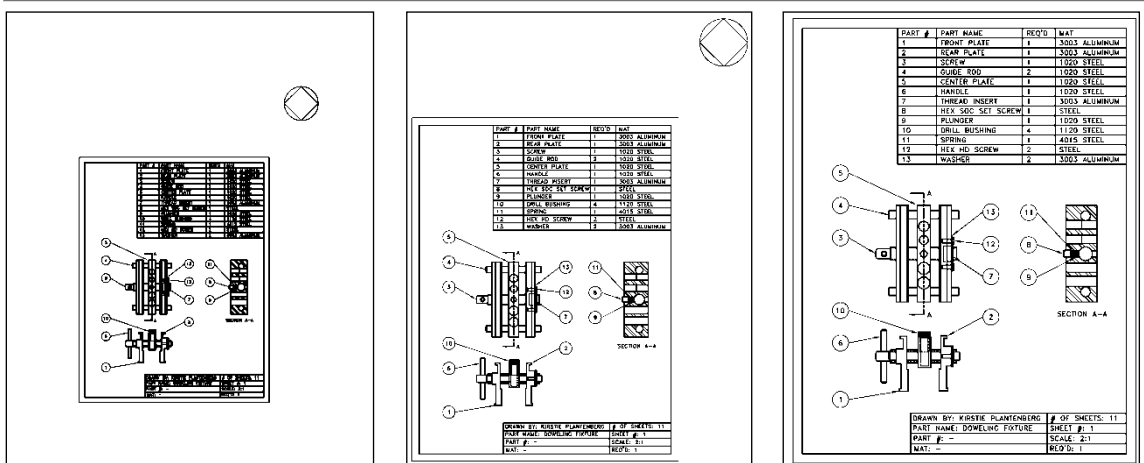


Figure 2.11-1: Drawing as displayed in AutoCAD®.



Printing the *Display*

Printing the *Extent*

Printing the *Limits*

Figure 2.11-2: Printing regions.

2.11.1) Page setup

Printing preferences for a particular drawing may be set in the *Page Setup - Model* window. These settings are drawing specific which means that they are stored in the drawing or template file and not in the program. To access the *Page Setup - Model* window select **Application button – Print – Page Setup...** and then click on the **Modify...** button. Figure 2.11-3 shows the *Page Setup - Model* window with the important features identified.

2.11.2) Plot

The *Plot – Model* window allows you to send your drawing to a printer or plotter. The default settings used to print your drawing are specified in the *Page Setup – Model* window. The *Plot – Model* window allows you to change some of these default settings without having to access the *Page Setup – Model* window again. To access the *Plot - Model* window select **Application button - Print - Plot...**

2.11.3) Printing to scale

Print scale expresses the ratio between the printed size of an object to its actual size. If a drawing is printed full-scale, it implies that a feature dimensioned as 1 inch measures 1 inch with a ruler on the printed drawing. This is referred to as a 1 to 1 scale. Printing full scale, in most cases, is difficult to achieve unless you have access to a large plotter. In a classroom setting, most engineering drawings are printed on a standard 8.5” x 11” sheet of paper regardless of the object’s size. The scale at which the part is printed should allow all details of the part to be seen clearly and accurately. **Even though a drawing may not be able to be printed full scale, they should always be drawn full scale in the CAD environment.**

Since it is impractical to print all drawings full scale, we employ printing to half scale, quarter-scale and so on. For example, if a drawing is printed half-scale, a feature

that is dimensioned 1 inch will measure 0.5 inch on the printed drawing. The scale at which the drawing is printed should be indicated on the drawing next to the text “SCALE” in the title block. On a drawing, half-scale may be denoted in the following ways.

1/2 or 1:2 or 0.5

Although it is nice to print to scale, the ASME standard states that no dimension should be measured directly from the printed drawing. For drawings that are not prepared to any scale, the word “NONE” should be entered after “SCALE” in the title block.

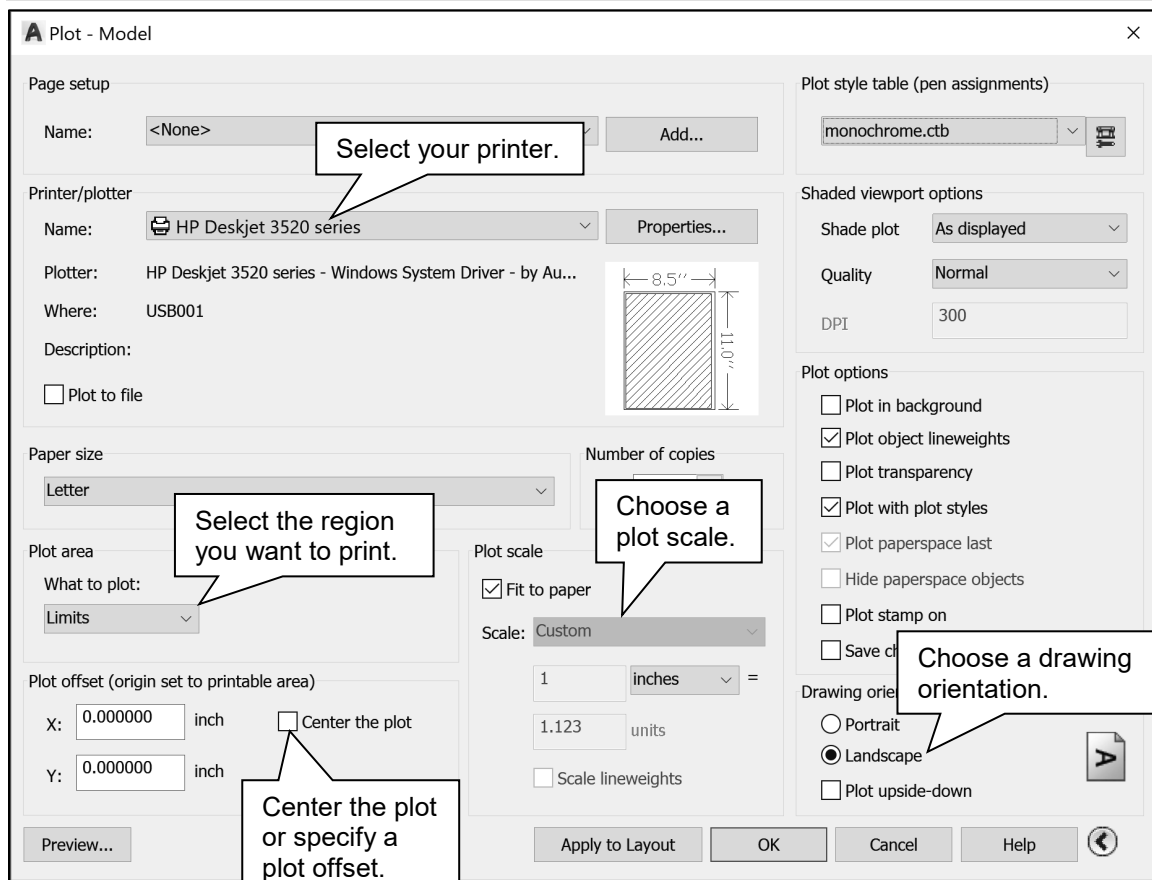


Figure 2.11-3: Page Setup - Model window

2.12) PRINTING TUTORIAL

The objective of this tutorial is to familiarize the user with different printing methods. We will set up a drawing that will be printed in three different ways: *limits*, *extents*, and *display*. These three methods may all give similar results or vastly different results depending on the drawing. We will also go through the steps used to print an object to scale.

2.12.1) Creating the drawing

- 1) Read section 2.11).
- 2) Open up your **set-mm.dwt** drawing template.
- 3) Check your paper size.

How?

- a) Command: **limits**
- b) Specify lower left corner or [ON/OFF] <0,0>: **Enter**
- c) Specify upper right corner <280,216>: **Enter** (Your paper size should be **297** by **210**. If it is not, set it to this size now. If it is 12 x 9 or 11 x 8.5, you are in an *inch* drawing.)

- 4)  **Printing Tut.dwg**

- 5) Turn your **OBJECT SNAP**  and your **DYNAMIC INPUT**  off (*Drawing Status bar*).

- 6) Enter your **World coordinate system** .


Where? View ribbon tab – *Coordinate* panel

- If the *Coordinate* panel is not shown, right click on one of the panel names and activate it.

- 7) Draw a **RECTangle** whose lower left corner is located at (0, 0) and is 297 mm long by 210 mm high (width). (This rectangle indicates the edge of the paper or limits.)

How?

- a) Command: **rec** or **rectangle**
- b) Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **0,0**
- c) Specify other corner point or [Area/Dimensions/Rotation]: **d**
- d) Specify length for rectangles <10>: **297**
- e) Specify width for rectangles <10>: **210**
- f) Specify other corner point or [Area/Dimensions/Rotation]:
Select a point in the middle of your drawing area.

8) . (**Zoom All**)

9) Draw a **Circle** of radius **50**, and center located at **350,100**. (Don't worry, this circle will be off of your paper.)

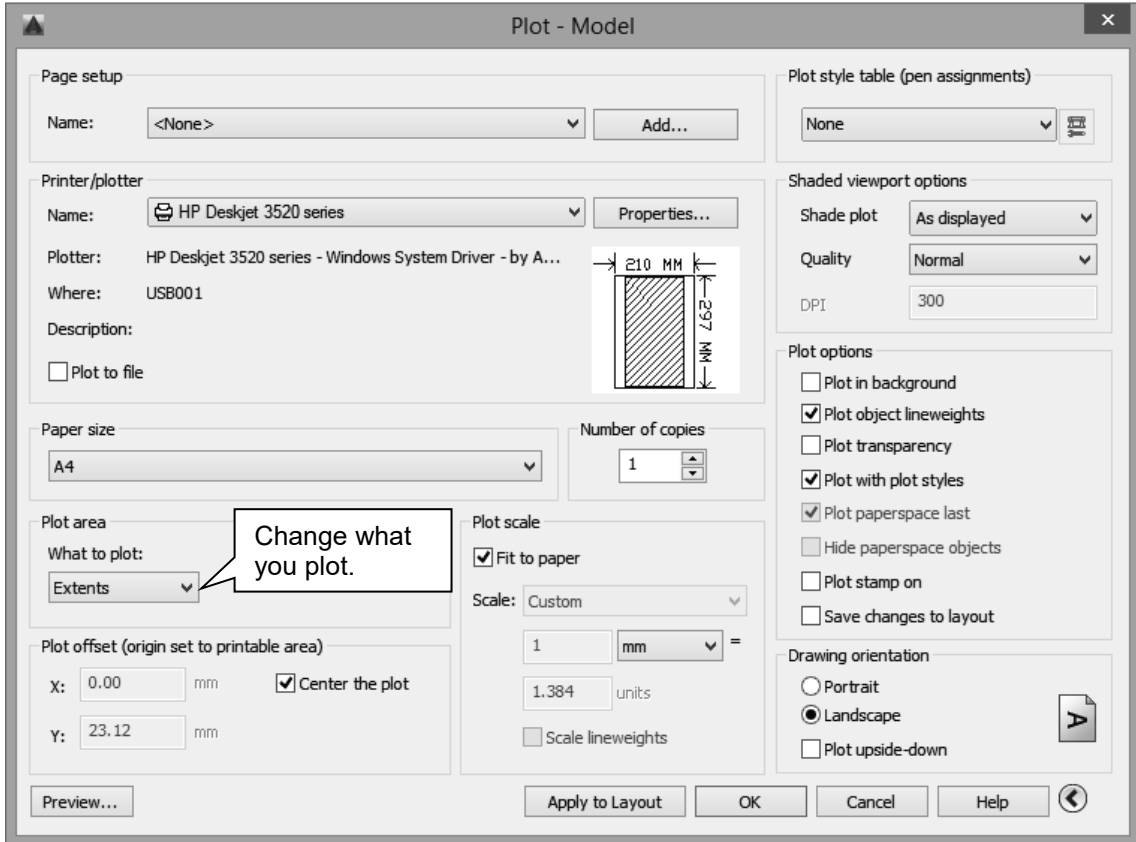
How?


- a) Command: **c** or **circle**
- b) Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **350,100**
- c) Specify radius of circle or [Diameter]: **50**

10) . (**Zoom All**) and  Save

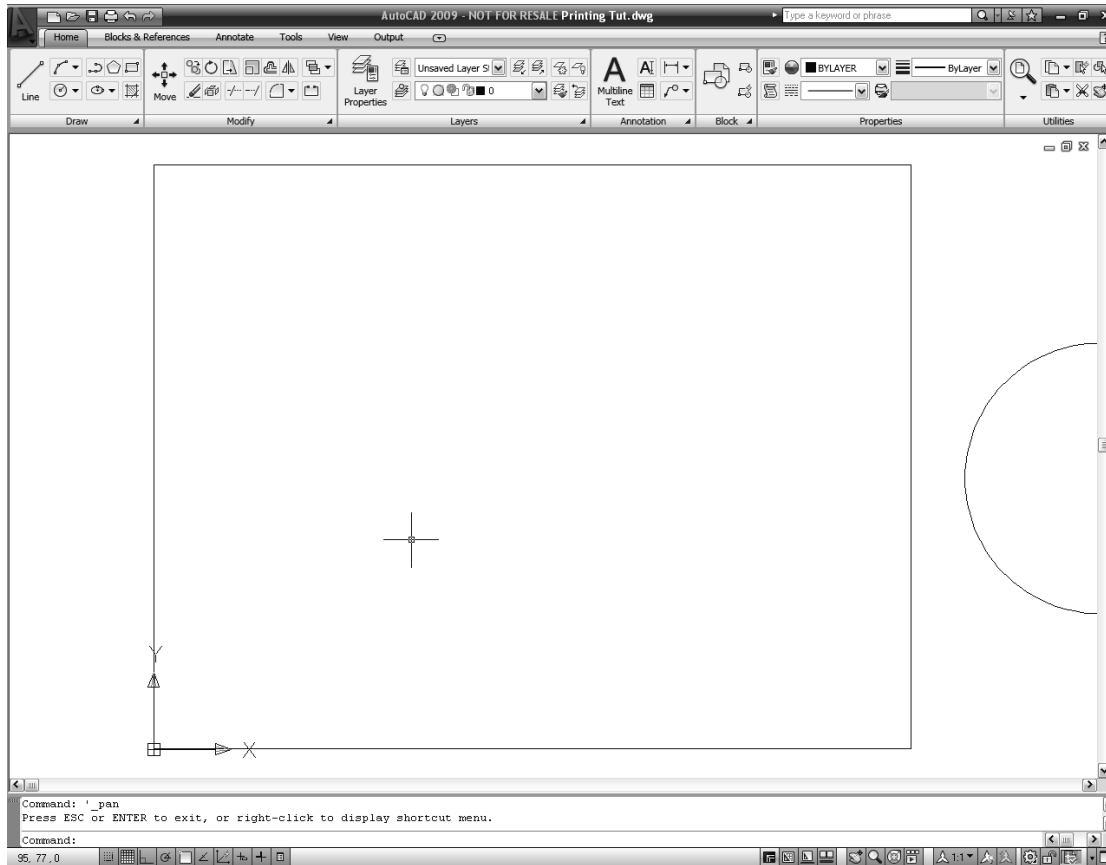
2.12.2) Printing

- 1) Application button: **Print – Plot...**
- 2) Plot – Model window: (See figure on next page.)
 - a) Choose a printer.
 - b) Activate the **Fit to Paper** toggle.
 - c) Select **Extents** as your plot area.
 - d) Activate the **Center the plot** toggle.
 - e) Select the **Preview...** button. Take note of what is being printed and then hit **Esc** to exit the plot preview.
 - f) Select **Display** as your plot area.
 - g) Select the **Preview...** button. Take note of what is being printed and then hit **Esc** to exit the plot preview.
 - h) Select **Limits** as your plot area.
 - i) Select the **Preview...** button. Take note of what is being printed and then hit **Esc** to exit the plot preview.
 - j) Select the **Cancel** button.



- 3) **Pan**  the drawing so that the rectangle is in the center of the drawing area and the circle is slightly off the screen.

Where? View ribbon tab – *Navigate* panel or use your mouse wheel.



- 4) Repeat step 2). and notice that this time the display printing method gives a different result.

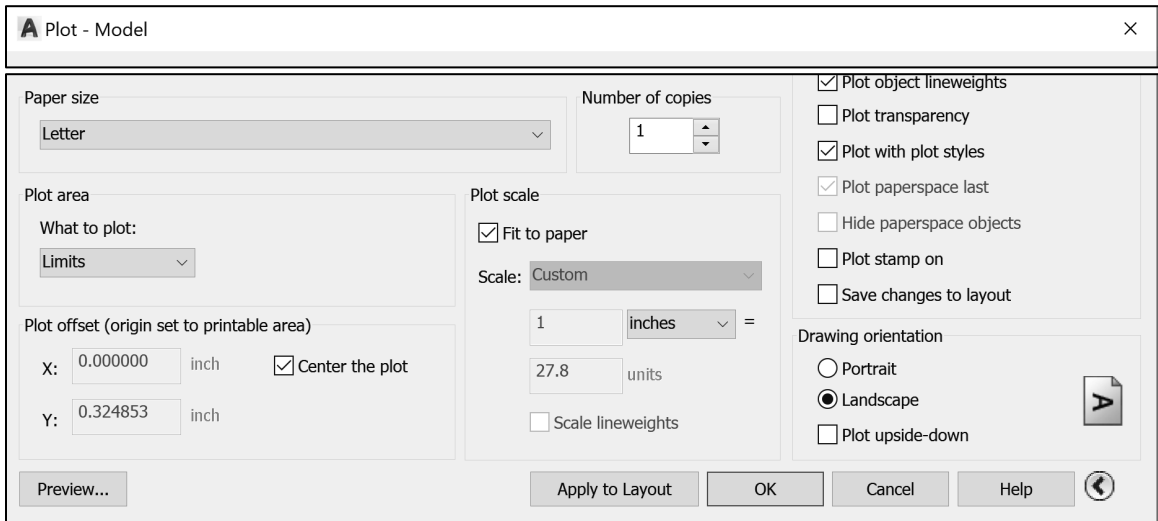
2.12.3) Printing to scale

You may find a situation where you need to print your drawings to scale. This means that the drawing must be printed at exactly the size that it was drawn, or at a specific ratio or scale to the exact size.

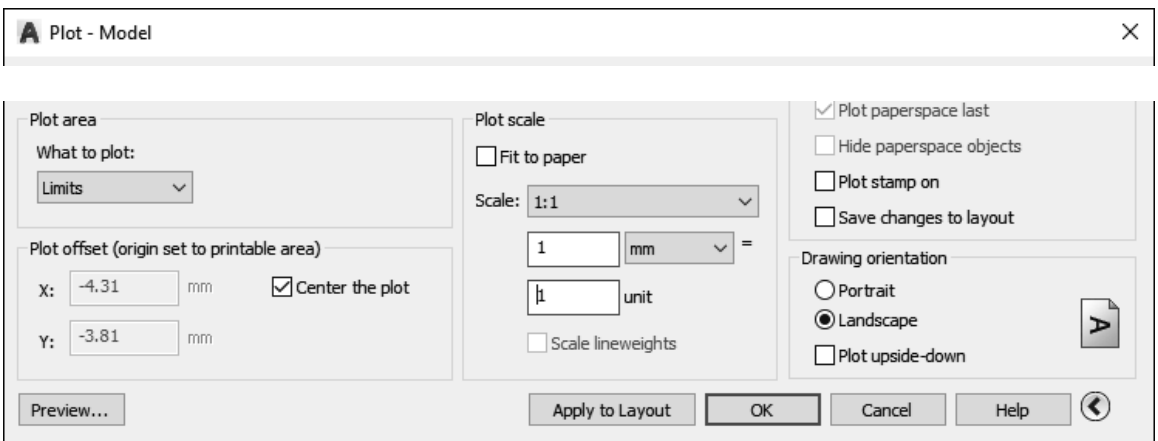
- 1) **Move** the circle anywhere inside the rectangle.
 - a) **Command:** **m** or **move**
 - b) **Select objects:** Place your cursor over the circumference of the circle and click your left mouse button.
 - c) **Select objects:** **Enter**
 - d) **Specify base point or [Displacement] <Displacement>:** Select a point inside the circle.
 - e) **Specify second point or <use first point as displacement>:** Move the circle inside the rectangle and click your left mouse button.

- 2) Type your name inside the circle.
 - a) **Command:** **text**
 - b) Specify start point of text or [Justify/Style]: **Select a point inside the circle.**
 - c) Specify height <3>: **3**
 - d) Specify rotation angle of text <0>: **Enter**
 - e) Type your name and hit **Enter** twice when you are done.

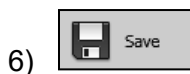
- 3) Print your drawing using **Limits** as your region, select a paper size of **Letter 8.5x11in**, **Center the plot** and the **Fit to paper** option.



- 4) Print your drawing using **Limits** as your region, select a paper size of **Letter 8.5x11in**, **Center the plot** and a **1:1 scale**. Note that not all of the rectangle will be printed.



- 5) Measure both printed circles and indicate their measured diameters.

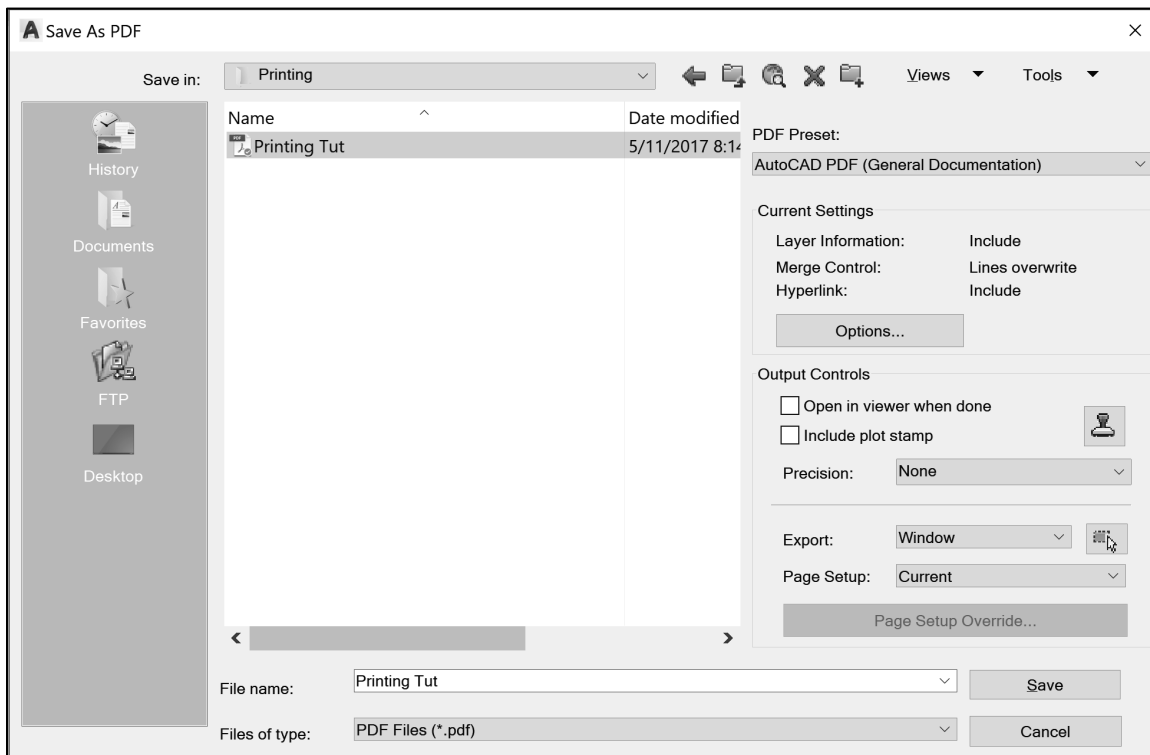


2.12.4) Printing to PDF

You may find a situation where you need to print your drawings to PDF or another format. AutoCAD allows you to do this using its export feature.

1) Application button: **Export – PDF...**

- a) Select the file location.
- b) File name: Name the file
- c) Export: **Window** (Select what you want to print)
- d) Page Setup: If you have already set up your page you can select **Current**. If you have not set up your page use **Override** to set up your page.
- e) If you want to view your print, select the **Open in viewer when done toggle**.



2.13) DRAW COMMANDS

The *Draw* panel (Figure 2.13-1) contains commands that allow you to draw standard geometries. The *Draw* commands can be used to create new objects such as lines and circles. Most AutoCAD® drawings are composed purely and simply from these basic components. A good understanding of the *Draw* commands is fundamental to the efficient use of AutoCAD®. To determine the name of the command associated with each icon in the *Draw* panel, place the cursor over each icon in turn and the associated command name will pop up and then in a few seconds an extended tooltip will appear.

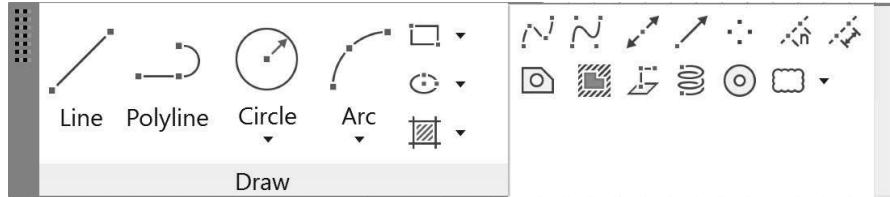


Figure 2.13-1: *Draw* panel


As is usual with AutoCAD®, the *Draw* commands can be accessed in one of several ways: typing the command in the *Command* window, from the *Draw* panel, and from the *Draw* tool palette.

2.13.1) Line

Lines are probably the most simple of AutoCAD's objects. Using the **LINE** command, a line can be drawn between any two points picked within the drawing area. A line drawn between two points is often called a vector. This terminology is used to describe the type of drawings that AutoCAD® creates. AutoCAD® drawings are generically referred to as "vector drawings." Vector drawings are extremely useful where precision is the most important criterion because they retain their accuracy irrespective of scale.

With the **LINE** command you can draw a simple line from one point to another or you can continue picking points and AutoCAD® will draw a straight line between each picked point and the previous point. Each line segment drawn is a separate object and can be moved or erased as required. While the **LINE** command is active, you can un-enter the last point by using the **UNDO** option available in the *Command* window. You can also close a sequence of lines (connect the start and end point) using the **CLOSE** option.


The **LINE** command may be accessed in the following way.

- *Draw* panel: 
- *Command* window: **l** or **line**

2.13.2) Construction line

The construction line (**XLINE**) command creates a line of infinite length which passes through two picked points. Construction lines are very useful for creating construction frameworks or grids. Construction lines are not normally used as objects in finished drawings. Therefore, it is usual to draw all your construction lines on a separate layer which will be turned off or frozen prior to printing. Because of their nature, the ZOOM EXTENTS command ignores construction lines.

The construction line command may be accessed in the following way.

- Draw panel: 
- Command window: **xl** or **xline**


Construction line options (Specify a point or [Hor/Ver/Ang/Bisect/Offset]:).

- Hor: Creates a horizontal construction line.
- Ver: Creates a vertical construction line.
- Ang: Creates a construction line at a specified angle.
- Bisect: Create a construction line that bisects an angle defined by 3 points.
- Offset: Creates a construction line that is offset from an existing line by a specified distance.

2.13.3) Ray

The **RAY** command creates a line starting at a point and going off to infinity through another specified point. The `through point:` prompt is repeated allowing you to create multiple rays. Because of their nature, the ZOOM EXTENTS command ignores rays.

The RAY command may be accessed in the following way.

- Draw panel: 
- Command window: **ray**

2.13.4) Polyline

Polylines (**PLINES**) differ from lines in that they are more complex objects. A single polyline can be composed of a number of straight-line or arc segments. Polylines can also be assigned line widths to make them appear solid. Figure 2.13-2 shows a number of polylines to give you an idea of the flexibility of this type of line. Because of their complexity, polylines use up more memory than the equivalent line. As it is desirable to keep file sizes as small as possible, it is a good idea to use LINES rather than polylines unless you have a particular requirement.

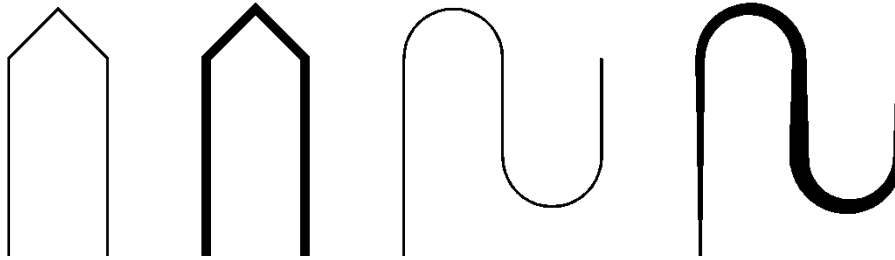



Figure 2.13-2: Polylines

The PLINE (polyline) command may be accessed in the following way.

- Draw panel: 
- Command window: **pl** or **pline**

Polyline LINE options (Specify next point or [Arc/Halfwidth/Length/Undo/Width]:).

- Halfwidth: Enables you to set the halfwidth of the start and end of the line.
- Length: Enables you to specify the length of the line.
- Undo: Removes the most recent segment added to the polyline.
- Width: Enables you to set the start and end widths of the line.

Polyline ARC options (Specify endpoint of arc or [Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]:).

- Angle: Enables you to specify the included angle of the arc segment.
- CEnter: Enables you to specify the center point of the arc.
- Direction: Enables you to specify the starting direction of the arc.
- Halfwidth: Enables you to set the halfwidth of the start and end of the arc.
- Radius: Enables you to specify the radius of the arc.
- Second pt: Enables you to specify the second point of a three point arc.
- Undo: Removes the most recent segment added to the polyline.
- Width: Enables you to set the start and end widths of the arc.

The **Undo** option is particularly useful. This allows you to unpick polyline vertices one at a time so that you can easily correct mistakes. Also, polylines may be edited after they are created using the command **PEDIT**.

2.13.5) Polygon

The **POLYGON** command can be used to draw any regular polygon from 3 sides up to 1024 sides. This command requires four inputs from the user, the number of sides, a pick point for the center of the polygon, whether you want the polygon inscribed or circumscribed and then a pick point which determines both the radius of this imaginary circle and the orientation of the polygon. This command also allows you to define the polygon by entering the length of a side using the **EDGE** option.

The POLYGON command may be accessed in the following way.

- Draw panel: 
- Command window: **pol** or **polygon**

2.13.6) Rectangle

The **RECTangle** command is used to draw a rectangle whose sides are, by default, vertical and horizontal. However, you may draw a rectangle at a specified angle.

The RECTANGLE command may be accessed in the following way.

- Draw panel: 
- Command window: **rec** or **rectangle**

RECTANGLE options (Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width] :).

- Chamfer: Creates a rectangle with chamfered corners.
- Elevation: Enables you to specify the elevation of the rectangle.
- Fillet: Creates a rectangle with filleted corners.
- Thickness: Enables you to specify the thickness of the rectangle.
- Width: Enables you to specify the line width of the rectangle.

RECTANGLE options after picking the first corner point (Specify other corner point or [Area/Dimensions/Rotation] :).


- Area: Enables you to specify the area and length of the rectangle.
- Dimension: Enables you to specify the length and width of the rectangle.
- Rotation: Enables you to specify the angle of the line that connects the first and second corner of the rectangle.

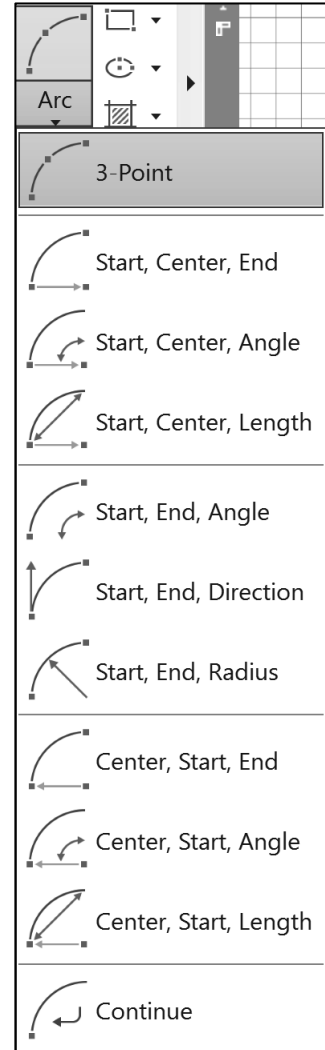
2.13.7) Arc

The **Arc** command allows you to draw an arc of a circle. There are numerous ways to define an arc; the default method uses three pick points, a start point, a second point and an end point. Using this method, the drawn arc will start at the first pick point, pass through the second point and end at the third point. Other ways of defining an arc can be accessed through the fly-out menu under the ARC icon.

Arcs, by default, travel in the counter clockwise direction. This default direction may be changed by holding down the **ctrl** key as you drag.

The ARC command may be accessed in the following way.


- Draw panel: 
- Command window: **a** or **arc**

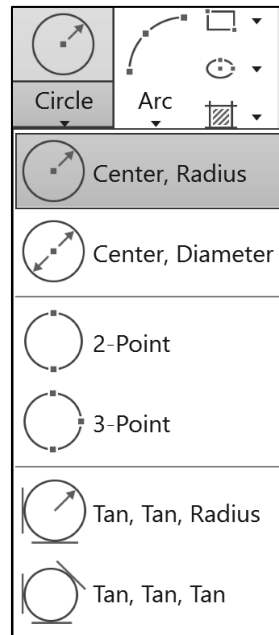


2.13.8) Circle

The **Circle** command is used to draw circles. There are a number of ways that you can define a circle. The default method is to pick the center point and then to either pick a second point on the circumference of the circle or to enter the circle's radius in the *Command* window. Other ways of defining a circle can be accessed through the fly-out menu under the CIRCLE icon.

The CIRCLE command may be accessed in the following way.

- Draw panel: 
- Command window: **c** or **circle**



2.13.9) Spline

A **SPLine** is a smooth curve that is fitted along a number of control points. Splines can be edited after they have been created using the **SPLINEDIT** command. Using this command, you can change the tolerance, add more control points, move control points and close a spline. However, if you just want to move spline control points, it is best to use the grips boxes.

You can create or edit splines using either *control vertices* or *fit points* (see Figure 2.13-3). The grip boxes are shown when you click on the spline. Creating a spline using *control vertices* has the advantage of fine control. The variables **CVSHOW** and **CVHIDE** control whether or not the control vertices are displayed. The options available within the spline command depend on the method used to create the spline.

