

Ministry of Higher Education & Scientific Research
Northern Technical University
Engineering Technical College / Mosul
Dept. of Power Mechanics Engineering Technology

Engineering Drawing
For
First year students

Contents

1	Module Objectives:	4
2	Module Delivery Plan (Weekly Lab. Syllabus).....	4
3	Getting Started with AutoCAD	4
3.1	Introduction to AutoCAD	4
3.2	Starting Up AutoCAD 2010	4
3.3	Workspaces Defined.....	6
3.4	Creating a New File in 2D Drafting & Annotation Workspace	8
3.4.1	Opening an Existing File	9
3.4.2	Keyboard Input	9
3.4.3	Special Keys.....	9
3.4.4	Function Keys.....	10
3.4.5	User Interface Layout.....	10
3.4.6	Ribbon Defined.....	10
3.4.7	Ribbon Controls.....	10
3.4.8	Add or Remove Tabs	11
3.4.9	Application Menu	11
3.4.10	Quick Access Toolbar.....	12
3.4.11	Ribbons	12
3.4.12	InfoCenter	15
3.4.13	Graphical Area.....	16
3.4.14	Status Bar	16
3.4.15	Points In AutoCAD	16
3.4.16	AutoCAD Default Settings	17
3.5	Things You should Know about AutoCAD	17
3.6	Drawing Area Setup.....	17
3.7	Designation of Sizes.....	18
3.7.1	Drawing Units Setup	18
3.8	Viewing Commands.....	19
4	Basic Draw Commands	21
4.1	Introduction.....	21
4.1.1	Typing Commands.....	21
4.1.2	Reissuing the Last Command	21
4.1.3	Pointing Device (Mouse).....	21
4.1.4	Undo and Redo	22
4.1.5	Point.....	22
4.2	Basic Commands	23
4.2.1	Line Command	23
4.2.2	Circles	26
4.2.3	Arc Command.....	27
5	Drawing Precision in AutoCAD	28
5.1	SNAP Command	28
5.2	Turn Snap On/OFF	28
5.3	Grid Command	28
5.4	Turn Grid On/Off.....	28
5.5	Object Snaps	29
5.5.1	Object Snaps Defined	29
5.5.2	Effect of Using Object Snaps.....	29
5.5.3	Running Object Snap.....	29
6	Modify commands	31
6.1	Move Command	31

6.2	Copy Command.....	31
6.3	Mirror Command.....	32
6.4	Rotate Command	33
7	Advanced Draw Commands	35
7.1	Polygon Command.....	35
7.2	Rectangle Command.....	35
7.3	ELLIPSE.....	36
7.4	Pline Command.....	38
8	Advanced Modify commands	39
8.1	Erase Command.....	39
8.2	TRIM.....	39
8.3	EXTEND.....	40
8.4	Offset Command.....	41
8.5	SCALE.....	41
8.6	Break Command.....	42
8.7	Stretch Command.....	43
8.8	Fillet.....	43
8.9	Chamfer.....	44
8.10	Array.....	45
8.10.1	Rectangular Array.....	45
8.10.2	Polar Array	45
9	Creating Complex Objects	47
10	Adding Dimensions.....	55
10.1	Linear Dimensions.....	55
10.2	Aligned Dimensions.....	55
10.3	Radial Dimensions.....	55
10.4	Angular Dimensions	56
10.5	Continued and Baseline Dimensions.....	56
10.6	Leaders.....	56
10.7	Quick Dimensions.....	57
11	Text, Hatching Objects and Layers.....	58
11.1	Text.....	58
11.1.1	Access Command	58
11.1.2	DTEXT (Dynamic Text).....	58
11.1.3	Text Styles	58
11.2	Hatching Objects	59
11.2.1	Type and Pattern.....	60
11.2.2	Gradient Hatch.....	60
11.2.3	Hatch and Gradient Dialog Box: Gradient Tab.....	60
11.3	Layers.....	61
11.3.1	Access Command.....	61
12	Setting Up a Layout	63
12.1	Title Block	63
12.2	Borders and Frames	63
12.3	Scales.....	64
13	Samples of Theoretical Exam Questions.....	65
13.1	Multiple Choice Questions (MSQ) Samples.....	65
13.2	Fill in the Blank Questions	71
13.2.1	Fill in the blank the correct AutoCAD shortcut commands.....	71
13.2.2	AutoCAD 2010 Shortcut Commands Table:	72
13.3	State (True or False) Sample Questions: -	73

1 Module Objectives:

The goal of engineering drawing is to learn how to use AutoCAD, a software used to create accurate technical drawings. First, students will get familiar with the AutoCAD program and learn to use basic tools like lines, circles, and rectangles to make 2D drawings. After that, they will learn how to change and adjust their drawings by using tools like move, copy, and mirror. Students will also learn how to add dimensions and text to their drawings to make them clear and easy to understand.

2 Module Delivery Plan (Weekly Lab. Syllabus)

Week no.	Material Covered
1	Getting Started with AutoCAD
2	Basic Draw Commands
3	
4	
5	Drawing Precision in AutoCAD
6	Modify commands
7	Advanced Draw Commands
8	
9	
10	Advanced Modify Commands
11	
12	Creating Complex Objects
13	Adding Dimensions
14	Hatching / Text
15	Setting Up a Layout

3 Getting Started with AutoCAD

3.1 Introduction to AutoCAD

AutoCAD is a computer-aided design (CAD) software used for creating 2D and 3D designs, developed by the American company Autodesk. The first version was released in 1980 under the name CAD (Computer-Aided Design). It is used in various fields such as civil engineering, architecture, mechanical engineering, interior design, and electrical engineering. AutoCAD is used to create all types of project designs and visualizations, as they would appear in real life.

3.2 Starting Up AutoCAD 2010

There are several ways to open the program, including:

1. To open AutoCAD, click Start menu (Windows) ► (All) Programs ► Autodesk ► AutoCAD 2010 ► AutoCAD 2010 - English.

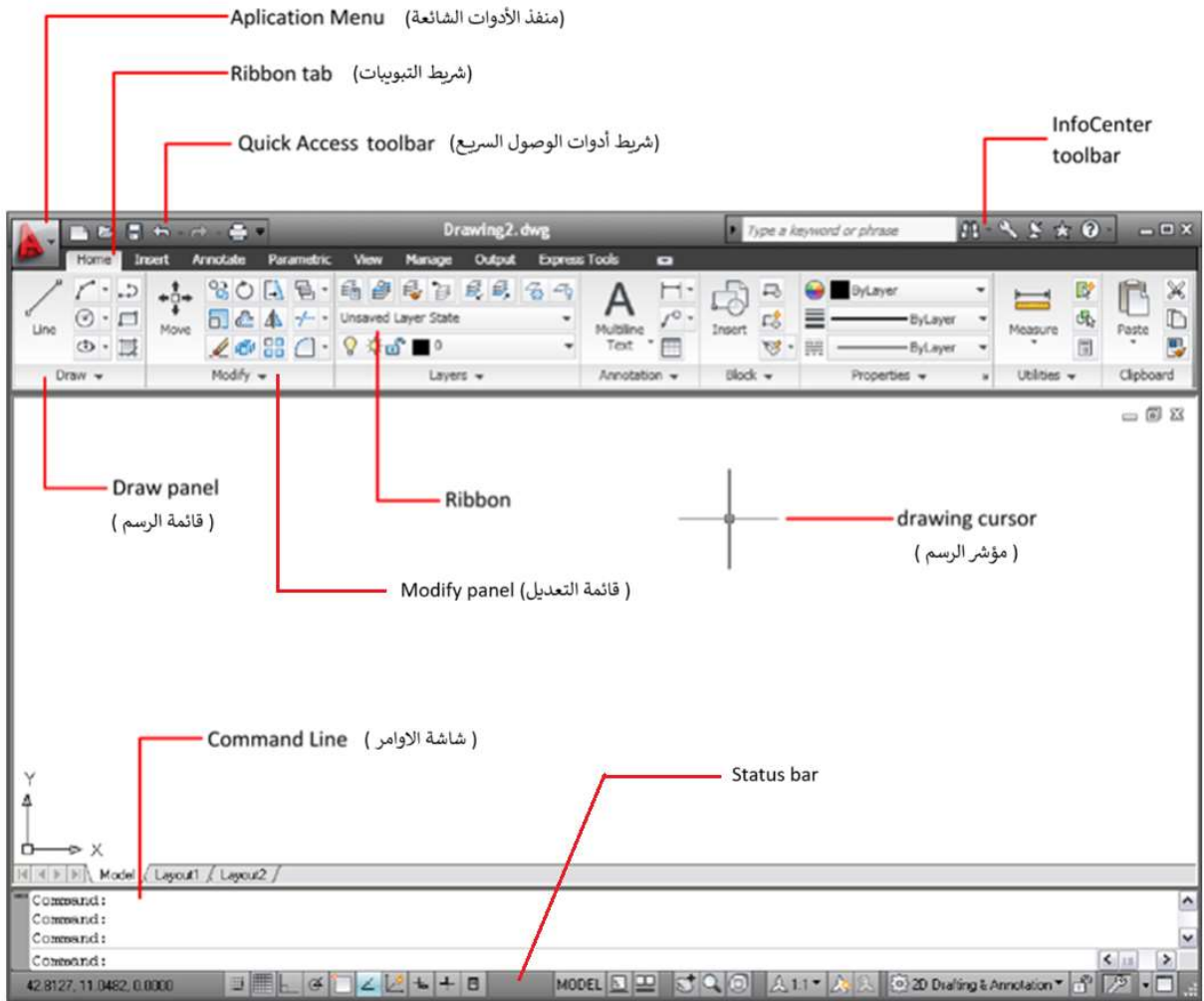
NOTE: If the New Features dialog box appears, select Maybe later and OK to close it. The AutoCAD window opens with an empty drawing file named Drawing1.dwg.

2. Double Click on the program's icon visible on the desktop

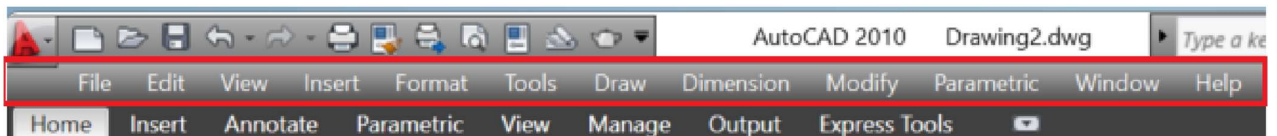


3. Before moving on to the next lesson, become familiar with the areas of the AutoCAD window labeled below.

Note that AutoCAD automatically assigns generic name, Drawing X, as new drawings are created. In our example, AutoCAD opened the graphics window using the default system units and assigned the drawing name Drawing1.



If necessary, click on the down-arrow in the Quick Access bar and select Show Menu to display the AutoCAD Menu Bar. The Menu Bar provides access to all AutoCAD commands.



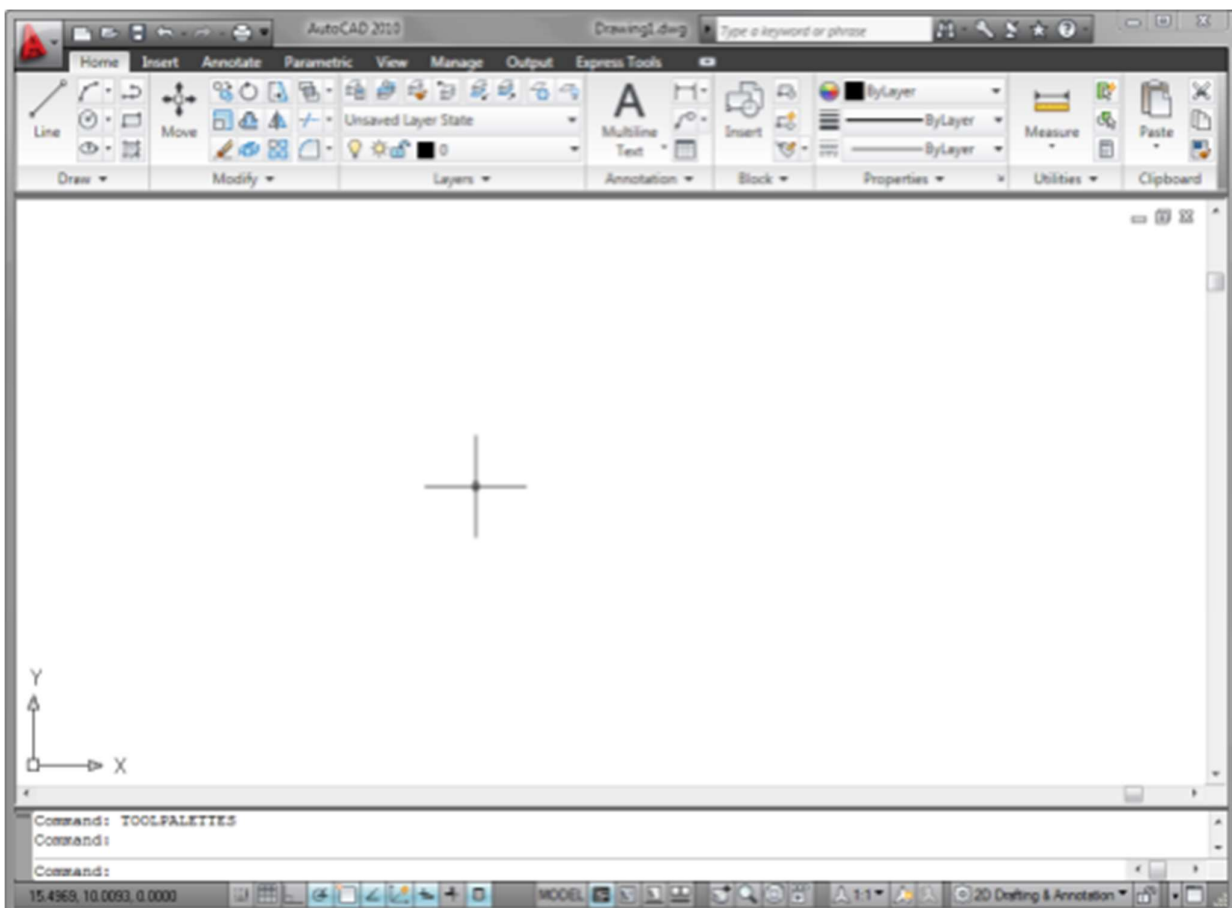
3.3 Workspaces Defined

When you launch the application, the interface elements displayed are only those associated with the active workspace. A workspace is a task-oriented drawing environment oriented in such a way as to provide you with only the tools and interface elements necessary to accomplish the tasks relevant to that environment.