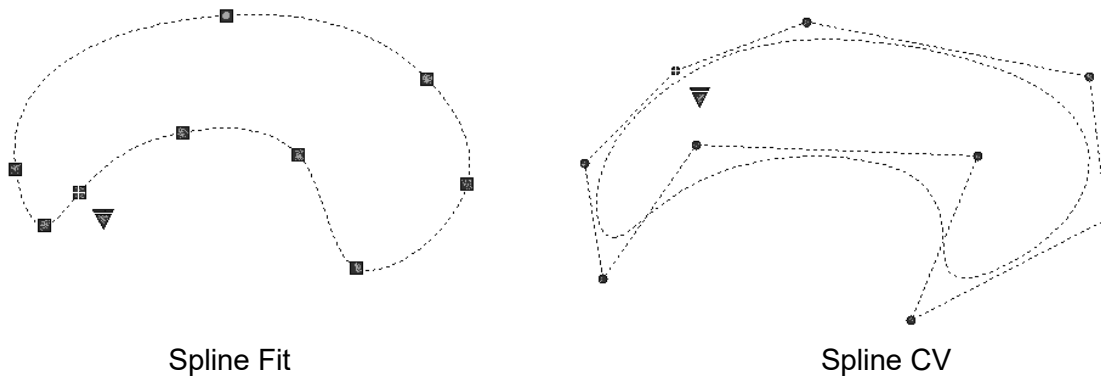


Figure 2.13-3: Spline methods

The SPLINE fit command may be accessed in the following way.

- **Draw panel:**  (fit)  (CV)
- **Command window:** **spl** or **spline**

Spline Fit options (Specify first point or [Method/Knots/Object]:)

- **Method:** Allows you to choose between the *fit* and *control vertices* methods.
- **Knots:** Allows you to choose the knot parameterization (chord, square root or uniform).

Spline Fit options (Enter next point or [start Tangency/toLerance]:)

- **start Tangency:** Allows you to specify the tangency of the first curved segment.
- **toLerance:** Allows you to control how closely the spline conforms to the control points. A low tolerance value causes the spline to form close to the control points. A tolerance of 0 (zero) forces the spline to pass through the control points.

Spline Fit options (Enter next point or [end Tangency/toLerance/Undo/Close]:)

- **end Tangency:** Allows you to specify the tangency of the last curved segment.
- **Undo:** Removes the last point.
- **Close:** Closes the spline.


Spline CV options (Specify first point or [Method/Degree/Object]:)

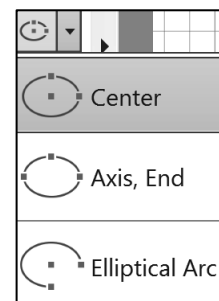
- **Degree:** Allows you to specify the degree of the spline.

2.13.10) Ellipse

The **ELLipse** command gives you a number of different creation options. The default option is to pick the two end points of an axis and then a third point to define the eccentricity of the ellipse. Other ways of defining an ellipse can be accessed through the fly-out menu under the ELLIPSE icon.

The ELLIPSE command may be accessed in the following way.


- **Draw panel:** 
- **Command window:** **el** or **ellipse**



2.13.11) Point

The **P**oint command will insert a point marker in your drawing at a position which you pick or at any coordinate location which you enter in the *Command* window. Other ways of defining a point can be accessed through the fly-out menu. The default point style is a simple dot, which is often difficult to see but you can change the point style to something more easily visible or elaborate using the point style dialogue box.

The POINT command may be accessed in the following way.

- *Draw* panel: 
- *Command* window: **po** or **point**

You can access the *Point Style* window with the command **DDPTYPE** or you can access it from the pull-down menu at **Format - Point Style...** To change the point style (**PDMODE**), just pick the picture of the style you want and then click the **OK** button (see Figure 2.13-4). You may also change the point size (**PDSIZE**) in this window. You will need to use the **REGEN** (regenerate) command to update your existing points. Any new points created after the style has been set will automatically display in the new style.

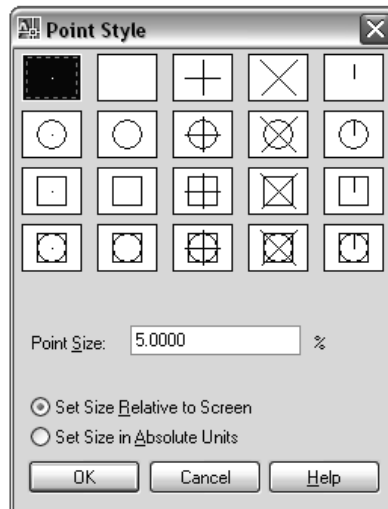


Figure 2.13-4: *Point Style* window

2.14) TEXT

Text may be added to any location of your drawing and is present in your drawing dimensions. AutoCAD's text is very flexible. It offers all of the standard Windows® fonts plus a few extra fonts. Various text commands and settings may be found in two locations. Text commands may be found in the *Annotation* panel in the *Home* tab and the *Text* panel in the *Annotate* tab shown in Figure 2.14-1.

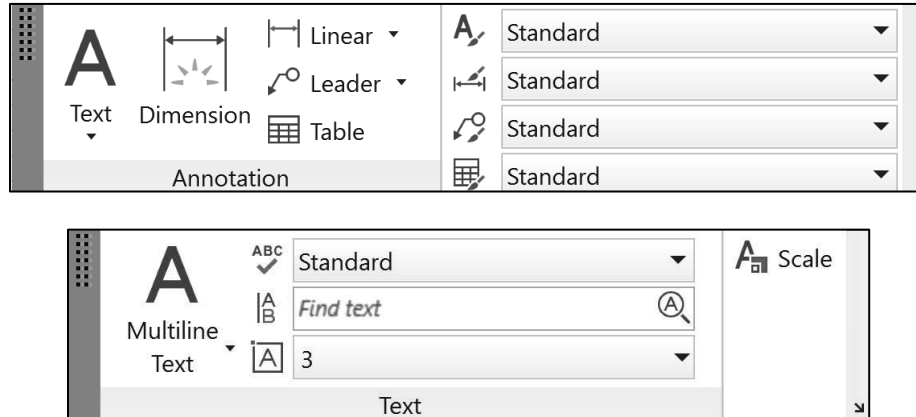


Figure 2.14-1: Annotation and Text panels


2.14.1) Style

The default text font and size may be set in the *Text Style* window. Many text styles may be created and saved under a style name. Properties such as the text direction and the text angle may also be set. The *Text Style* window shown in Figure 2.14-2 may be accessed by typing the command **Style** or **ST**, or from the pull-down menu at **Format – Text Style...**. Figure 2.14-2 also shows some examples of different text effects performed on an Arial font.

2.14.2) Single-line text

The **TEXT** command creates a single-line of text. When creating text, you can click anywhere in a drawing to create a new text block. The advantage of using single line text is its wide variety of justification options. If **TEXT** was the last command entered, pressing **Enter** at the *Specify Start Point of Text:* prompt skips the prompts for height and rotation angle and enters the text directly underneath the previously entered text. The command **DDEDIT** may be used to edit existing text.

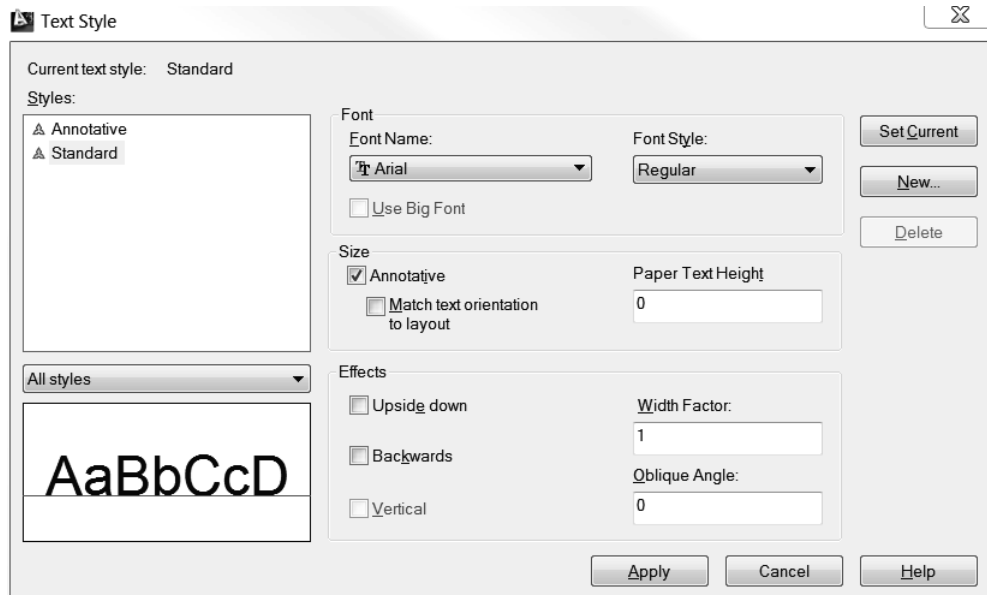
The **TEXT** command may be accessed in the following way.

- Annotation panel: 
- Command window: **text** or **dt** or **dtext**

JUSTIFY options (Enter an option

[Left/Center/Right/Align/Middle/Fit/TL/TC/TR/ML/MC/MR/BL/BC/BR] :).

- **Left:** Justify left.
- **Center:** Allows you to specify the center of the text.
- **Right:** Justify right.
- **Align:** Allows you to specify the left and right boundary of your text. The text will automatically adjust its height to fit the boundary.
- **Middle:** Allows you to specify the middle of the text.
- **Fit:** Allows you to specify the left and right boundary of your text. The text will automatically adjust its width to fit the boundary.
- **TL:** Top left.
- **TC:** Top center.
- **TR:** Top right.
- **ML:** Middle left.
- **MC:** Middle center.
- **MR:** Middle right.
- **BL:** Bottom left.
- **BC:** Bottom center.
- **BR:** Bottom right.



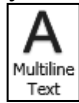
Text **Text** *Text* ǀǂǃǄ ǂǃǄǀ

Figure: 2.14-2: *Text Style* window and text effects examples**2.14.3) Multi-lined text**

Using the multi-lined text (**MText**) command activates an in-place text editor and adds a *Text Editor* tab to the ribbon. The *Text Editor* tab includes several different panels that allow you to change your text properties (i.e., font, size, justification). Figure 2.14-3 shows the *Text Editor* tab and in-place text editor. To edit multi-lined text you can double click on the text or use the commands **DDEDIT** or **MTEDIT**. Also, the command

TEXTTOFRONT may be used to move text and dimensions in front of all other objects in the drawing.

The MTEXT command may be accessed in the following way.



- Annotation panel:
- Command window: **t** or **mt** or **mtext**

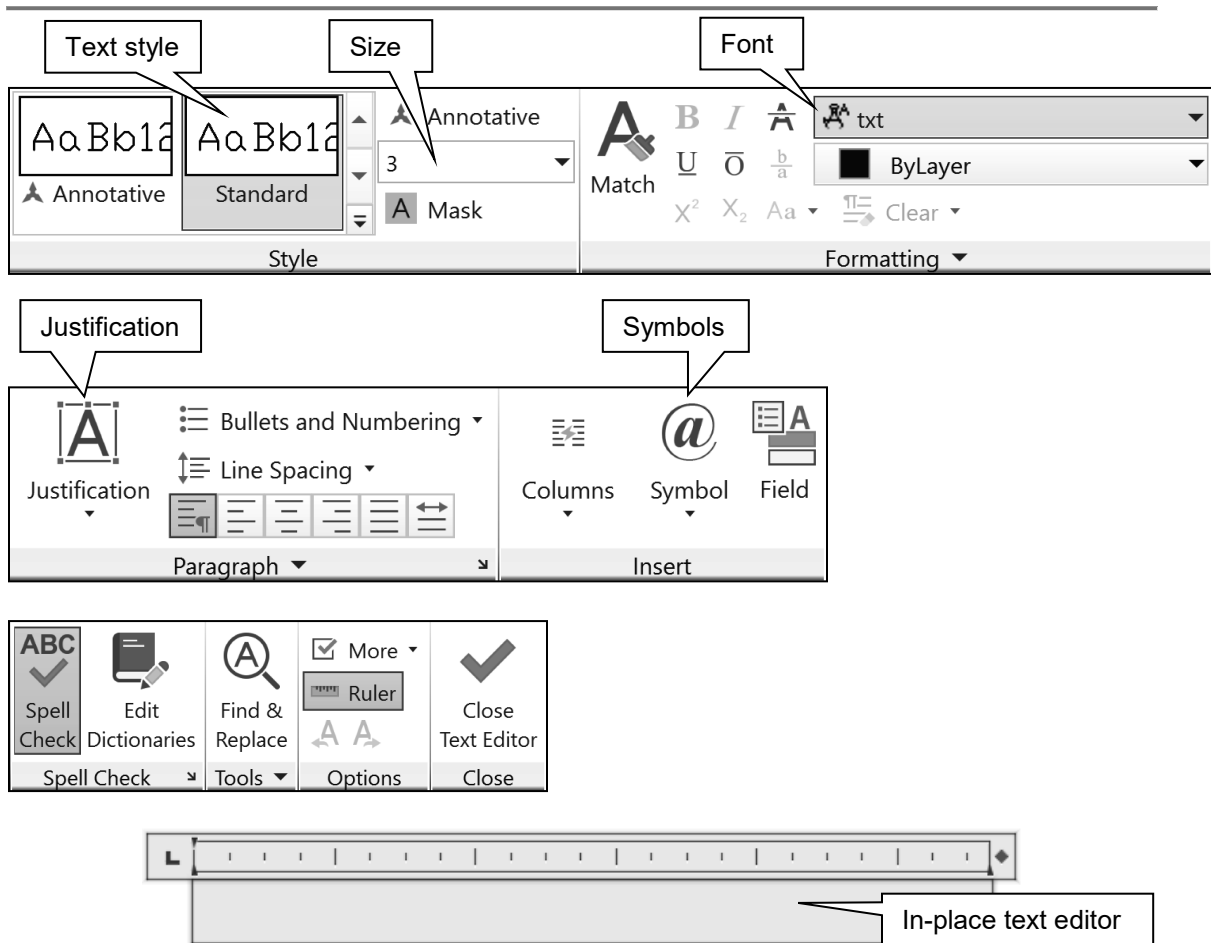


Figure 2.14-3: In place text editor.

2.15) MODIFY COMMANDS

AutoCAD® drawings are rarely completed simply by drawing lines, circles and other geometries available within the *Draw* commands. It is more than likely that you will need to modify these basic objects, in some way, to create the shape you need. The *Modify* panel (Figure 2.15-1) contains commands that allow you to change standard geometries. AutoCAD provides a wide range of *Modify* commands such as MOVE, COPY, ROTATE and MIRROR. As you can see, the command names are easily understandable. However, the way these commands work is not always obvious. It is very important to read the prompt displayed in the *Command* window while applying a *Modify* command.

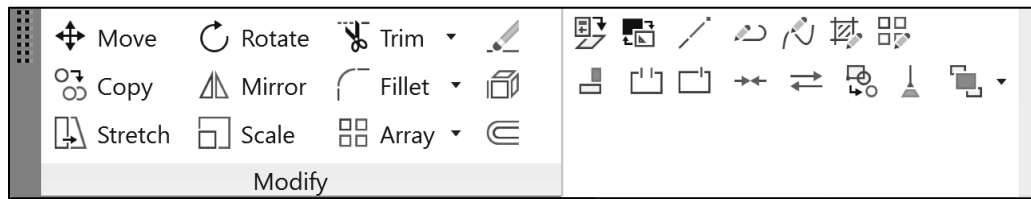


Figure 2.15-1: Modify panel

As is usual with AutoCAD®, the *Modify* commands may be accessed in one of several ways: typing the command in the *Command* window, from the *Application button*, and from the *Modify* panel.


2.15.1) Selecting objects

All *Modify* commands require you to select the object or objects you wish to modify. The simplest way to select an object is to place your cursor over the object and click your left mouse button. The selected object will become dashed or dotted. You may select several objects if necessary. If you accidentally select an object that should not be included in the *Modify* command, hold down the **SHIFT** key and select the unwanted object or type **R** or **Remove** in the *Command* window and select the object again. It will be removed from the selection set. If you then need to add an object to the selection set after applying the REMOVE option, you need to type **A** or **Add** and then select the object.

2.15.2) Erase

The **Erase** command is one of the simplest AutoCAD commands and is one of the most used. The command erases or deletes any selected object(s) from the drawing.


The ERASE command may be accessed in the following way.

- Modify panel: 
- Command window: **e** or **erase**


2.15.3) Copy

The **COpy** command can be used to create one or more duplicates of any object(s) which have been previously created.

The COPY command may be accessed in the following way.

- Modify panel: 
- Command window: **co** or **cp** or **copy**

Copying an object(s)

- 1) **Command:** **co** or **copy** or **Modify panel:** 
- 2) **Select objects:** Select an object that you want to copy.
- 3) **Select objects:** Select an object that you want to copy or hit **Enter** to stop selecting objects.
- 4) **Specify base point or [Displacement/mOde] <Displacement>:** **o**
- 5) **Enter a copy mode option [Single/Multiple] <Multiple>:** **m**
- 6) **Specify base point or [Displacement/mOde] <Displacement>:** Select a base point (see selecting base points).
- 7) **Specify second point or [Array] <use first point as displacement>:** Select a second base point.
- 8) **Specify second point or [Array/Exit/Undo] <Exit>:** To make another copy, select another second base point or hit **Enter** to exit.

Selecting base points

- **Copying an object(s) a specified distance:** The two base points are simply used to indicate the distance and direction of the copied object from the original object. The first *base point* does not have to be picked on or near the object, just select a point anywhere on the drawing area. The *second base point* is specified as a relative (@) coordinate. This relative coordinate gives the specified distance and direction.
- **Copying an object(s) to a specific location:** In this situation, the base points need to have a geometric relationship with the object and its subsequent final location. The first *base point* should be a geometric location on the object and the *second base point* should be either a coordinate point or a geometric location on an existing object.

2.15.4) Mirror

The **Mirror** command allows you to mirror selected objects in your drawing by picking them and then defining the position of an imaginary mirror line using two points. To create perfectly horizontal or vertical mirror lines turn the ORTHO command on. Figure 2.15-2 shows examples of mirroring.

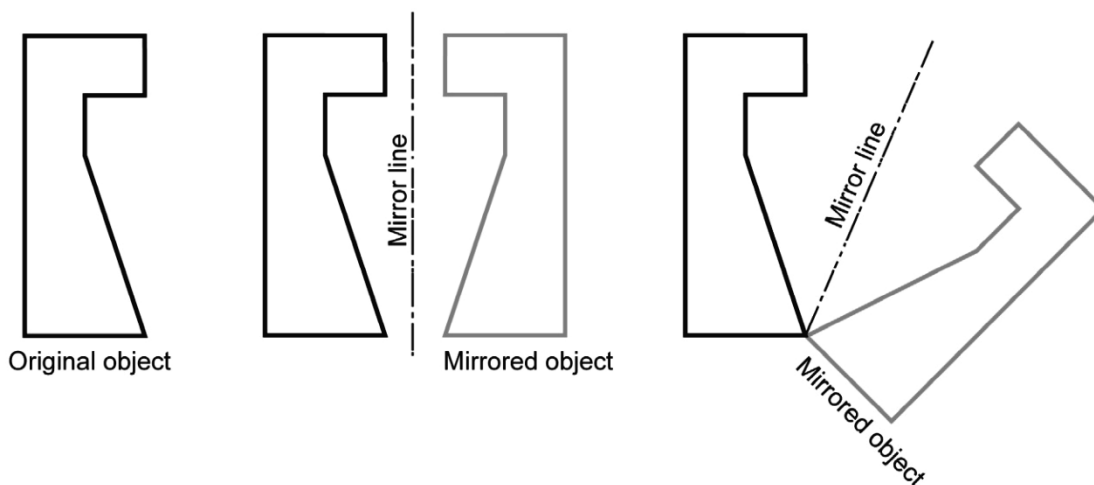



Figure 2.15-2: Mirroring an object

The MIRROR command may be accessed in the following way.

- Modify panel: 
- Command window: **mi** or **mirror**

Mirroring an object(s)

- 1) Command: **mi** or **mirror** or Modify panel: 
- 2) Select objects: Select an object that you want to mirror.
- 3) Select objects: Select an object that you want to mirror or hit **Enter** to stop selecting objects.
- 4) Specify first point of mirror line: Select a point
- 5) Specify second point of mirror line: Select a point
- 6) Erase source objects? [Yes/No] <N>: (Entering **Y** will erase your originally selected objects)

2.15.5) Offset

The **OFFSET** command creates a new object parallel to or concentric with a selected object. The new object is drawn at a user defined distance (the offset) from the original and in a direction chosen. The OFFSET command may only be used on one object or entity at a time. Figure 2.15-3 shows a rectangle being offset. This effect could not be obtained by offsetting a rectangle that was created using four lines.

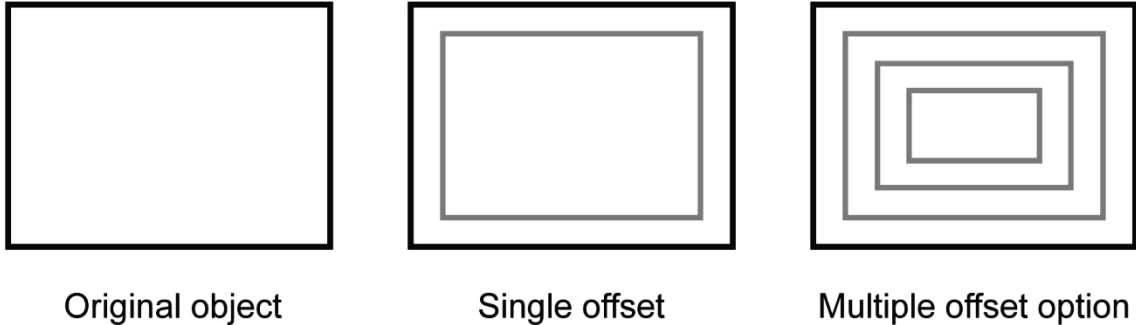




Figure 2.15-3: Offsetting a rectangle

The OFFSET command may be accessed in the following way.

- Modify panel: 
- Command window: **offset**

Offsetting an object

- 1) **Command:** **offset** or **Modify panel:** 
- 2) Specify offset distance or [Through/Erase/Layer] <1.0000>: Specify the distance to offset.
- 3) Select object to offset or [Exit/Undo] <Exit>: Select an object.
- 4) Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: Select a point on either side of the object to specify the direction of offset. Select the option MULTIPLE if you want to perform multiple offsets. The offset will be relative to the last offset object.
- 5) Select object to offset or [Exit/Undo] <Exit>: Select another object to be offset by the distance specified above or hit **Enter** to exit the command.

2.15.6) Array

The **ARray** command makes multiple copies of selected objects in a rectangular pattern (columns and rows) or a polar (circular) pattern. Figure 2.15-4 shows an example of a rectangle and a polar array. The system variable **ARRAYTYPE** sets the type of array (Arraytype = 0 (rectangular), 1 (path array), 2 (Polar array)).

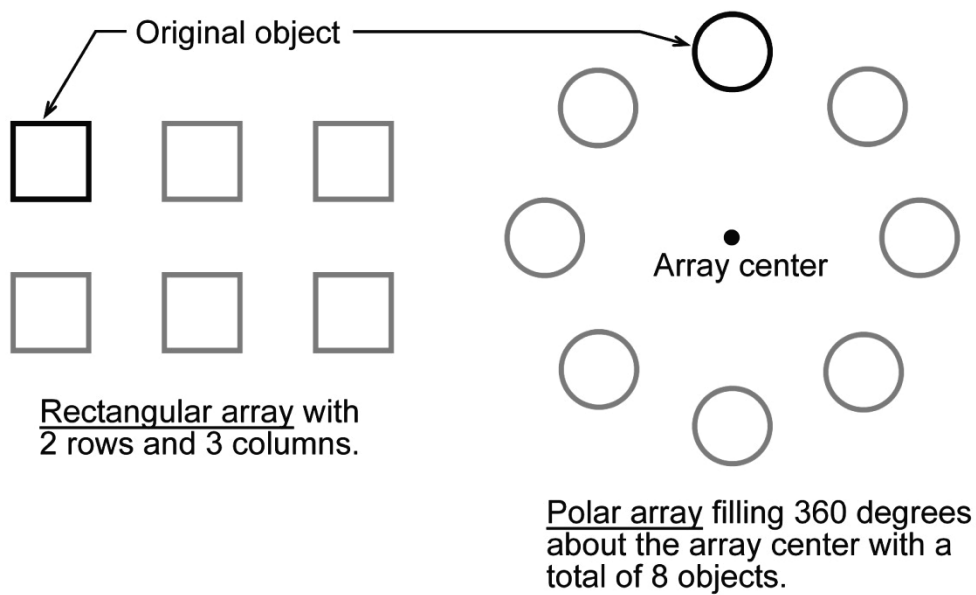

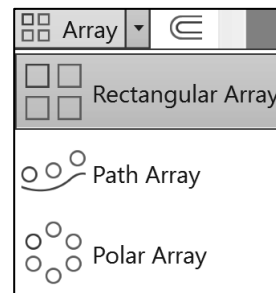


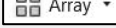
Figure 2.15-4: Rectangular and polar array














The ARRAY command may be accessed in the following way.

- **Modify panel:**  Array
- **Command window:** **ar** or **array**

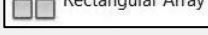


Creating an array

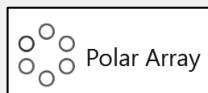
- 1) **Command:** **ar** or **array** or **Modify panel:**  Array ▾
- 2) Select objects: **Select the object you wish to array.**
- 3) Select objects: **Enter**
- 4) Select grip to edit array or [ASsociative/Base point/COunt/Spacing/COlumns/Rows/Levels/eXit] <eXit>: **Select array options or enter the options in the *Array Creation* tab.**

 Rectangular	 Columns: <input type="text" value="4"/>	 Rows: <input type="text" value="3"/>	 Levels: <input type="text" value="1"/>	 Associative	 Base Point	 Close Array
	 Between: <input type="text" value="825"/>	 Between: <input type="text" value="600"/>	 Between: <input type="text" value="1"/>			
	 Total: <input type="text" value="2475"/>	 Total: <input type="text" value="1200"/>	 Total: <input type="text" value="1"/>			
Type	Columns	Rows ▾	Levels	Properties		Close

Creating a Rectangular array

- 1) **Command:** **arrayrect** or **Modify panel:**  Rectangular Array
- 2) Select objects: **Select the object you wish to array.**
- 3) Select objects: **Enter**
- 4) Select grip to edit array or [ASsociative/Base point/COunt/Spacing/COlumns/Rows/Levels/eXit] <eXit>: **col**
- 5) Enter number of columns or [Expression] <4>: **Specify the number of rows in the array.**
- 6) Specify the distance between columns or [Total/Expression] <58>: **Specify the distance between the columns.**
- 7) Select grip to edit array or [ASsociative/Base point/COunt/Spacing/COlumns/Rows/Levels/eXit] <eXit>: **r**
- 8) Enter number of rows or [Expression] <4>: **Specify the number of columns in the array.**
- 9) Specify opposite corner to space items or [Spacing] <Spacing>: **s**
- 10) Specify the distance between rows or [Total/Expression] <48>: **Specify the distance between the rows.**
- 11) Specify the incrementing elevation between rows or [Expression] <0>: **Specify the row elevation. For a 2D drawing, there is no need for elevation.**
- 12) Select grip to edit array or [ASsociative/Base point/COunt/Spacing/COlumns/Rows/Levels/eXit] <eXit>: **x**

Creating a Polar array



- 1) Command: **arraypolar** or Modify panel:
- 2) Select objects: **Select the object you wish to array.**
- 3) Select objects: **Enter**
- 4) Specify center point of array or [Base point/Axis of rotation]: **Select the center point.**
- 5) Select grip to edit array or [ASsociative/Base point/Items/Angle between/Fill angle/ROws/Levels/ROtate items/eXit] <eXit>: **i**
- 6) Enter number of items or [Expression] <3>: **Enter the number of items.**
- 7) Select grip to edit array or [ASsociative/Base point/Items/Angle between/Fill angle/ROws/Levels/ROtate items/eXit] <eXit>: **f**
- 8) Specify the angle to fill (+=ccw, -=cw) or [EXpression] <360>: **Enter the angle to fill.**
- 9) Select grip to edit array or [ASsociative/Base point/Items/Angle between/Fill angle/ROws/Levels/ROtate items/eXit] <eXit>: **x**

Creating a Path array



- 1) Command: **arraypolar** or Modify panel:
- 2) Select objects: **Select the object you wish to array.**
- 3) Select objects: **Enter**
- 4) Select path curve: **Select the curve that the array will follow**
- 5) Select grip to edit array or [ASsociative/Base point/Items/Angle between/Fill angle/ROws/Levels/ROtate items/eXit] <eXit>: **i**
- 6) Specify the distance between items along the path or [Expression] <1>: **10**
- 7) Specify number of items or [Fill entire path/Expression] <29>: **20**
- 8) Select grip to edit array or [ASsociative/Base point/Items/Angle between/Fill angle/ROws/Levels/ROtate items/eXit] <eXit>: **x**

Once an array has been created, it becomes a group of objects. That means that the individual objects that make up the array cannot be edited independently. The array as a whole may be modified by clicking on the group. After the array has been selected, an *Array Edit* panel will appear. The array parameters may be changed in this panel as shown in Figure 2.15-5. Alternatively, you may enter the command prompt editing options by using the ARRAYEDIT command. If you want to edit an individual array object, you may **ctr1**-click to Erase, Move, ROtate or SScale the selected object without affecting the array as a whole. Alternatively, you may EXPLODE the array and break it up into its individual objects.

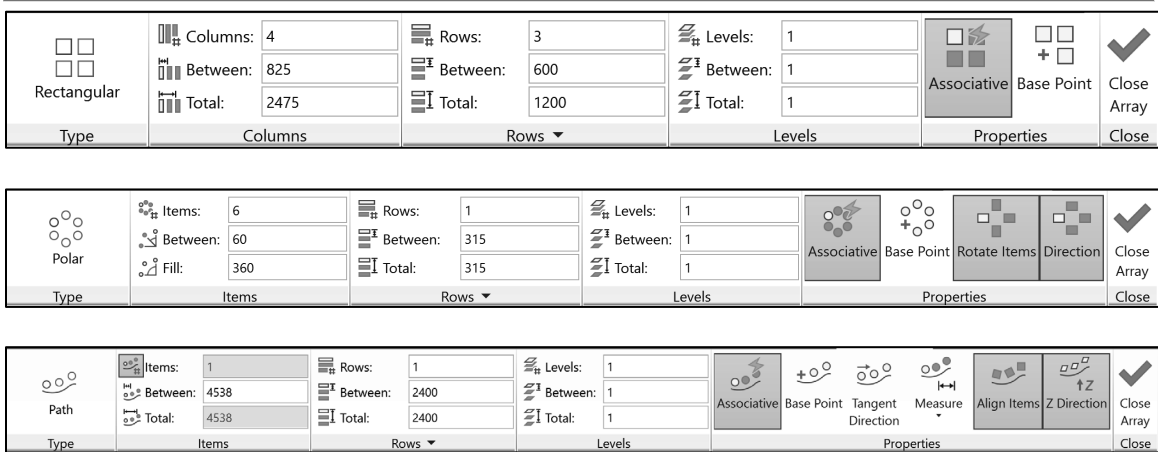



Figure 2.15-5: Array edit panel

2.15.7) Move

The **Move** command works in a similar way to the COPY command except that no copy is made; the selected object(s) is simply moved from one location to another.

The MOVE command may be accessed in the following way.

- Modify panel:  Move
- Command window: **m** or **move**

Moving an object(s)

- 1) Command: **m** or **move** or Modify panel:  Move
- 2) Select objects: Select an object that you want to move.
- 3) Select objects: Select an object that you want to move or hit **Enter** to stop selecting objects.
- 4) Specify base point or [Displacement] <Displacement>: Select a base point (see selecting base points).
- 5) Specify second point or <use first point as displacement>: Select a second base point.

Selecting base points

- Moving an object(s) a specified distance: The two base points are simply used to indicate the distance and direction of the final location from the original location. The first *base point* does not have to be picked on or near the object, just select a point anywhere on the drawing area. The *second base point* is specified as a relative (@) coordinate. This relative coordinate gives the specified distance and direction.
- Moving an object(s) to a specific location: In this situation, the base point needs to have a geometric relationship with the object and its subsequent final location. The first *base point* should be a geometric location on the object and the *second base point* should be either a coordinate point or a geometric location on an existing object.

2.15.8) Rotate

The **ROtate** command allows an object or objects to be rotated about a point selected by the user. AutoCAD prompts for a second rotation point or an angle which can be typed in the *Command* window. Figure 2.15-6 shows examples of rotating.

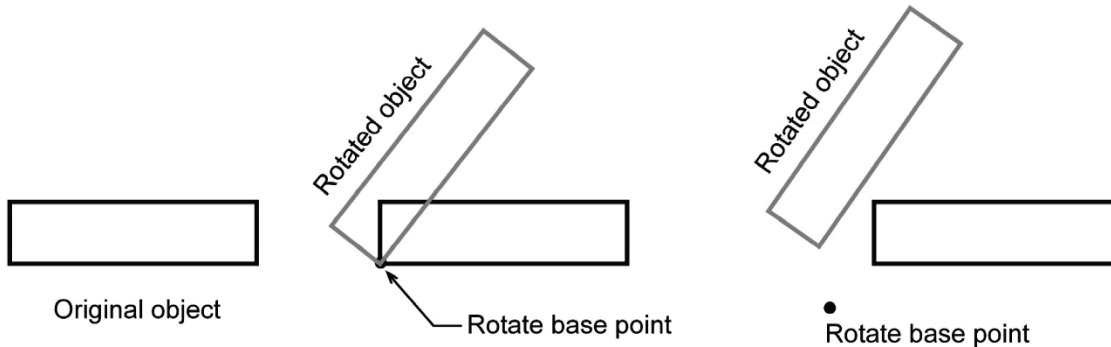




Figure 2.15-6: Rotate effects

The ROTATE command may be accessed in the following way.

- Modify panel: 
- Command window: **ro** or **rotate**

Rotating an object(s)

- 1) Command: **ro** or **rotate** or Modify panel: 
- 2) Select objects: Select an object that you want to rotate.
- 3) Select objects: Select an object that you want to rotate or hit **Enter** to stop selecting objects.
- 4) Specify base point: Select the point to rotate about.
- 5) Specify rotation angle or [Copy/Reference] <0>: Enter the angle of rotation or select the **COPY** option if you want to keep the original object.

2.15.9) Scale

The **Scale** command can be used to change the size of an object or group of objects. You are prompted for a base point about which the selection set will be scaled. Scaling can then be completed by picking a second point or by entering a scale factor. Figure 2.15-7 shows examples of scaling.

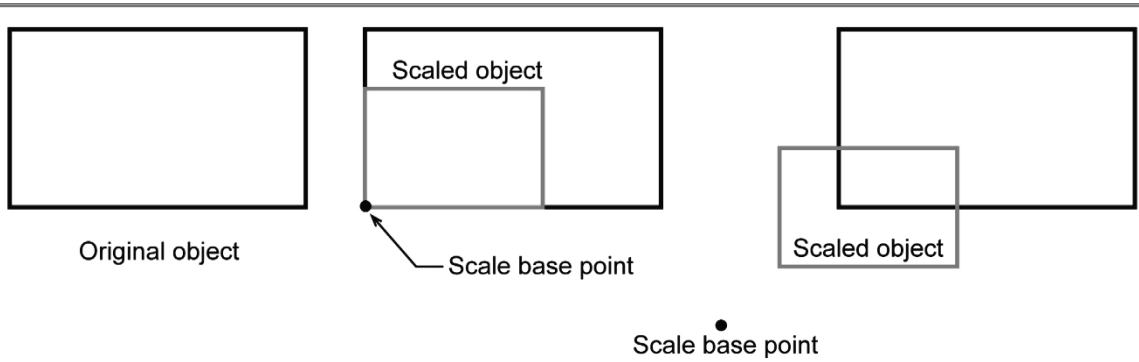




Figure 2.15-7: Scale effects

The SCALE command may be accessed in the following way.

- Modify panel: 
- Command window: **sc** or **scale**

Scaling an object(s)

- 1) Command: **sc** or **scale** or Modify panel: 
- 2) Select objects: Select an object that you want to scale.
- 3) Select objects: Select an object that you want to scale or hit **Enter** to stop selecting objects.
- 4) Specify base point: Select a scale reference point.
- 5) Specify scale factor or [Copy/Reference] <1.000>: Enter the scale factor or select the COPY option if you want to keep the original object.

2.15.10) Stretch

The **STRETCH** command can be used to move one or more vertices of an object while leaving the rest of the object unchanged. In Figure 2.15-8, a rectangle has been stretched by moving one vertex to create an irregular shape.

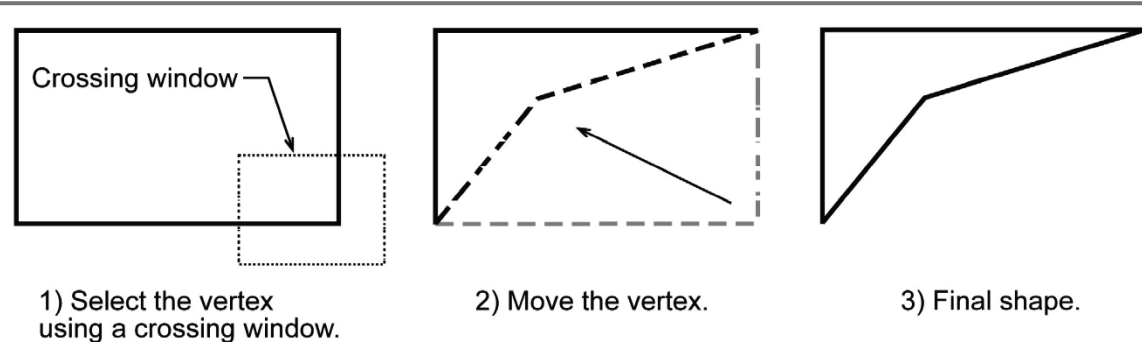




Figure 2.15-8: Using the STRETCH command

The STRETCH command may be accessed in the following way.

- Modify panel: 
- Command window: **stretch**


Stretching a vertex

- 1) Command: **stretch** or Modify panel: 
- 2) Select objects: **c** (for crossing window)
- 3) Specify first corner: Select the first corner of the window.
- 4) Specify opposite corner: Select the opposite corner of the window.
- 5) Select objects: **PRESS Enter** or **Space** to discontinue selecting objects.
- 6) Specify base point or [Displacement] <Displacement>: Select the vertex to stretch.
- 7) Specify second point or <use first point as displacement>: Select the final position of the vertex or enter relative coordinates to specify the distance to move relative to its original position.

2.15.11) Lengthen

The **LENgthen** command is used to change the length of objects and the included angle of arcs. The **LENgthen** command maintains the current orientation of the object, unlike the **STRETCH** command. The **LENgthen** command allows you to specify the total length/angle of the object, or change the length by a percentage of the original length or by a specified amount (delta amount).

The **LENgthen** command may be accessed in the following way.

- Modify panel: 
- Command window: **len** or **lengthen**

2.15.12) Trim

The **TRim** command can be used to trim off part of an object. In order to trim an object you must draw a second object which forms the cutting edge. Cutting edges can be lines, xlines, rays, polylines, circles, arcs or ellipses. Blocks and text cannot be trimmed or used as cutting edges. At each trimming step you are given the option to **UNDO** the previous trim. This can be very useful if you inadvertently pick the wrong object. Figure 2.15-9 shows an example of trimming.

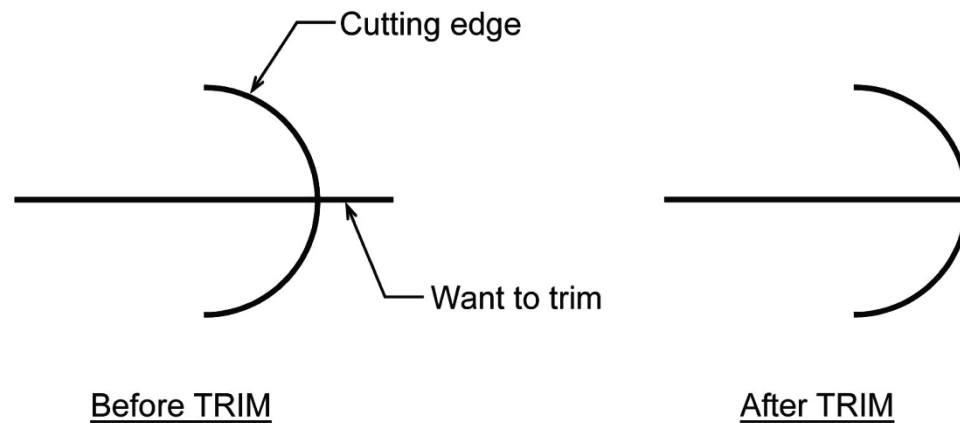




Figure 2.15-9: An example of trimming

The TRIM command may be accessed in the following way.

- Modify panel:  Trim
- Command window: **tr** or **trim**

Trimming an object

- 1) Command: **tr** or **trim** or Modify panel:  Trim
- 2) Select cutting edges ...
Select objects or <select all>: Select an object that will be used as a cutting edge (this object usually is not trimmed).
- 3) Select objects: Select another cutting edge or hit **Enter** or **Space** to discontinue selecting cutting edges.
- 4) Select object to trim or shift-select to extend or
[Fence/Crossing/Project/ Edge/eRase/Undo]: Select the portion of the object that you wish to trim.
- 5) Select object to trim or shift-select to extend or
[Fence/Crossing/Project/ Edge/eRase/Undo]: Select another object to trim or select the Undo option to undo your last trim.

2.15.13) Extend

The **EXtend** command extends a line, polyline or arc to meet an existing object (known as the boundary edge) as illustrated in Figure 2.15-10. You can tell AutoCAD which direction to extend the object by picking a point to one side or the other of the midpoint. If the object does not extend, it means that you are either picking the wrong end of the object or the object you are trying to extend will not meet the boundary edge. The solution is to either pick nearer the end you want to extend, move the boundary edge so that the extended object will intersect it, or use the EDGE option. The EDGE/EXTEND option will create an imaginary boundary for the object to intersect. Figure 2.15-10 shows an example extending an object.

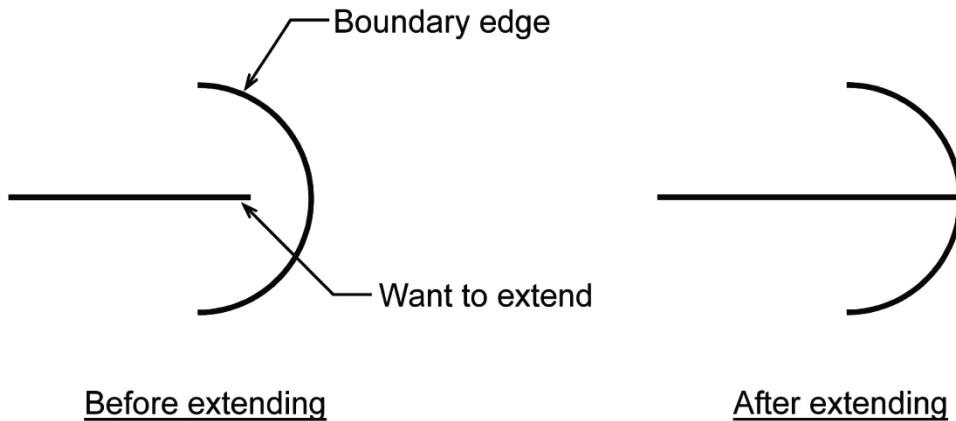
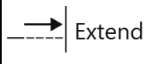



Figure 2.15-10: Extending an object

The EXTEND command may be accessed in the following way.


- Modify panel:  Extend
- Command window: **ex** or **extend**

Extending an object

- 1) Command: **ex** or **extend** or Modify panel:  Extend
- 2) Select boundary edges ...
Select objects or <select all>: Select the object to be used as the boundary edge.
Select objects: Select another boundary edge object or hit **Enter** or **Space** to end the selection.
- 3) Select object to extend or shift-select to trim or
[Fence/Crossing/Project/ Edge/Undo]: Select the object to extend or select the **EDGE** option if your extended object and boundary object will not intersect.
- 4) Select object to extend or shift-select to trim or
[Fence/Crossing/Project/ Edge/Undo]: Select the object to extend or hit **Enter** or **Space** to end the command.

2.15.14) Break

The **BReak** command enables you to break (remove part of) an object by defining two break points. In Figure 2.15-11, a corner of a rectangle has been removed using the **BREAK** command. The remaining lines are still part of one entity. By default, AutoCAD assumes that the point used to select the object is the first break point. However, you can use the **FIRST POINT** option to override this. If you need to break an object into two

without removing any part of it, use the **BREAK AT POINT** command .

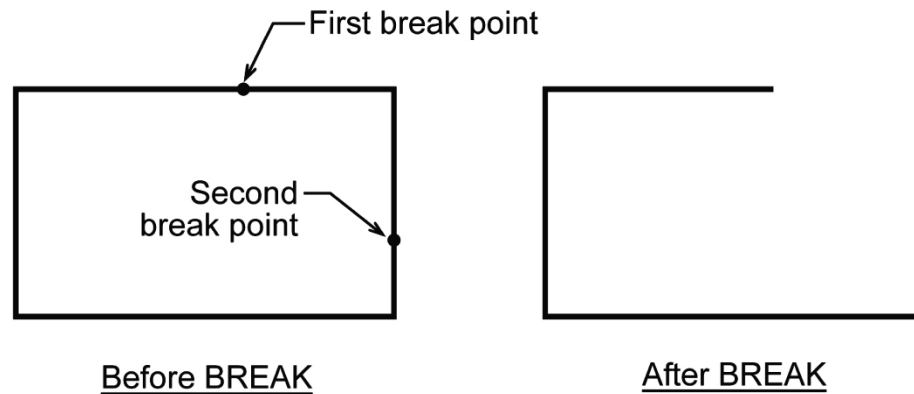



Figure 2.15-11: Breaking an object

The BREAK command may be accessed in the following way.


- Modify panel: 
- Command window: **br** or **break**

2.15.15) Join

The **Join** command joins to separate objects into one object. The command prompt will change depending on the source object. These are the restrictions of the JOIN command for the following source objects:

- Line: The lines must be collinear (lying on the same infinite line) but can have gaps between them.
- Polyline: The joined objects can be lines, polylines, or arcs. The objects cannot have gaps between them and must lie on the same plane parallel to the UCS xy plane.
- Arc: The arcs must lie on the same imaginary circle, but can have gaps between them. The CLOSE option converts the source arc into a circle. The arcs are joined counterclockwise beginning from the source object.
- Elliptical Arc: The elliptical arcs must lie on the same ellipse, but can have gaps between them. The CLOSE option closes the source elliptical arc into a complete ellipse. The elliptical arcs are joined counterclockwise beginning from the source object.
- Spline: The spline or helix objects must be contiguous (lying end-to-end). The resulting object is a single spline.
- Helix: The helix must be contiguous (lying end-to-end). The resulting object is a single spline.


The JOIN command may be accessed in the following way.

- Modify panel: 
- Command window: **j** or **join**

Joining a line to a line

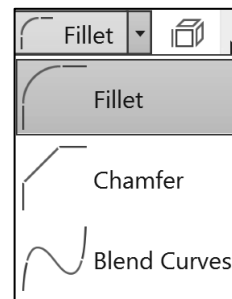
- 1) Command: **j** or **join** or Modify panel: 
- 2) Select source object or multiple objects to join at once: **Select the first line.**
- 3) Select objects to join: **Select a line to join to the first line.**
- 4) Select objects to join: **Select a line to join or hit **Enter** or **Space** to end selection.**
- 5) If the join was successful the prompt will read: **1 line joined to source.**

Joining an arc to create a complete circle

- 1) Command: **j** or **join** or Modify panel: 
- 2) Select source object or multiple objects to join at once: **Select the arc.**
- 3) Select arc to join to source or [cClose]: **close**
- 4) If the join was successful the prompt will read: **Arc converted to a circle.**

2.15.16) Chamfer

The **CHAMfer** command enables you to create a chamfer (an angled corner) between any two non-parallel lines or any two adjacent polyline segments. A chamfer is usually applied to intersecting lines. The lines do not have to intersect, but their separation cannot be more than the chamfer distance. The **FILLET**, **CHAMFER** and **BLEND CURVES** commands reside in the same icon stack.



The **CHAMFER** command may be accessed in the following way.

- Modify panel:  Chamfer
- Command window: **cha** or **chamfer**

CHAMFER options (Select first line or

[Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple]:).

- Undo: Enables you to undo the last chamfer while in the multiple mode.
- Polyline: Enables you to chamfer polylines.
- Distance: Enables you to specify two distances that defines the chamfer size.
- Angle: Enables you to specify an angle and a distance that defines the chamfer size.
- Trim: Enables you to specify whether you want the original line trimmed or not trimmed.
- mEthod: Controls whether a chamfer is created using two distances or a distance and an angle.
- Multiple: Allows you to create several chamfers within the same command.

2.15.17) Fillet

The **Fillet** command is a very useful tool which allows you to draw a tangent arc between two objects. The objects are usually intersecting. The objects do not have to intersect, but their separation cannot be more than the fillet radius. It's worth experimenting with this command. It can save you a lot of time and enables you to construct shapes which otherwise would be quite difficult.

The FILLET command may be accessed in the following way.

- Modify panel:  Fillet
- Command window: **f** or **fillet**


FILLET options (Select first object or [Undo/Polyline/Radius/Trim/Multiple] :).

- Undo: Enables you to undo the last fillet while in the multiple mode.
- Polyline: Enables you to fillet polylines.
- Radius: Enables you to specify the radius of the fillet.
- Trim: Enables you to specify whether you want the original line trimmed or not trimmed.
- Multiple: Allows you to create several fillets within the same command.

2.15.18) Blend curves

The **BLEND curves** command creates a tangent or smooth spline between two open curves. The selected curves remain unchanged and the shape of the connecting spline depends on the specified continuity.

The BLEND CURVES command may be accessed in the following way.

- Modify panel:  Blend Curves
- Command window: **bl** or **blend**

2.15.19) Explode

The **EXPLODE** command is used to break apart single objects into their constituent parts. In other words, the command is used to return blocks, polylines, rectangles, etc... (which may be composed of a number of component objects) back to their individual component parts.

The EXPLODE command may be accessed in the following way.

- Modify panel: 
- Command window: **explode**

2.16) OBJECT SNAP COMMANDS

The *Object Snap* commands (Osnaps for short) are drawing aids which are used in conjunction with other commands to help you draw accurately. Osnaps allow you to snap to specific geometric locations on an existing object. For example, you can accurately pick the end point of a line or the center of a circle. Osnaps are so important that you cannot draw quickly or accurately without them. For this reason, you must develop a good understanding of what the Osnaps are and how they work.

The *Object Snap* panel shown in Figure 2.16-1 contains commands that allow you to snap to the geometry of an existing object. The *Object Snap* panel is not usually displayed by default, but you can add it through the CUI.

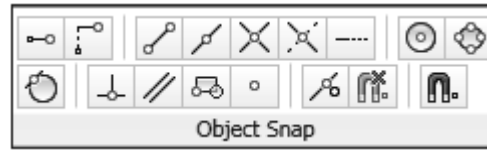



Figure 2.16-1: The *Object Snap* panel

When using Osnaps, you only need to pick a point which is near the desired point because AutoCAD automatically snaps to the location implied by the particular Osnap you are using. While drawing, notice that when you move the cursor close enough to an Osnap location, it is highlighted with an Osnap marker. Each Osnap has a different marker. Figure 2.16-2 shows examples of some of the different Osnap markers.



Endpoint Midpoint Intersection Center Quadrant Tangent Perpendicular Parallel Nearest

Figure 2.16-2: Object snap markers

Object snaps are so important that AutoCAD automatically detects an object's geometric locations while you are drawing. This automatic feature may be turned on and off in the *Status Bar* by clicking on the **Object Snap**  icon or by using the command **OSNAP**. To override an automatically chosen Osnap, you can select your preferred Osnap in the *Object Snap* panel or type the command in the *Command* window.

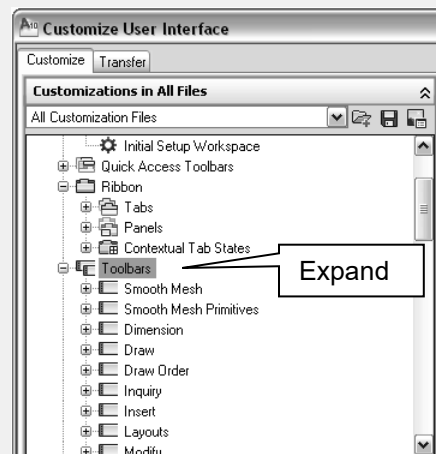
Adding the *Object Snap* panel to the *Home* tab

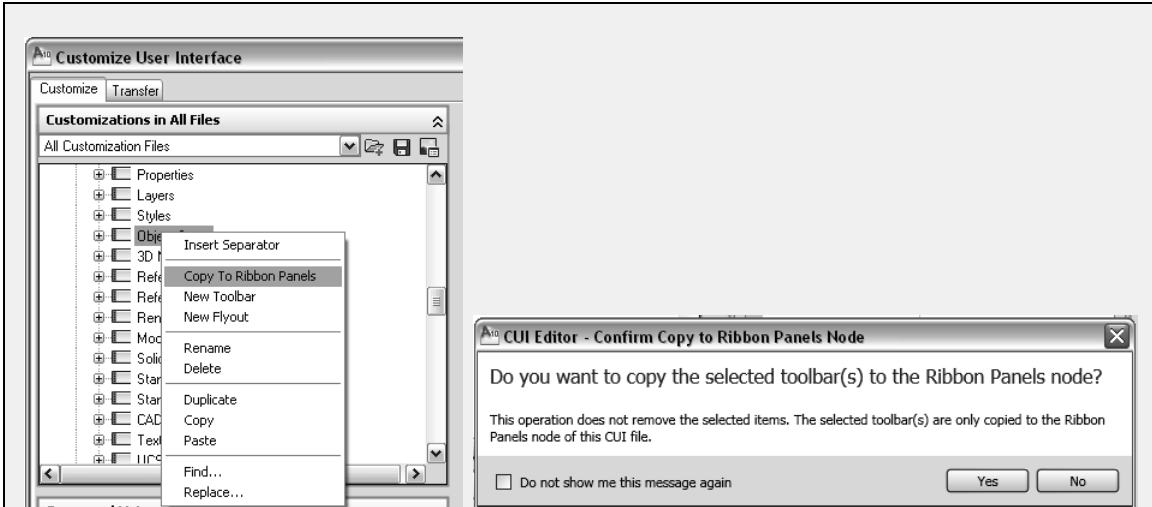
1. Command: **cui** or



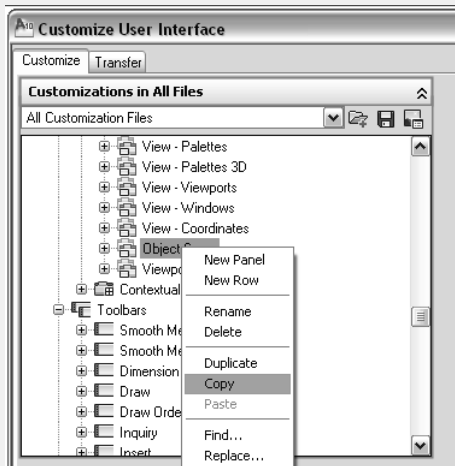
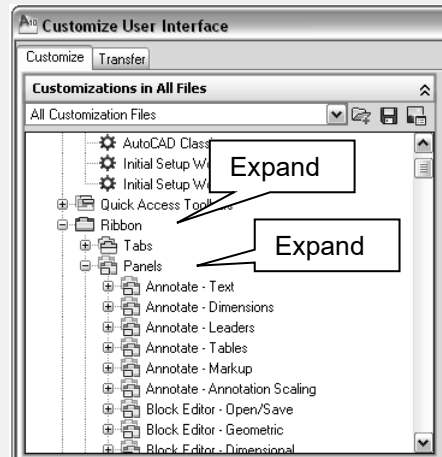
Manage tab – Customization panel:

2. Customize User Interface window: Expand the *Toolbars* in the tree view on the left side of the window.
 - a) Scroll down until you see the *Object Snap* toolbar. Right click on *Object Snap* and select **Copy to Ribbon Panels**.
 - b) Select **Yes** in the *CUI Editor – Confirm Copy to Ribbon Panels Node Window*, if one appears.

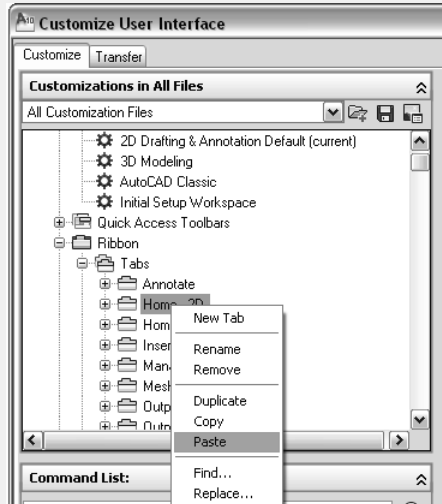




- c) Scroll up and expand the *Ribbon* and *Panels* in the tree view on the left side of the window.
- d) Scroll down until you see the *Object Snap* panel. Right click and select **Copy**.
- e) Within the *Tabs* root, right click on the *Home 2D* tab and select **Paste**.
- f) Select **OK**.




- g) The *Object Snap* panel should automatically show up in the *Home* tab. You can hide or show panels by right clicking on an existing panel and selecting panel. The check mark means that the panel is showing.



2.16.1) Endpoint

The **ENDpoint** Osnap snaps to the end points of lines and arcs and to polyline vertices.


The ENDPOINT command may be accessed in the following way.

- Object Snap panel: 
- Command window: **end** or **endpoint**

2.16.2) Midpoint

The **MIDpoint** Osnap snaps to the midpoint of a line, an arc and a polyline segment.


The MIDPOINT command may be accessed in the following way.

- Object Snap panel: 
- Command window: **mid** or **midpoint**

2.16.3) Intersection

The **INTersection** Osnap snaps to the physical intersection of any two drawn objects (i.e., where lines, arcs or circles cross each other) and to polyline vertices. However, this Osnap can also be used to snap to intersection points which do not physically exist. This feature is called the *Extended Intersection*. To use the extended intersection feature, you must pick two points to indicate which two objects should be used.


The INTERSECTION command may be accessed in the following way.

- Object Snap panel: 
- Command window: **int** or **intersection**

2.16.4) Apparent intersection

The Apparent Intersection (**APPINT**) Osnap snaps to the point where objects (on different planes) appear to intersect in the current view. For example, you may be looking at a drawing in plan view where two lines cross; however, since AutoCAD is a 3 dimensional drawing environment, the two lines may not physically intersect.

The Apparent Intersection (APPINT) command may be accessed in the following way.

- Object Snap panel: 
- Command window: **appint**

2.16.5) Extension

The **EXTension** Osnap enables you to snap to some point along the imaginary extension of a line, arc or polyline segment. To use this Osnap, you must hover the cursor over the end of the object. When the end is found, a small cross appears at the endpoint and a dashed extension line is displayed from the endpoint to the cursor. The tool tip for

the extension displays the relative polar coordinate of the current cursor position. This can be a useful guide for positioning your next point. In the case of the arc extension, the tool tip displays the distance along the arc. Once the extension is visible the distance from the endpoint may be entered. Figure 2.16-3 shows the extension snap in action.

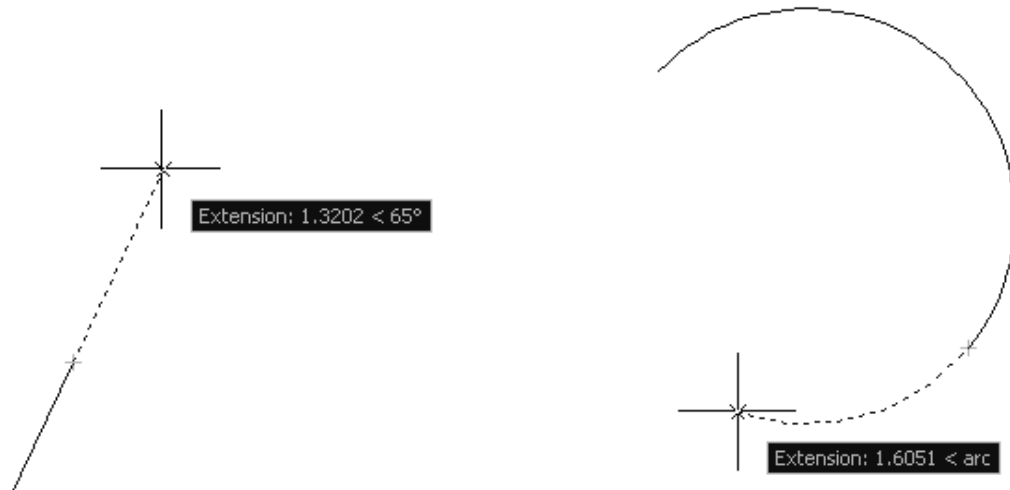



Figure 2.16-3: Extension snap


The EXTENSION command may be accessed in the following way.

- Object Snap panel: 
- Command window: **ext** or **extension**

2.16.6) Center

The **CENter** Osnap snaps to the center of a circle, arc or polyline arc segment. The cursor must pass over the circumference of the circle or the arc before the center can be found.


The CENTER command may be accessed in the following way.

- Object Snap panel: 
- Command window: **cen** or **center**

2.16.7) Quadrant

The **QUADrant** Osnap snaps to one of the four circle quadrant points located at north, south, east and west or 90, 270, 0 and 180 degrees respectively.


The QUADRANT command may be accessed in the following way.

- Object Snap panel: 
- Command window: **quad** or **quadrant**

2.16.8) Tangent

The **TANgent** Osnap snaps to a tangent point on a circle or arc. This Osnap works in two ways. For example, you can either draw a line from a point to the tangent point or you can draw a line between two tangent points. The second method does not give you a rubber band line to view when selecting the second point.


The TANGENT command may be accessed in the following way.

- Object Snap panel: 
- Command window: **tan** or **tangent**

2.16.9) Perpendicular

The **PERpendicular** Osnap snaps to a point which forms a 90 degree angle between the selected object and the object being drawn.


The PERPENDICULAR command may be accessed in the following way.

- Object Snap panel: 
- Command window: **per** or **perpendicular**


2.16.10) Parallel


The **PARallel** Osnap is used to draw a line parallel to any other line in your drawing.

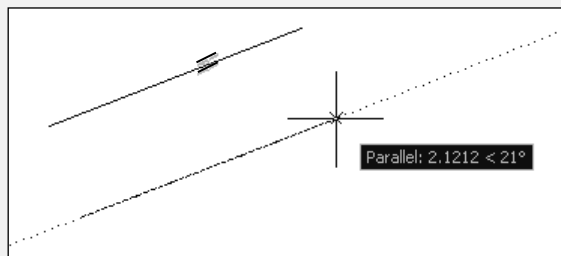
The PARALLEL command may be accessed in the following way.

- Object Snap panel: 
- Command window: **par** or **parallel**

Drawing a line parallel to another line

- 1) Command: **l** or 
- 2) Specify first point: Select or enter your first point.
- 3) Specify next point or [Undo]: **par** or


Object Snap panel: 
to Move your cursor over the reference line. The parallel marker will appear. Move your cursor to a position that is approximately parallel to the reference line. A dashed line will appear that indicates you will create a line parallel to the reference line.



2.16.11) Nearest

The **NEARest** Osnap snaps onto an existing object. This Osnap is useful if you want to make sure that the point lies on an object but you don't necessarily mind exactly where it is located.


The NEAREST command may be accessed in the following way.

- Object Snap panel: 
- Command window: **nea** or **nearest**

2.16.12) Snap from




The SNAP **FROM** Osnap is a little more complicated than the other object snaps but it can be very useful. The SNAP FROM Osnap does not snap to an object, rather it snaps to a point at some distance or offset from an object snap location.

The SNAP FROM command may be accessed in the following way.

- Object Snap panel: 
- Command window: **from**

Using the SNAP FROM Osnap


The following command sequence creates a line that starts 1 unit away from the midpoint of an existing line.

- 1) Command: **l** or 
- 2) Specify first point: **from** or Object Snap panel: 
- 3) Base point: **mid** or Object Snap panel: 
of Select the midpoint of the reference line.
- 4) <Offset>: **1**
- 5) Specify next point or [Undo]: Select the second point.

2.16.13) Insert

The **INSert** Osnap snaps to the insertion point of a block, text or an image.


The INSERT command may be accessed in the following way.

- Object Snap panel: 
- Command window: **ins** or **insert**

2.16.14) Node

The **NODE** Osnap snaps to the center of a point, a dimension definition point or a dimension text origin. This Osnap can be useful if you have created a number of POINTS with the MEASURE OR DIVIDE commands.



The NODE command may be accessed in the following way.

- Object Snap panel: 
- Command window: **node**

2.16.15) Geometric Center

The *Geometric Center* osnap snaps to the geometric center of a closed object (e.g. rectangle, polygon, closed spline). The *Geometric Center* snap is not accessible in the *Object Snap* panel, but can be automatically detected or access using the command **GCE**.

2.16.16) Object snaps settings

Individually picking or entering the *Object Snaps* commands can sometimes be a time consuming process. With *Object Snap*  set to on, AutoCAD will automatically snap to a geometric location of an object. The specific snaps that AutoCAD will detect can be set in the *Object Snap* tab of the *Drafting Settings* window (Figure 2.16-4). This can be accessed by selecting the **Object Snap Settings**  icon, by right clicking on OSNAP in the *Status* bar and selecting **Settings...** or by entering **OSNAP** in the *Command* window. The snaps that have a check mark next to them will be automatically detected. You can set the *AutoSnap Markers Size* and *Aperture Size* by selecting the *Options...* button (Figure 2.16-5). The aperture size controls the size of the area around the current cursor position used to search for object snaps.

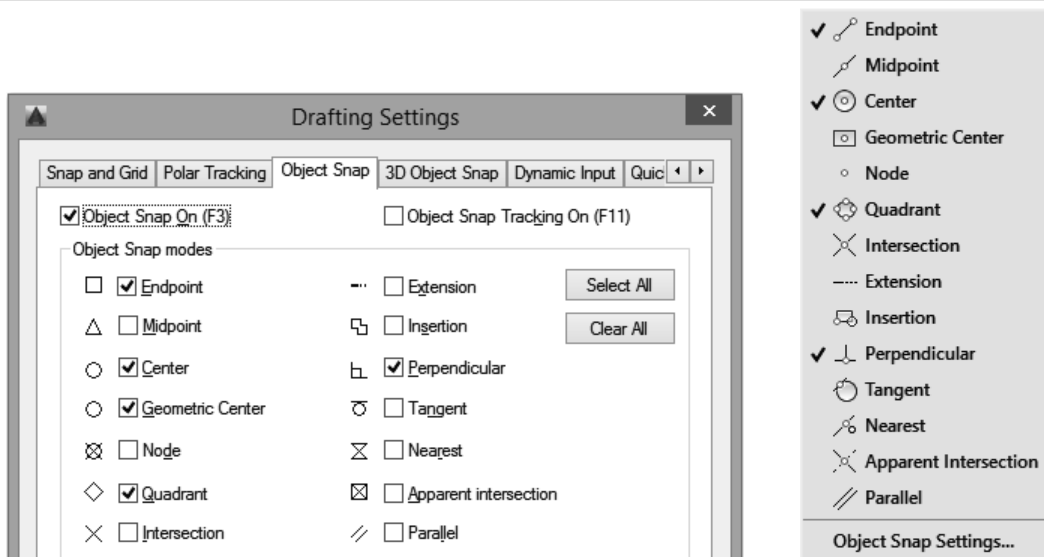
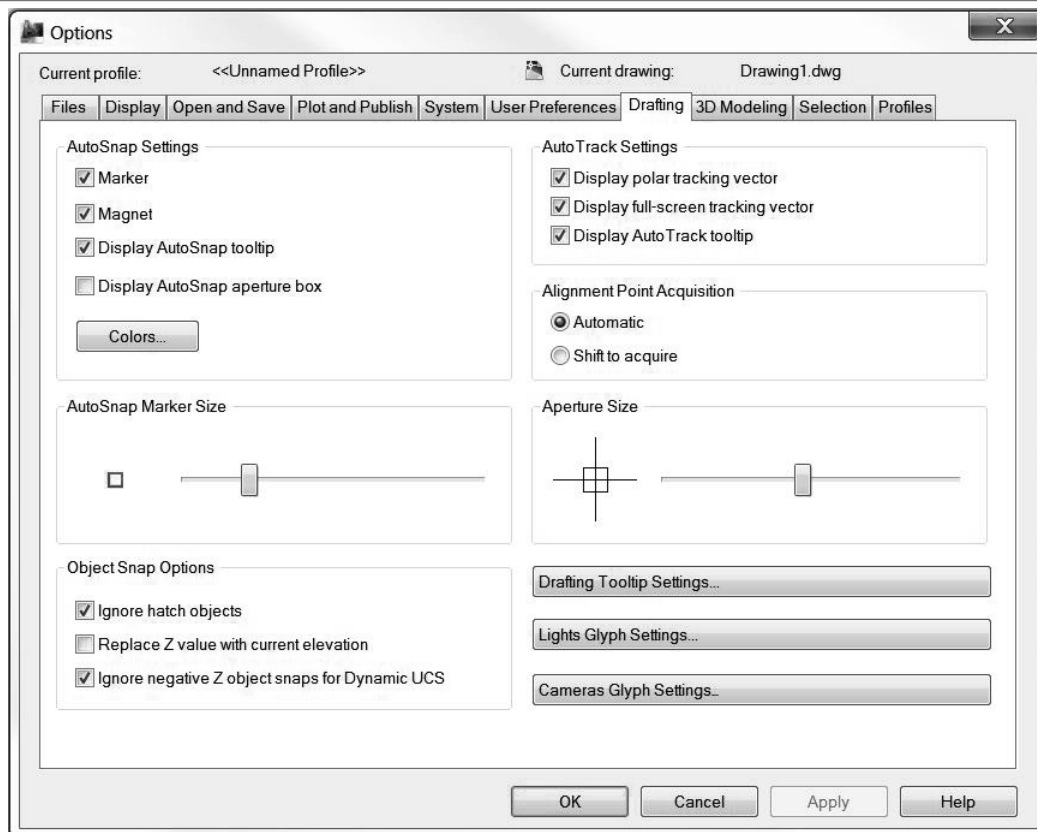



Figure 2.16-4: *Object Snap* settings

Figure 2.16-5: *Object Snap* options

2.16.17) Object snap cycling and aperture

Using object snaps is a great way to construct accurate drawings. However, when a drawing becomes very complex, it can be quite difficult to pick the exact point you want. This is particularly problematic if there are a number of possible snap points in close proximity. There are several ways to circumvent this problem. You can identify the particular Osnap that you wish to use by selecting it in the *Object Snap* panel or entering it in the *Command* window, or you can decrease the size of your aperture box.