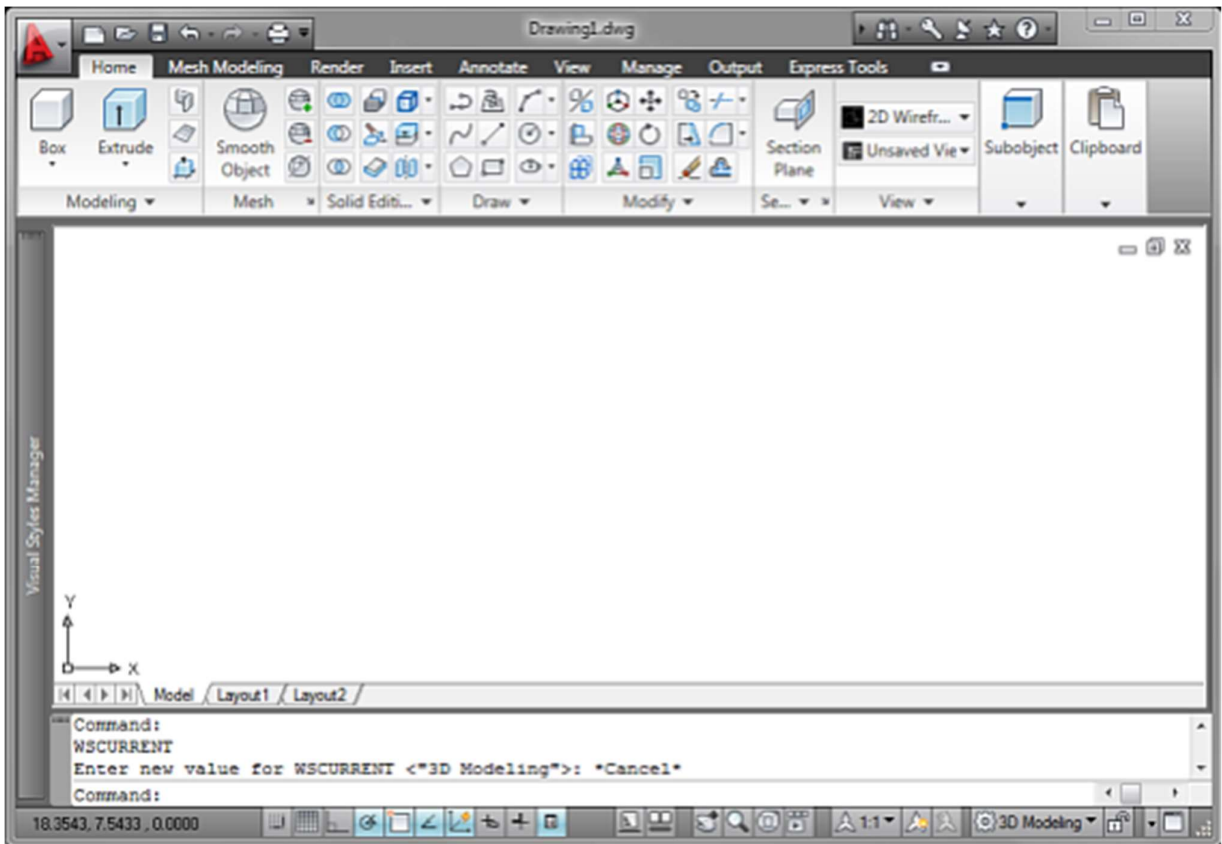
By default, AutoCAD has four workspace configurations:

- 2D Drafting & Annotation
- 3D Modeling
- AutoCAD Classic
- Initial Setup Workspace

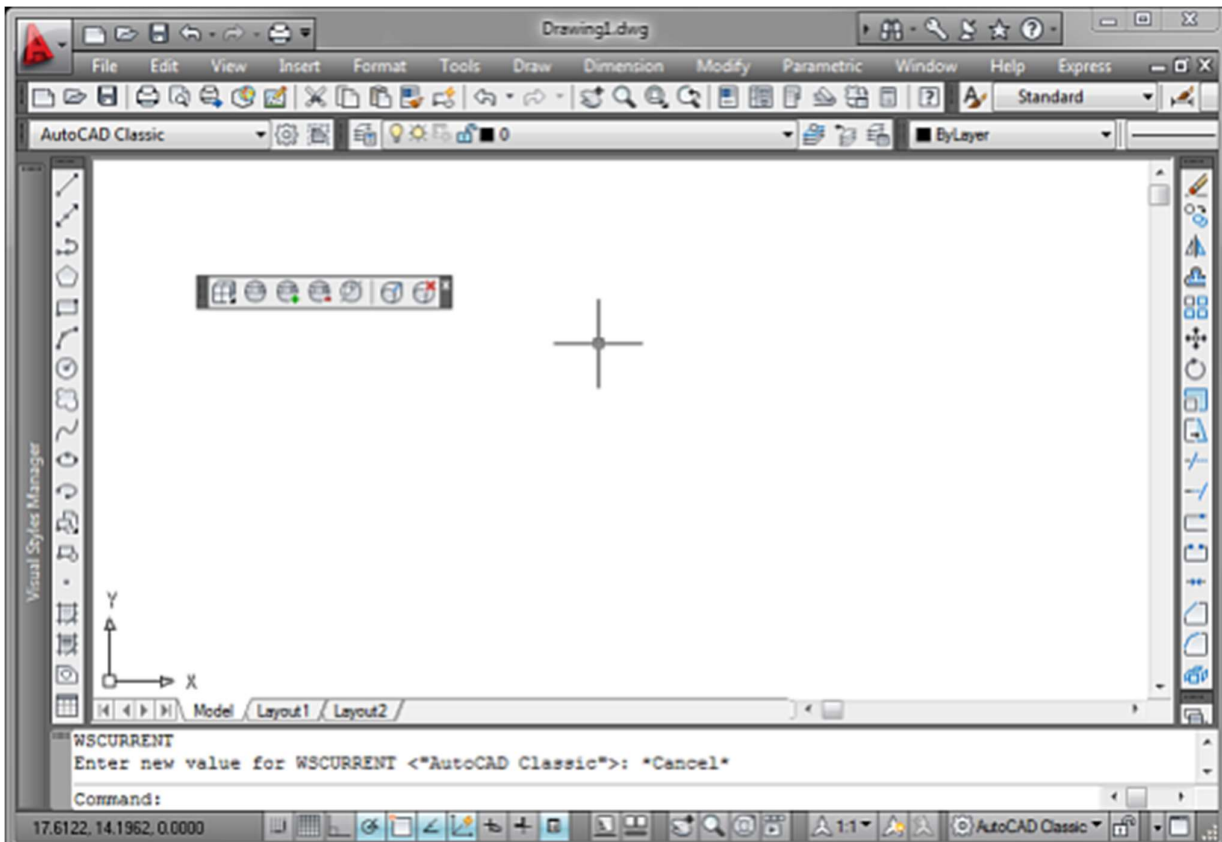
AutoCAD is shown here with the 2D Drafting & Annotation workspace active.



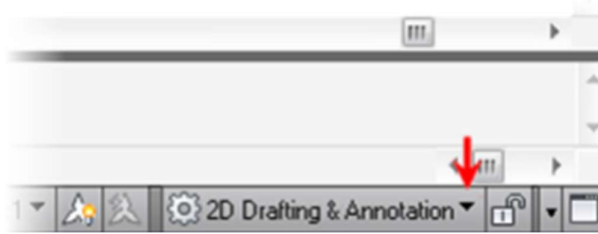
AutoCAD is shown here with the 3D Modeling workspace active.



AutoCAD is shown here with the AutoCAD Classic workspace active.



After you start the program, you can switch to the desired workspace. The application will open with the last workspace used. The Workspace Switching dropdown list is accessed in the lower right corner of the AutoCAD window on the Status Bar.



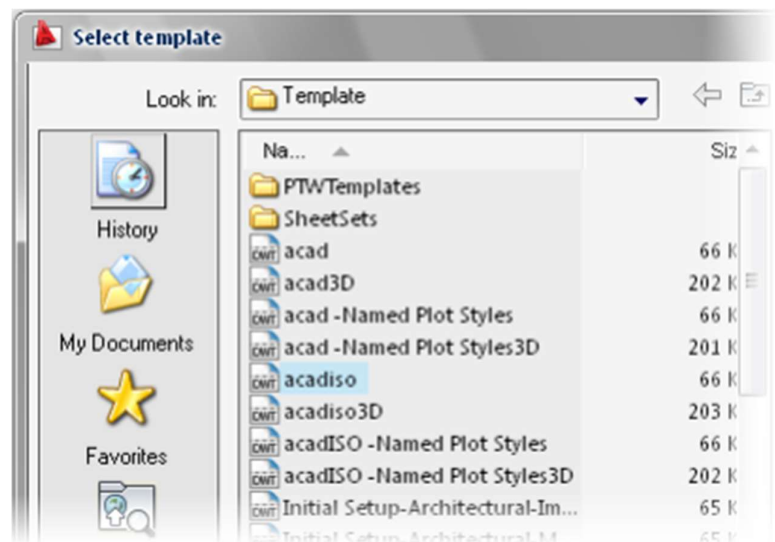
3.4 Creating a New File in 2D Drafting & Annotation Workspace

The following steps give an overview of activating the 2D Drafting & Annotation workspace.

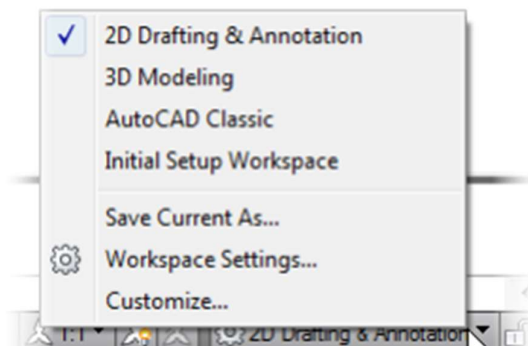
1. Start AutoCAD.
2. Use the default drawing toolbar or on the Quick Access toolbar, click New.



3. Select acad.dwt (imperial) or acadiso.dwt (metric) as the template file.

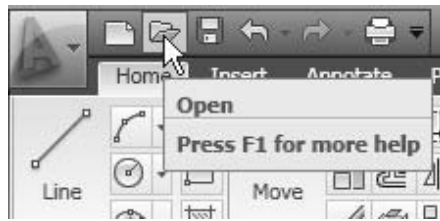


4. Click the Workspaces settings icon, located at the bottom right corner of the AutoCAD window. Select 2D Drafting & Annotation.

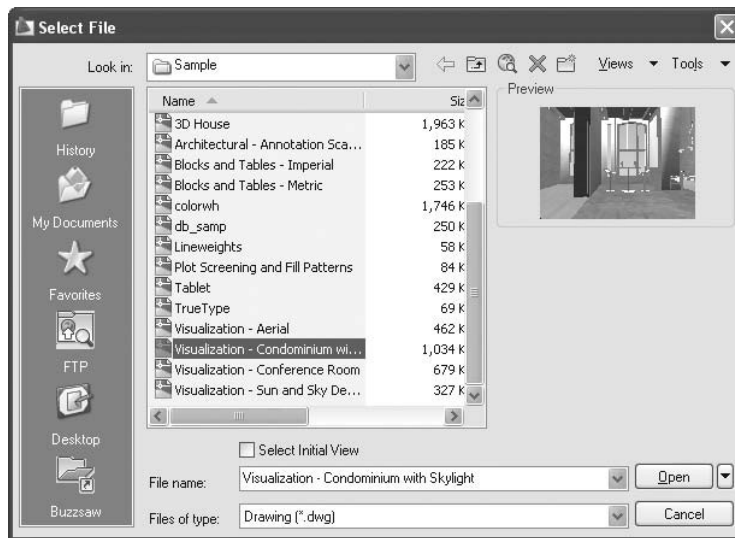


3.4.1 Opening an Existing File

- To open an existing file for further editing, click the **Open** button from the **Quick Access Toolbar**.



- The following dialog box will appear:



- Specify the hard drive, and the folder your file resides in.
- AutoCAD drawing files have the extension *.dwg.
- If you want to open a single file, select the file and click open (you can also double-click on the file's name).
- If you want to open more than one file, select the first file name, then hold the [Ctrl] key on the keyboard, and click the other file names.
- You can open as many files as you wish.
- When you are done, click **Open**.

3.4.2 Keyboard Input

Using the keyboard is familiar to everyone who works with computers. For much of the work that you do in AutoCAD you use the keyboard, but you use a few keystrokes more often than others.

3.4.3 Special Keys

You use the following keys most often. These keys have special meaning to the software.

- Use the ESC key to cancel all current actions and return to the Command: prompt.
- Press the ENTER key following all keyboard input. You also complete many commands by pressing ENTER.
- Pressing the SPACEBAR is equivalent to pressing the ENTER key and is often easier to use.

- Pressing the SPACEBAR or ENTER at the Command: prompt repeats the last command used.
- Pressing the UP and DOWN arrow keys will cycle through previous commands used.
- The TAB key is especially useful to navigate in a dialog box. You should use the TAB key to move from field to field. Be careful not to press ENTER.

3.4.4 Function Keys

The use of each of the function keys can be duplicated in other ways with the exception of F2. You may find that the on-screen equivalents to the function keys are easier and allow you to keep your eyes on the screen.

Option	Description
F1	Displays Help
F2	Toggles Text Window
F3	Toggles OSNAP
F4	Toggles TABMODE
F5	Toggles ISOPLANE
F6	Toggles UCSDETECT
F7	Toggles GRIDMODE
F8	Toggles ORTHOMODE
F9	Toggles SNAPMODE
F10	Toggles Polar Tracking
F11	Toggles Object Snap Tracking
F12	Toggles Dynamic Input

3.4.5 User Interface Layout

There are interface elements common to other Windows applications such as ribbon panels, toolbars, and menus. If you have used other Windows applications, these user interface elements should appear familiar. However, there are interface elements such as the command line and the status bar, which are unique to AutoCAD.

3.4.6 Ribbon Defined

The ribbon is a special tool palette associated with each workspace containing only the tools and controls relevant to that workspace. For example, the ribbon for the 2D Drafting & Annotation workspace contains tools relevant to 2D drawing, dimensioning, and annotating, but does not contain tools for 3D geometry creation.

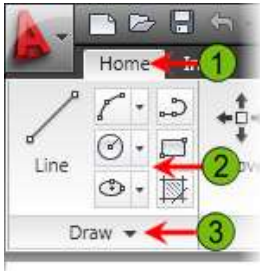
3.4.7 Ribbon Controls

The ribbon is turned on by default when you start the software in either the 2D Drafting & Annotation or the 3D Modeling workspace. The ribbon is organized into a series of tabs. Each tab includes a different set of panels with related commands and controls that may be found on the Classic AutoCAD toolbars and dialog boxes.

You can turn the tabs and associated panels on the ribbon on or off by right-clicking on the ribbon area and selecting Tabs or Panels to select the desired options. You can also turn Panel Titles on or off by right-clicking the Panel tabs. Additionally, you can save your Ribbon configuration.

Each tab on the ribbon has its own set of panels that contain groups of related tools, such as those used for 2D drawing, adding text, or adding dimensions. Some panels can be expanded

to display more tools. Likewise, some tools can be expanded for more options, such as the Circle tool as indicated by an arrow in the corner of the icon.



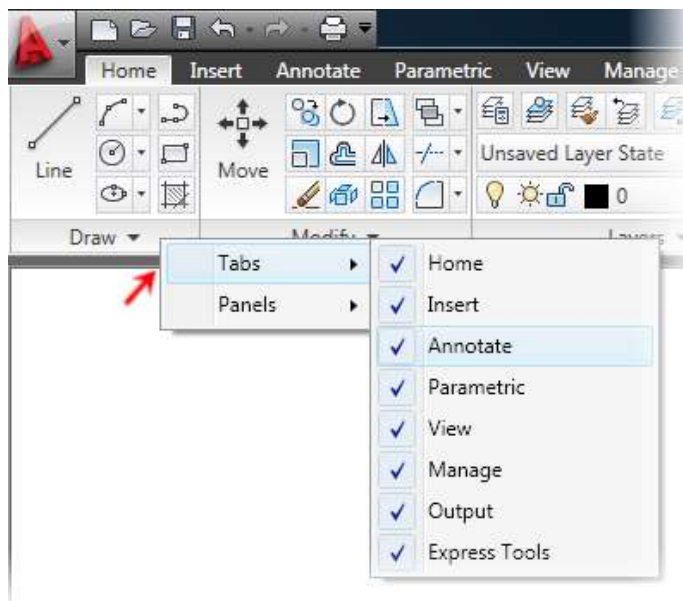
1 Tabs : Identifies the purpose and name of the control panel.