Another option is to use the object snap cycling feature. This allows you to cycle through all valid snap points within the aperture area until you find the one you want. This

feature only works when *Object Snap*  is turned on. Once a snap marker appears, you can cycle through other local snap points by pressing the **Tab** key. Each time **Tab** is pressed, the next snap point is highlighted along with the object or objects to which it belongs. Using this feature, you can be absolutely sure that you are selecting the point you want, no matter how complex the arrangement of objects.

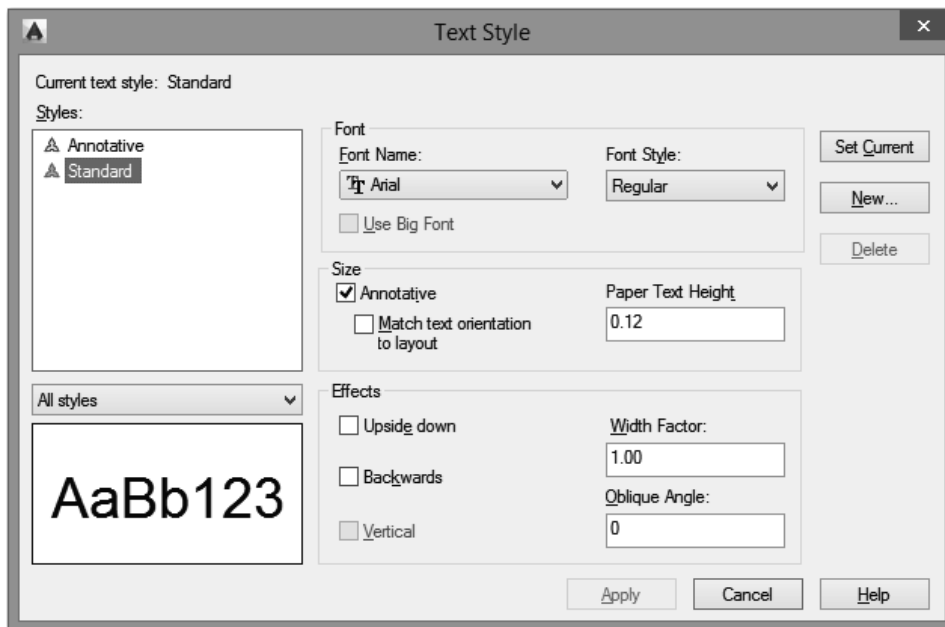
2.17) WAGON TUTORIAL


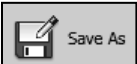
The objective of this tutorial is to draw several different entities such as LINES, CIRCLES, POLYGONS, etc..., and use several different modifying commands such as MOVE and COPY.

AutoCAD® allows you to draw many predefined entities or shapes. Some entities can be defined in many different ways. For example, a circle can be defined by a center point coordinate and a radius, or by two tangent points and a radius. During each command, look at the command prompt and make note of the different options. Also, notice the commands enclosed in angled brackets < >. They are the default values or options and are selected by pressing the **Enter** or **Space** key.

2.17.1) Preparing to draw


- 1) View the *Modify, Move & Copy* and *Text* videos and read sections 2.13) to 2.16).
- 2) Open your **set-inch.dwt** template.
- 3) Set your default text font and size.
 - **Command: SStyle**
 - **Text Style window:** Set your text *Font Name* to **Arial**, check the **Annotative** toggle, and set your text *Height* to **0.12**, select **Apply** and then **Close**.



- 4)  (You have just added features to your template file.)
- 5)  **Wagon Tut.dwg.**

6) Activate a **GRID** that is **0.5 x 0.5**.

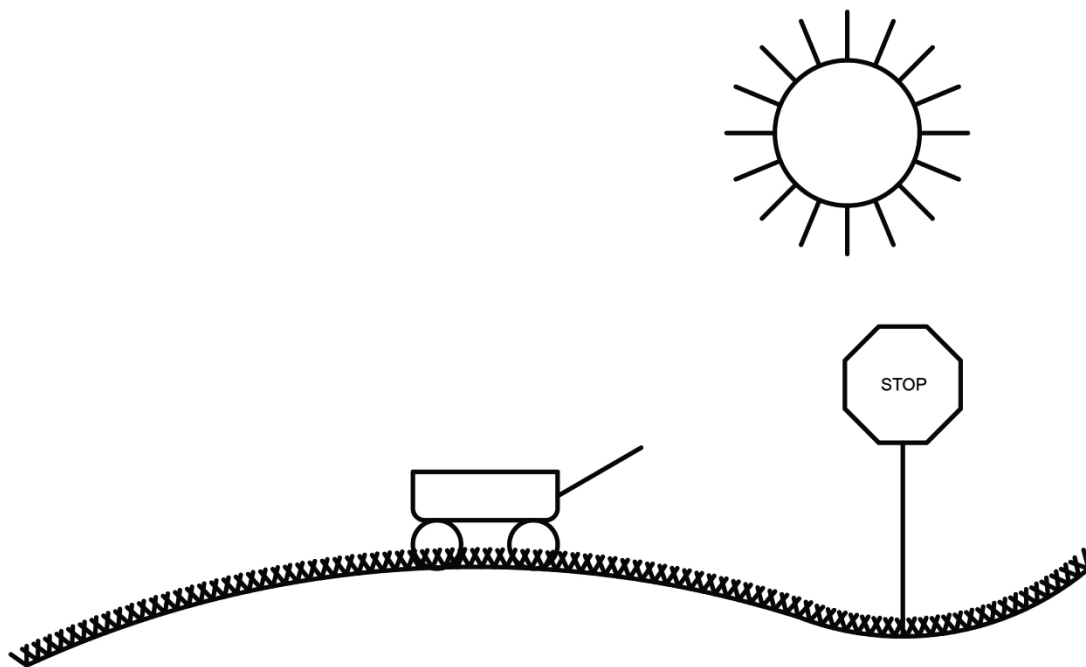
7)  (**Zoom All**).

8) Enter your WCS  (*View tab - Coordinates panel*) or (**UCS - W**).

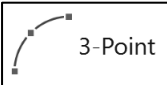
9) Turn your **Dynamic input** off and your **Object Snap** off in the *Status bar*.

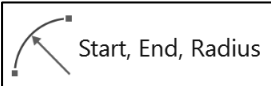
2.17.2) Drawing

You will be drawing the following scene.



1) Draw the ground using two **Arcs**.

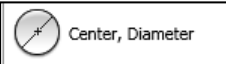
- a) Home tab - Draw panel:  3-Point
- b) Specify start point of arc or [Center]: **0,1**
- c) Specify second point of arc or [Center/End]: **4,2**
- d) Specify end point of arc: **8,1.5**

- e) Home tab - Draw panel:  Start, End, Radius
- f) Specify start point of arc or [Center]: **end**
- g) of **Select the right end of the existing arc (right side).**
- h) Specify end point of arc: **11,2**
- i) Specify center point of arc (hold Ctrl to switch direction): **3**

Note: By selecting a specific type of arc from the pull-down menu, AutoCAD® automatically inputs the correct options.

- 2) **Join the two arcs.**
 - a) Command: **j** or **join**
 - b) Select source object or multiple objects to joint at once: **Select one of the arcs.**
 - c) Select objects to joint: **Select the other arc.**
 - d) Select objects to join: **Enter** (If the join was successful, it will say 2 objects converted to 1 polyline.)


- 3) **Add grass to your ground.**
 - a) Draw a **line** that is **0.2** units long at an angle of **145°** starting at the left **end** of the ground.
 - i. Command: **l** or **line**
 - ii. Specify first point: **end**
 - iii. of **Select the left end of the arc.**
 - iv. Specify next point or [Undo]: **@0.2<145**
 - v. Specify next point or [Undo]: **Enter**
 - b) Draw a **line** that is **0.2** units long at an angle of **90°** starting at the left **end** of the ground.
 - c) **Array** the grass blade along the **path** of the ground.
 - i. Command: **ar** or **array**
 - ii. Select objects: **Select the two lines that you just drew.**
 - iii. Select objects: **Enter**
 - iv. Enter array type [Rectangular/PAtH/POLar] <Rectangular>: **pa**
 - v. Select path curve: **Select the ground.**
 - vi. Select grip to edit array or [ASsociative/Method/Base point/Tangent direcction/Items/Rows/Levels/Align items/Z direction/eXit] <eXit>: **i**
 - vii. Specify the distance between items along path or [Expression] <0.25>: **0.1**
 - viii. Specify number of items or [Fill entire path/ Expression] <114>: **f**
 - ix. Select grip to edit array or [ASsociative/Method/Base point/Tangent direcction/Items/Rows/Levels/Align items/Z direction/eXit] <eXit>: **x**


- 4) **Create the wagon wheels.**
 - a) Draw a **Circle**  that has a diameter of **0.5** inches and whose center is located at **4.25,2.25**.

How?


- ii. Command: **c** or **circle**
- iii. Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **4.25,2.25**
- iv. Specify radius of circle or [Diameter] <1.00>: **d**
- v. Specify diameter of circle <2.00>: **0.5**

b) **COPY** the circle 1 inch to the right.

- i. Command: **co** or **copy** or Modify panel: 
- ii. Select object: Use the square cursor to select the circle with the mouse. Place the square on the border of the circle and click the left mouse button.
- iii. Select object: **Enter**
- iv. Specify base point or [Displacement/mOde] <Displacement>: **o**
- v. Enter copy mode option [Single/Multiple] <Multiple>: **s**
- vi. Specify base point or [Displacement/mOde/Multiple] <Displacement>: **Pick a point anywhere within the drawing area.**
- vii. Specify second point or <use first point as displacement>: **@1<0** (This moves the circle 1 inch to the right or at an angle of 0 degrees.)


5) Create the body of the wagon by drawing a **1.5 x 0.5 RECTangle**  whose lower left corner starts at **(4,2.5)**.

How?

- a) Command: **rec** or **rectangle** or 
- b) Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **4,2.5**
- c) Specify other corner point or [Area/Dimensions/Rotation]: **d**
- d) Specify length for rectangles <10.00>: **1.5**
- e) Specify width for rectangles <10.00>: **0.5**
- f) Specify other corner point or [Area/Dimensions/Rotation]: **Select a point above the two wheels.**


6) Draw the handle of the wagon using a **Line** that is **1** inch long at a **30** degree angle and starts at the midpoint of the right side of the wagon.



- a) Command: **line** or **l** or 
- b) **LINE** Specify first point: **mid**
- c) **of** Select the midpoint of the right side of the wagon.
- d) Specify next point or [Undo]: **@1<30**
- e) Specify next point or [Undo]: **Enter**

7) Round the corners of the wagon body using a **0.125** radius **FILLET**.



- a) Command: **fillet** or Modify Panel: 
- b) Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **r**
- c) Specify fillet radius <0.00>: **0.125**
- d) Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **Select the bottom line of the wagon body and then select one of the vertical line of the wagon body.**
- e) Hit the **space** bar and then fillet the other bottom corner.

8) Create the sun body using a **Circle** of radius **0.75** and whose center is located at **8.5,6.5**.

- 9) Create the sun rays using a polar **ARray**. Each sun ray is **0.5** inches long.



- a) Command: **l** or **line** or
- b) Specify first point: **quad**
- c) of **Select the right side quadrant of the sun body circle.**
- d) Specify next point or [Undo]: **@0.5<0**
- e) Specify next point or [Undo]: **Enter**



- f) Command: **arraypolar** or Modify panel:
- g) Select objects: **Select the newly created line.**
- h) Select objects: **Enter**
- i) Specify center point of array or [Base point/Axis of rotation]: **cen**
- j) of **Select the circumference of the sun.**
- k) Select grip to edit array or [ASsociative/Base point/Items/Angle between/Fill angle/ROws/Levels/ROtate items/eXit] <eXit>: **i**
- l) Enter number of items in array or [Expression] <6>: **16**
- m) Select grip to edit array or [ASsociative/Base point/Items/Angle between/Fill angle/ROws/Levels/ROtate items/eXit] <eXit>: **x**

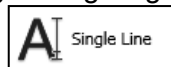
- 10) Create the stop sign post using a **Line** that starts at the **QUADrant** of the second ground arc and is **2** inches long and travels vertically upward.

- 11) Create the stop sign using an octagon (**POLygon**) that is circumscribed in a circle of radius **0.75** at the top of the sign post.



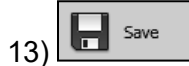
- a) Command: **pol** or **polygon** or Draw panel:
- b) Enter number of sides <4>: **8**
- c) Specify center of polygon or [Edge]: **ext** or **extension**
of **Place your cursor over the top end of the sign post. Once AutoCAD locates the end, move your cursor up to create an extension line. Enter 0.6 in the Command window without losing the extension line.**
- d) Enter an option [Inscribed in circle/Circumscribed about circle] <I>: **c**
- e) Specify radius of circle: **0.6**

- 12) Enter the word **STOP** in the middle of the stop sign using single line **text**.



- a) Command: **text** or Annotation panel:
- b) Select Annotation Scale Window: Select **OK** (Note: If you do not get this window, your text is not annotative. You need to fix this in the *Style Manager* window (**STYLE**).)
- c) Specify start point of text or [Justify/Style]: **j**
- d) Enter an option [Left/Center/Right/Align/Middle/Fit/TL/TC/TR/ML/MC/MR/BL/BC/BR]: **m**
- e) Specify middle point of text: Use the **EXTension** snap to locate a point **0.6** inches above the stop sign post like we did before.
- f) Specify rotation angle of text <0>: **Enter**

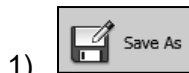
g) Type the word **STOP** and hit **Enter** twice.








2.17.3) Using modify commands

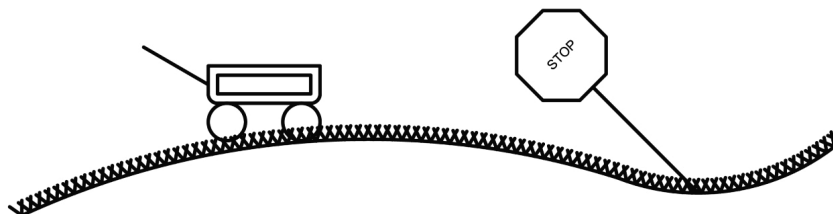
For the following commands, you will need to select objects that contain more than one entity. The easiest way to do this is to use a window. When the command asks you to *select object*, you will draw a window around the entire object using the following steps. Click your left mouse button and release, drag the mouse so that the box encloses the entire object, and click the left mouse button again. If an entity was not selected, use the mouse to select the entities individually. If an entity was selected and you do not want it to be selected, type an **R** for *remove* and select the entity.

IMPORTANT! While performing the following commands, **READ THE COMMAND PROMPT** to see what AutoCAD® is wanting.



1) **Wagon Tut Modified.dwg**

- 2) On your own, **Move**  the sun directly to the left by **6** units and **SCALE**  the sun by half using the center of the sun body as the base point.
- 3) On your own, **OFFSET**  the wagon body to the inside by **0.125** to produce a double line.
- 4) On your own, **Mirror**  the entire wagon using the left vertical edge of the wagon as the mirror line. Delete the original wagon.
- 5) On your own, **ROtate**  the stop sign **45** degrees using the bottom of the sign post as the base point.

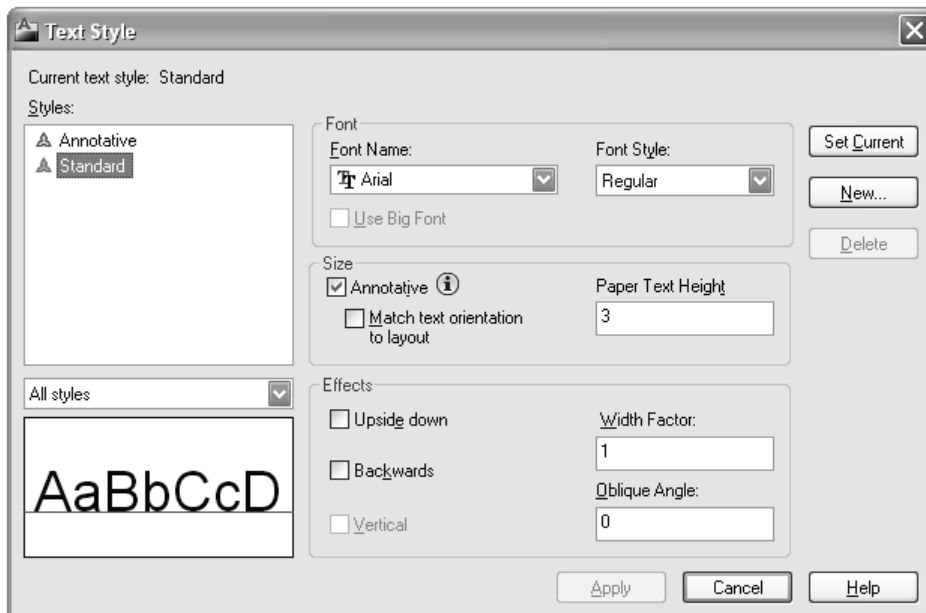


- 6) Print the completed drawings using **Limits** as your print area and the **Scale to Fit** option.

- 7)  Save

Exercise 2.17-1: Text Style

- 1) Open your **set-mm.dwt** template file. Set your text font to **Arial**, text height to **3 mm** and make the text **Annotative**. Resave your file.



2.18) SELECTING OBJECTS

When you start a *Modify* command such as ERASE, two things happen. First, the cursor changes from the usual *crosshairs* to a *pickbox* and second, you will see the `Select objects:` prompt in the *Command* window. Both of these cues are there to let you know that AutoCAD® is expecting you to select one or more objects. When an object has been picked it is highlighted to show that it is part of the current selection and the *Command* window reports `1 found`. The `Select objects:` prompt will continue to appear so that you can continue adding more objects to the current selection or you can press **Enter** or **space** to complete the selection. The most commonly used selection options are explained below.

- Selecting objects by picking: To select an object, place the *pickbox* over part of the object and click your left mouse button.
- Window selection: The WINDOW option is invoked by typing **W** in response to the `Select objects:` prompt or by clicking and releasing the left mouse button and then moving the mouse to the right and up or down. The WINDOW option allows you to define a rectangle using two points. Once the window is defined, all objects that lie entirely within the window will be selected.
- Crossing window selection: The CROSSING WINDOW option is invoked by typing **C** in response to the `Select objects:` prompt or by clicking and releasing the left mouse button and then moving the mouse to the left and up or down. Once the window is defined, all objects that lie inside or that cross the window will be selected.
- Lasso window selection: The LASSO WINDOW option is invoked by clicking and holding the left mouse button. If the mouse is moved to the right, only objects that lie entirely within the window will be selected. If the mouse is moved to the left, all objects that lie inside or that cross the window will be selected.

If you accidentally selected an object that you do not want to include in the modify command, type **R** or **Remove** in the *Command* window and select the object again. It will be removed from the selection group. If you then need to add another object to the selection group, you need to type **A** or **Add** and then select it. You can also quickly remove an object from the selection set by holding the **SHIFT** key down and selecting the object.

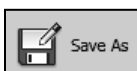
The **PICKADD** command controls whether subsequent selections replace the current selection group or add to it. If `PICKADD = 0`, then the object will replace the selected object. If `PICKADD = 1`, the object will be added to the selection group. If `PICKADD = 2`, the object will also be added to the selection group but with the added feature that the selected objects will remain selected when you exit the **SELECT** command. The size of the pickbox may be set in the *Options* window – *Selection* tab. If you click on the *Visual Effect Settings...* button, you can set the selection preview filters and the area selection effect.

2.19) OBJECT SELECTION TUTORIAL

AutoCAD® has a whole range of tools which are designed to help you select just the objects you need. This tutorial is designed to familiarize you with these selection options.

2.19.1) Selecting objects using the pickbox.

- 1) View the *Selection* video and read section 2.18).
- 2) Open your **set-mm.dwt** template file.

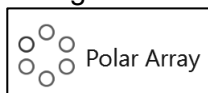


- 3) **Selection Tut.dwg**.

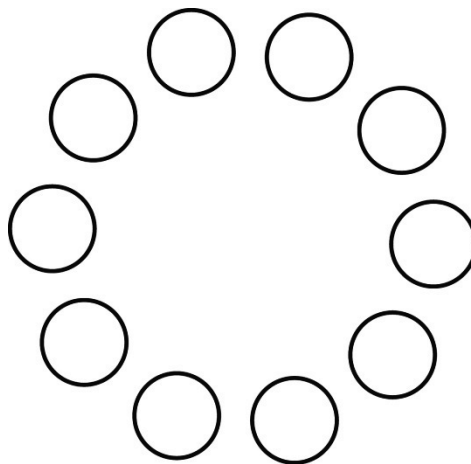
- 4) Get into your **World Coordinate System**.

- 5) Draw a **Circle** of radius **20** mm whose center is located at **75,100**.

- 6) **ARray** the *Circle* using the following settings.



- a) **Polar Array**
- b) Center point = **150,150**
- c) Total number of items = **10**
- d) Fill angle = **360**



- 7) **Zoom All**

- 8) **EXPLODE** the array group.

- a) Command: **explode** or

- b) Modify Panel:



- c) Select objects: Select the array group.
- d) Select objects: **Enter**



- 9)

- 10) Use the pickbox to **Erase** objects.

- a) Command: **e** or **erase** or

Modify panel: 

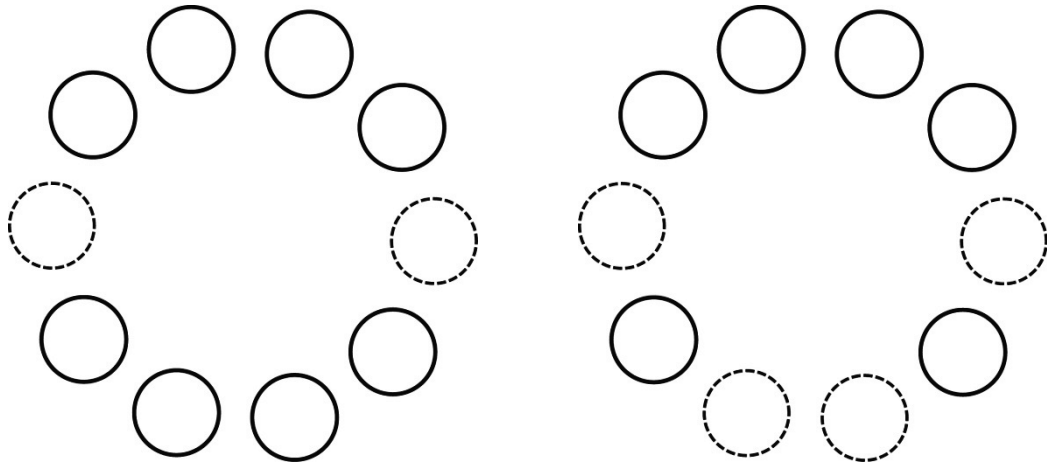
- b) Select objects: **all**

- c) Select objects: **r** (Note: This command allows you to remove an object from your selection set. You may also remove objects from the selection set by holding down the **shift** key.)


- d) Remove objects: Select all but the left and right circles using the pickbox as shown in the figure on the left.

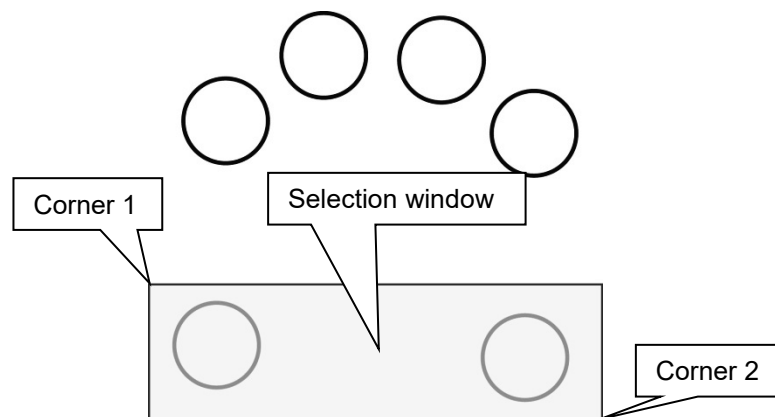
- e) Remove objects: **a**

- f) **Select objects:** Select the bottom two circles using the pickbox as shown in the figure on the right.
- g) **Select objects:** **Enter**




2.19.2) Selecting objects using a window

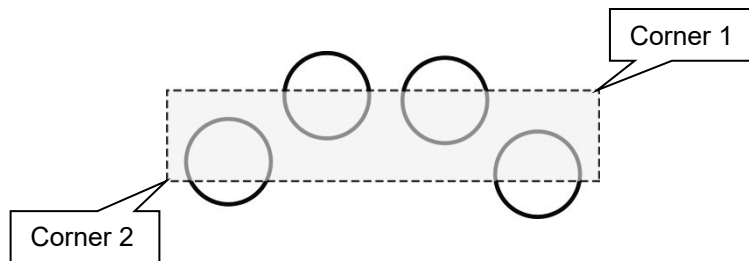
- 1) **Erase** the bottom two circles using a window.
- Command:** **e** or **erase** or 
 - Select objects:** Move your pickbox to the approximate location of *Corner 1* and click your left mouse button once.
 - Specify opposite corner:** Move the mouse down and to the right and click your left mouse button again at the approximate location of *Corner 2*. (Only objects that are completely enclosed in the window will be selected.)
 - Select objects:** **Enter**



2) **Erase** the top four circles using the crossing window option.

- a) **Command:** **e** or **erase** or 
- b) **Select objects:** Move your pickbox to the approximate location of *Corner 1* and click your left mouse button once.
- c) **Specify opposite corner:** Move the mouse down and to the left and click your left mouse button again at the approximate location of *Corner 2*. (Objects that are completely enclosed in or cross the window will be selected.)
- d) **Select objects:** **Enter**


Note: Clicking left to right will create a standard window, and clicking right to left creates a crossing window.

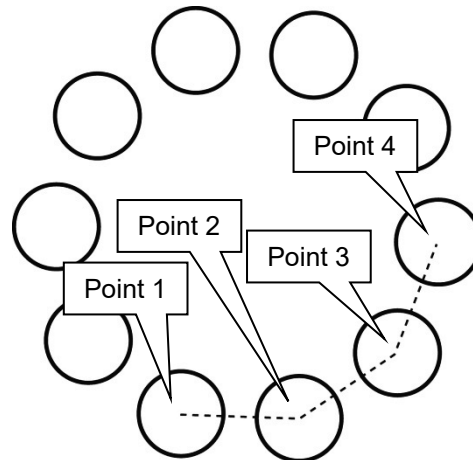


2.19.3) Selecting objects using advanced techniques


1) **Close** and **don't** save your drawing and reopen **Selection Tut.dwg**. This will give you your full array of circles back. If you forget to save or saved in between steps, you will need to draw the circle array again.

2) **Erase** selected circles using a **Fence** line.

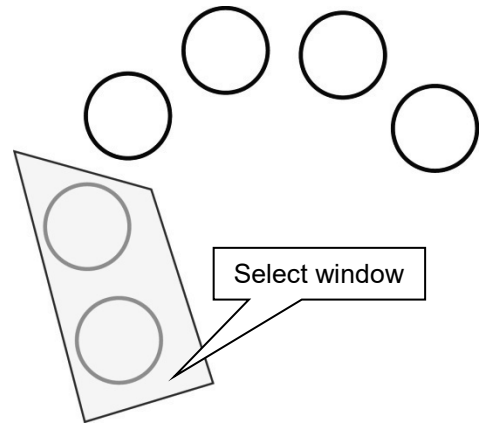
- b) **Command:** **e** or **erase** or 
- c) **Select objects:** **f**
- d) **Specify first fence point:** Select a point near *Point 1*.
- e) **Specify next fence point or [Undo]:** Select a point near *Point 2*.
- f) **Specify next fence point or [Undo]:** Select a point near *Point 3*.
- g) **Specify next fence point or [Undo]:** Select a point near *Point 4*.
- h) **Specify next fence point or [Undo]:** **Enter** (Note: All objects that cross the fence line are selected.)
- i) **Select object:** **Enter**




3) **Erase** selected circles using a **Window Polygon**.

- a) Command: **e** or **erase** or 
- b) Select objects: **wp**
- c) On your own, create the selection window shown. **IMPORTANT!** To end the polygon window hit **Enter**.

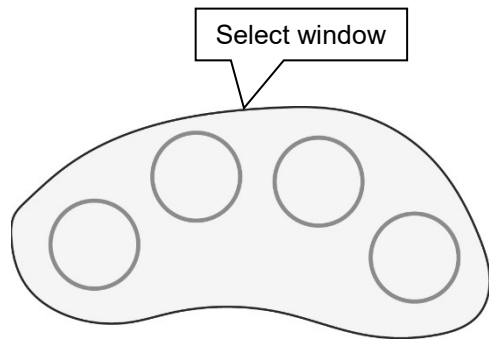
Note: A crossing polygon window option may be invoked by typing **CP**.



4) **Erase** selected circles using a **Lasso** window.

- d) Command: **e** or **erase** or 
- e) Select objects: Click and hold your left mouse button while dragging your mouse to create the selection window shown.

Note: If you drag your mouse to the left, you will create a crossing window.

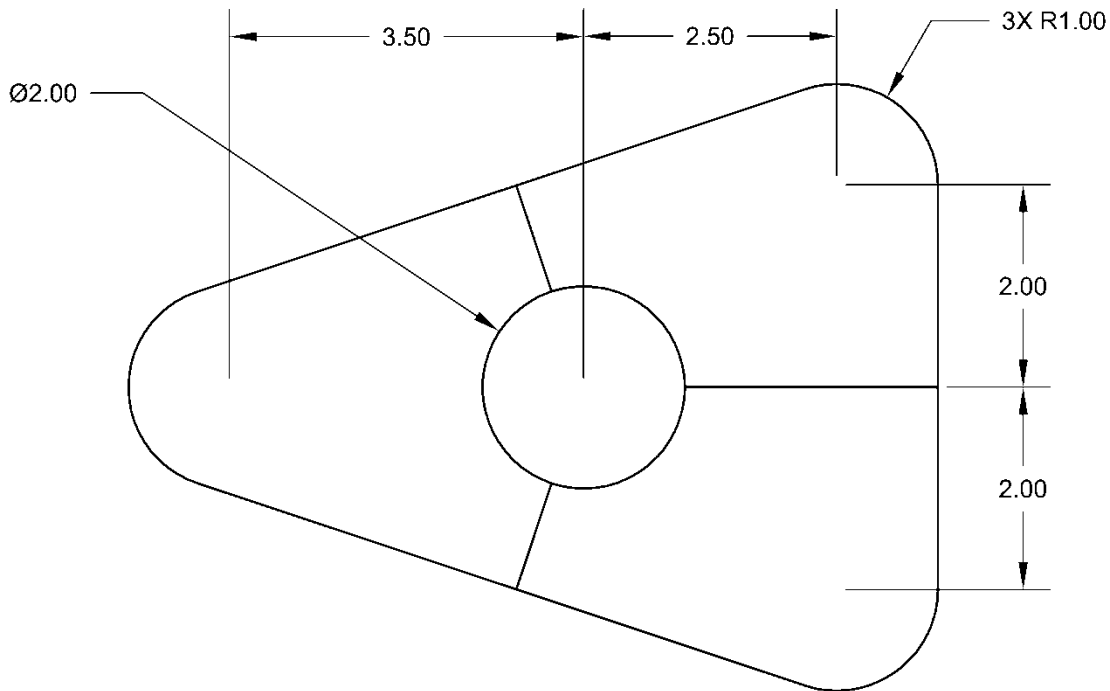


2.20) OBJECT SNAP TUTORIAL

The objective of this tutorial is to familiarize the reader with the OSNAP and TRIM commands. Object snaps automatically select specified geometric locations on an existing object such as the endpoint or the center. Osnaps help you draw accurately and quickly. In this tutorial we will be using the Automatic Object Snaps. Therefore, you will not have to type the OSNAP command in the *Command* window. For the most part, AutoCAD will automatically detect them

2.20.1) Drawing using osnaps

We will be drawing the following object.

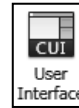


1) View the *Object Snaps* video and review section 2.16.

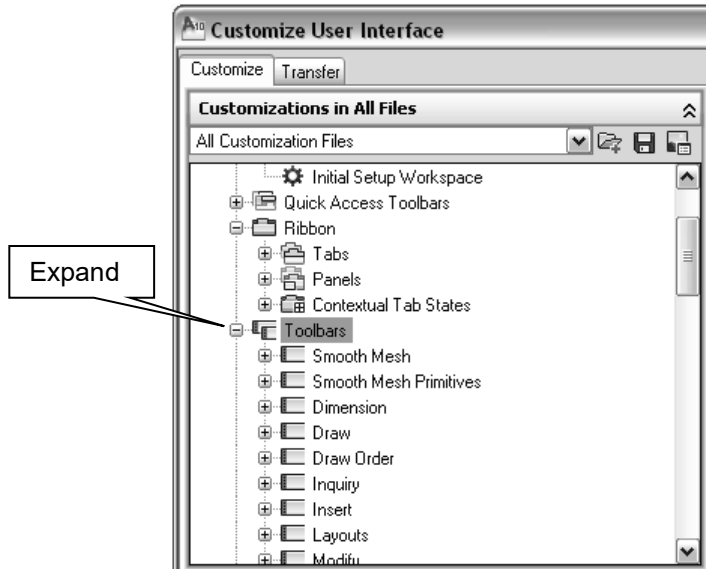
2)  **set-inch.dwt.**

3)  **OSNAP Tut.dwg.**

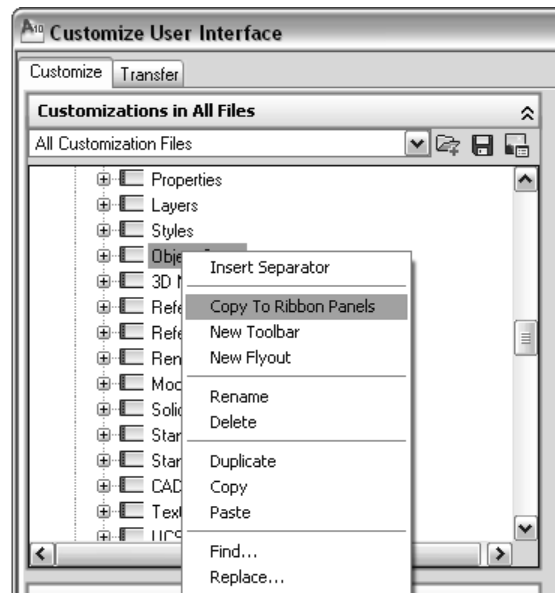
4) Add the *Object Snap* panel to the *Home* tab.



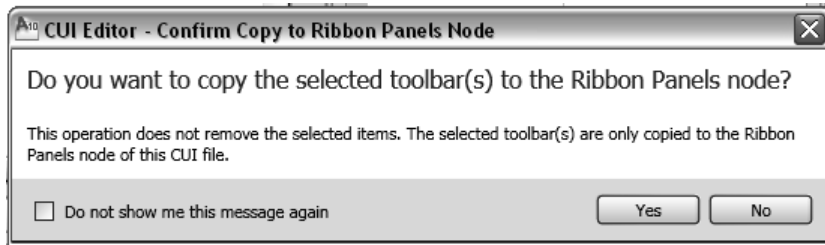
- a) Command: cui or Manage tab – Customization panel:
- b) Customize User Interface window: Expand the *Toolbars* in the tree view on the left side of the window.



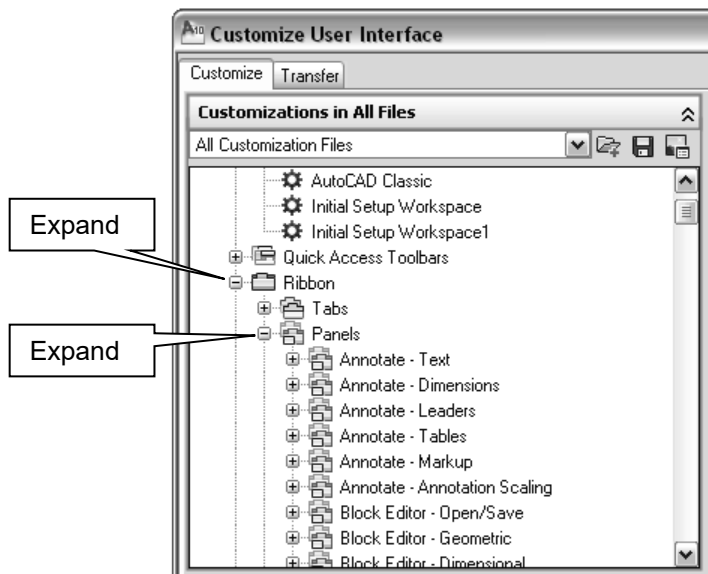
- c) Scroll down until you see the *Object Snap* toolbar. Right click on *Object Snap* and select **Copy to Ribbon Panels**.



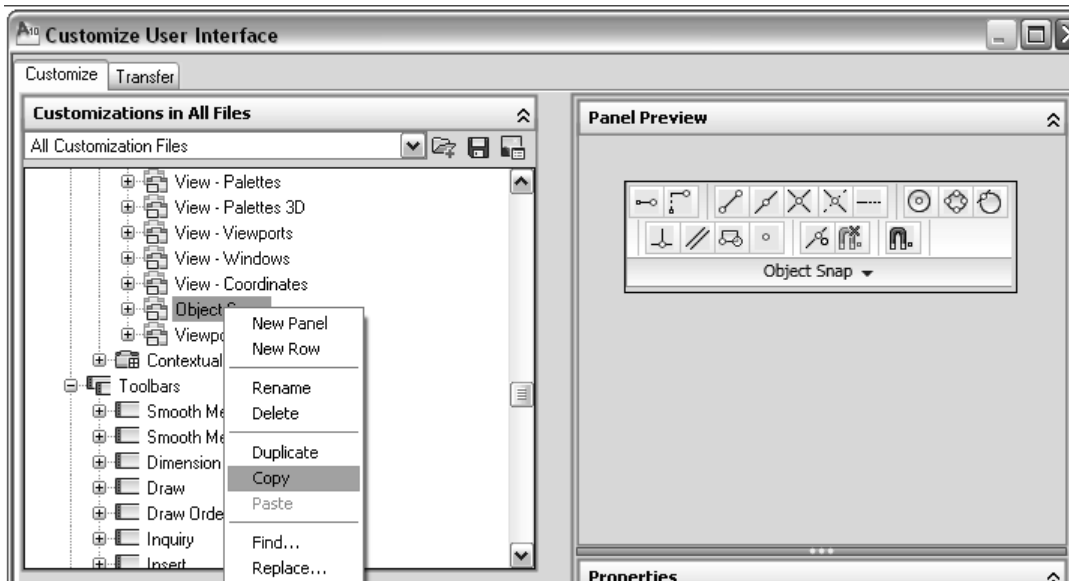
- d) Select **Yes** in the *CUI Editor – Confirm Copy to Ribbon Panels Node* Window, if one appears.



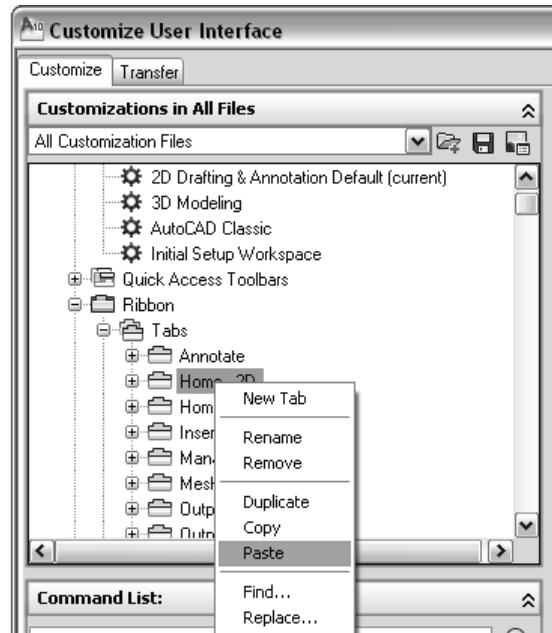
- e) Scroll up and expand the *Ribbon* and *Panels* in the tree view on the left side of the window.



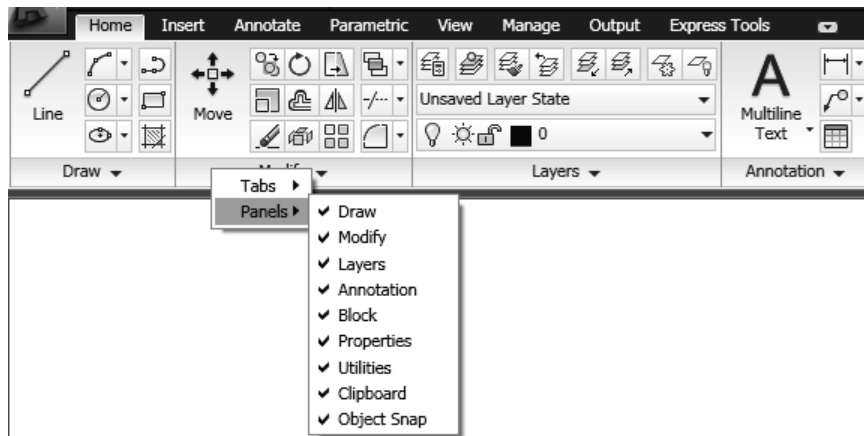
- f) Scroll down until you see the *Object Snap* panel. Right click and select **Copy**.



- g) Within the *Tabs* root, right click on the *Home 2D* tab and select **Paste**.
 h) Select **OK**.




- i) The *Object Snap* panel should automatically show up in the *Home* tab. You can hide or show panels by right clicking on an existing panel and selecting *Panels*. The check mark means that the panel is showing.



Note: You may rearrange the panels by clicking on the *Panel name* and dragging it to a new location.

- 5) Enter your **World coordinate system**.
- 6) Draw a **Circle** of diameter 2 whose center is located at **5,4**.
- 7) **COPY** the circle to create the three outer circles. Look at the initial drawing to get the distances. Try to do this without looking at the How? box.

How?

a) Command: **co** or **copy** 

b) Select objects: **Select the circle.**

c) Select objects: **Enter**

d) Specify base point or [Displacement/mOde/Multiple]
<Displacement>: **m**

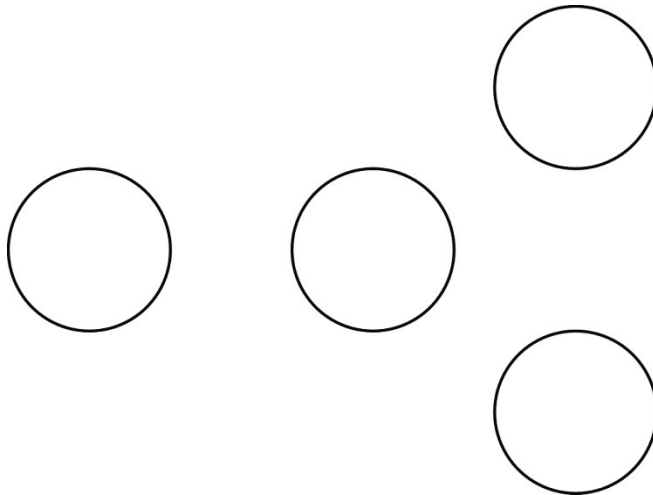
e) Specify base point or [Displacement/mOde/Multiple]
<Displacement>: **Select any point in the drawing area.**

f) Specify second point or [Array] <use first point as displacement>: **@2.5,2**


g) Specify second point or [Array/Exit/Undo] <Exit>: **@2.5,-2**

h) Specify second point or [Array/Exit/Undo] <Exit>: **@-3.5,0**

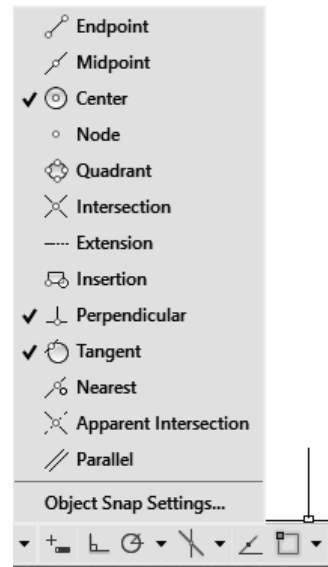
i) Specify second point or [Array/Exit/Undo] <Exit>: **Enter**



- 8) **Zoom All**

9) Turn your **Object Snap** on  in the *Status Bar* and tell AutoCAD® to automatically detect centers, tangencies and perpendicularities.

a) Set up your *Object Snap* options by clicking on the arrow next to the *Object Snap* icon in the *Status bar*. Select **Center**, **Perpendicular** and **Tangent**.

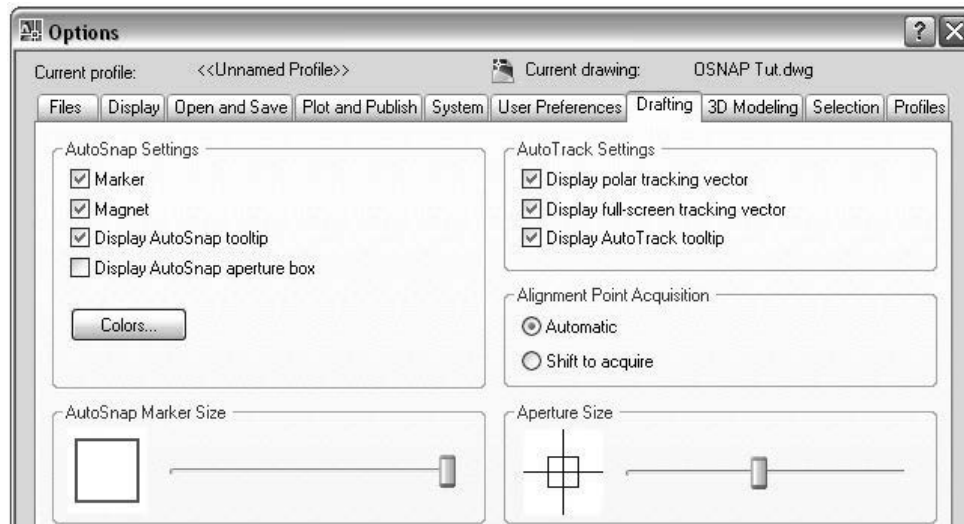


10) Set the size and color of the AutoSnap marker.

a) Application - Options window – Drafting tab: Set the *AutoSnap Marker Size* to maximum and then click the **Colors...** button.

b) Drawing Window Colors: Set the autosnap marker color to **red** and then click on the **Apply & Close** button.

c) Click **OK** in the *Options* and *Drafting Settings* windows.

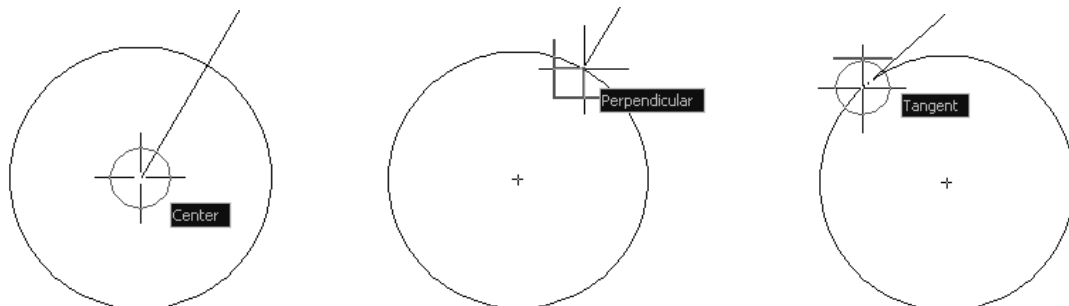


11) Visualize the autosnap markers.

a) Start a **Line** anywhere in the drawing area.




b) Move your cursor around the circles and locate each one of the automatic snap markers (Center, Perpendicular, Tangent).

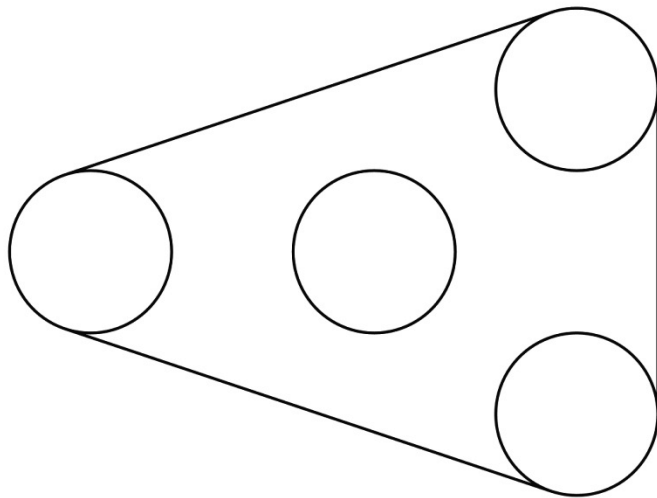
c) Hit **Esc** to exit the **LINE** command.



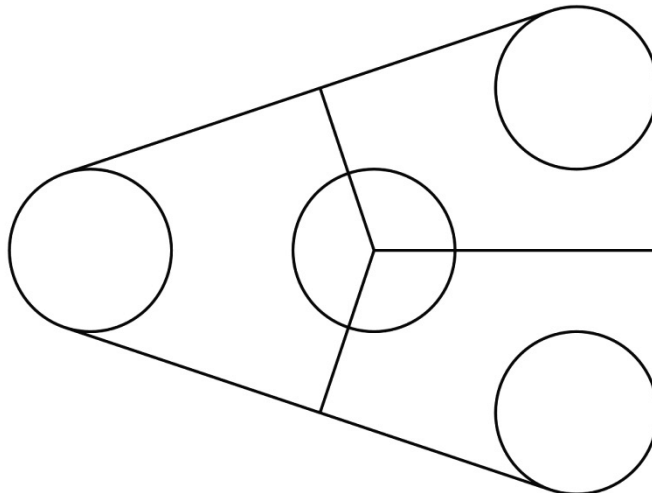
12) Set the autosnap marker size back to 25% of maximum and the color to your preference.

13) Draw 3 **TANgent Lines** around the outer circles.

- a) **Command:** **I** or 
- b) **Specify first point:** Move your cursor to one of the circles. Notice that the tangent Osnap marker does not appear. To specifically tell AutoCAD® that you want to detect the tangent, type **tan** or select .
- c) **to** Select the circle.
- d) **Specify next point or [Undo]:** Move your cursor to the circle at the other end of the line. Again, AutoCAD® will not detect the tangent. Type **tan** or select .
- e) **to** Select the circle.
- f) **Specify next point or [Undo]:** **Enter**
- g) Complete the other two lines.



14) Draw the 3 **Lines** from the **CENter** point of the middle circle to **PERpendicular** to the outer tangent lines. Use the Osnap markers to guide you. (Note: To acquire the center of the circle you need to hover over its circumference.)

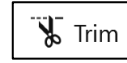


2.20.2) Trimming

1) **TRim** the unwanted lines.

a) **Command:** **TRim** or

Modify panel:



b) Select cutting edges ...

Select objects or <select all>: **Select the 3 tangent lines and the middle circle.**

Problems?

- If AutoCAD® will not let you select multiple items, change your PICKADD variable to 1.

c) Select objects: **Enter**

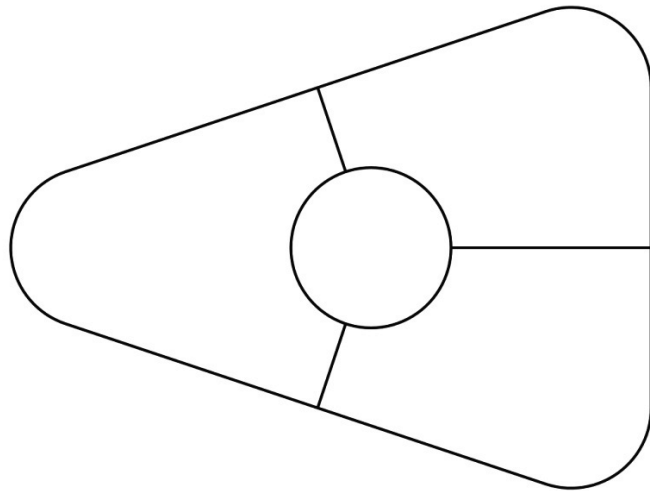
d) Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: **Select the inner portion of the outer circles and the lines inside the middle circle.**

Problems?

- If AutoCAD® will not let you trim the outer circles, you did not draw the tangent lines correctly.

e) Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: **Enter**

2) **Save** your drawing and **print** your **extents** using the **scale to fit** option.



2.21) POLAR TRACKING

Polar tracking restricts cursor movement to specified angles. When you are creating or modifying objects, you can use polar tracking to display temporary alignment paths defined by the polar angles you specify. Figure 2.21-1 shows a polar tracking path restricting a line to be along a 15-degree angle. Once a polar tracking path has been snapped to, a distance from the previous point may be entered in the *Command* window. This will place the next point along the polar path at the specified distance.

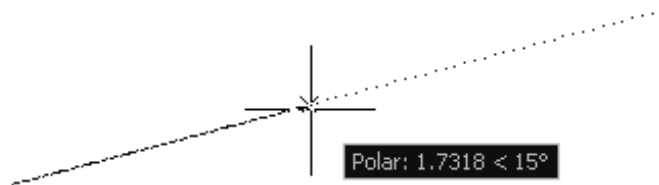



Figure 2.21-1: Polar tracking path

Polar angles are relative to the orientation of the WCS or the current user coordinate system (UCS). You can turn your polar tracking on and off by clicking on the

Polar Tracking  icon in the *Status Bar*. To change the increment of the tracking angle you can use the **POLARANG** command, or click on the arrow next to the *Polar Tracking* icon and select **Tracking Settings....** Figure 2.21-2 shows the *Polar Tracking* settings window.

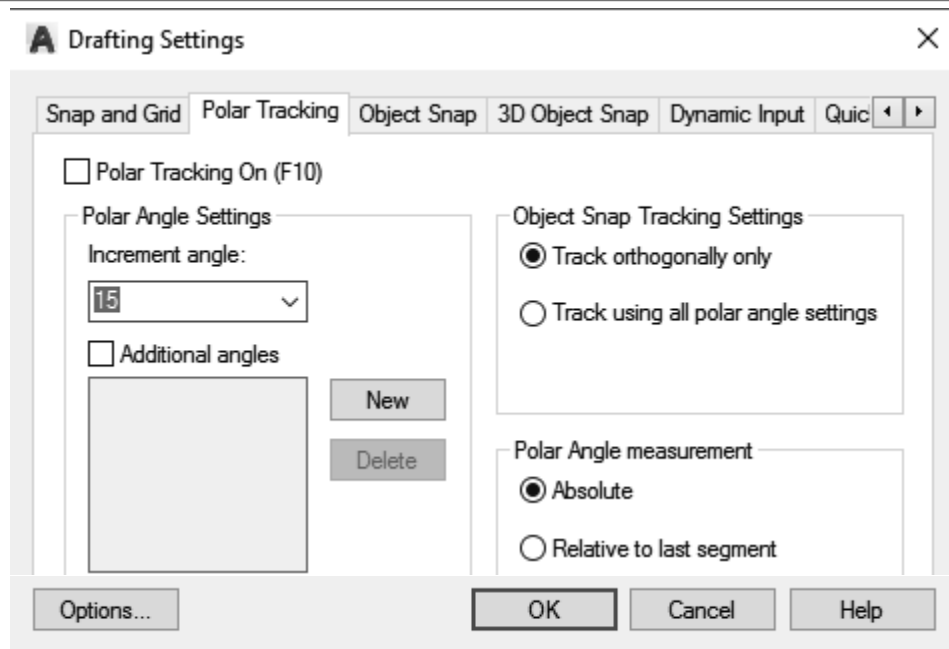


Figure 2.21-2: Polar tracking settings


Polar snap restricts cursor movement to specified increments along a polar tracking path. You can turn on the polar snap by setting your **SNAPTYPE** variable to 1 (a value of 0 is for standard snap). You can set the polar snap distance with the **POLARDIST** command.

2.22) OBJECT SNAP TRACKING

Object snap tracking enables you to track along alignment paths that are based on object snap points. After you acquire a point, horizontal, vertical, or polar alignment paths relative to the point are displayed as you move the cursor. You can acquire up to seven snap points at a time. Acquired snap points are displayed as a small plus sign (+). You can turn your object snap tracking on and off by clicking on the **Object Snap Tracking**



icon in the *Status Bar* or by hitting **F11**. Figure 2.22-1 shows a line being drawn with the help of object snap tracking. The end of the line will be drawn where the center of the circle aligns with the endpoint of the existing line.

A temporary track point is a point that is not associated with the geometry directly, but with the alignment paths that come off the geometry. You can select a temporary track point by typing **TT** or selecting the **Temporary Track Point**  icon in the *Object Snap* panel and then selecting the point.

Object snap tracking is set to create paths at 0, 90, 180, and 270 degrees. However, you can use polar tracking angles instead by setting the **POLARMODE** variable to 2.

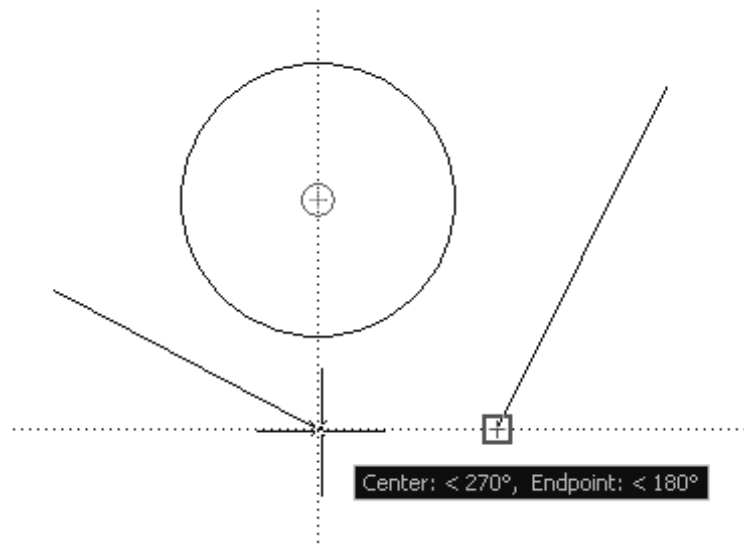


Figure 2.22-1: Using object snap tracking

2.23) TRACKING TUTORIAL

The objective of this tutorial is to familiarize the user with AutoCAD's tracking ability. AutoCAD® allows you to track along polar paths. Polar paths are created at angle increments that you specify. Distances along a polar path may be entered to identify the location of the next point. Object snap tracking is also available. This allows you to place a point at the intersection of multiple paths. These paths extend from acquired object snap points.

2.23.1) Polar tracking

1) View the *Tracking* video and read sections 2.21) and 2.22).

2)  **set-mm.dwt** and  **Tracking Tut.dwg**

3) Status Bar:

a) Turn your **Object Snap**  and **Polar Tracking**  on.

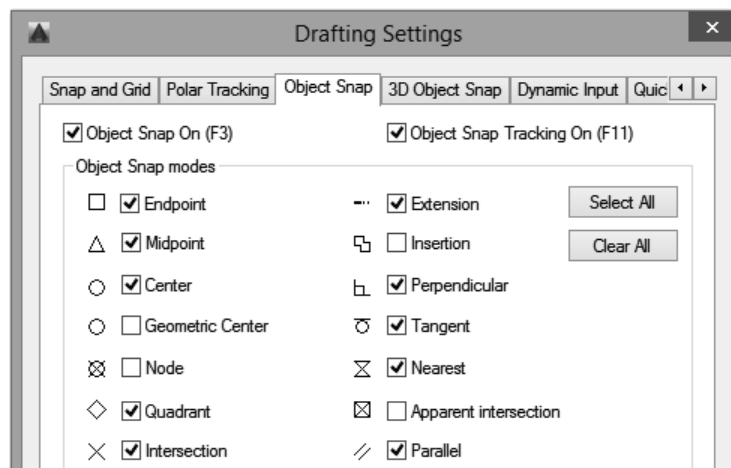
b) Turn your **Dynamic Input**  off.

4) Set your polar tracking angle increment to 15 degrees.

a) Command: **polarang**

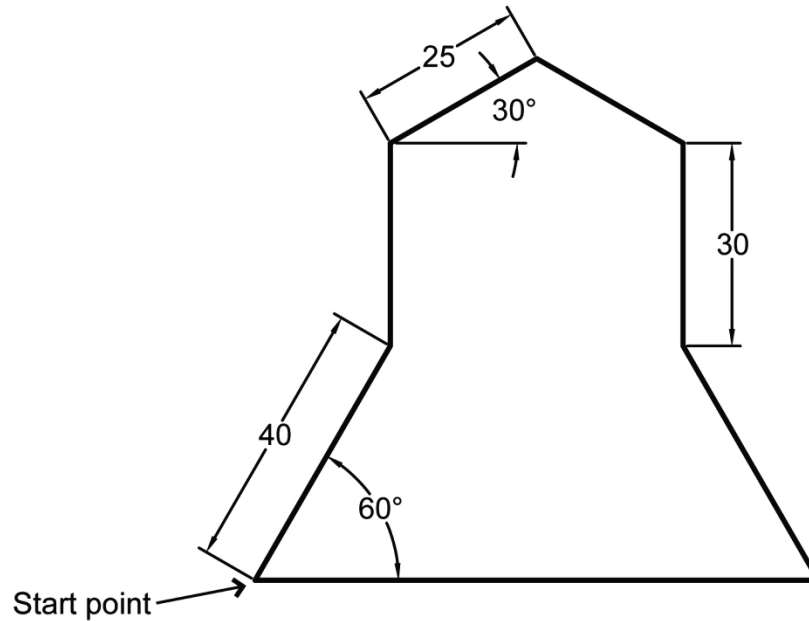
b) Enter new value for POLARANG <90>: **15**

5) Access the **Osnap settings**  and activate the snaps shown.

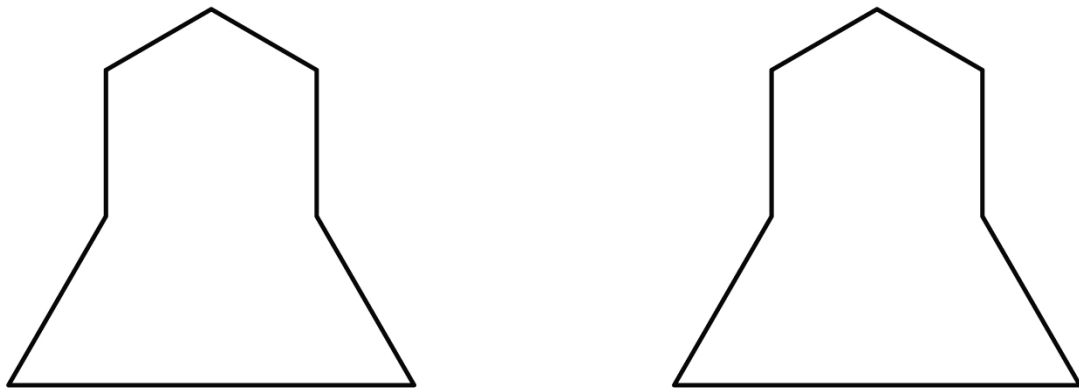


6) Enter your **World Coordinate System** and **Zoom All**

- 7) Draw the following object using polar tracking. The object is symmetric.
 - a) Start a **Line** at **30,30**.
 - b) Move your cursor around the drawing area until you snap to the 60 degree polar path.
 - c) Type **40** and then hit **Enter**.
 - d) Use polar tracking to complete the object.




- 8) **Mirror** your entire drawing. The first point of the mirror line should be **140,20**. Use the **90°** polar path to snap the second point of the mirror line. Do not erase the original drawing.

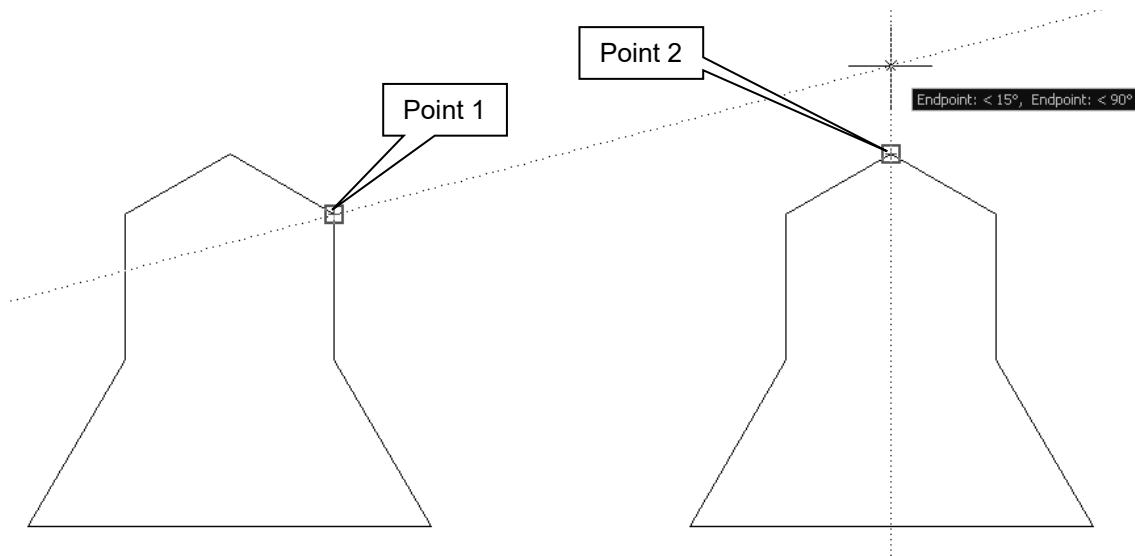


- 9) Status Bar. Turn your **Object Snap Tracking**  on.

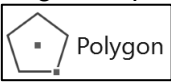

- 10) Set your **POLARMODE** variable to **2**. (This allows you to track using polar paths.)