2 Panels : Contains groups of related tools associated with the selected tools.

3 More Tools : Click and hold the down arrow to display more tools and options in the selected panel.

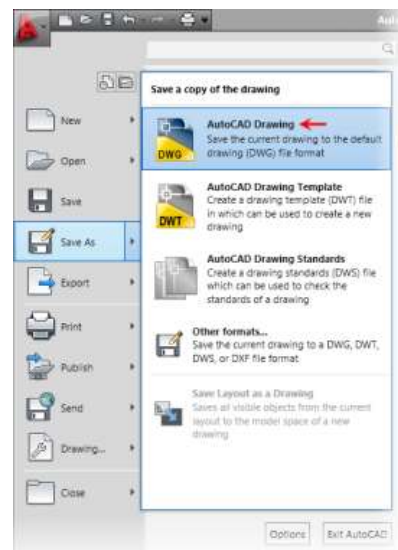
3.4.8 Add or Remove Tabs

To turn specific tabs on or off, right-click in the ribbon and select Tabs. Choose to display or remove tabs from the ribbon. Tabs currently displayed are indicated with a check mark.



3.4.9 Application Menu

You can use the Application Menu to access several key commands such as New, Open, Save, Print, and Close. Most of these commands lead to submenus that give you more detailed options.



Using this menu, you can:

- Create a new file
- Open an existing file
- Save the current file
- Save the current file under a new name (Save As) and/or in a different folder
- Export the current file to a different file format
- Print the current file
- Publish the current file
- Send the current file to eTransmit or email
- Use all the functions related to your drawing
- Close the current file
- Exit AutoCAD

3.4.10 Quick Access Toolbar

The Quick Access Toolbar is the small toolbar located at the top left of the screen:

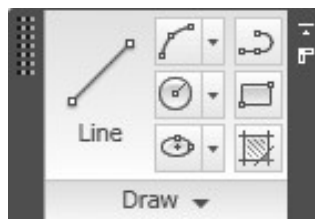


Using this toolbar, you can:

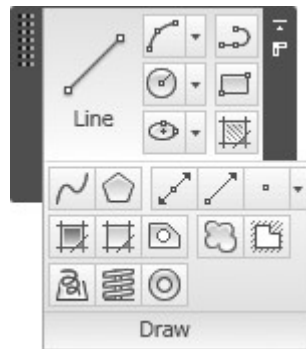
- Create a new file
- Open an existing file
- Save the current file
- Undo and redo
- Print the current file

3.4.11 Ribbons

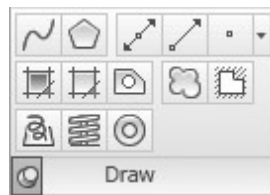
- Ribbons consist of two parts:
 - Tabs
 - Panels
- For example, the Home tab consists of eight panels: Draw, Modify, Layers, Annotation, Block, Properties, Utilities, and Clipboard.
- In each tab you will see different panels.
- The following is the Draw panel:



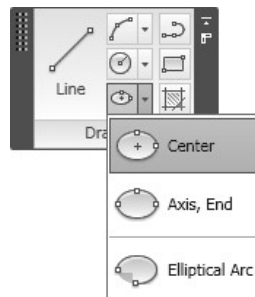
- Some panels (such as the Draw panel) have a small triangle near the title, which indicates that there are more buttons available. If you click on it you will see the following:



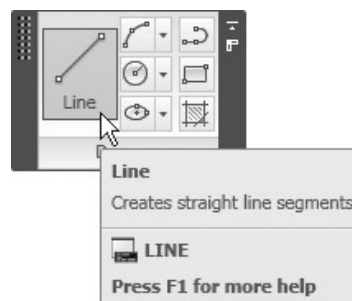
- At the lower left-hand corner of the panel you will see a small pushpin. If you click on it, this will be the default view. To return to the previous view, simply click the pushpin again.



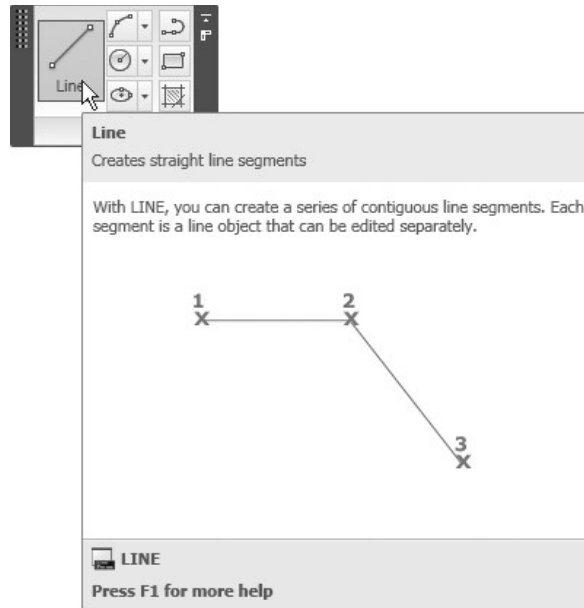
- Some panel buttons may have a small triangle just to the right, which means there are additional options, as shown here:



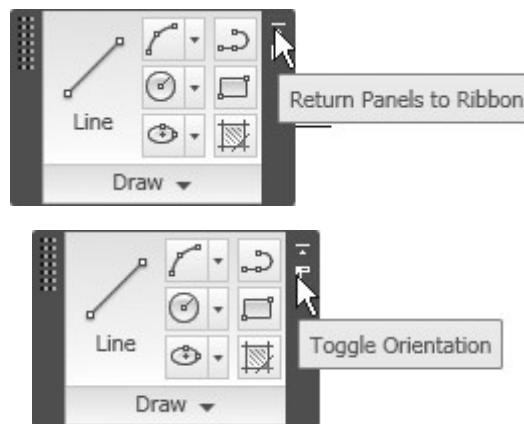
- If you hold your cursor over any button for one second, a small help screen appears:



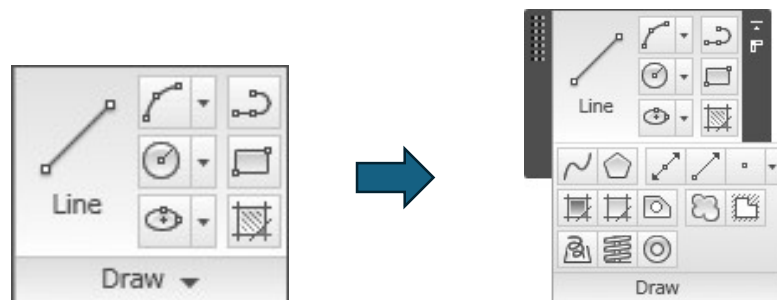
- However, if you hold your cursor over the button for three seconds, you will see an extended help screen:



- Panels can be docked or floating. By default, all panels are docked. To make a panel a floating panel, simply click on the name of the panel, hold it, and drag it to its new location.
- If all panels remain docked, you will not see any panel outside the tab it belongs to. But if you make any panel a floating panel, then you will be able to see it in all other tabs.
- While the panel is floating, you will see two small buttons on the right-hand side. The following image shows the function of each one:

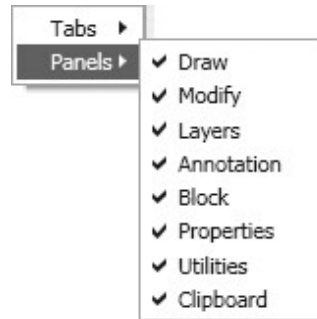


- All panels have two orientations: a vertical orientation and a horizontal orientation. The following illustration shows the tabs and panels that appear for both orientations.

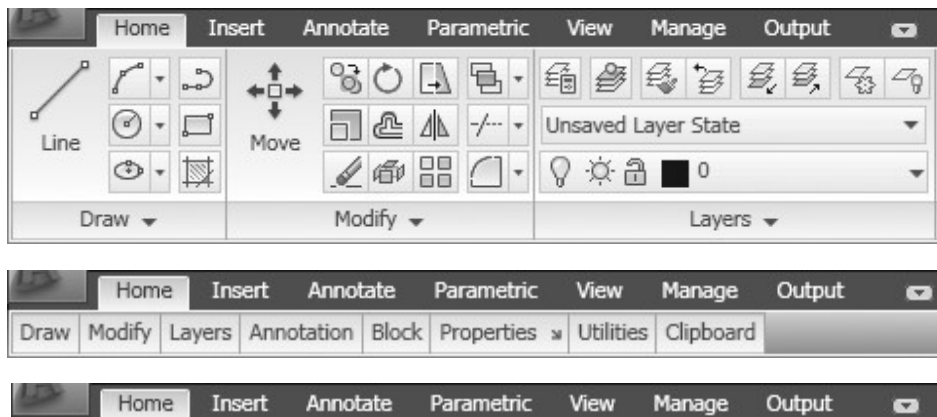




- The number of tabs and panels are predefined by default.
- You can turn off/on any tab/panel by right-clicking on any tab/panel and then selecting the desired tab/panel to be turned on or off.



- Ribbons have three different shapes. You can alter the shapes by clicking the small arrow at the right:

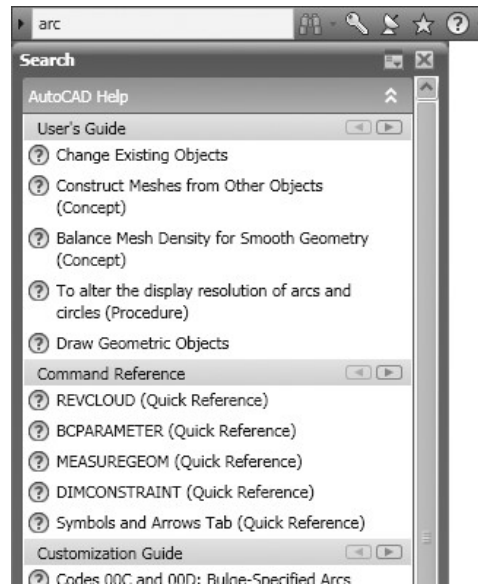


3.4.12 InfoCenter

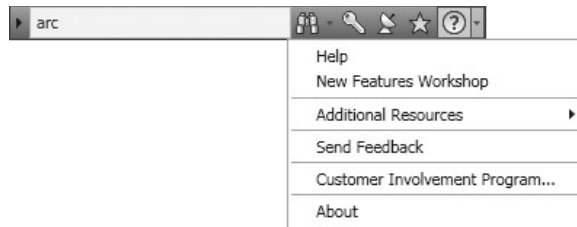
- At the top-right part of the screen, you will see the InfoCenter:



- The InfoCenter gives you the opportunity to type in keywords that will enable AutoCAD to search both online and offline resources and provide you with a list of related help topics. See the following example:



- You may still use the conventional Help search by clicking the arrow at the right-hand side of the InfoCenter:



3.4.13 Graphical Area

- The **Graphical Area** is where you do your actual drawing. You use the crosshairs to specify points in the X,Y plane.
- You can monitor the coordinates of the crosshairs using the left side of the **Status Bar**.

3.4.14 Status Bar

- The Status Bar in AutoCAD contains many functions that will help you draft more precisely. We will discuss most of the buttons on the Status Bar throughout this book.
- There are two views for the **Status Bar**:

- By default (Icons)



- Buttons



- To switch views, right-click the **Status Bar** to alternate between buttons and icons.

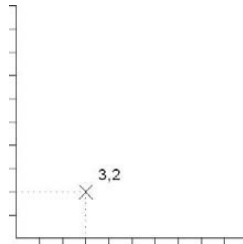


- If you select **Use Icons**, the view will change accordingly.

3.4.15 Points In AutoCAD

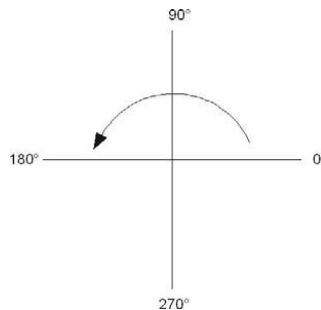
- Points are defined (and saved) in AutoCAD using the Cartesian coordinate system.

- The coordinates will look something like 3.25,5.45, which is the format of X,Y.
- So the first and most traditional way of specifying points in AutoCAD is to type the coordinates whenever you are asked to do so, by typing X,Y (pronounced X comma Y). See the following illustration:



3.4.16 AutoCAD Default Settings

- Sign convention: positive is up and right.
- Angle convention: positive is counterclockwise (CCW) starting from the east Sign convention: positive is up and right. (i.e., 0 angle). See the following illustration:



3.5 Things You should Know about AutoCAD

- The left mouse button is always used to select and/or click.
 - The mouse is the primary input device:
 - The right mouse button, when clicked, offers a drop-down menu.
- The mouse wheel has zooming functions:
 - Zoom in on your drawing by moving the wheel forward.
 - Zoom out of your drawing by moving the wheel backward.
 - Pan (i.e., move through the drawing) by pressing the wheel and holding it and then moving the mouse.
 - Zoom to the edges of your drawing by double-clicking the wheel.
- If you type an AutoCAD command or any input in the **Command Window**, you have to press the [Enter] key to execute it.
- [Enter] = [Spacebar] in AutoCAD.
- To cancel any AutoCAD command, press [Esc].
- To repeat the last AutoCAD command, press [Enter] or [Spacebar].

3.6 Drawing Area Setup

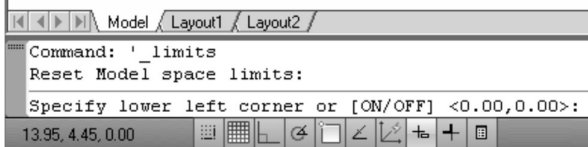
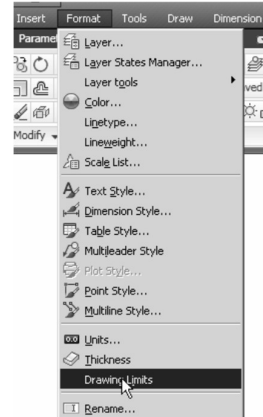
In AutoCAD, the drawings must be drawn full scale and therefore, the limits are needed to size up a drawing area. The limits of the drawing area are usually determined by the following factors.

- The actual size of the drawing.
- The space needed for putting down the dimensions, notes, bill of materials.
- The space for the borders and title block.

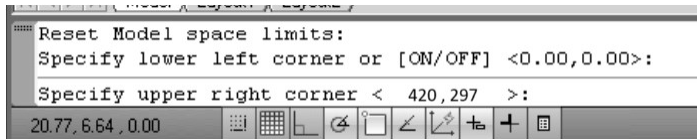
AutoCAD offers users an infinite drawing sheet on all sides.

The limits command allows the user to change the upper and lower limits of the drawing area. Turn limits, checking ON or OFF.

1. In the Menu Bar select:
[Format] → [Drawing Limits]
2. In the command prompt area, the message “Reset Model Space Limits: Specify lower left corner or [On/Off] <0.00,0.00>:” is displayed. Press the ENTER key once to accept the default coordinates <0.00,0.00>.



3. In the command prompt area, the message “Specify upper right corner <420,297>:” is displayed. Press the ENTER key again to accept the default coordinates <420,297>.



3.7 Designation of Sizes

The original drawing should be made on the smallest sheet, permitting the necessary clarity and resolution. The preferred sizes according to ISO-A series of the drawing sheets are given in Table below.