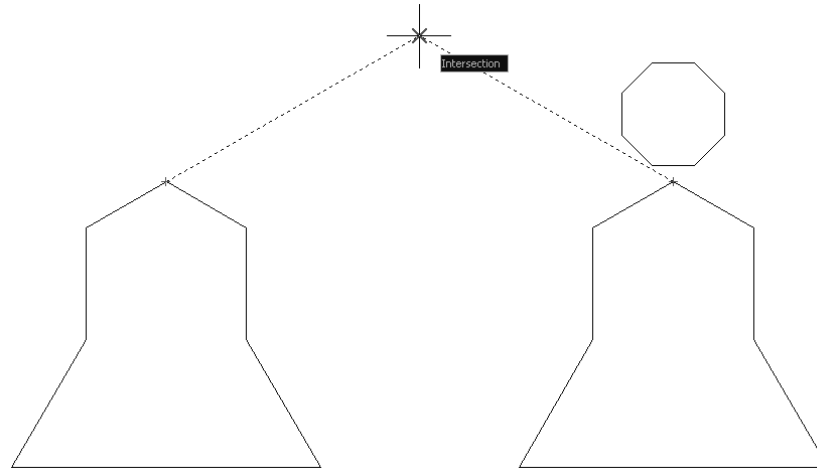
11) Draw an octagon that is aligned on a 15 degree path from *Point 1* and a 90 degree path from *Point 2*.

- a) **Command:** POLYgon or **Draw panel:** 
- b) Enter number of sides <4>: **8**
- c) Specify center of polygon or [Edge]: Acquire *Point 1* by moving your cursor over it. A small cross will appear when the point is acquired. Move the cursor out along the 15 degree polar path. Acquire *Point 2*. Move the cursor along the 90 degree polar path until the 15 degree path appears. Select the point where the paths intersect.
- d) Enter an option [Inscribed in circle/Circumscribed about circle] <I>: **i**
- e) Specify radius of circle: **15**

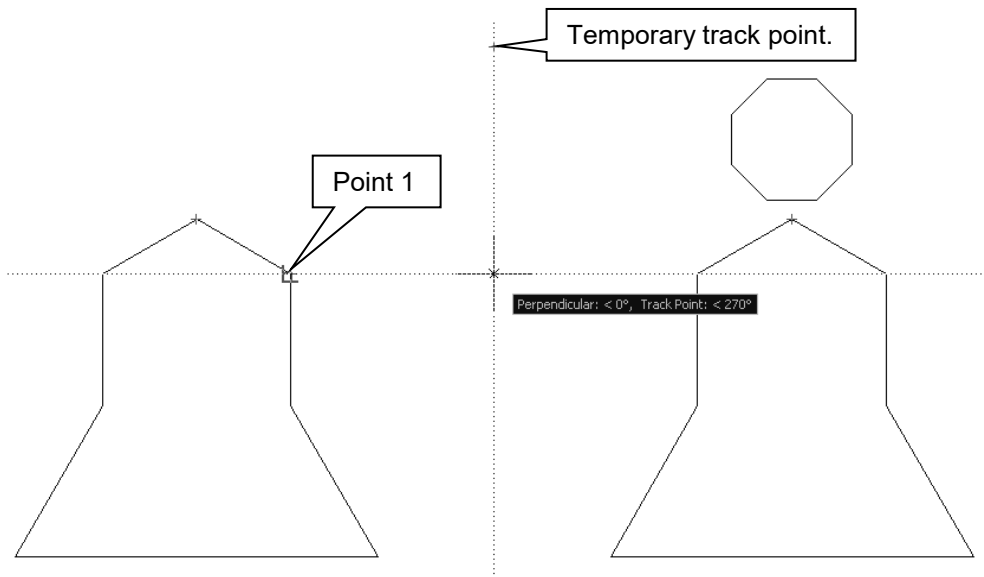


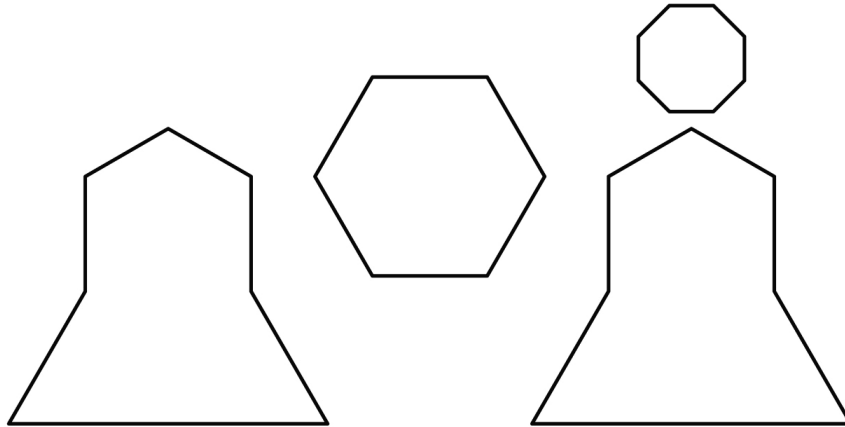
12) Draw a hexagon in the middle of the two main objects using a temporary tracking point.

- a) **Command:** POLYgon or **Draw panel:**  Polygon
- b) Enter number of sides <8>: **6**
- c) Specify center of polygon or [Edge]: Select the **Temporary Track Point**  icon in the **Object Snap** panel
- d) Specify temporary OTRACK point: Acquire the intersection of the two extensions shown in the following figure and then select the point where the extensions cross.



- e) Specify center of polygon or [Edge]: Use the 90 degree path off of the temporary track point and the 0 degree path off of *Point 1* to establish the center of the polygon.
- f) Enter an option [Inscribed in circle/Circumscribed about circle] <I>: **i**
- g) Specify radius of circle: **30**





- 13)  and **print** your drawing.

2.24) DYNAMIC INPUT

Dynamic input provides a command interface near the cursor to help you keep your focus in the drawing area. When the dynamic input is on, a tooltip displays information near the cursor that is dynamically updated as the cursor moves. You can enter a response in the tooltip area instead of in the *Command* window. If you press the **Down Arrow** key, you can view and select options. Pressing the **Up Arrow** key displays recent input. Figure 2.24-1 shows the use of dynamic inputs in the creation of a line.

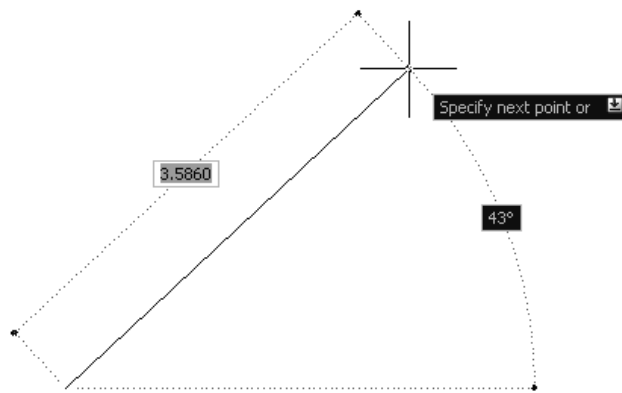



Figure 2.24-1: An example of dynamic inputs

The dynamic input may be turned on and off by clicking on the **Dynamic Input**  icon in the *Status Bar* or by using the **F12** key. To set the dynamic input controls, right click on the *Dynamic Input* icon in the *Status Bar* and select **Dynamic Input Settings...** or from the *Application button*, **Tools – Drafting Settings... - Dynamic Input** tab. Figure 2.24-2 shows the *Dynamic Input* settings window.

When the *pointer input* (see Figure 2.24-2) is on and a command is active, the location of the crosshairs is displayed as coordinates in a tooltip near the cursor. Use the *pointer input* settings to change the default format for coordinates and to control when *pointer input* tooltips are displayed.

When the *dimensional input* (see Figure 2.24-2) is on, the tooltip displays distance and angle values when prompted for a second point. The values in the dimensional tooltips change as you move the cursor. You can press **Tab** to move to the value you want to change. When you use grip boxes to edit an object, the dimensional input tooltips can display the following information: original length, a length that updates as you move the grip, the change in the length and angle, and the radius of an arc. Which one of these is displayed may be set in the *dimension input* settings.

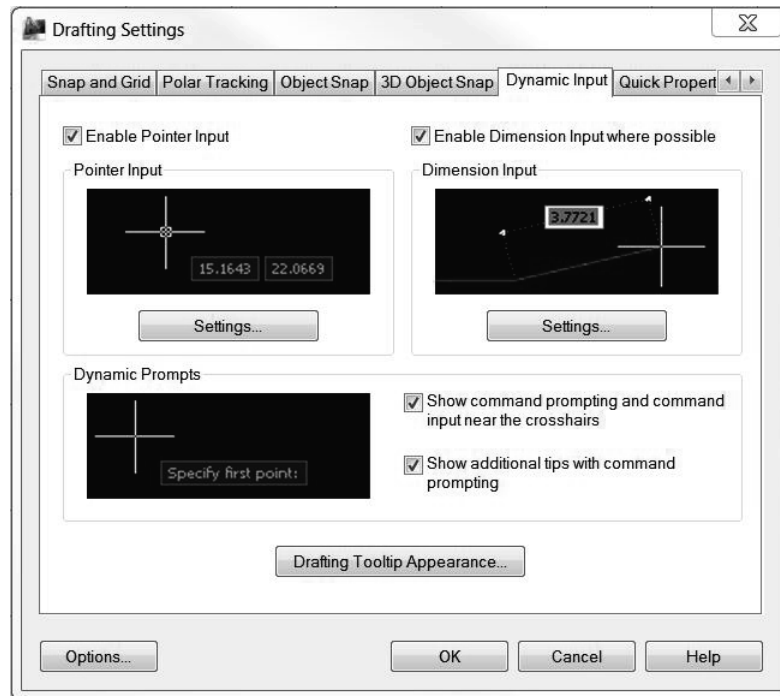


Figure 2.24-2: Dynamic input settings

2.25) DYNAMIC INPUT TUTORIAL




The objective of this tutorial is to familiarize the user with using dynamic inputs and the different settings that are available. The dynamic input allows you to enter commands and select options in an area near the cursor and the object being drawn.

2.25.1) Dynamic input settings

1) Read section 2.24)

2)  **set-inch.dwt** and  **Dynamic input Tut.dwg**

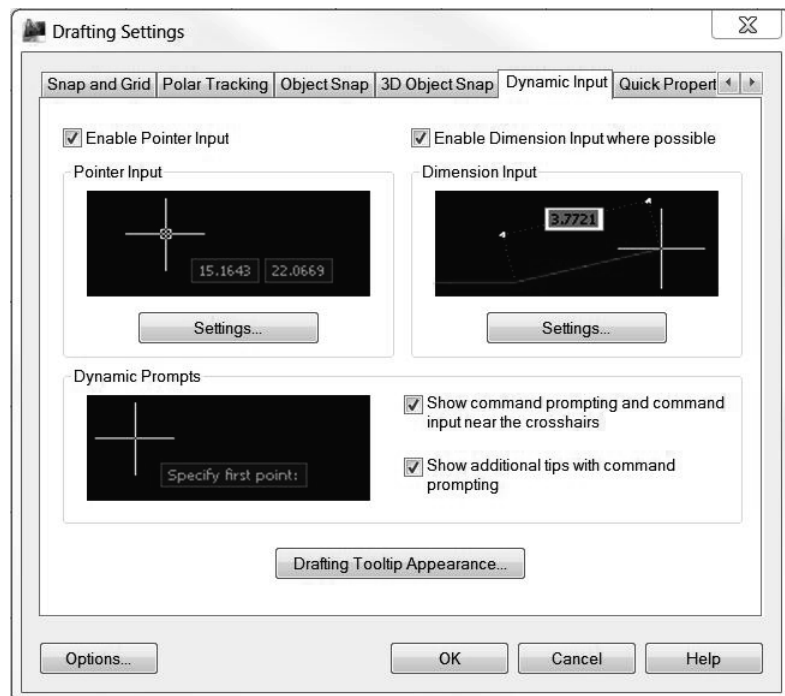
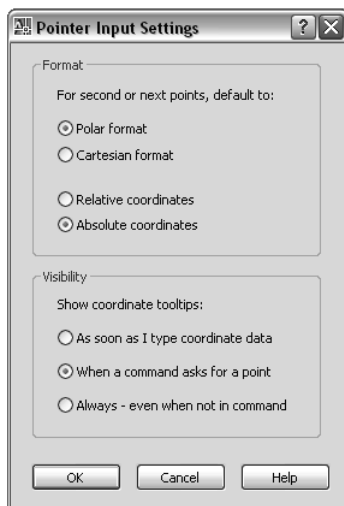
3) Status Bar:

- a) Turn your **Dynamic Input**  on.
- b) Turn your **Polar Tracking**  and **Object Snap Tacking**  off.

4) Activate a 1 x 1 **GRID** and **Zoom All**


5) Change the dynamics input settings.

- a) Status Bar: Right click on the **Dynamic Input** icon and select **Dynamic Input Settings...**
- b) Drafting Settings window – Dynamic Input tab: Select the **Settings...** button under **Pointer Input**.
- c) Pointer Input Settings window: Activate the **Polar format**, **Absolute coordinates** and **When a command asks for a point** radio buttons.
- d) Select **OK** in both windows.



- 6) Draw a **Line** from (2,2) to (3,3) with the *dynamic input* on.
 - a) Command: **l** or **line**
 - b) Specify first point: Move your cursor around the drawing area. Notice the coordinates near the cursor. When finished, enter the first point without taking your cursor down to the *Command* window **2,2**.
 - c) Specify next point or [Undo]: Move your cursor around the screen and see the length of the line and angle change. When finished enter the second point **3,3** without taking your cursor down to the *Command* window and then **Enter** to exit the command.
 - d) Specify next point or [Undo]: **Enter**
 - e) Note the coordinate of the second point is relative to the WCS.
- 7) Pointer Input Settings window: Change *Absolute coordinates* to **Relative coordinates**.
- 8) Draw a **Line** from **4,4** to **5,5** as in step 6).
- 9) **Zoom All**. (Notice that the second point is relative to the first point and not to the WCS.)
- 10) Pointer Input Settings window: Change *Relative coordinates* to **Absolute coordinates**.

2.25.2) Locking a coordinate

- 1) We will be drawing a **Line** from 3,2 that is 3 inches long at a 45 degree angle. From the end of the previous line, draw a Line that is 2 inches long at a 30 degree angle.
 - a) Command: **l** or 
 - b) Specify first point: **3,2**
 - c) Specify next point or [Undo]: Press the **Tab** key until the angle dimension turns red (box color should be white) and then type **45** and hit the **Tab** key again. Notice the lock that appears on the angle dimension.
 - d) Specify next point or [Undo]: Move your cursor around the drawing area. Notice that the length of the line changes, but the angle does not. When finished, type **3** and hit **Enter**.
 - e) Specify next point or [Undo]: Press the **Tab** key until the linear dimension turns red (box color should be white) and then type **2** and hit the **Tab** key again.
 - f) Specify next point or [Undo]: Move your cursor around the drawing area. Notice that the angle of the line changes, but the length does not. When finished, type **30** and hit **Enter** twice.

- 2) 

2.26) GRIP BOXES

Object grips or grip boxes are usually blue boxes that appear on a selected object if no command is active. Once the grip boxes appear, you can use them to MOVE, COPY, SCALE, ROTATE, MIRROR and STRETCH the object. To understand how grip boxes work, let's look at a simple line (Figure 2.26-1). If you click on the line, grip boxes will appear at the ends and the middle. If you click on the middle grip box, you can move the entire line. If you click on one of the end grips, you can stretch the line. For more advanced options, click on a grip and then right click to access a shortcut menu.

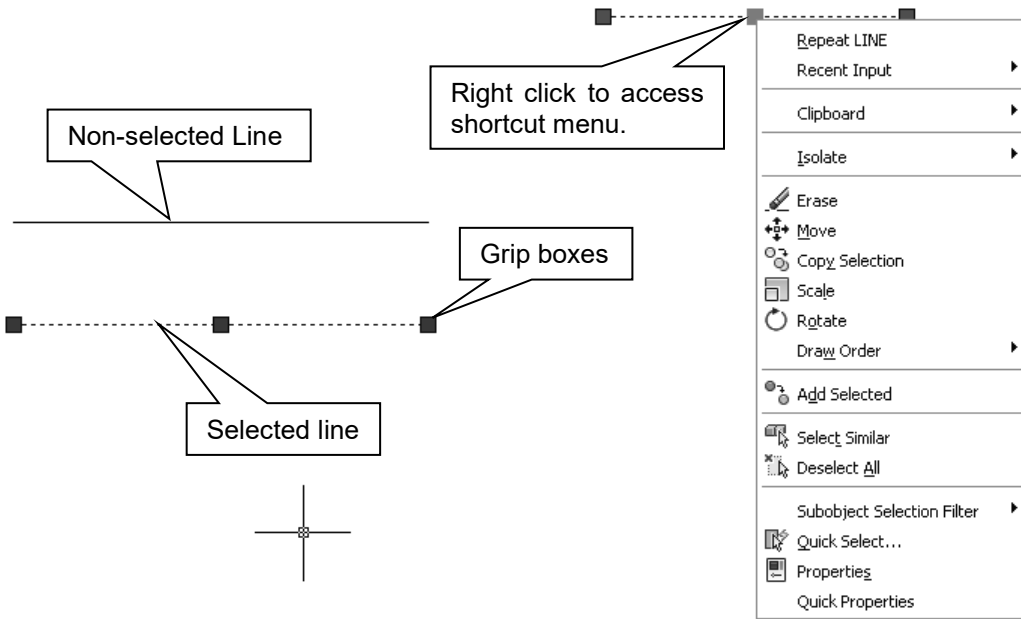


Figure 2.26-1: Grip boxes

The grip box mode may be turned off by setting the **GRIPS** variable to 0 and turned on by setting it to 1. Additional midpoint grips on polyline segments may be shown by setting GRIPS to 2. The size of the grip boxes are controlled by the **GRIPSIZE** variable. Grip box settings are also available in the *Selections* tab of the *Options* window.

2.27) GRIP BOX TUTORIAL

The objective of this tutorial is to use object grips or grip boxes and the *Properties* window to edit a drawing. Grip boxes allow you to quickly modify an object without having to enter a command.

2.27.1) Preparing to draw

- 1) Read section 2.26).

- 2)  the preexisting file ***grip_boxes_student_2018.dwg***. This is located on the disc that comes with the book.

- 3)  **Grip boxes Tut.dwg**

- 4) Status Bar:

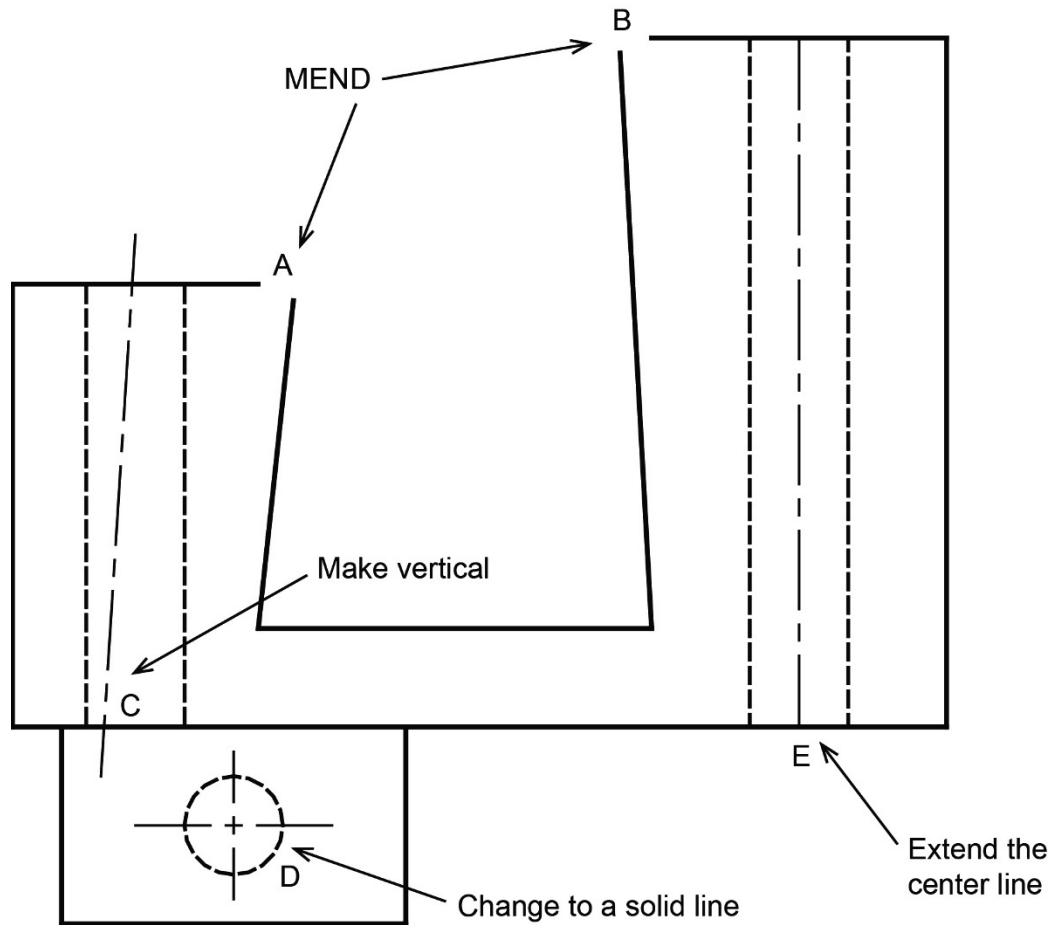
- a) Turn your **Object Snap** , **Object Snap Tracking**  and **Polar Tracking**  on.

- b) Turn your **Dynamic Input**  off.

- 5) Deactivate all but the following automatically detected object snaps: **Endpoint**, **Center**, and **Perpendicular**.

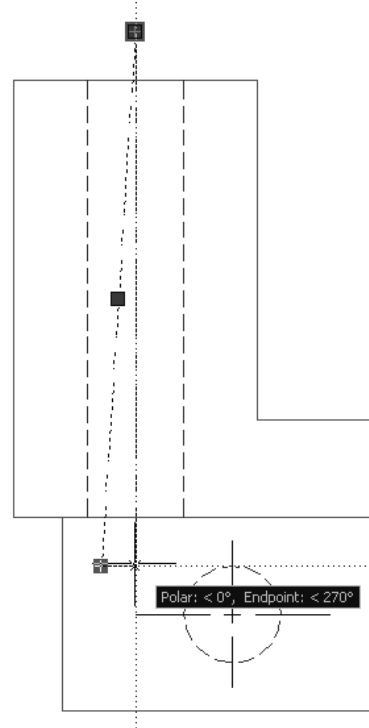
2.27.2) Fixing the drawing

We need to fix the areas indicated in the figure.

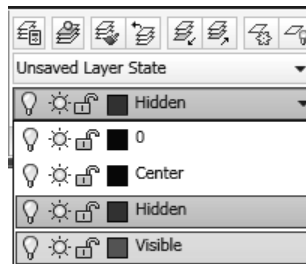


- 1) Repair corner A.
 - a) Select the almost vertical line at corner A.
 - b) Click on the top grip box.
 - c) Move your cursor and see the end of the line moves.
 - d) Move your cursor over to right end of the horizontal line at corner A. Once the END snap marker appears, click your left mouse button.
 - e) Hit the **Esc** to deselect the line.
- 2) Using the same procedure to repair corner B.


- 3) Make line C vertical.
 - a) Select line C (the centerline).
 - b) Click on the bottom grip box.
 - c) Acquire the top end of the centerline by moving your cursor over it.
 - d) Move your cursor vertically downward until the 0 degree tracking path appears and then click your left mouse button.
 - f) Hit the **Esc** to deselect the line.

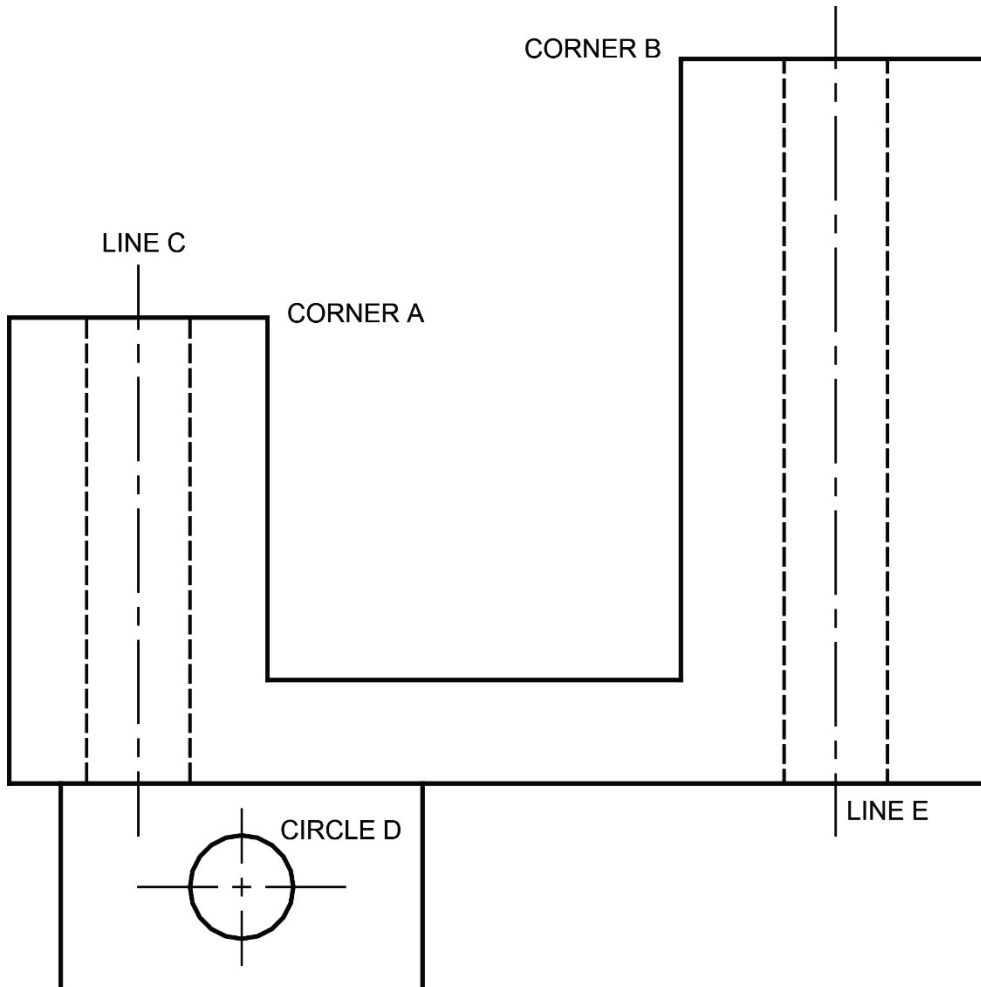


- 4) Change circle D to a solid line.
 - a) Select circle D.
 - b) Locate the *Layers* panel in the *Home* tab.
 - c) Select the layers pull down selection window by clicking on the arrow.
 - d) Move your cursor down the list and select **Visible**.
 - e) Hit the **Esc** to deselect the circle.



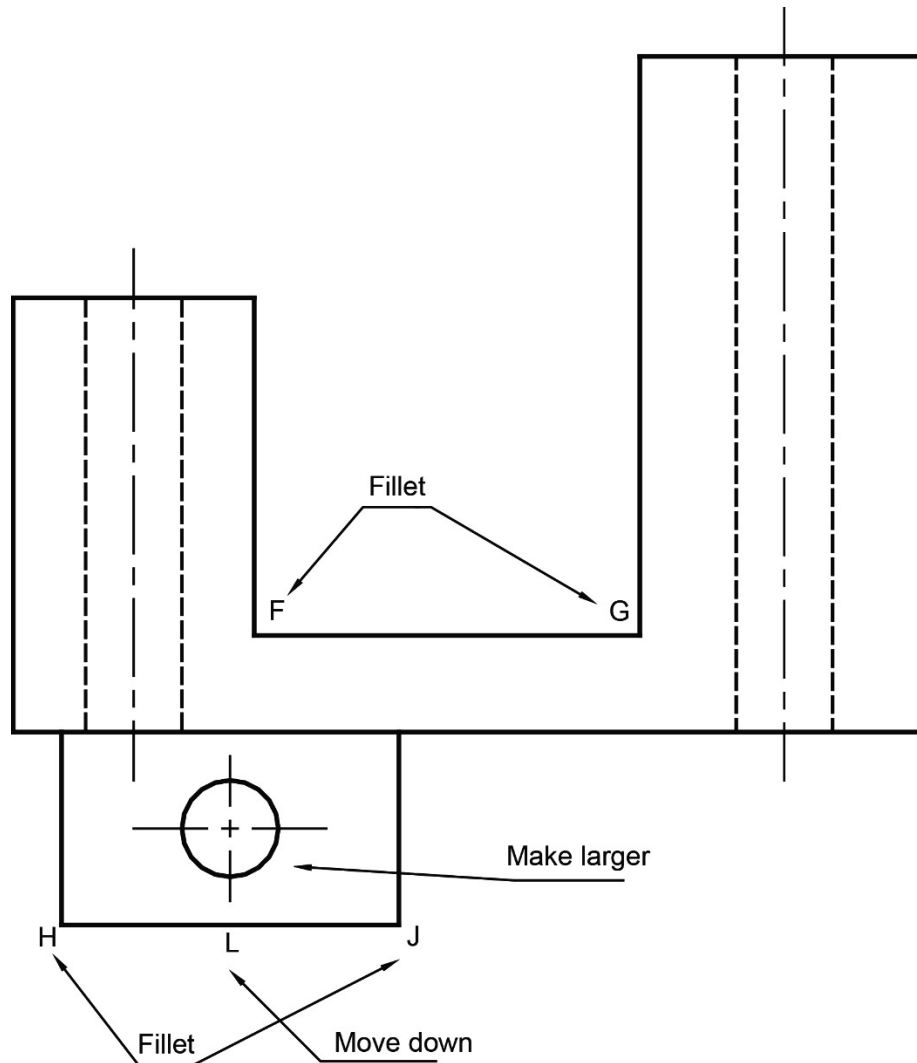
5) Extend Line E past the top and bottom horizontal visible/solid lines.


- a) Command: **len** (lengthen) or Modify Panel: 
- b) Select on object or [DElta/Percent/Total/DYnamic]: **de**
- c) Enter delta length or [Angle] <0.0000>: **10**
- d) Select an object to change or [Undo]: **Select Line E near the top.**
- e) Select an object to change or [Undo]: **Select Line E near the bottom.**
- f) Select an object to change or [Undo]: **Enter**



2.27.3) Modifying the drawings shape

The following figure lists the modifications that we will perform.



- 1) Apply an R5 (radius = 5) **Fillet** to corners F and G.
 - a) Command: **Fillet** or Modify panel: 
 - b) Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **r**
 - c) Specify fillet radius <10.0000>: **5**
 - d) Select first object or [Undo/Polyline/Radius/Trim/ Multiple]: **Select one of the two lines that make up corner F.**
 - e) Select second object or shift-select to apply corner: **Select the other line that makes up corner F.**
 - f) Command: Hit the **space** bar to repeat the last command and apply the same fillet to corner G.

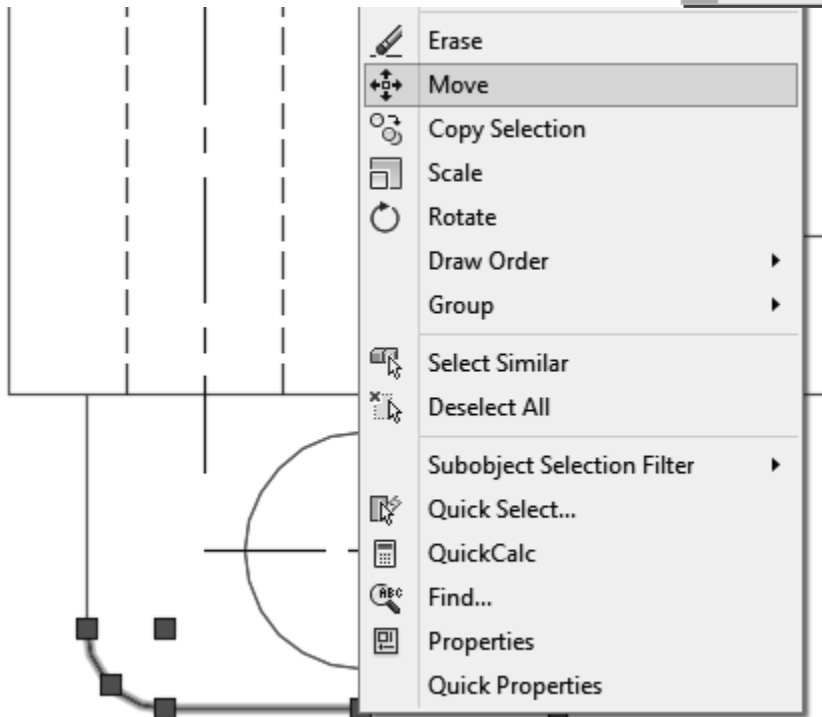
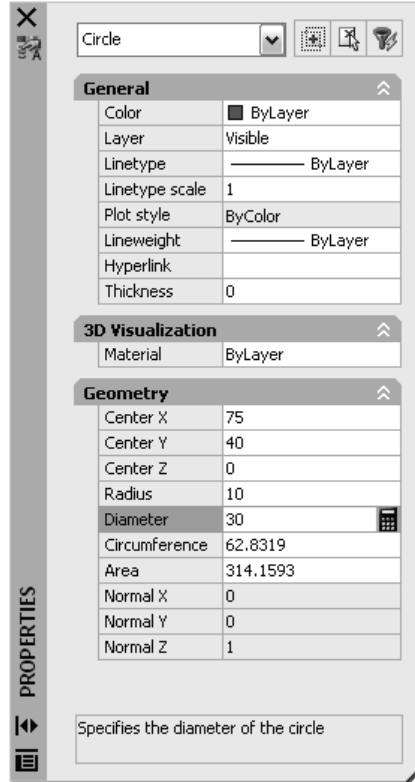
- 2) Apply a **R10 Fillet** to corners H and J. This time use the **Multiple** option.

- 3) Change the size of Circle D.
 - a) Select Circle D.

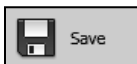


- b) View tab – Palettes panel:
 - c) Properties window: Change the circle's **Diameter** to **30** and click on the graphics screen to watch the changes occur. Close the window when you are done.
 - d) Hit the **Esc** to deselect the circle.