Designation	Dimension (mm)
A0	841 x 1189
A1	594 x 841
A2	420 x 594
A3	297 x 420
A4	210 x 297

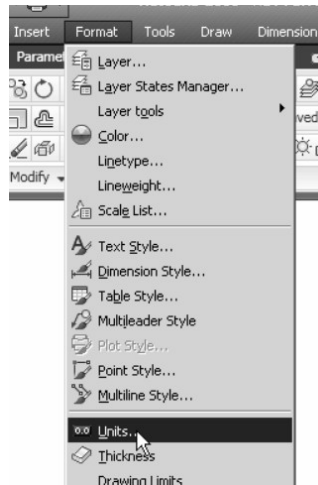
3.7.1 Drawing Units Setup

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

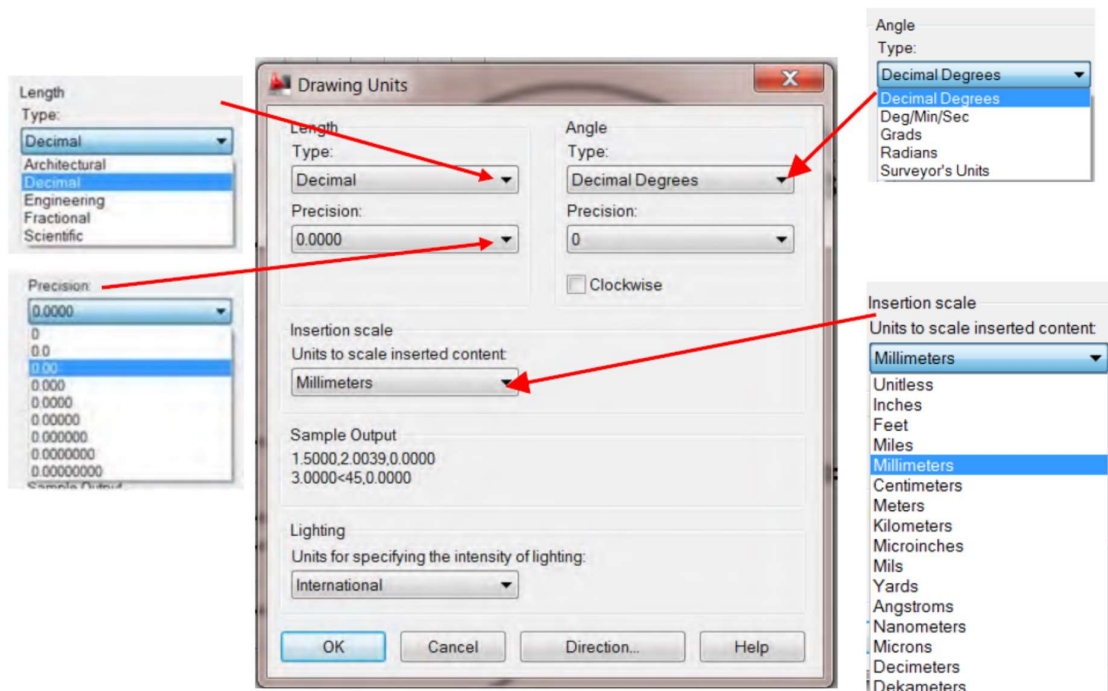
1. In the Menu Bar select:

[Format] → [Units]

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

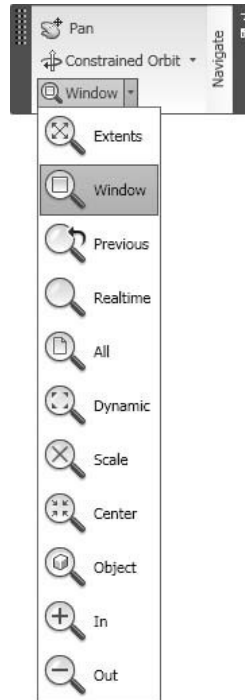


2. Click on the Length Type option to display the different types of length units available. Confirm the Length Type is set to Decimal.
3. In the Drawing Units dialog box, set the measurement to the SI (metric) or Imperial (Inches) units.
4. Set the Precision to appropriate digits after the decimal point as shown in the above figure.

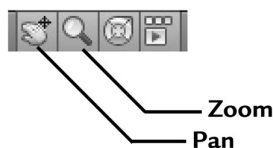


3.8 Viewing Commands

- We already discussed using the mouse wheel for zooming in, zooming out, and panning.
- You can also zoom in, zoom out, and pan using the zooming and panning commands.
- If using **Ribbons**, make sure you are in the **View** tab, and, using the **Navigate** panel, select the first button on the left, or the small arrow, to see a list of zooming commands:



- **Zoom Extents** is used to zoom to all objects.
- **Zoom Window** is used to specify a rectangle. By specifying two opposite corners, whatever is inside the rectangle will look larger.
- **Zoom Previous** is used to restore the previous view, up to the last ten views.
- **Zoom Realtime** is done by clicking the left button on the mouse and holding it. If you move forward, you are zooming in; if you move backward, you are zooming out.
- **Zoom Dynamic** is used with the Zoom Window first. You will see the whole drawing and your current place (shown as a dotted green line), go to the new location, and press [Enter].
- **Zoom Scale** is used to input a scale factor. If you type in a number less than 1, you will see the drawing smaller. If the scale factor you type in is greater than 1, you will see the drawing larger. If you put the letter x after the number (e.g., 2x) the scale will be relative to the current view.
- **Zoom Center** is used to specify a new center point for the zooming, along with a new height.
- **Zoom Object** is used to zoom to certain selected objects. AutoCAD will ask you to select objects and the selected objects will fill the screen.
- **Zoom In** is not really a zoom option, but rather a programmed option equal to the Zoom Scale with a scale factor of 2x.
- **Zoom Out** is just like Zoom In, but with a zoom factor of 0.5x.
- Also on the Status Bar, you can use the following two buttons:



4 Basic Draw Commands

4.1 Introduction

Starting with the AutoCAD sketching, with 2D systems recognize only flat shapes defined by points, lines, curves contained in two-dimensional plane. 2D systems are primitive by comparison with 3D system, but 2D system is sufficient for a wide variety application and at its lower cost, is an attractive choice for many companies. The majority of CAD orthographic engineering drawings and electrical circuit drawing are created on 2D system as shown in below Figure.



4.1.1 Typing Commands

All AutoCAD commands can be typed in at the command line. Many commands also have one or two letter aliases that can also be typed as shortcuts to the commands.

1. Type the desired command at the command prompt.

Command: **LINE**

or

2. Type the command's alias.

Command: **L**

3. Press **ENTER**.

4. Type an option at the command prompt.

TIP: Many AutoCAD commands require you to press ENTER to complete the command. You know you are no longer in an AutoCAD command when you see a blank command line.

4.1.2 Reissuing the Last Command

The last used AutoCAD command can be re-entered by one of the following three methods of ENTER. The ENTER key on the keyboard will always act as ENTER, the SPACEBAR and RIGHT MOUSE will act as enter most of the time.

1. Press the **ENTER** key on the keyboard

or

2. Press the **Space bar** on the keyboard.

or

3. Click the right mouse button.

4.1.3 Pointing Device (Mouse)

AutoCAD uses either a mouse or digitizing tablet to select objects in a drawing.

- **Left Mouse Button**

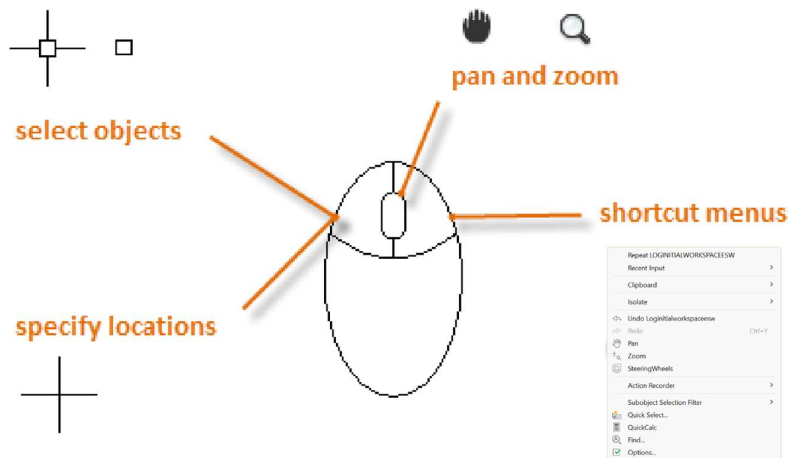
Used to pick or select objects

- **Click the left mouse button** to select an object area in the drawing.
- **Press ESC twice** to deselect an object (or to cancel a command).

- **Right Mouse Button**

Used to enter a **command**, **repeat last command**, or **access shortcut menus**.


- Click the right mouse button.



TIPS: SHIFT + the right mouse button brings up the object snap menus.


4.1.4 Undo and Redo

Reverses the last action.

1. Choose **Edit, Undo**.
or
2. Click the Undo icon. 
- or
3. Press **CTRL + Z**.
4. Type U at the command prompt to undo the last command.
Command: **U**

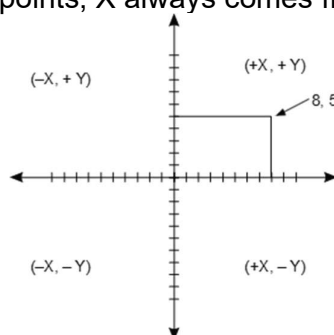
Redo

Reverses the effects of a single UNDO or U command.

1. Choose **Edit, Redo**.
or
2. Click the Redo icon. 
- or
3. Type REDO at the command prompt to redo the last undo command.
Command: **REDO**

4.1.5 Point

AutoCAD uses points to determine where an object is located. There is an origin where it begins counting from. This point is (0, 0). Every object is located in relation to the origin. If user were to draw a line straight out to the right from the origin, this would be considered the positive X-axis. If user were to draw a line straight up, this would be the positive Y-axis. The below figure shows a point located at (8, 5). This means that the point is 8 units over in the axis and 5 units up in the Y-axis. When user is working with points, X always comes first.

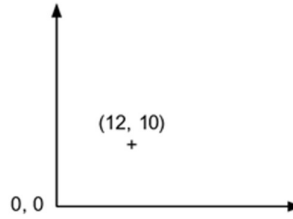


Ex: Plot a point at the location (12, 10).

Solution: See the below figure

Command: Point

point (12, 10)




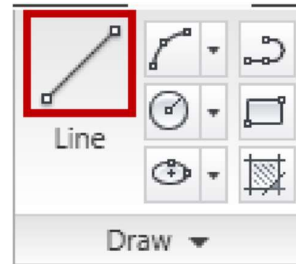
This places the given point in the drawing at location (12, 10).

4.2 Basic Commands

4.2.1 Line Command

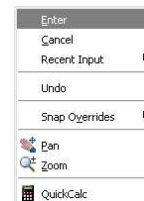
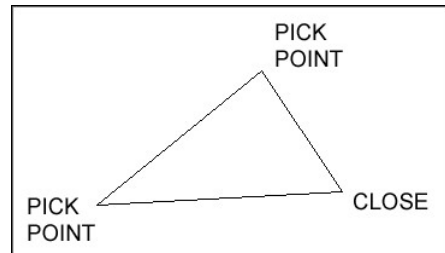
Creates single straight-line segments

1. Choose Draw, Line.
or
2. Click the Line icon. 
3. Type LINE from the command prompt
Command: **LINE** or **L**
4. Press **ENTER**
5. Pick From point: **(point)**
6. Pick Specify next point or [Close/Undo]:**(point)**
7. Pick Specify next point or [Close/Undo]:**(point)**
8. Press **ENTER** to end line sequence
or
9. Type U to undo the last segment
To point: **U** (undo)
or
10. Type C to create a closed polygon
To point : **C** (close)








TIPS:

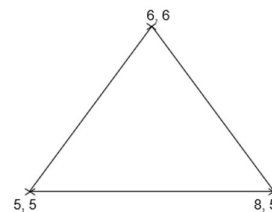
- You can continue the previous line or arc by responding to the From point: prompt with a space or ENTER.
- Choose the right mouse button for the line pop-up menu to appear while in the line command



Line can be drawn by anyone of the following three methods using LINE commands

- I. **Absolute co-ordinate system:** x,y coordinate when AutoCAD asks for a point

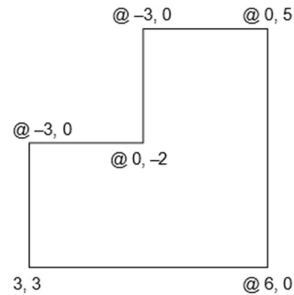
COMMAND: LINE 
 Specify first point: 5, 5 
 Specify next point: 8, 5 
 Specify next point: 6, 6 
 Specify next point: C 



NOTE: If dynamic input (F12) is on, you must type the # sign before entering absolute coordinates (e.g.#1,1).

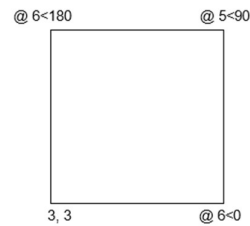
II. **Relative co-ordinate system:** @deltax,deltay when AutoCAD asks for a point.

COMMAND: LINE ↵
Specify first point: 3, 3 ↵
Specify next point: @ 6, 0 ↵
Specify next point: @ 0, 5 ↵
Specify next point: @ -3, 0 ↵
Specify next point: @ 0, -2 ↵
Specify next point: @ -3, 0 ↵
Specify next point: c ↵



III. **Polor co-ordinate system:** it uses a distance and an angle with reference to a previous point to locate a point. Angles are measured in anticlock direction, taking 0° towards right

COMMAND: LINE ↵
Specify first point: 3, 3 ↵
Specify next point: @ 6 < 0 ↵
Specify next point: @ 5 < 90 ↵
Specify next point: @ 6 < 180 ↵
Specify next point: c ↵



4.2.1.1 Dynamic Input

Dynamic Input provides a command interface near the cursor to help you keep your focus in the drafting area.

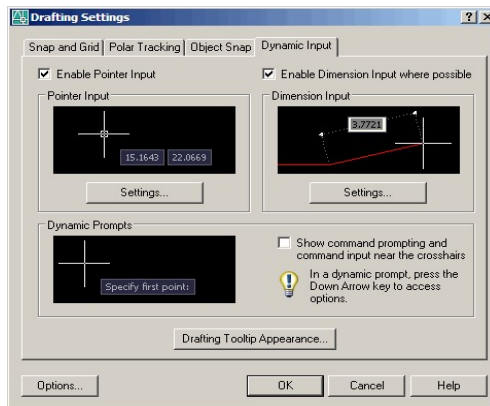
When Dynamic Input is on, tooltips display information near the cursor that is dynamically updated as the cursor moves. When a command is active, the tooltips provide a place for user entry.

Turning Dynamic Input ON/OFF

- 1. Click **Dyn** on the status bar
or
- 2. Press **F12**



TIP: Right-click Dyn and click Settings to control what is displayed by each component when Dynamic Input is on.



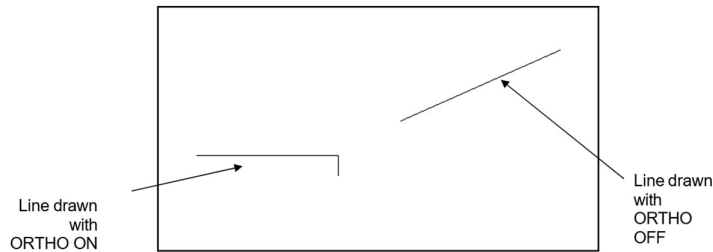
4.2.1.2 Orthogonal Lines

Controls lines from being drawn at various angles to straight lines. When the snap grid is rotated, ortho mode rotates accordingly.