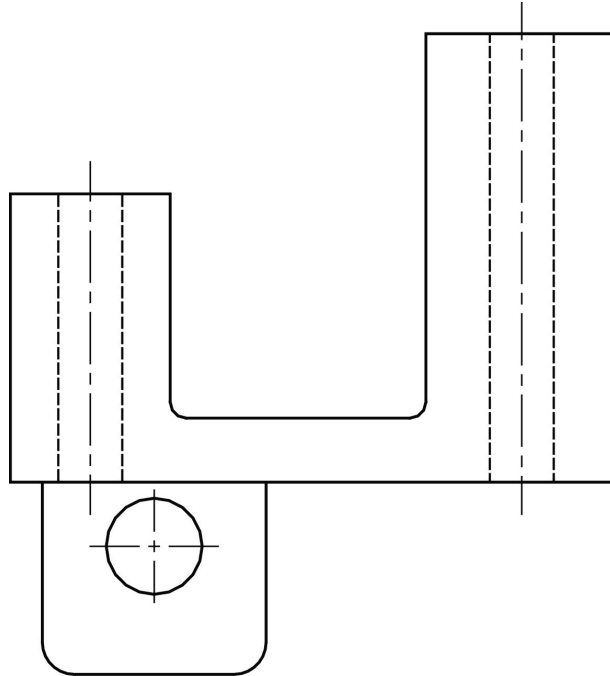
- 4) On your own, **LENGTHen** the vertical centerline of circle D in both directions by **2.5** units.
- 5) Move line L and the fillets down by 20 mm.
 - a) Using a window to select line L and both fillets.
 - b) Left click on the middle grip box.
 - c) Right click on the middle grip box and select **Move** from the shortcut menu.
 - d) Specify move point or [Base point/Copy/Undo/eXit]: Use polar tracking to move line L and the fillets down **20** mm.



- 6) Use grip boxes to connect the ends of the vertical lines to the fillets.



- 7) and **print** your drawing.



2.28) PARAMETRIC DRAWING

The commands located in the *Parametric* panel allow you to apply geometric and dimensional constraints to 2D drawings (shown in Figure 2.28-1). The power of parametric drawing becomes apparent during the design phase of a part. Parametric drawing allows you to apply key functional constraints and play with or adjust the rest of the part design without worrying about losing the constraints that you have already applied.

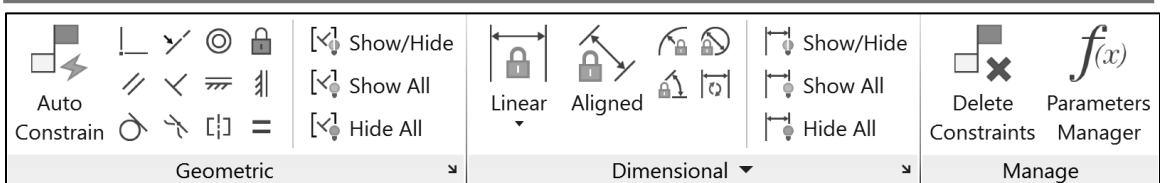


Figure 2.28-1: Parametric panel

2.28.1) Geometric constraints

Geometric constraints control the orientation relationship between two elements or of a single element. For example, you are able to force an element to be horizontal, or you can make two lines be perpendicular to each other. Table 2.28-1 describes the available geometric constraints. This command may be accessed through the *Parametric* ribbon tab or by using the **GEOMCONSTRAINT** command.













Constraint Type	Icon	Purpose
Coincident		Constrains two points to coincide, or a point to lie on an object or an extension of an object.
Colinear		Constrains two lines to lie on the same infinite line.
Concentric		Constrains two circles, arcs or ellipses to have the same center point.
Fix		Constrains a point or a curve to a fixed location and orientation with respect to the WCS.
Parallel		Constrains two lines to maintain the same angle.
Perpendicular		Constrains two lines or polyline segments to maintain a 90 degree angle to each other.
Horizontal		Constrains a line or a pair of points to lie parallel to the x-axis of the current UCS.
Vertical		Constrains a line or a pair of points to lie parallel to the y-axis of the current UCS.
Tangent		Constrains two curves to maintain a point of tangency to each other or their extensions.
Smooth		Constrains a spline to be contiguous and maintain G2 continuity with another spline, line, arc or polyline.
Symmetric		Constrains two curves or points on objects to maintain symmetry about a selected line.
Equal		Constrains two line or polyline segments to maintain equal lengths, or circles and arcs to maintain equal radii.

Table 2.28-1: Geometric Constraints

2.28.2) Dimensional constraints

Dimensional constraints control size and angle. You can apply a dimensional constraint between two elements, two points on one element or to control the overall size or angle of an element. The size of the element will automatically change if the value of the dimension is changed. The dimension contains both a name and a value. This allows you the ability to relate one dimension to another through an equation (see Figure 2.28-2). Dimensional constraints may be accessed in the *Parametric* ribbon tab or by using the **DIMCONSTRAINT** command.

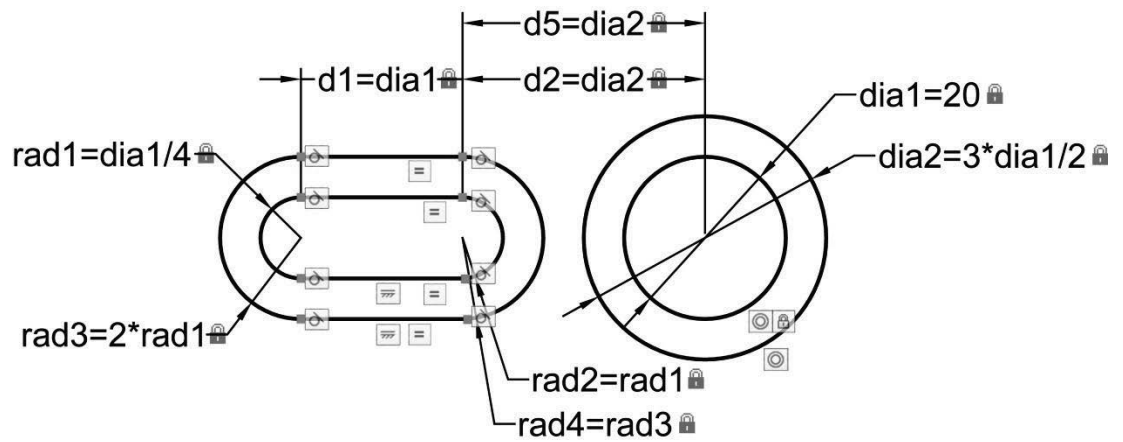


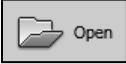
Figure 2.28-2: Example of parametric constraints.

2.29) APPLYING PARAMETRIC CONSTRAINTS TUTORIAL

The objective of this tutorial is to familiarize you with applying geometric and dimensional constraints to a pre-drawn part.

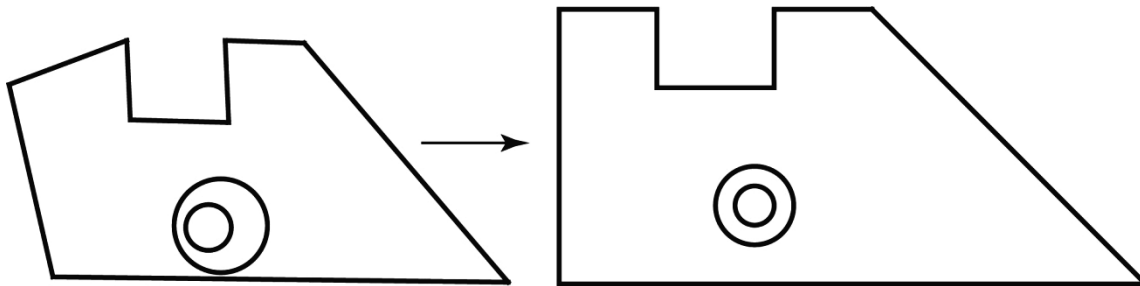
2.29.1) Applying geometric constraints

- 1) Read section 2.28).


- 2)  the preexisting drawing *parametric_student_2018.dwg*.

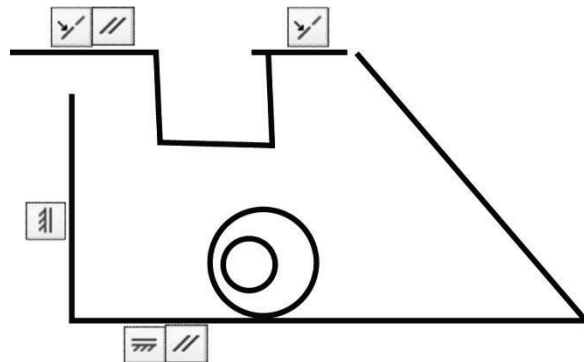
- 3)  **Apply Parametric Tut.dwg**

- 4) Our goal is to make the drawing on the left look like the drawing on the right without having to redraw it.






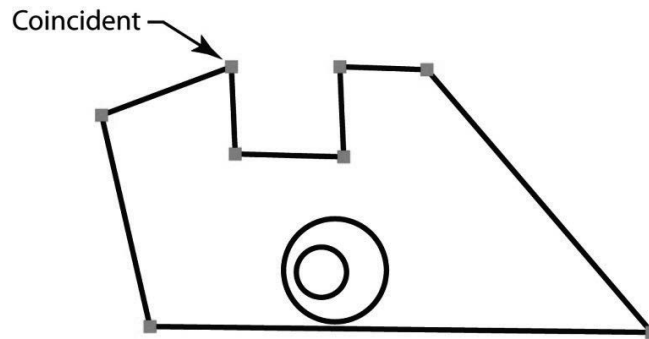
- 5) We start by applying the geometric constraints. The first thing you might think about doing is to make the bottom line horizontal, the left line vertical, followed by a few more orientation constraints. However, as you can see in the figure, that doesn't work out too well. I have found the best constraint to start with is the

coincident constraint . Find everything that needs to be fixed together and apply that first and then apply the orientation constraints.





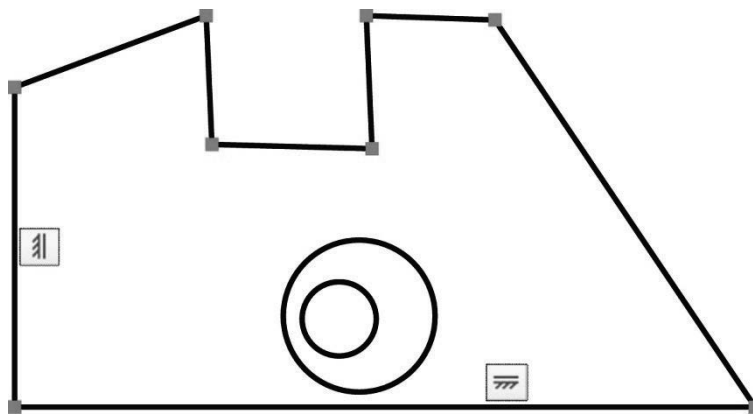
6) Apply a **COINCIDENT** constraint to every corner.

- a) Parametric ribbon tab – Geometric panel: 
- b) Select the first point or [Object/Autoconstrain] <Object>: Hover over the bottom line near the bottom left corner of the part. Once you get this symbol  at the corner, select the line.
- c) Select the second point or [Object] <Object>: Hover over the left vertical line near the bottom left corner of the part. Once you get this symbol  at the corner, select the line. A blue square should appear at the corner.
- d) Apply a coincident constraint to the remaining corners using a similar procedure.










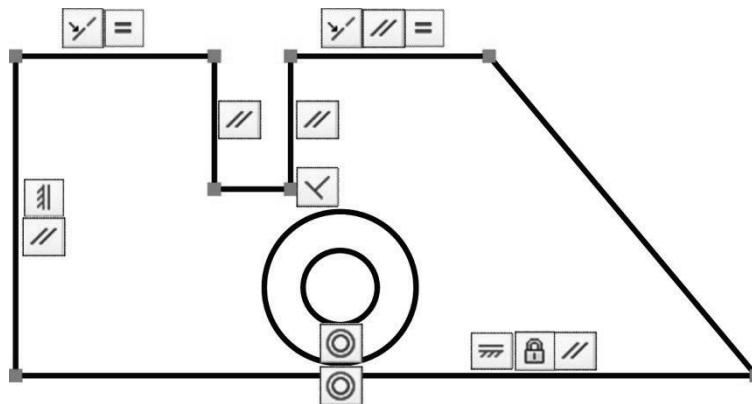
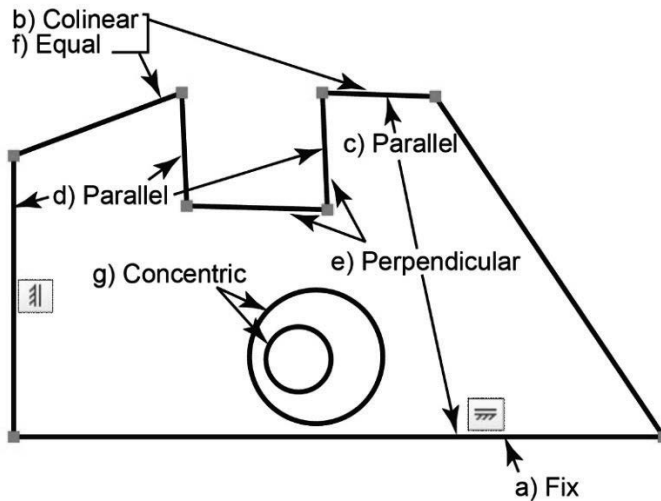
7) Make the bottom line **HORIZONTAL** and the left line **VERTICAL** as shown in the following figure.

- a) Parametric ribbon tab – Geometric panel: 
- b) Select an Object or [2Points] <2Points>: Select the bottom line.
- c) Apply a vertical  constraint to the left line using a similar procedure.



8) On your own, apply the following constraints. See the figures below and on the following page to guide you.

- a) **FIX**  the bottom horizontal line.
- b) Apply a **COLINEAR**  constraint to the top two lines.
- c) Make the right side top line **PARALLEL**  to the bottom line.
- d) Make left the two short nearly vertical lines **PARALLEL**  to the left vertical line.
- e) Make the short nearly horizontal line **PERPENDICULAR**  to the right side short vertical line.
- f) Make the two top lines **EQUAL**  in length.
- g) Make the two circles **CONCENTRIC** .

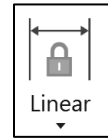




9)  Save

2.29.2) Applying dimensional constraints

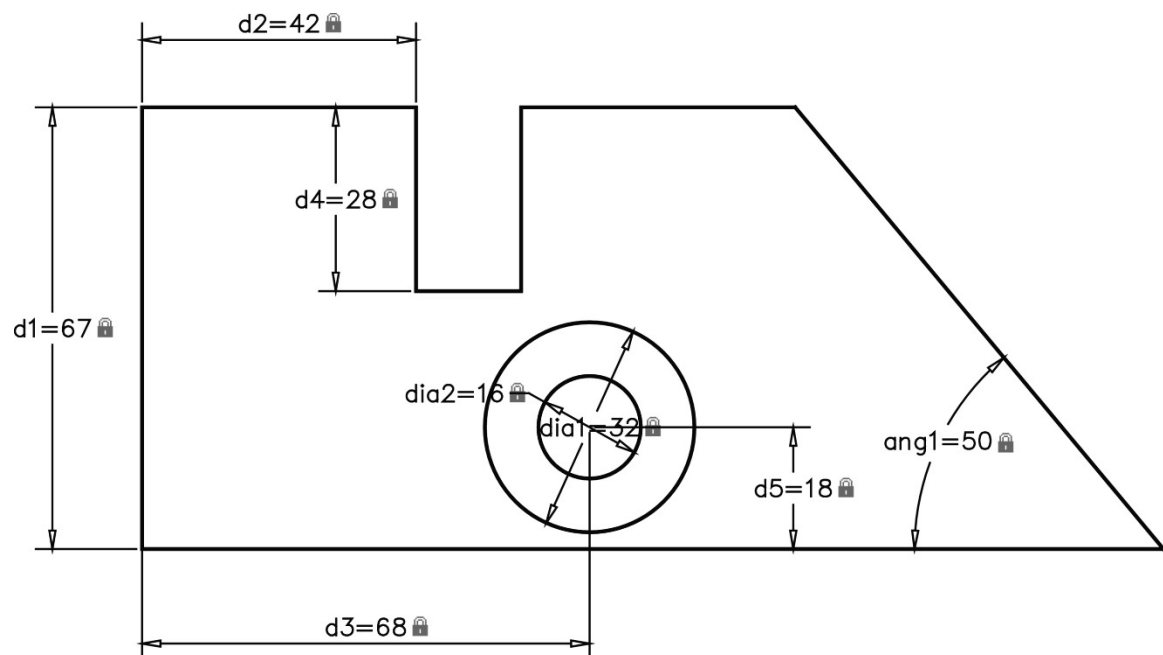
Now that our part is completely constrained geometrically, let's add some size constraints.



1) Add **LINEAR** dimensions to the part.



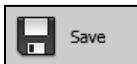
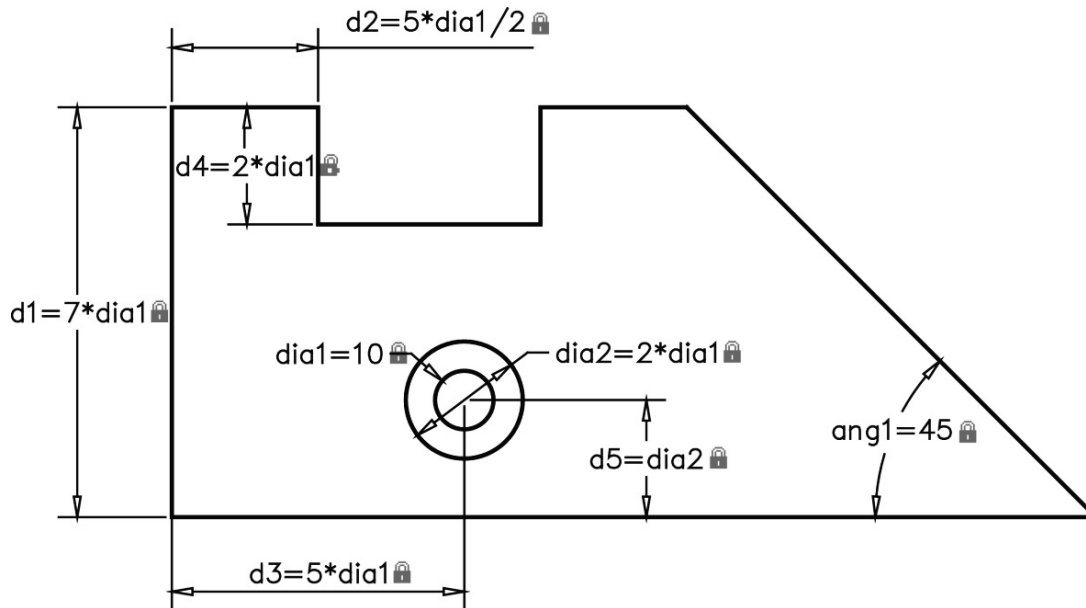
- Command:** `dclinear` or Parametric ribbon tab – Dimensional panel:
- Specify first constraint point or [Object] <Object>: Hover over the top horizontal line near the left corner of the part. Once you get this symbol  at the corner, select the line.
- Specify second constraint point: Hover over the bottom horizontal line near the left corner of the part. Once you get this symbol  at the corner, select the line.
- Specify dimension line location: Pull the dimension away from the part and click the left mouse button.
- Dimension text = 67 Don't worry about the dimension value right now, just hit **Enter**.
- Place the rest of the linear (**DCLINEAR**) dimensions shown in the figure using a similar procedure. To place the linear dimensions between the lines and the center of the circle, select the line first and then the border of the circle.

Don't worry if your dimensions are named differently than the following figure.



- On your own, add the angular (**DCANGULAR**)  and diameter (**DCDIAMETER**)  dimensions. Read the command prompt for guidance.

- 3) Double click on the angular dimension and type **ang1=45** and then hit **Enter**.
- 4) Use the same procedure to set both the dimension NAME and VALUE of all the constraints to be the same as those shown in the figure below. NOTE: You may need to create some temporary dummy names if you have a duplicate dimension name. You should start with the diameter dimension since every other dimension is based off of those dimensions.



- 5)
- 6) As you can see, we have related every dimension to *dia1*. Therefore, if we change the value of *dia1*, the entire part should change size. Change *dia1* to **30**. As you can see, something went terribly wrong. This is a clear sign that we did not completely constrain our part. Can you figure out what dimension we missed? **Undo** until the part is back to its correct shape and size.
- 7) Add the LINEAR dimension that controls the total length of the bottom line and set its length to **15*dia1**.

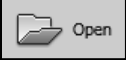
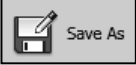
- 8) Change *dia1* to **30**. This time the result is completely different.



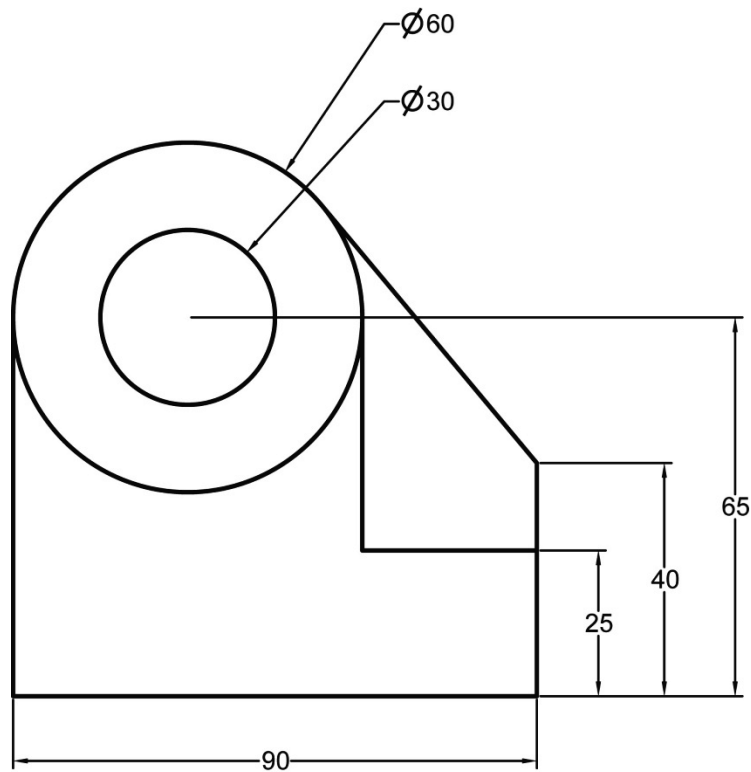
- 9) and **print** your drawing using the **scale to fit** option.

2.30) AUTOMATIC PARAMETRIC CONSTRAINTS TUTORIAL

The objective of this tutorial is to familiarize the reader with applying parametric constraints to an already correctly drawn part.

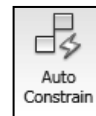
- 1)  **set-mm.dwt** and  **Auto Parametric Tut.dwg**

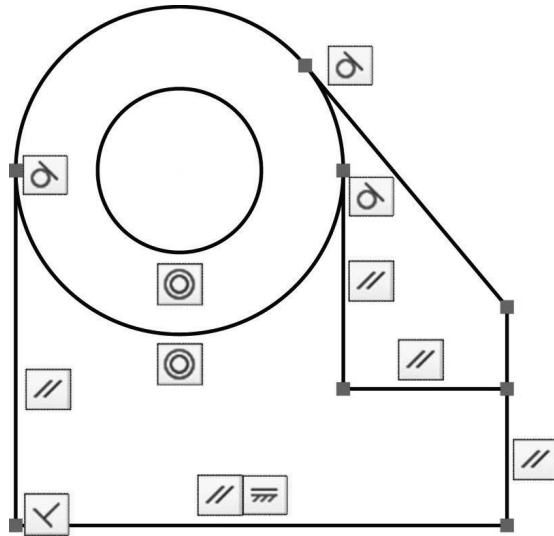
- 2) Draw the following object using the proper OSNAPS. Do not include the dimensions, but draw it to the correct size.



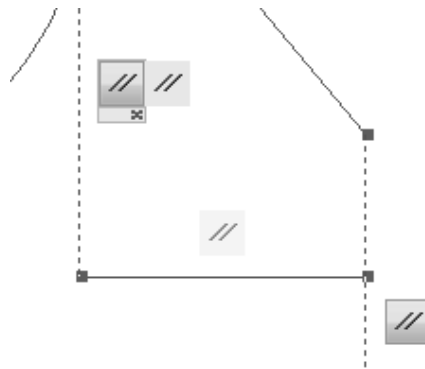
- 3) Automatically apply the geometric constraints based on your drawing.

- a) Command: **autoconstrain** or Parametric tab – Geometric panel:
 b) Select objects or [Settings]: **all**
 c) Select objects or [Settings]: **Enter**

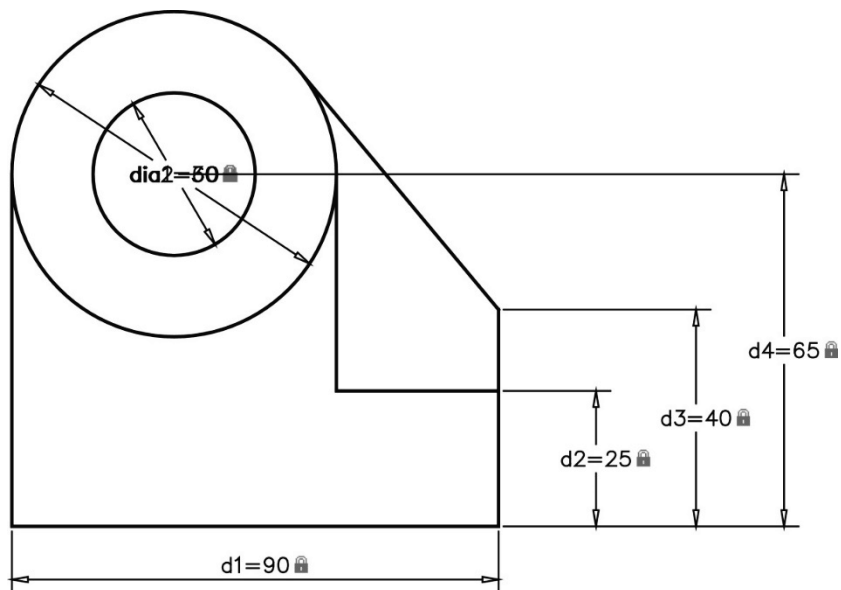




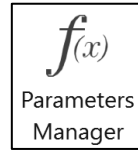
d) Notice the constraints that were made. Your constraints may be slightly different depending on how you drew the object. Most of the constraints are obvious. However, the PARALLEL constraints are not so clear. Which elements are parallel to which elements? To find out, hover your mouse over the constraint and connecting constraints will highlight. You will also notice that an X will appear below the constraint. If you click on this X, the constraint will be deleted (don't do this right now).



4) Apply the following dimensional constraints. If you name your dimensions the same as shown in the figure, it will make it easier later on. It is hard to see in the figure, but **dia1 = 30** and **dia2 = 60**.



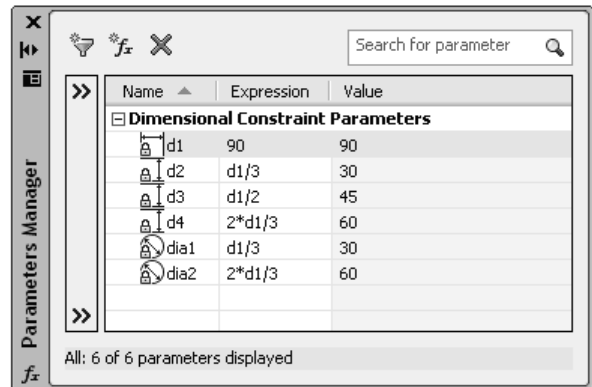
5) Apply functional relationships between the dimensional constraints.



- a) Command: **parameters** or Parametric tab – Geometric panel:
- b) Parameters Manager window: Input the following functional relationships between the dimensional constraints. The *Parameters Manager* allows you to change the name and value of every dimension (double click to change the value). Refer to the above figure to relate the name to the original value.
- c) **Close** the *Parameters Manager* window.

6) Change **d1** to **60** and make sure all the constraints are properly applied.

7) **Save** and **print** your drawing using the **scale to fit** option.



NOTES:

DRAWING IN AUTOCAD QUESTIONS

Name: _____ Date: _____

User interface

Q2-1) The shortcut menu is accessed by

- a) Hitting the space bar
- b) Left clicking
- c) Right clicking
- d) Hitting the enter key

Q2-2) The window and tab that you need to access in order to change the drawing area's color?

- a) Options-Selection
- b) Options-Drafting
- c) Options-Files
- d) Options-Display

Q2-3) This key allows you to quickly repeat your last command.

- a) space bar
- b) esc
- c) tab
- d) enter

Q2-4) Circle all the ways that you can use to access AutoCAD's commands.

- a) Ribbon panels
- b) Command window
- c) Tool palettes
- d) Application button search box

Q2-5) A place that gives you quick access to commands that are frequently turned on and off.

- a) Ribbon panels
- b) Command window
- c) Tool palettes
- d) Status bar

Starting and setting up a drawing

Q2-6) The system variable that determines whether the *Select Template* window or the *Create New Drawing* window appears when starting a new drawing.

- a) drawing
- b) startup
- c) template
- d) create

Q2-7) The command that is used to select your unit of measure and precision.

- a) limits
- b) precision
- c) measure
- d) units

Q2-8) The command that is used to set your paper size.

- a) limits
- b) precision
- c) measure
- d) units

Name: _____ Date: _____

Coordinates

Q2-9) Which coordinate system is stationary and cannot be translated or rotated?

- a) WCS
- b) UCS
- c) CCS
- d) LCS

Q2-10) Which coordinate system may be translated and rotated by the user to make drawing easier?

- a) WCS
- b) UCS
- c) CCS
- d) LCS

Q2-11) An absolute coordinate is measured relative to the

- e) origin
- f) last point entered
- g) bottom left corner of your paper
- h) cursor

Q2-12) A relative coordinate is measured relative to the

- a) origin
- b) last point entered
- c) bottom left corner of your paper
- d) cursor

Q2-13) A relative coordinate is indicated by placing the following symbol before your coordinate.

- a) #
- b) &
- c) @
- d) <

Printing

Q2-14) Which printing option prints everything that you can currently see?

- e) limits
- f) extents
- g) display
- h) window
- i) scale to fit

Q2-15) Which printing option will expand/reduce your drawing to fit the paper?

- a) limits
- b) extents
- c) display
- d) window
- e) scale to fit

Q2-16) Which printing option prints everything within your defined paper size?

- a) limits
- b) extents
- c) display
- d) window
- e) scale to fit

Q2-17) Which printing option prints everything that has been drawn?

- a) limits
- b) extents
- c) display
- d) window
- e) scale to fit

Name: _____ Date: _____

Draw commands

Q2-18) An arc may be defined by its START, END and (circle all that apply)

- a) radius
- b) direction
- c) diameter
- d) angle

Q2-19) What is the typed command used to access text settings?

- a) textsettings
- b) edit
- c) style
- d) font

Q2-20) A polygon that is drawn on the outside of a circle is said to be ...

- a) inscribed
- b) circumscribed
- c) outward
- d) inward

Q2-21) A circle may be defined by its CENTER and (circle all that apply)

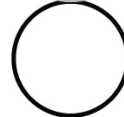
- a) radius.
- b) diameter.
- c) circumference.
- d) three points.

Modify commands

Q2-22) What is the rotation basepoint used in the example shown?

- a) circle center
- b) top quadrant
- c) right quadrant
- d) bottom quadrant

Rotated circle



Original circle

Q2-23) What is the scale basepoint used in the example shown?

- a) lower left corner
- b) upper left corner
- c) rectangle center
- d) upper right corner

Original rectangle



Scaled rectangle

Q2-24) When moving an object using relative coordinates, the first base point may be
(circle all that apply)

- a) the origin.
- b) a point on the object.
- c) the point that you are moving the object to.
- d) any point.

Name: _____ Date: _____

Selecting objects

Q2-25) What type of selection window selects everything touching and inside the defined window?

- a) standard window
- b) crossing window
- c) lasso window
- d) polygon window

Q2-26) Clicking, and dragging the mouse from right to left produces a

- a) standard window.
- b) crossing window.
- c) polygon window.
- d) fence.

Q2-27) Clicking and dragging the mouse from left to right produces a

- a) standard window.
- b) crossing window.
- c) polygon window.
- d) fence.

Q2-28) Clicking, holding and dragging the mouse produces a

- a) standard window.
- b) crossing window.
- c) lasso window.
- d) fence.

Q2-29) When AutoCAD is waiting for you to select an object, the crosshairs turn into a

- a) hand.
- b) pickbox.
- c) arrow.
- d) lasso.

Q2-30) You know when an object is selected because it becomes

- a) highlighted
- b) invisible
- c) blinks
- d) bigger

Q2-31) After selecting several objects to erase, what command would you type if you wanted to remove an object(s) from the selection set?

- a) add
- b) delete
- c) remove
- d) eliminate

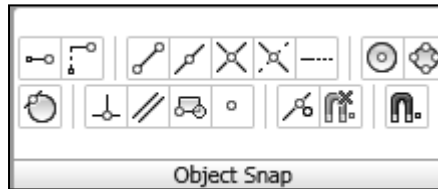
Q2-32) What key do you need to hold down if you want to remove an object(s) from the current selection set?

- a) ctrl
- b) esc
- c) tab
- d) shift

Name: _____ Date: _____

Object snaps

Some of the following questions refer to the object snap panel shown.



Q2-33) Object snaps not only speed up the drawing process, you cannot draw without them unless you enter coordinates.

- a) accurately
- b) neatly
- c) efficiently
- d) with the mouse

Q2-34) You may access the object snap settings by right clicking on the *Object Snap* icon in the status bar, clicking on the arrow next to the icon, or by typing what command?

- a) snapset
- b) snap
- c) osnap
- d) object

Q2-35) *Object Snap* panel: Top row, ninth icon from the left?

- a) endpoint
- b) quadrant
- c) nearest
- d) tangent

Q2-36) *Object Snap* panel: Top row, third icon from the left?

- a) endpoint
- b) quadrant
- c) nearest
- d) tangent

Q2-37) *Object Snap* panel: Second row, sixth icon from the left?

- a) endpoint
- b) quadrant
- c) nearest
- d) tangent

Q2-38) Which is the window and tab where you can access the *Autosnap Marker* size?

- a) options-files
- b) options-display
- c) options-drafting
- d) options-selection

Q2-39) The object snap that will not show where the first point of a line is located until the second point is selected.

- a) endpoint
- b) quadrant
- c) nearest
- d) tangent

Name: _____ Date: _____

Tracking and dynamic input

Q2-40) This enables you to snap along angled track lines.

- a) temporary tracking
- b) object snap tracking
- c) polar tracking
- d) dynamic input

Q2-41) What typed command allows you to change the polar tracking angle.