- 1. Press Function Key **F8**.
or
- 2. Double Click **ORTHO** from the Status Bar.



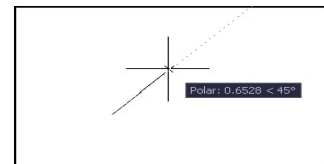
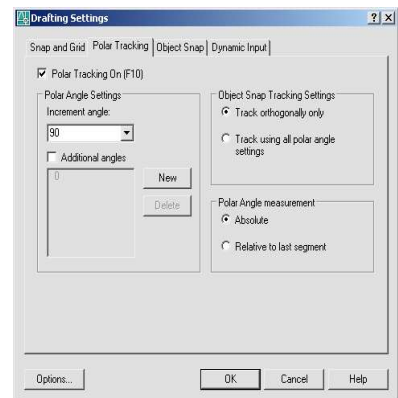
3. Press **CTRL + L.**



4.2.1.3 Polar Tracking

Polar Snaps work independently from snaps. With Polar Snaps on, AutoCAD shows the distances and angles being displayed as the cursor moves.

1. Choose **Tools, Drafting Settings** or **DDSETTINGS** at the command prompt.
3. Choose the Polar tracking TAB from the dialog box.
4. Select the desired incremental angle from the dropdown list (or create a new angle).
5. Pick **OK** to exit the dialog box.
6. Draw a **LINE** using the Polar Snap references.



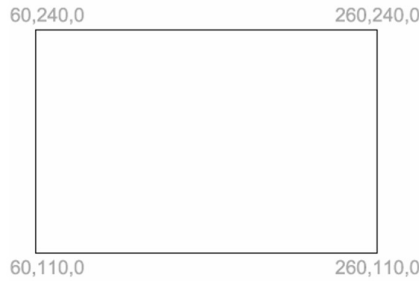
Ex (1) :- Line tool

1. Open AutoCAD. The drawing area will show the settings of the **acadiso.dwt** template Limits set to **420,297**, **Grid** set to 10, **Snap set** to 5
2. Left-click on the Line tool in the 2D Draw control panel



3. Make sure **Snap** is on by either pressing the **F9** key or the **SNAP** button in the **status bar**. **_Snap on_** will show in the command palette.
4. Move the mouse around the drawing area. The cursor's pick box will jump from point to point at 5 unit intervals. The position of the pick box will show as coordinate numbers in the status bar (left-hand end).
5. Move the mouse until the coordinate numbers show **60,240,0** and press the **Pick** button of the mouse (left-click).
6. Move the mouse until the coordinate numbers show **260,240,0** and left-click.
7. Move the mouse until the coordinate numbers show **260,110,0** and left-click.
8. Move the mouse until the coordinate numbers show **60,110,0** and left click.

- Move the mouse until the coordinate numbers show **60,240,0** and left click. Then press the Return button of the mouse (right-click).
The below figure appears in the drawing area.



Ex (2):- Line tool

- Close the drawing and open a new acadiso.dwt window.
- Left-click on the Line tool icon and enter figures as follows at each prompt of the command line sequence:

Command: _line Specify first point: 70,230

Specify next point: @220,0

Specify next point: @0, -70

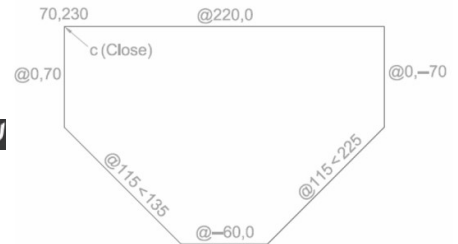
Specify next point or [Undo]: @115 <225

Specify next point or [Undo]: @ -60,0

Specify next point or [Close/Undo]: @115 < 135

Specify next point or [Close/Undo]: @0,70

Specify next point or [Close/Undo]: c (Close)



4.2.2 Circles

With this command user can draw a circle as required. User can also invoke the circle command from the draw, toolbar, from the DASHBOARD. Circle may be defined by a center point and either a radius or diameter; by two or three points on the diameter, arc and a radius value. Following are the various methods to draw circles:

- Using center and radius**

EX: Draw a circle with center (5, 5) and radius 16 units

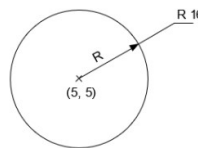
Solution:

COMMAND : CIRCLE

Select the circle tool from the Draw toolbar.

3P/2P/TTR/Centre point > : 5,5

Diameter/ <Radius> : 16



- Using center and diameter**

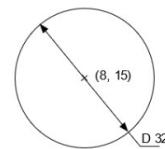
EX: Draw a circle with center (8, 15) and diameter 16 units

Solution:

COMMAND : CIRCLE

3P/2P/TTR/<centre point > : 8, 15

Diameter/ < Radius > : 16



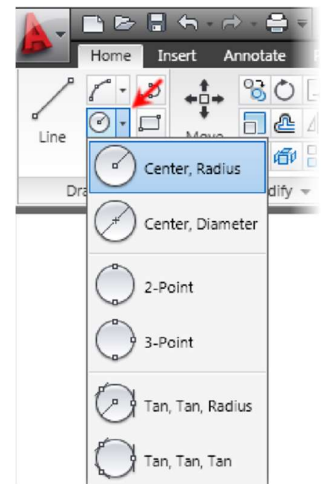
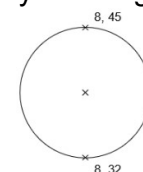
- 2 points (2p):** This allows to enter two diametrically opposite points of the circle.

EX: Draw a circle using the given 2 points (8, 32) and (8, 45) by entering 2nd points of the circle diameter.

Solution:

COMMAND : CIRCLE

3P/2P/TTR/< centre point> 16p

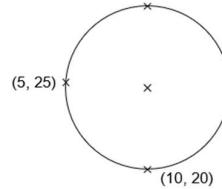


First point on diameter: 8, 32 ↵
Second point on diameter 8, 45 ↵

- **3 points (3p):** This allows to enter any three points on the circumference of the circle. EX: Draw a circle using the given 3 points (5, 20),

Solution:

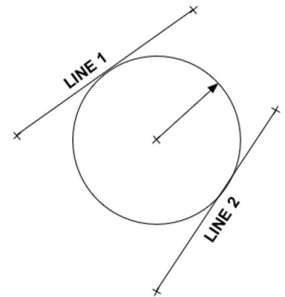
COMMAND : CIRCLE ↵
3p/2p/TTR/<centre point> : 3p ↵
First point: 5, 20 ↵
Second point: 5, 25 ↵
Third point: 10, 20 ↵



- **Tangent, Tangent and Radius (TTR):** This option allows to select two objects as tangent and fits a circle between them for a specified radius.

Problem 6: Draw a circle with radius 13 units and two existing lines as tangent.
Solution:

Take: For line 1, from point (17, 5) to point (20, 10)
For Line 2, from point (20, 2) to point (20, 6)
COMMAND : CIRCLE ↵
3p/2p/ TTR < centre point > : TTR ↵
Enter tangent specification : Select first object T1,
Enter second tangent specification: Select second object T2 ↵
Radius < current > : 13 ↵



TIPS:

- To create circles that are the same size, press ENTER when asked for the circle radius.
- When selecting a circle with a pickbox, be sure to select the circumference of the circle.

4.2.3 Arc Command

The Arc command creates an arc based on three points. Using the default method for creating an arc, you specify a start point, a second point, and an end point. The arc radius and center point are calculated based on the position of the three consecutive points you specify.

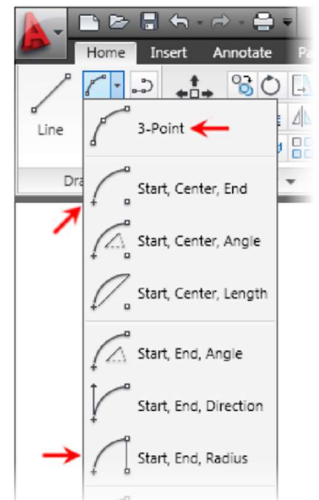
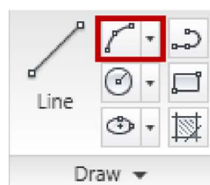
The following illustration represents an arc being created through three points.



- Command Line: ARC, A
- Menu Bar: Draw > Arc > 3 Points
- Ribbon: Home tab > Draw panel > Arc

Command Options

Arc options can be accessed from the drop-down menu next to the Arc button. The most common Arc options are indicated by red arrows in the following illustration.

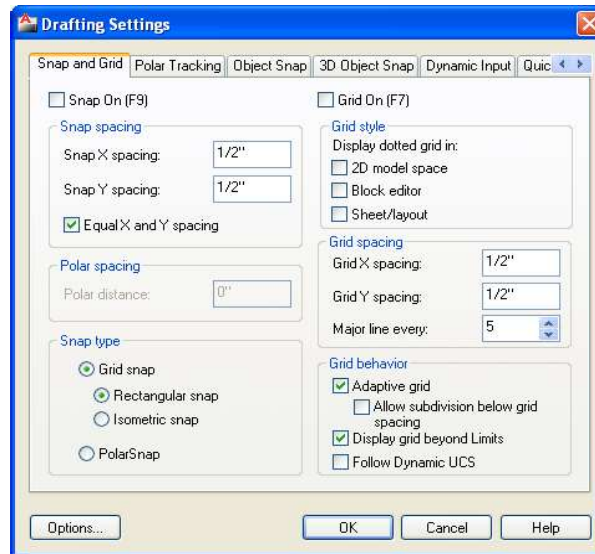


TIP: Except for 3 point arcs, arcs are drawn in a COUNTERCLOCKWISE direction.

5 Drawing Precision in AutoCAD

5.1 SNAP Command

- Choose Tools, Drafting Settings...
or
- Type SNAP at the command prompt.
Command: **SNAP** or **SN**
- Type One of the following options: Snap spacing
or [ON/OFF/Aspect/Style/Type]:



5.2 Turn Snap On/OFF

- Press Function Key F9 to turn the snap ON/OFF.
or
- Double Click SNAP on the Status Bar.
or
- Press CTRL + B.

TIP: Click with the right mouse button on the SNAP option from the status bar as a shortcut to changing the snap settings

5.3 Grid Command

- Choose Tools, Drafting Settings...
or
- Type DSETTINGS at the command prompt.
Command : **DSETTINGS (DS)**
or
- Type GRID at the command prompt.
Command: **GRID**
- Type One of the following options: Grid spacing(X) or
ON/OFF/Snap/Aspect <0000>:

5.4 Turn Grid On/Off

- Press Function Key F7 to turn the grid ON/OFF.
or
- Double Click GRID on the Status Bar.
or
- Press CTRL + G.

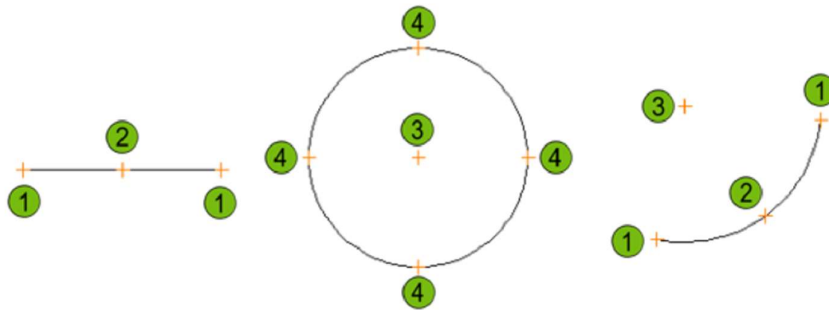
5.5 Object Snaps

Every object you create has various selectable points that you can use to position other objects. Every time you create an object you are required to specify a point or location. It is critical that these points be defined accurately if you expect your drawing to be accurate.

5.5.1 Object Snaps Defined

The following image shows three of the most basic types of objects that you can create: a line, a circle, and an arc. In this example, the line has two unique snap types with three possible locations that you can select; the circle has two unique snap types with five possible locations that you can select; the arc has three unique snap types with four possible locations that you can select.

1. Endpoint
2. Midpoint
3. Center point
4. Quadrant

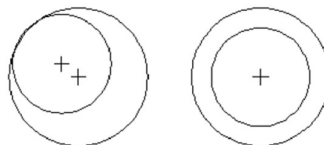


Object snaps exist for each of the previously mentioned points and more. You use them to select those points accurately. Attempting to accurately select these points without using object snaps would be very time-consuming and inaccurate.

5.5.2 Effect of Using Object Snaps

In the following image, the smaller circle needs to be concentric within the larger circle. By using the Center object snap, you can place the smaller circle precisely in the center of the larger circle.

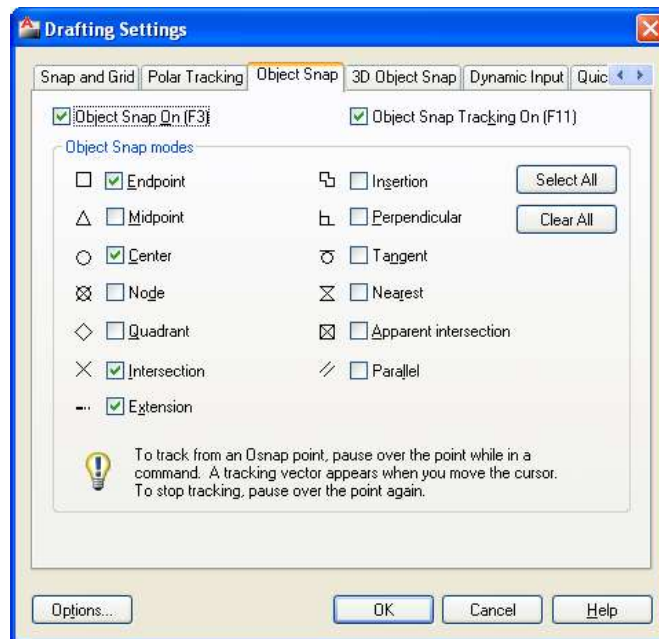
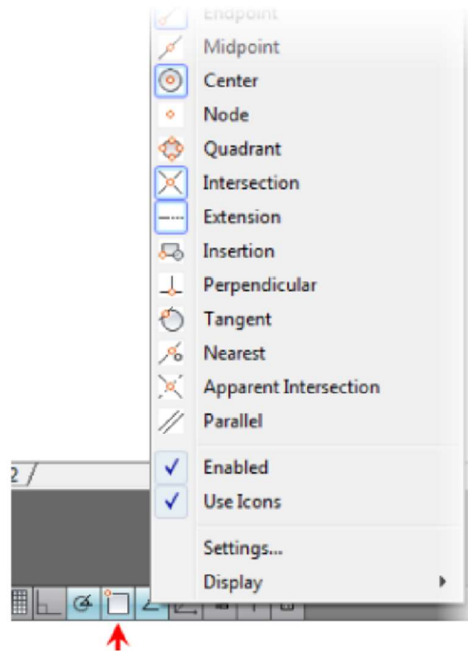
By examining each circle's coordinates, you can verify that they both share the same center point.



5.5.3 Running Object Snap


- Command Line: OSNAP
- Menu Bar: Tools > Drafting Settings > Object Snap Tab
- Keyboard Shortcut: F3
- Toolbar: Object Snap
- Status Bar: Object Snap

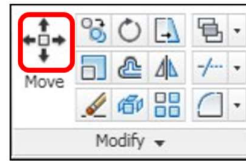




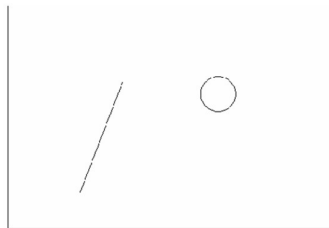
6 Modify commands

6.1 Move Command

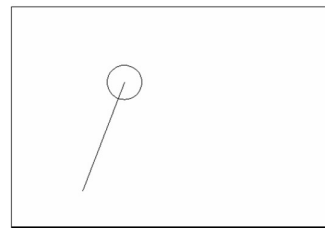
- Choose Modify, Move.
- Click the Move icon. 
- Type MOVE at the command prompt
Command: MOVE or M
- Pick Objects to move
Select objects: (select)
- Pick A point to move from
Base point or displacement: (pick point)
- Pick A point to move to
Second point of displacement: (pick point)



Circle before move




Circle after move



TIP: To move an object a specified distance, type a distance at the second point of displacement prompt: @1<0

6.2 Copy Command

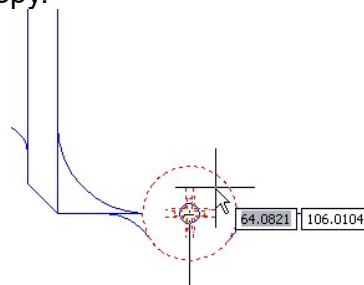
- Ribbon: Draw tab > Modify panel > Copy
- Command: COPY, CO
- Menu Bar: Modify > Copy
- Shortcut Menu: Select objects then right-click anywhere in the drawing window and select Copy.
- Click the Copy icon. 
- Pick Objects to copy.
Select objects: (select)
- Pick A point to move from.
Base point or displacement/Multiple: (pick point).
- Pick A point to copy to.
Second point of displacement: (pick point) or
- Type A point to copy to.
Second point of displacement: @ 1<0



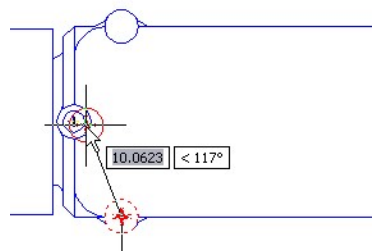
Procedure: Copying Objects with the Copy Command

The following steps describe an overview for copying objects in the drawing.

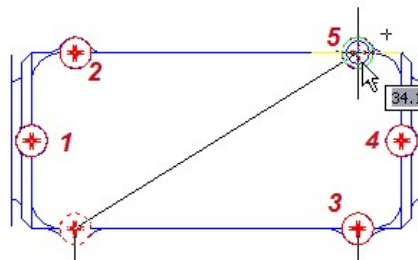
1. On the ribbon, click Home tab > Modify panel > Copy.
2. Select the object(s) to copy and press ENTER.
3. Specify a base point for the copy.



4. Specify a second point or displacement.



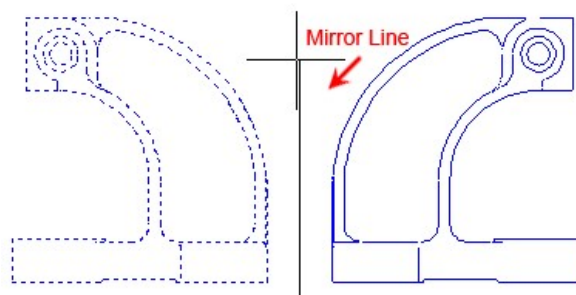
5. Continue to specify second points to create additional copies.



6.3 Mirror Command

Using the Mirror command, you can create symmetric versions of geometry. When you activate the Mirror command, you are prompted to select the objects to mirror and to define a mirror line. The mirror line defines a vector across which all selected geometry is mirrored. After defining the mirror line, you can decide whether to erase or retain the source geometry. You can also control the Mirror command using grips.

Mirror works for all object types.



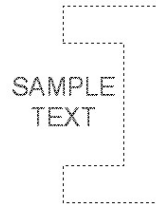
- Command Line: MIRROR, MI
- Ribbon: Home tab > Modify panel > Mirror
- Menu Bar: Modify > Mirror



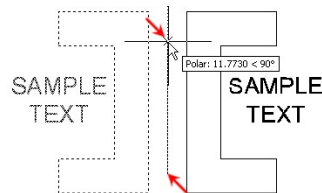
Procedure: Mirroring Objects

The following is an overview of mirroring objects in the drawing with the MIRRTEXT system variable set to 0.

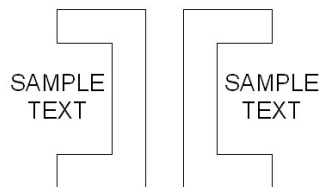
1. Activate the Mirror command.
2. Select the objects to be mirrored. Press ENTER.



3. Select two points to define the mirror line.



4. Enter Y to erase source objects or N to keep the source objects. Notice that the orientation of the text remains the same.



6.4 Rotate Command

You use the Rotate command to rotate objects in the drawing. Command options enable you to rotate the objects simply by entering an angle for the rotation or by defining a reference angle and then entering a new angle for the reference angle. If you need to make a copy of the object while rotating, you can specify the Copy option. When the Copy option is selected, a copy of the object is rotated, leaving the original object in its current position. You can also use grips to rotate the object.

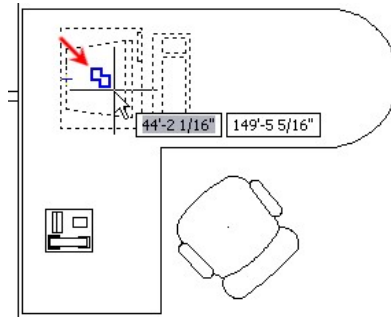
- Command Line: ROTATE, RO
 Ribbon: Home tab > Modify panel > Rotate
 Menu Bar: Modify > Rotate
 Shortcut Menu: Select objects then right-click anywhere in the drawing window. Click Rotate.



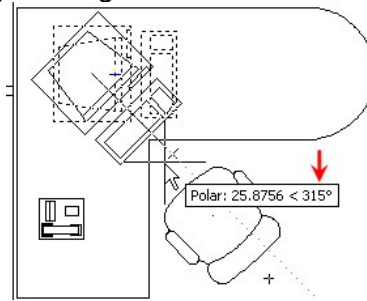
Procedure: Rotating Objects

The following steps provide an overview of rotating objects using the Rotate command.

1. On the ribbon, click Home tab > Modify Panel > Rotate.
2. Select the object(s) to rotate and press ENTER.
3. Specify a base point for the rotation. The object will rotate around this point.



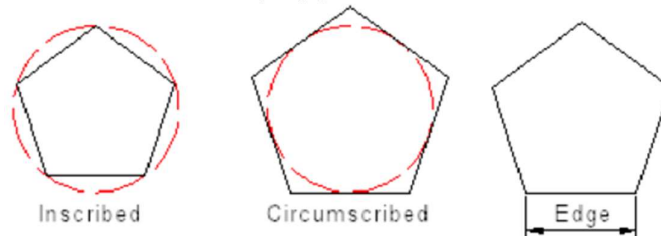
4. Specify a rotation angle by moving the cursor or entering an angle.



7 Advanced Draw Commands

7.1 Polygon Command

Use the Polygon command to create regular polygon geometry by specifying the center point and radius of an imaginary circle, or the start point and endpoint of one of the polygon edges. Regardless of the method you choose to define the polygon, all of its sides are equal in length. The default method for creating polygons is to specify a center point and radius. When you choose this method, you must choose either the Inscribed or Circumscribed option. Depending on the option you choose, the size of the polygon is calculated as shown in the following image.



Command: POLYGON, POL

Menu Bar: Draw > Polygon

Ribbon: Home tab > extended Draw panel > Polygon

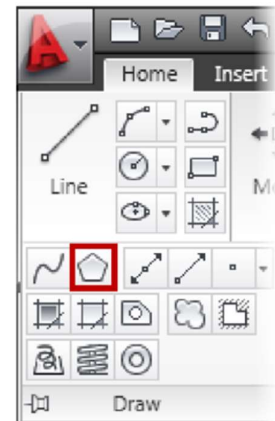
Type The number of sides for the polygon (3-1024)

Number of sides <default>: **number**

Pick The center of the polygon. Edge/<Center of polygon>: **pick**
or

Type **E** to define the polygon by two edges.

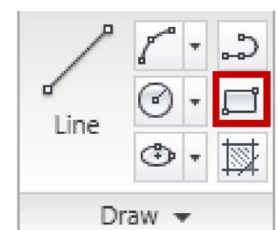
Type **I** or **C** to place the polygon inside or outside of an imaginary circle. Inscribed in circle/Circumscribed about circle (I/C):



7.2 Rectangle Command

Use the Rectangle command to create rectangular objects. A single polyline object is created with this command. The simplest method for creating a rectangle is to specify the first corner, then the opposite corner. Other options for creating the rectangle include the Area, Dimension, and Rotation options.

This illustration shows a rectangle with the point used to create it specified.



Command: RECTANGLE, REC

Menu Bar: Draw > Rectangle

Ribbon: Home tab > Draw panel > Rectangle

Type: Rectang at the command prompt

Command: RECTANG Chamfer/Elevation/Fillet/ Thickness/Width/<First corner>:

Pick: first corner.

Pick: other corner or type coordinates (i.e. @4,2).

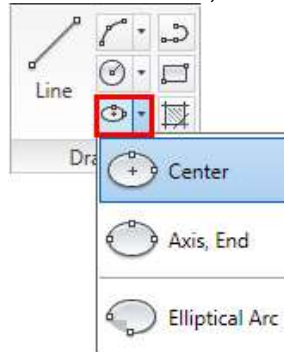
7.3 ELLIPSE

There are three options associated with the Ellipse tool. These three options allow the user to define the Ellipse.

- Ellipse is defined as the full length of one axis and half length of the other axis.
- Define as the center point of ellipse and half length of other two axis.

Command: ELLIPSE

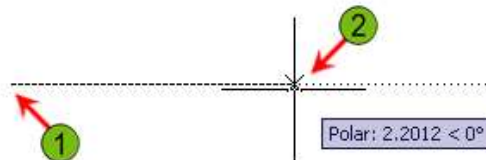
Ribbon: Home tab > Draw panel > Center/Axis, End/Elliptical Arc



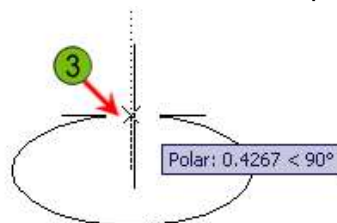
Procedure: Creating an Ellipse

The following steps give an overview of creating an ellipse by defining the axis endpoints.

1. On the ribbon, click Home tab > Draw panel > Ellipse.
2. Click to specify the first axis endpoint (1) and the other axis endpoint (2).



3. Click to specify the distance to the other axis endpoint (3).



The completed ellipse.

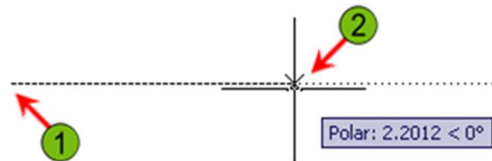


Note: To use the other Ellipse options, follow the command line prompts. Right-click to specify the options from the shortcut menu or enter the capitalized letter on the command line.

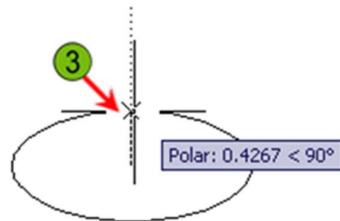
Procedure: Creating an Elliptical Arc

The following steps give an overview of creating an elliptical arc. The first few steps are identical to creating an ellipse.

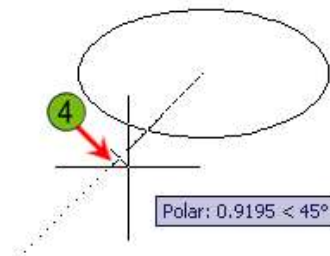
1. On the ribbon, click Home tab > Draw panel > Ellipse Arc.
2. Click to specify the first axis endpoint (1) and the other axis endpoint (2).



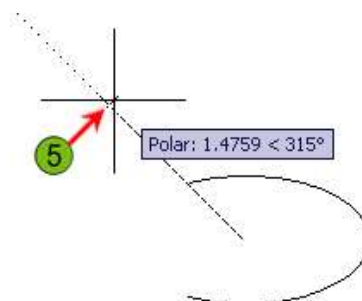
3. Click to specify the distance to the other axis endpoint (3).



4. Move the cursor to specify the start angle position (4).




5. Move the cursor to specify the end angle (5). Notice that the polar angle is set to 45 degrees.

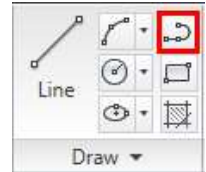


Note: The order in which the start and end angle are specified will determine which part of the ellipse is removed.

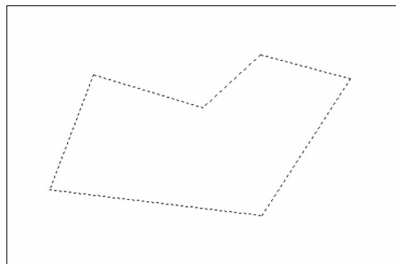
7.4 Pline Command

A polyline is a connected sequence of line segments created as a single object. You can create straight line segments, arc segments, or a combination of the two.

- Command: **PLINE, PL**
- Ribbon: **Home tab > Draw panel > Polyline**
- Pick the Pline icon. 
- Type **PLINE** at the command prompt Command : **PLINE** or **PL**
- Pick A point on the drawing to start the polyline Frompoint:(select)
- Type One of the following options Arc/Close/Halfwidth/Length/Undo/Width or
- Pick A point to continue drawing Arc/Close/Halfwidth/Length/Undo/Width/ <endpoint of line>: (pickpoint)




Polyline as one segment

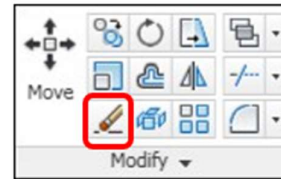


8 Advanced Modify commands

8.1 Erase Command

Deletes objects from a drawing.

- Choose Modify, Erase.
or
- Click the Erase icon. 
- Type ERASE at the command prompt.
Command: ERASE or E
- Pick Object at the select object prompt.
Select objects: (pick object)
- Press ENTER when you are done choosing objects.
Select objects: ENTER



8.2 TRIM

The TRIM command allows you to trim objects in a drawing so they end precisely at a cutting edge defined by one or more other objects in the drawing.

- Command: TRIM, TR
or
- Ribbon: Home tab > Modify panel > Trim
or
- Menu Bar: Modify > Trim
- Click the Trim icon
- Type TRIM at the command prompt
Command: TRIM
Select cutting edge(s)...
- Pick The CUTTING edge to extend to
Select objects: (select)
- Press ENTER to accept the cutting edge
Select objects: (press enter)
- Pick Objects to trim
<Select object to trim> / Project / Edge / Undo: Select an object,
enter an option, or press enter
- Press ENTER when you are done choosing objects
Select object to trim/Undo: (press enter)



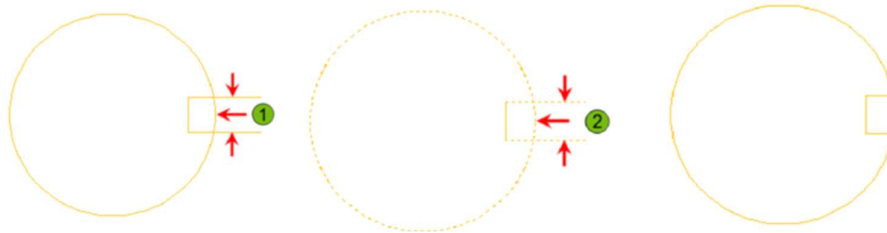
TIP: Hold the SHIFT key to interactively extend instead of trim.