- a) pangle
- b) anglepo
- c) setang
- d) polarang

Q2-42) This icon, located in the status bar, allows you to create tracking lines off of object snap points.

- a) temporary tracking
- b) object snap tracking
- c) polar tracking
- d) dynamic input

Q2-43) What letters do you type to indicate that you want to create a temporary track point?

- a) tm
- b) tt
- c) tp
- d) pt

Q2-44) What key do you press to cycle through the dynamic input dimensions?

- a) esc
- b) shift
- c) ctrl
- d) tab

Grip boxes

Q2-45) Clicking once on an object, while not currently in a command, accesses the

- a) grip boxes
- b) properties window
- c) the command used to create the object
- d) modify commands

Q2-46) The grip box options may be accessed by on one of the boxes.

- a) left clicking
- b) right clicking
- c) double clicking
- d) dragging

Q2-47) The command used to make a line longer by a prescribed length.

- a) extend
- b) lengthen
- c) stretch
- d) grow

DRAWING IN AUTOCAD PROBLEMS

Name: _____ Date: _____

P2-1) (May be attempted after completing the *Drawing using Coordinates Tutorial* in section 2.10) Write down the **relative Cartesian coordinates** needed to create the following object in the direction indicated in the figure. Assume that the start point is created by clicking the mouse; therefore, we do not know its exact coordinate position.

Start point: Click mouse anywhere

Next point:

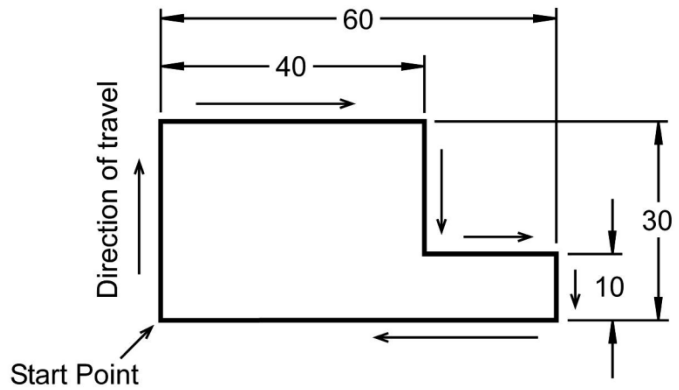
Next point:

Next point:

Next point:

Next point:

Next point:



Write down the **relative polar coordinates** needed to create the following object.

Start point: Click mouse anywhere

Next point:

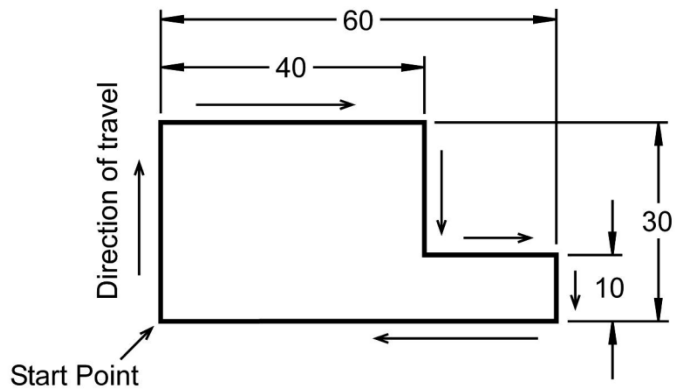
Next point:

Next point:

Next point:

Next point:

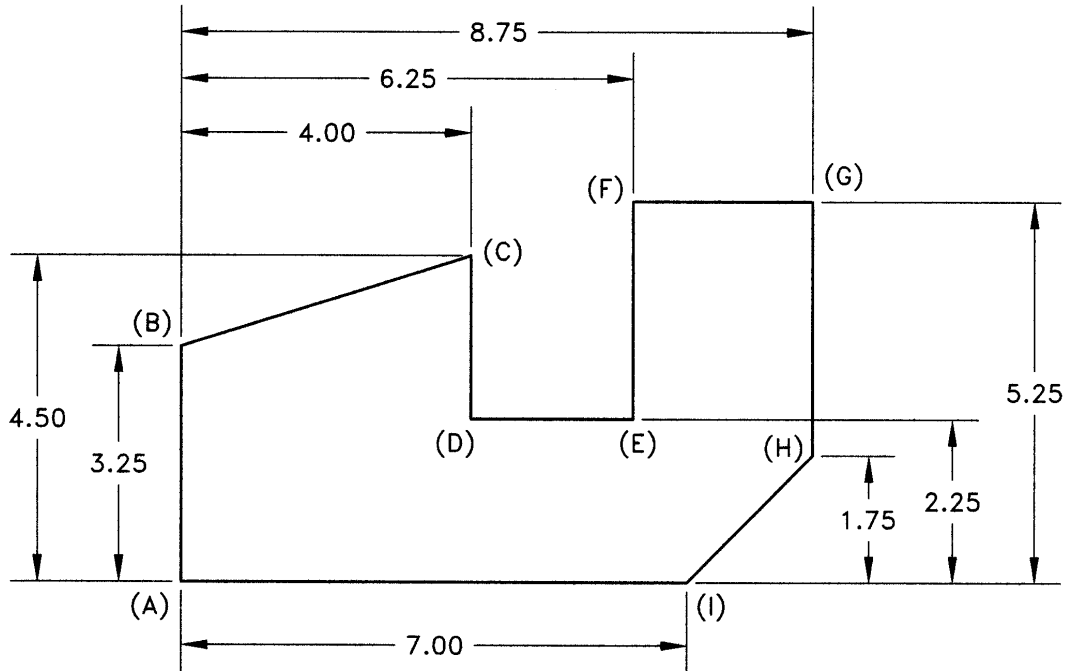
Next point:



NOTES:

Name: _____ Date: _____

P2-2) May be attempted after completing the *Drawing using Coordinates Tutorial* and the *Printing Tutorial* in sections 2.10 and 2.12. Fill in the table below with the appropriate relative coordinates needed to draw this object in AutoCAD and then draw the object in AutoCAD. Print using a window and the scale to fit option. Dimensions are in inches.

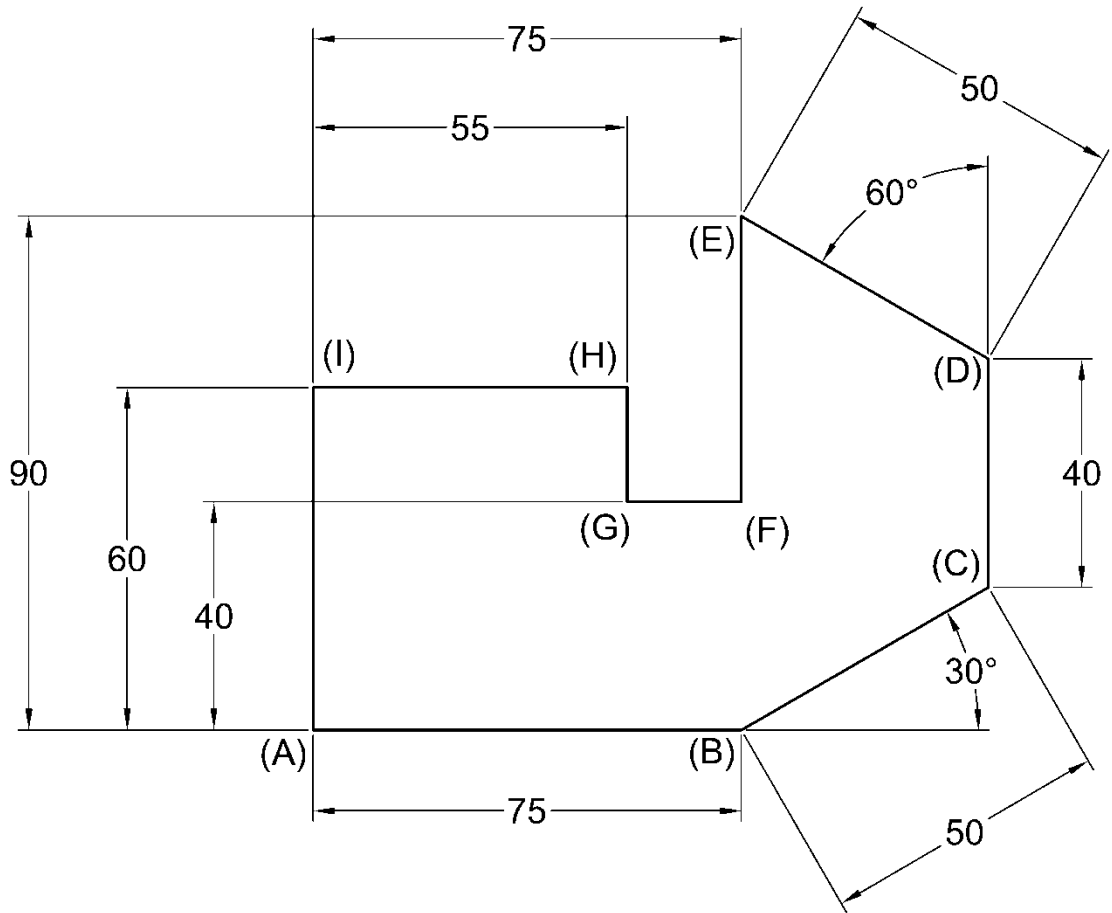


To Point	Relative Coordinate
A to B	@
B to C	@
C to D	@
D to E	@
E to F	@
F to G	@
G to H	@
H to I	@
I to A	@

NOTES:

Name: _____ Date: _____

P2-3) May be attempted after completing the *Drawing using Coordinates Tutorial* and the *Printing Tutorial* in sections 2.10 and 2.12. Fill in the table below with the appropriate relative coordinates needed to draw this object in AutoCAD and then draw the object in AutoCAD. Print using a window and the scale to fit option. Dimensions are in millimeters.

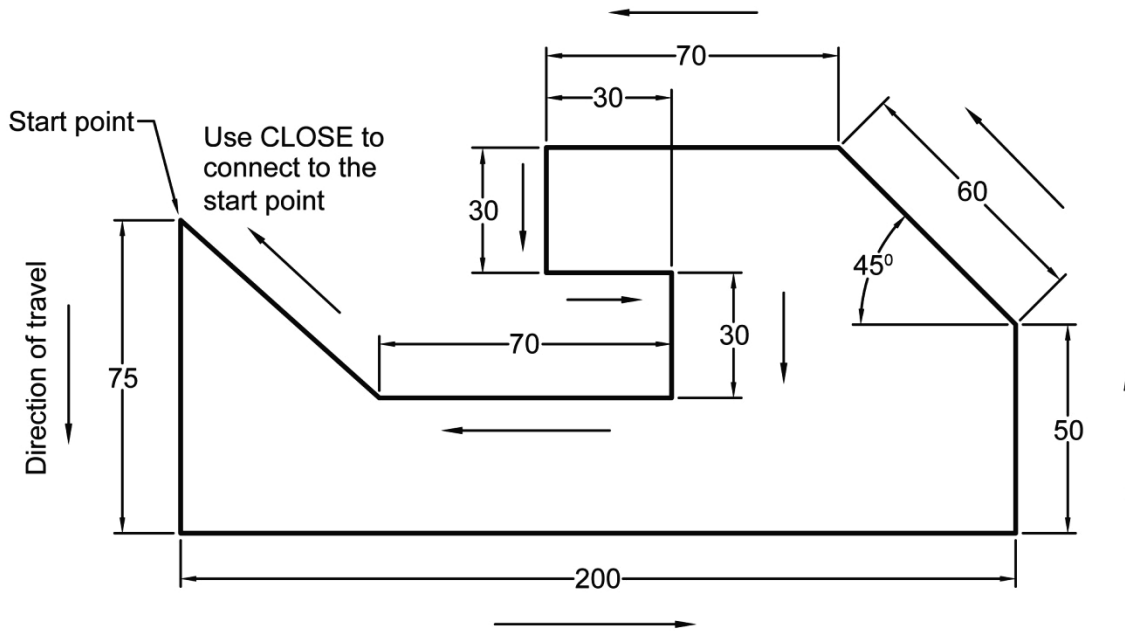


To Point	Relative Coordinate
A to B	@
B to C	@
C to D	@
D to E	@
E to F	@
F to G	@
G to H	@
H to I	@
I to A	@

NOTES:

Name: _____ Date: _____

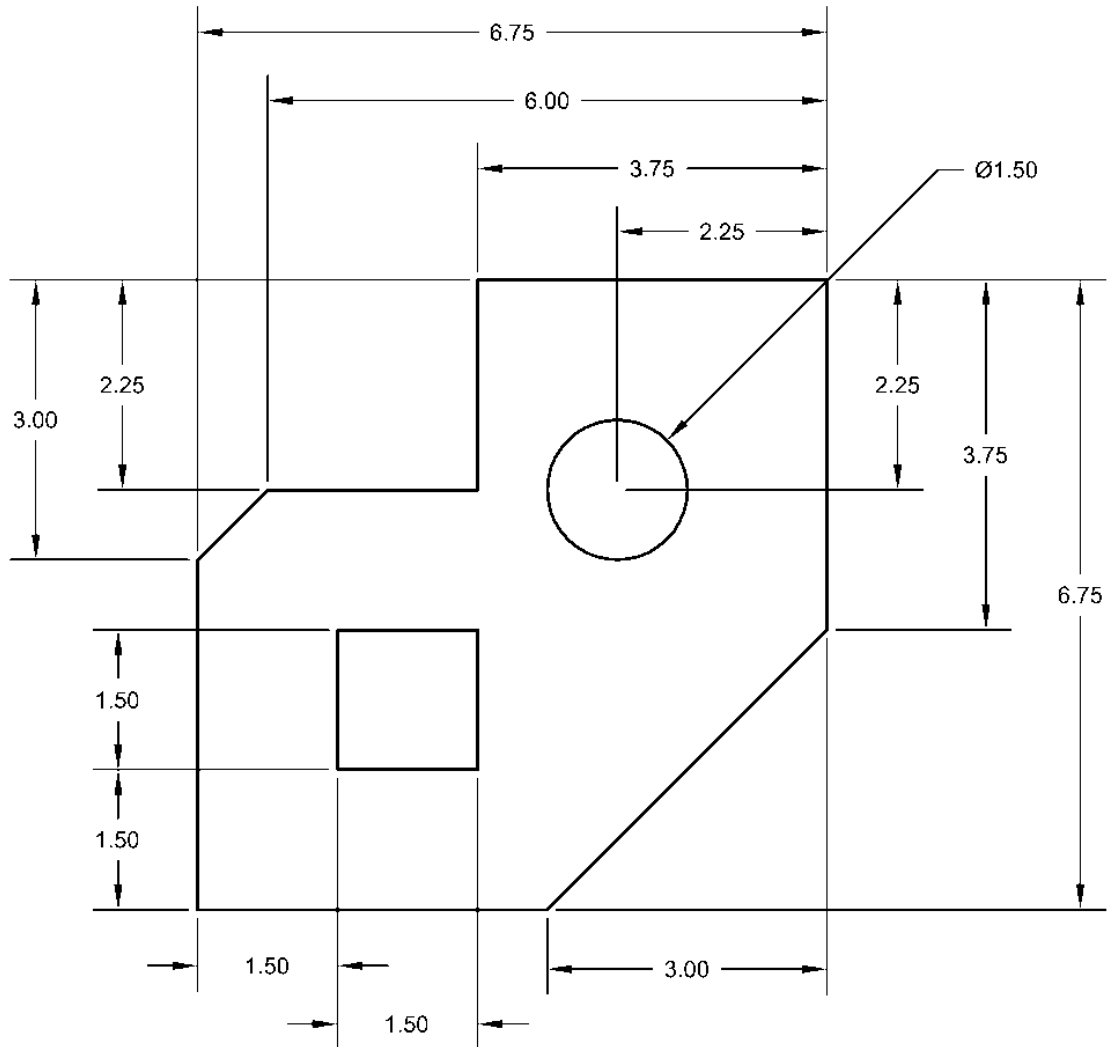
P2-4) May be attempted after completing the *Drawing using Coordinates Tutorial* and the *Printing Tutorial* in sections 2.10 and 2.12. Draw the following object using relative Cartesian and/or relative polar coordinates. Draw the object starting at the point indicated and in the direction indicated. To connect the start point and end point, type CLOSE while still in the LINE command. Write down the relative coordinates used to create the object in the table. Print using a window and the scale to fit option. Dimensions are in millimeters.



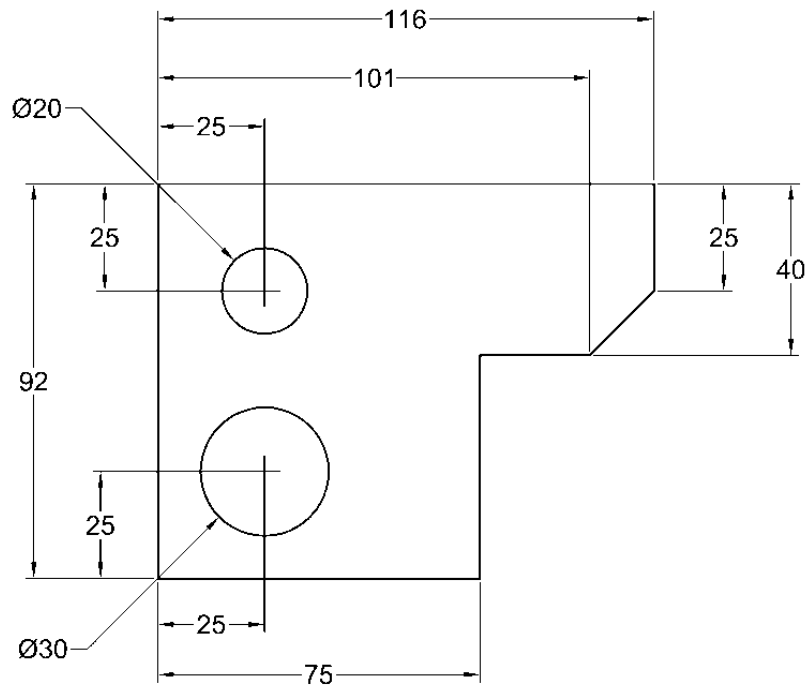
To Point	Relative Coordinate
Start point to next point	@
to Next point	@
to Next point	@
to Next point	@
to Next point	@
to Next point	@
to Next point	@
to Next point	@
to Next point	@
to Next point	CLOSE

NOTES:

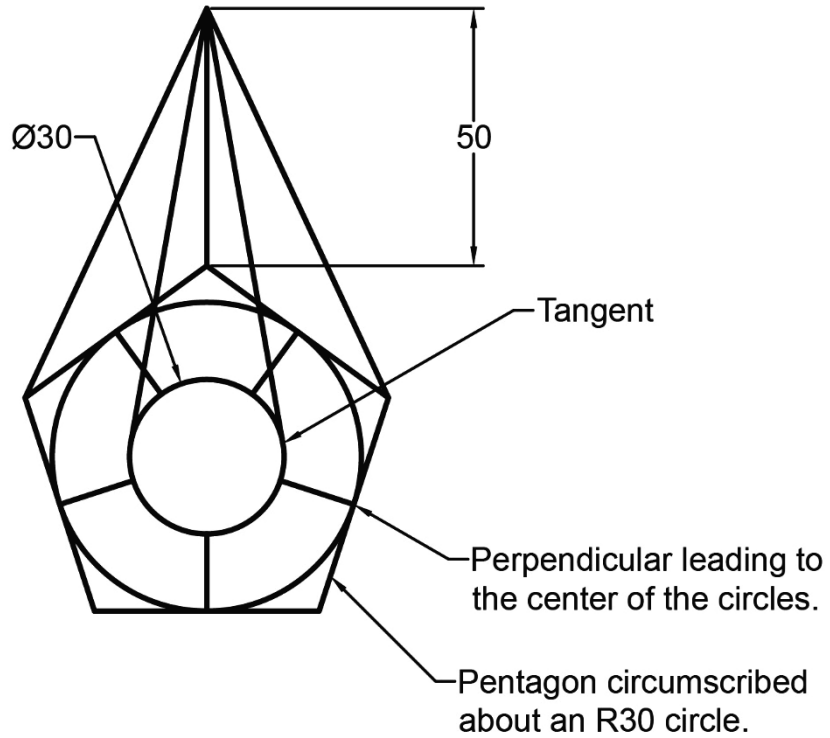
P2-5) May be attempted after completing the *WCS/UCS Coordinate Systems, Drawing using Coordinates* and the *Printing Tutorials* in sections 2.9, 2.10 and 2.12. Draw the following object in AutoCAD. HINT: To draw the circle and square, try changing your UCS to one of the corners of the main body. Print using a window and the scale to fit option. Dimensions are in inches.



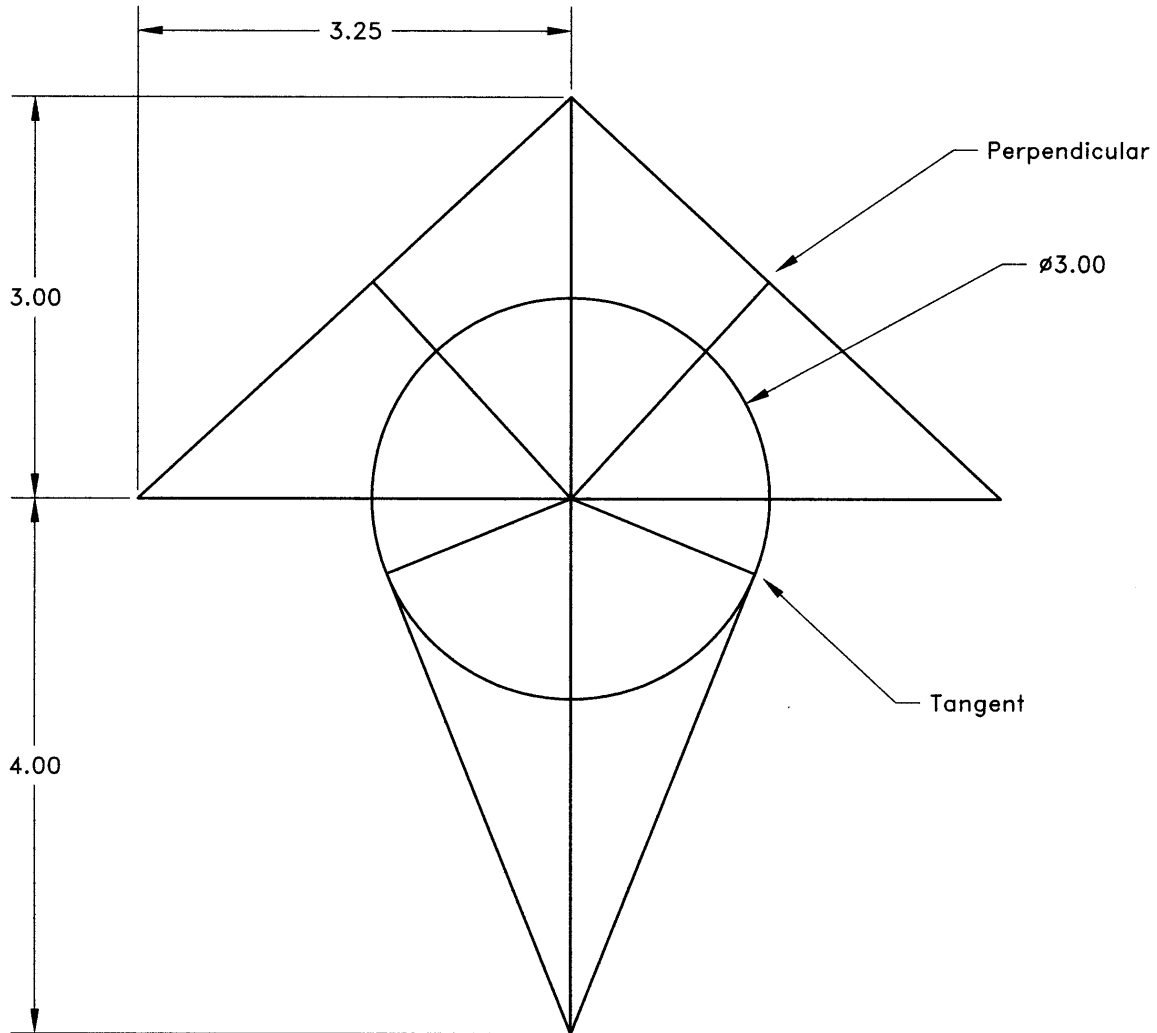
P2-6) May be attempted after completing the *WCS/UCS Coordinate Systems, Drawing using Coordinates* and the *Printing Tutorials* in sections 2.9, 2.10 and 2.12. Draw the following object in AutoCAD. HINT: To draw the circles, try changing your UCS to the corners of the main body. Print using a window and the scale to fit option. Dimensions are in millimeters.



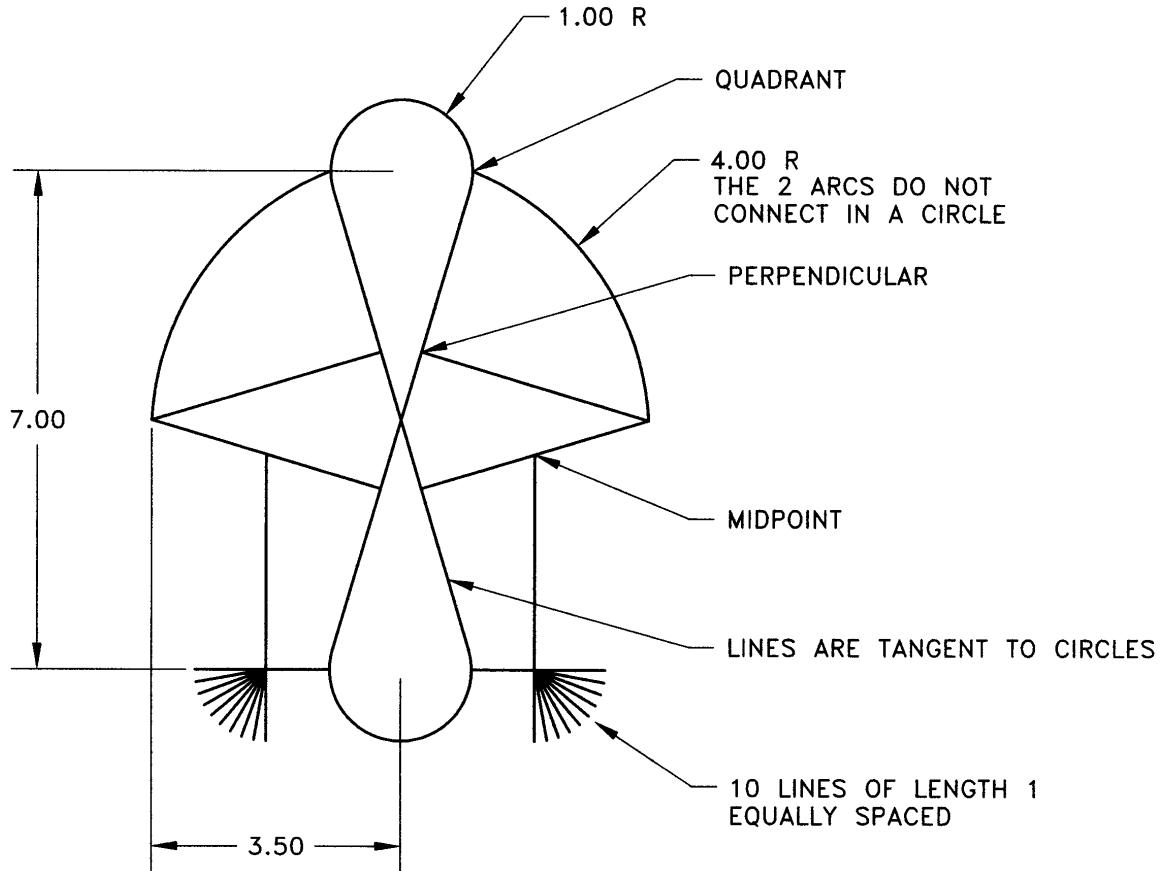
P2-7) May be attempted after completing the *Object Snap Tutorial* in section 2.20. Draw the following object using object snap commands where necessary. Put your name on the drawing and then print it using a *Window* and the *Scale to Fit* option. Dimensions are in millimeters.



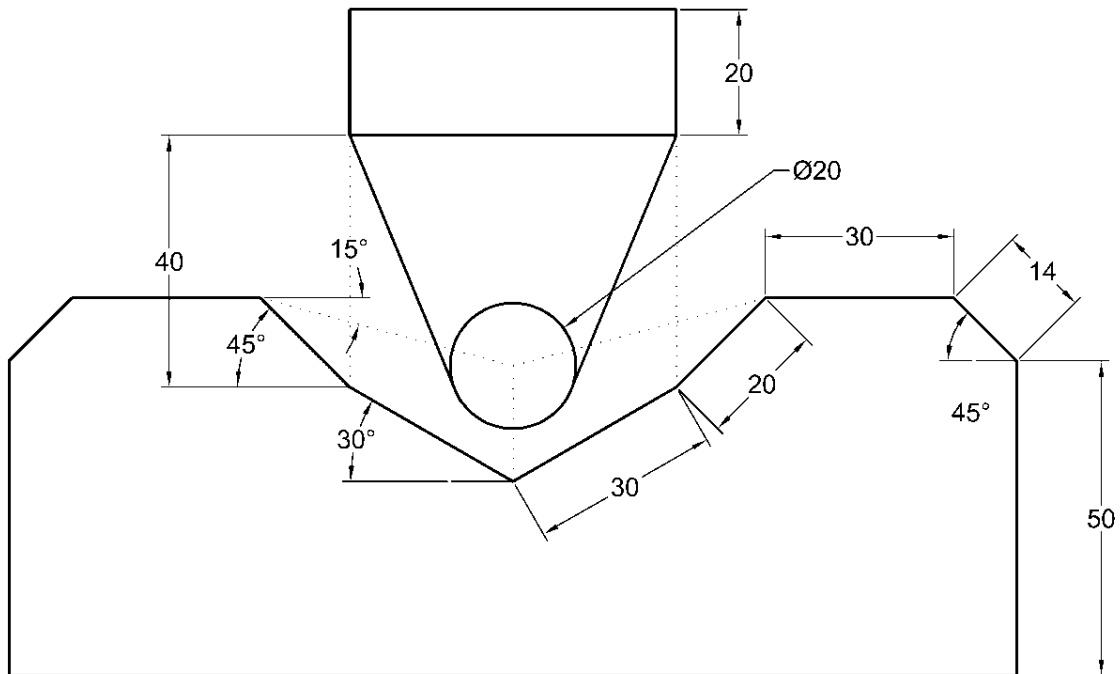
P2-8) May be attempted after completing the *Object Snap Tutorial* in section 2.20. Draw the following object in AutoCAD using object snap commands where necessary. Put your name on the drawing and then print it using a *Window* and the *Scale to Fit* option. Dimensions are in inches.



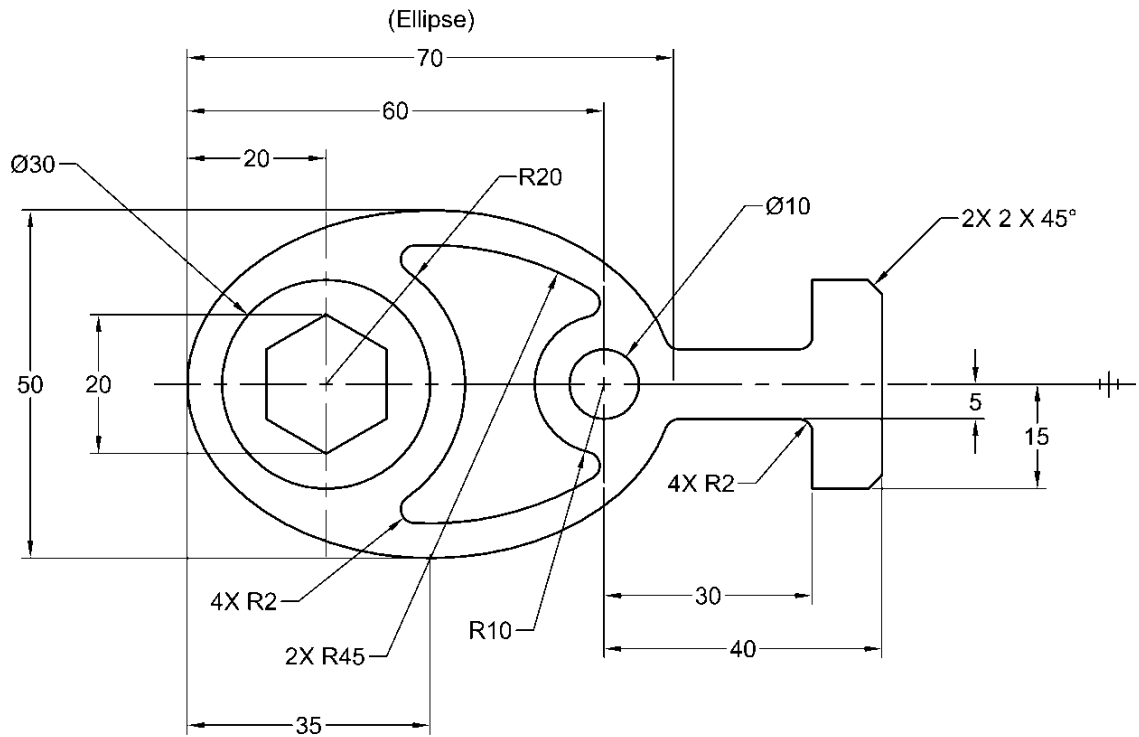
P2-9) May be attempted after completing the *Object Snap Tutorial* in section 2.20. Draw the following object in AutoCAD using object snap commands where necessary. Put your name on the drawing and then print it using a *Window* and the *Scale to Fit* option. Dimensions are in inches. Hint: To create the arc, choose the start point, then the end point, and then the radius. Remember that arcs are drawn according to the right hand rule.



P2-10) May be attempted after completing the *Tracking Tutorial* in section 2.23. Draw the following object with the help of track points. Put your name on the drawing and then print using a 1 to 1 scale. Dimensions are in millimeters.



P2-11) May be attempted after completing the *Object Snap Tutorial* in section 2.20. Draw the following object in AutoCAD. Put your name on the drawing and then print it using a *Window* and the *Scale to Fit* option. Dimensions are in millimeters.



P2-12) May be attempted after completing the *Object Snap Tutorial* in section 2.20. Draw the following object in AutoCAD. Put your name on the drawing and then print it using a *Window* and the *Scale to Fit* option. Dimensions are in millimeters.