Procedure: Trimming Objects

The following steps give an overview of using the Trim command to shorten objects to cutting edges.

1. On the ribbon, click Home tab > Modify panel > Trim.
2. Either select the objects to serve as cutting edges (1) and then press ENTER, or press ENTER without selecting any objects. Pressing ENTER without selecting activates implied selection, where all suitable objects in the drawing are treated as potential cutting edges.
3. Select the objects to trim (2).

4. Press ENTER to end the command and view your completed operation.



8.3 EXTEND

- Command: EXTEND, EX
or
- Ribbon: Home tab > Modify panel > Extend
or
- Menu Bar: Modify > Extend
- Click the Extend icon.
- Pick The BOUNDARY edge to extend to
Select objects: (select)
- Press ENTER to accept the boundary edge
Select objects: (press enter)
- Pick The objects to extend
<Select object to extend> / Project / Edge / Undo: Select an object, enter an option, or press enter : (select)
- Press ENTER when you are done choosing objects



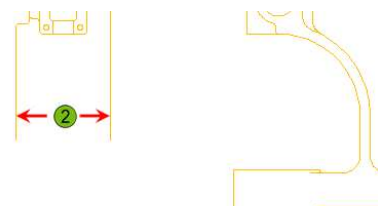
Procedure: Extending Objects

The following steps give an overview of using the Extend command to lengthen objects to boundary edges.

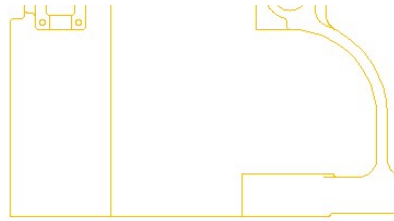
1. On the ribbon, click Home tab > Modify panel > Extend.
2. Either select the objects to serve as boundary edges (1) and then press ENTER, or press ENTER without selecting any objects. Pressing ENTER without selecting, activates implied selection, where all suitable objects in the drawing are treated as potential boundary edges.



3. Select the objects to extend (2).

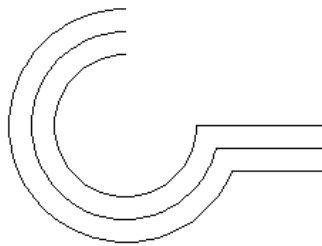



4. Press ENTER to end the command and view your completed operation.



8.4 Offset Command

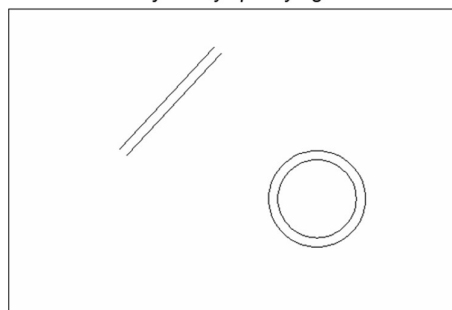
The Offset command creates a new object whose shape parallels the shape of a selected object. For example, in the following image, the inside shape has been offset twice using the Offset command with the Multiple option.




- Command: OFFSET, O
or
- Ribbon: Home tab > Modify panel > Offset
or
- Application Menu: Modify > Offset
- Choose the Offset icon. 
- Type the distance to offset.
Offset distance or <Through point>: (number)
- Pick The object to offset.
Select object to offset: (select object)
- Pick A side to offset object to. Side to offset: (pick side)
- Pick Another object to offset
Select object to offset: (pick side)
or
- Press Enter to end the command.



Offset objects by specifying a distance

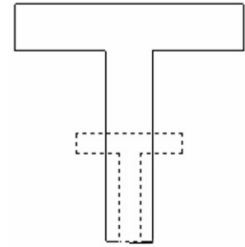


8.5 SCALE

- Choose Modify, Scale.
Or
- Click the Scale icon. 
or



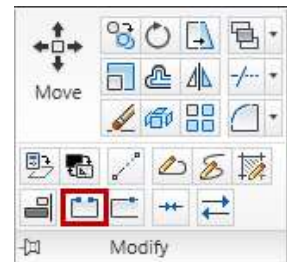
- Type SCALE at the command prompt
Command: SCALE
Select objects: (select objects)
- Pick A pivot point to scale about Base point: (point)
- Type A rotation angle<Scale factor>/Reference:(number)
or
- Pick A scale factor<Scale factor>/Reference:(point)
Scale factor/Reference: (points)



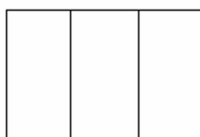
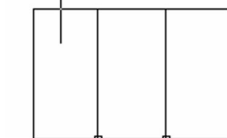
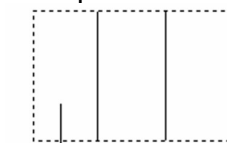
8.6 Break Command

Using the Break command, you can break a single object into two independent objects. You accomplish this by first selecting the object to break and then specifying the break points.

- Command: BREAK, BR
or
- Ribbon: Home tab > Modify panel > Break
or
- Menu Bar: Modify > Break
- Pick Object to break.
Select object: (select one object)
- Pick A second break point. Enter second point: (point)



- Type F to choose a different break point
Enter second point (or F for first point)
- Pick The first break point on the object
Enter first point: (point)
- Pick A second break point



TIPS:

- You can also type coordinates instead of picking a break point. Enter second point (or F for first point) @3'<0

- If you break a circle, it changes to an arc by deleting the portion from the first point to the second, going counterclockwise.
- Breaking a Polyline with nonzero width will cause the ends to be cut square.

8.7 Stretch Command

You use the Stretch command to change the shape of objects in the drawing

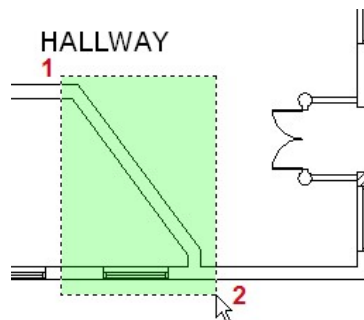
- Command: STRETCH, S
 or
- Ribbon: Home tab > Modify panel > Stretch
 or
- Menu Bar: Modify > Stretch



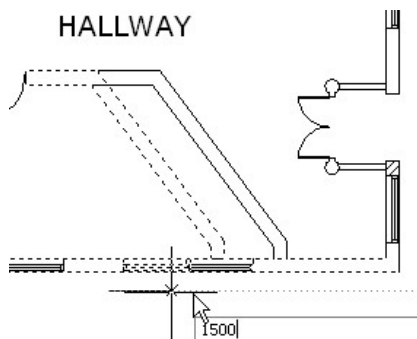
Procedure: Stretching Objects

The following steps give an overview of stretching objects in a drawing.


1. Start the Stretch command.
2. Select the objects to be stretched by defining a crossing window or crossing polygon selection.



3. Select the base point for the stretch.
4. Select the second point or enter a value for the stretch distance.



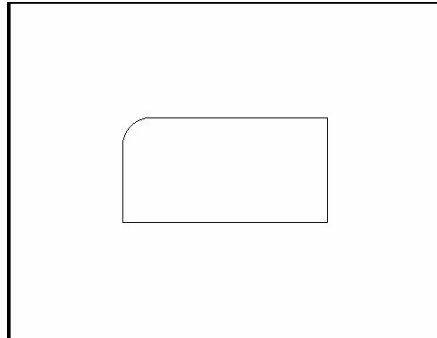
8.8 Fillet

- Choose Modify, Fillet.
 or
- Click the Fillet icon. 
- Type FILLET at the command prompt.



Command: FILLET


- Pick First object to fillet. Polyline/Radius/Trim<Select two objects>: select first object.
- Pick Second object to fillet.
Select second object:select second object. or
- Type One of the following options:
P Fillets an entire Polyline.
R Sets the fillet radius.
T Sets the trimmode (trim cuts the fillet corner and no trim keeps the fillet corner)

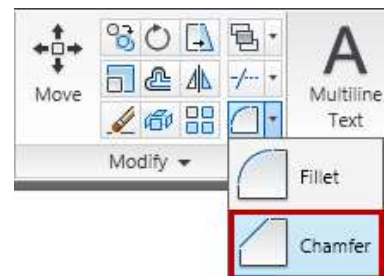


TIPS:

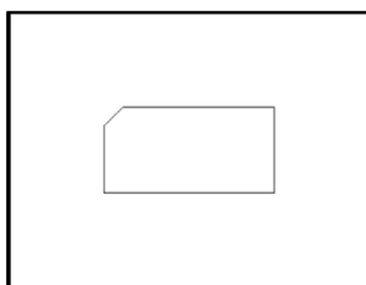
- You can also fillet PARALLEL lines as well as PLINES with LINES
- Type a radius of zero (0) to create a clean 90 degree corner

8.9 Chamfer

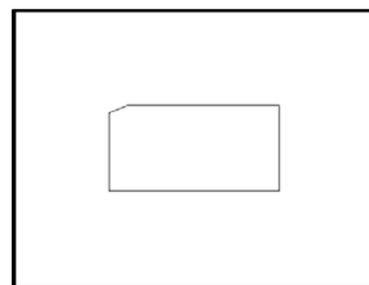
- Choose Modify, Chamfer.
or
- Click the Chamfer icon. 
- Type CHAMFER at the command prompt.
Command: CHAMFER
- Pick First object to chamfer. Polyline/Distance/Angle/Trim/Method<Select first line>: select first object
- Pick Second object to chamfer.
Select second object:select second object. or
- Type One of the following options:
P Chamfers entire Polyline.
D Sets chamfer distances.
A Uses a distance and angle method instead of two distances.
T Sets the trim mode
M Sets the method to distance or angle.



Chamfer with equal distances




Chamfer with different distances

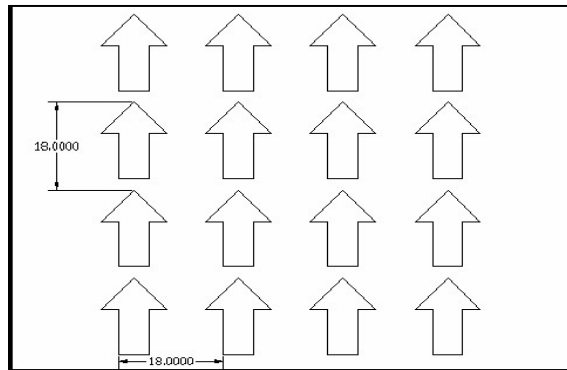


8.10 Array

8.10.1 Rectangular Array


To draw rectangular array:

- Choose Modify, Array.
or
- Click the Array icon. 
or
- Type ARRAY at the command prompt.
Command : ARRAY
- Pick Objectstoarray. Select objects : (select)
- Type R for a rectangular array.
Enter array type [Rectangular/PAth/POlar] <Rectangular>: R
Type = Rectangular Associative = Yes Select grip to edit array or
[ASsociative/Base
point/COunt/Spacing/COLumns/Rows/Levels/eXit]<eXit>:
S to change the spacing distance between each column and/or row.
• Type COL to change the number of columns.
• Type R to change the number of rows.

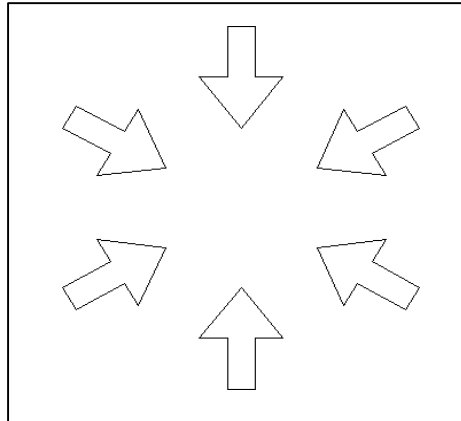


8.10.2 Polar Array

To draw a polar array:

- Choose Modify, ARRAY.
or
- Click the Array icon. 
or
- Type ARRAY at the command prompt.
Command: ARRAY
- Pick Objects to array.
Select objects:(select)
- Type P to draw a polar array. Enter array type
[Rectangular/PAth/POlar] <Rectangular>: PO (R/P): PO
- Pick A center point for the array. Center point of array:pick point
- Type one of the following options or press ENTER

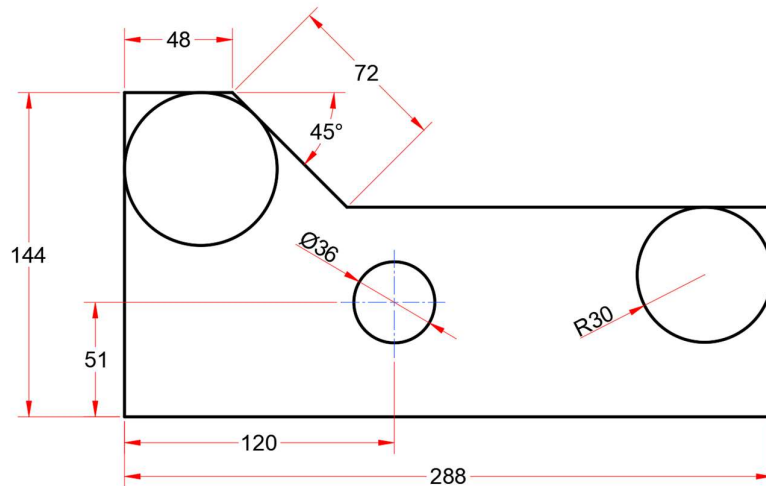
Select grip to edit array or [ASsociative/Base point/Items/Angle
between/Fill angle/ROWS/Levels/ROTate items/eXit]<eXit>:



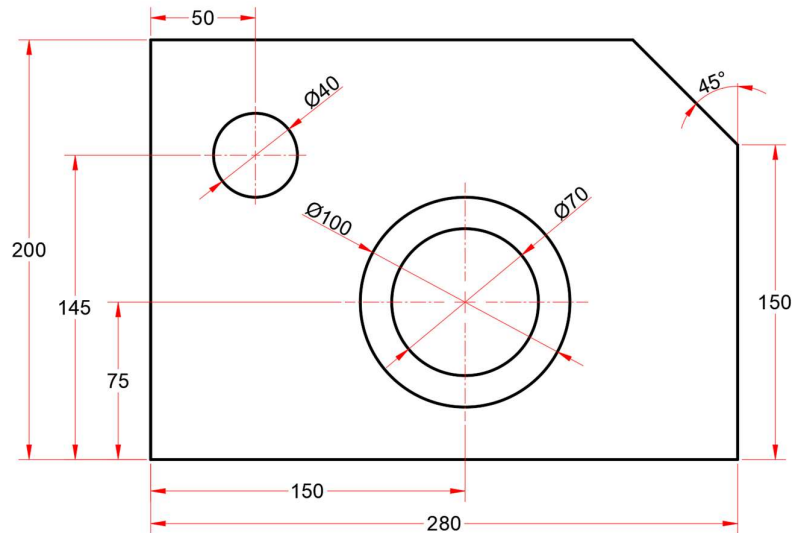
9 Creating Complex Objects

The following exercises demonstrate how to draw complex drawings by using the learned AutoCAD commands

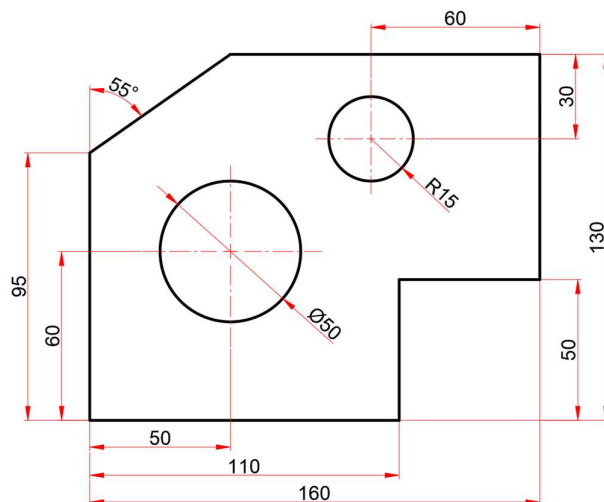
Exercise (1)



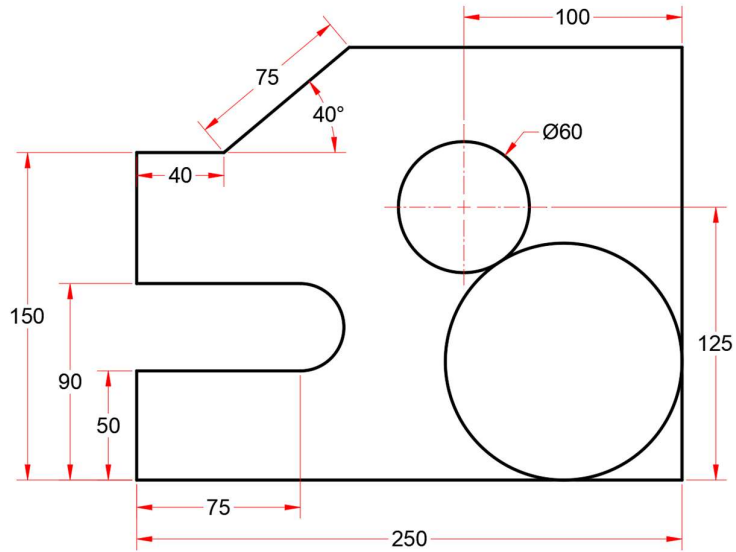
Exercise (2)



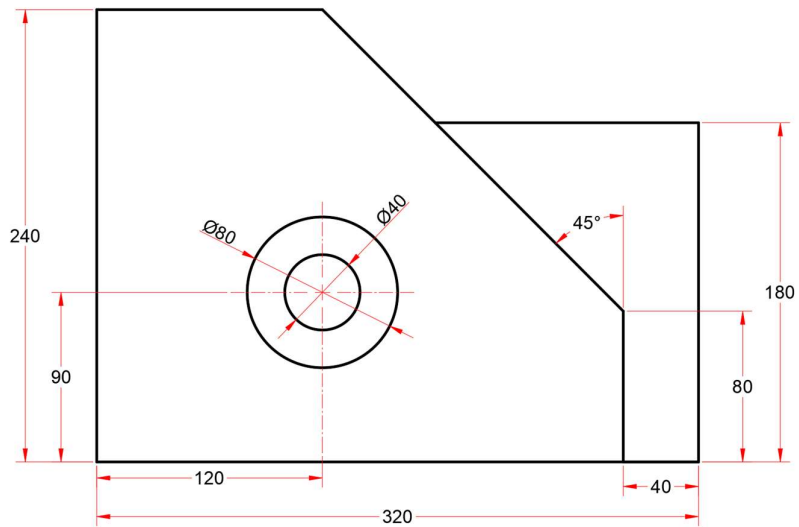
Exercise (3)



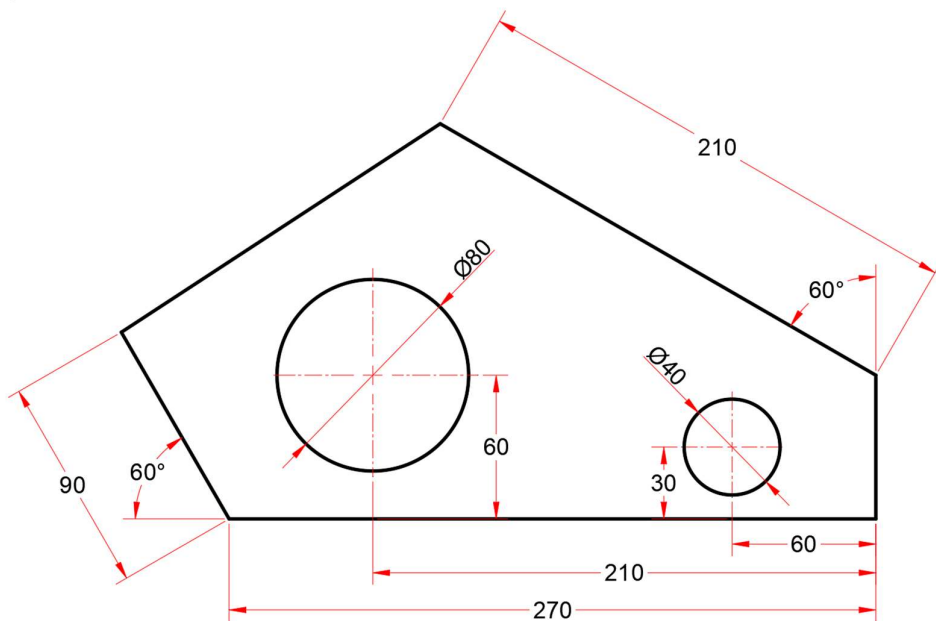
Exercise (4)



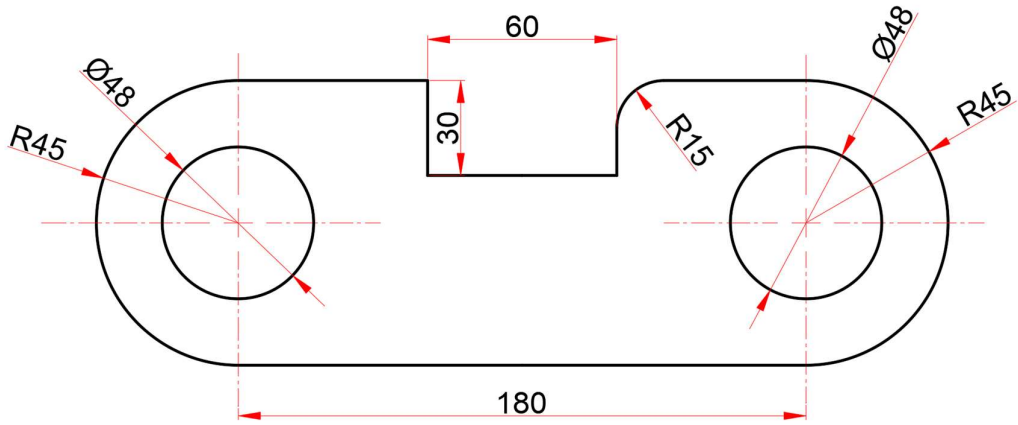
Exercise (5)



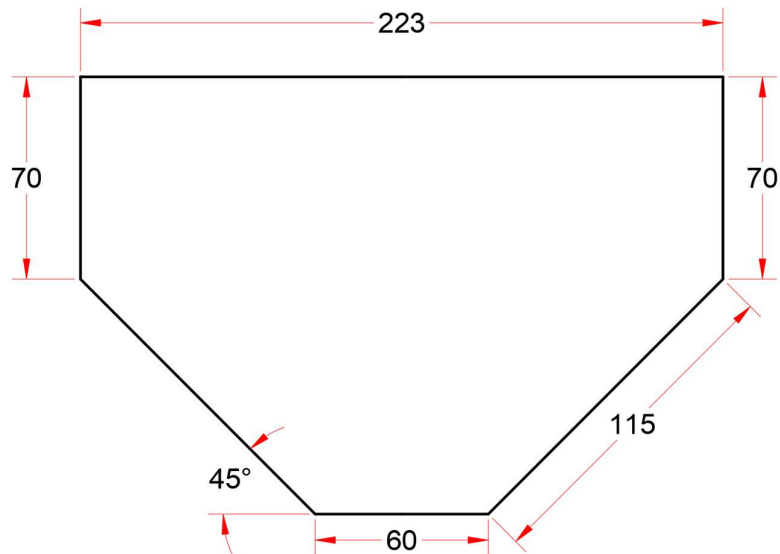
Exercise (6)



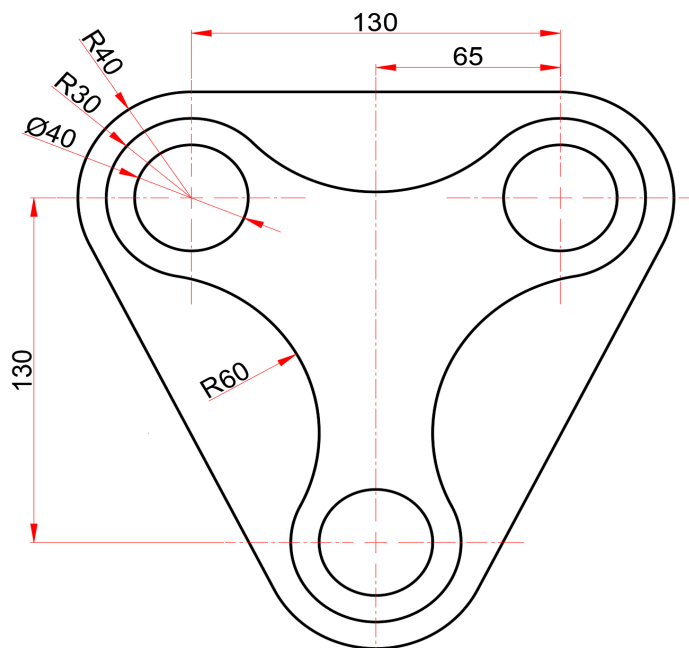
Exercise (7)



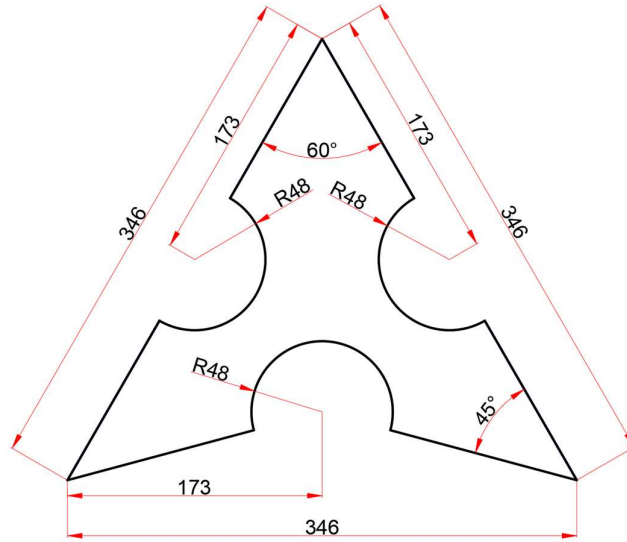
Exercise (8)



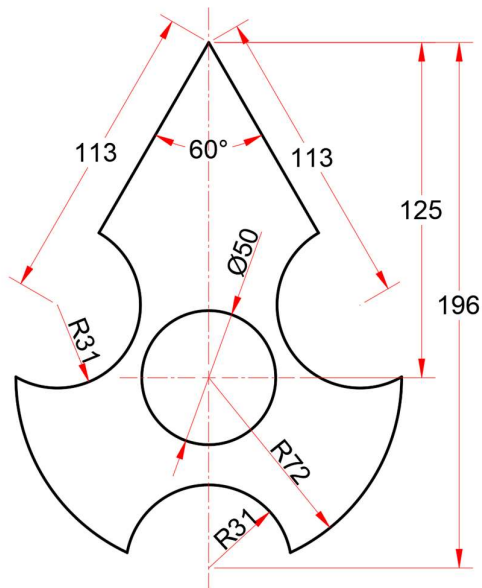
Exercise (9)



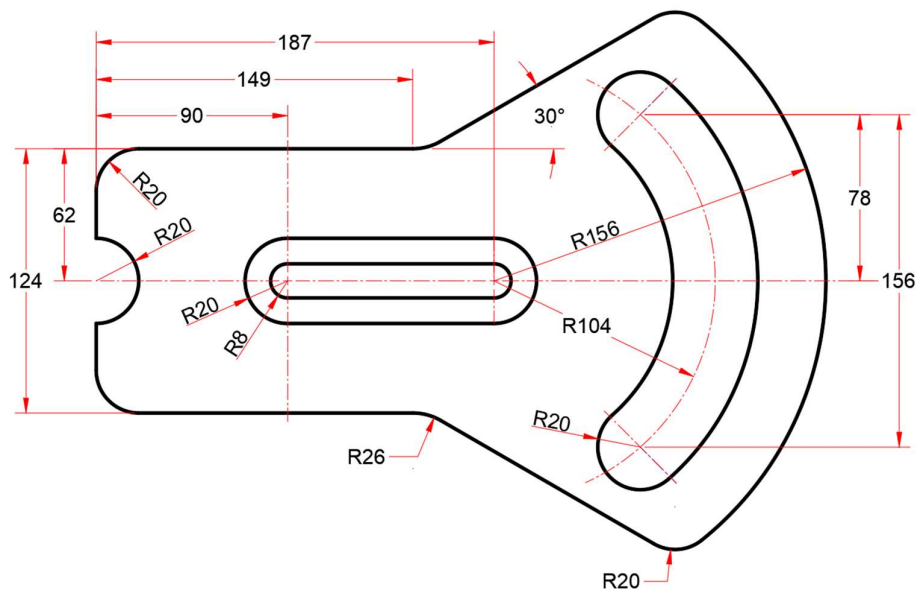
Exercise (10)



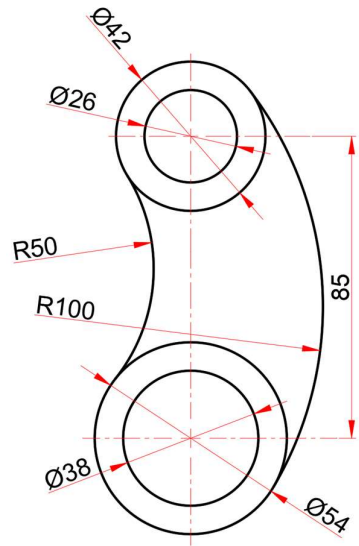
Exercise (11)



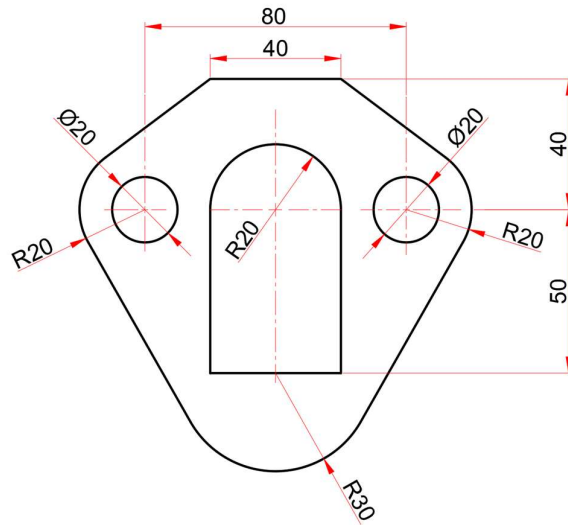
Exercise (12)



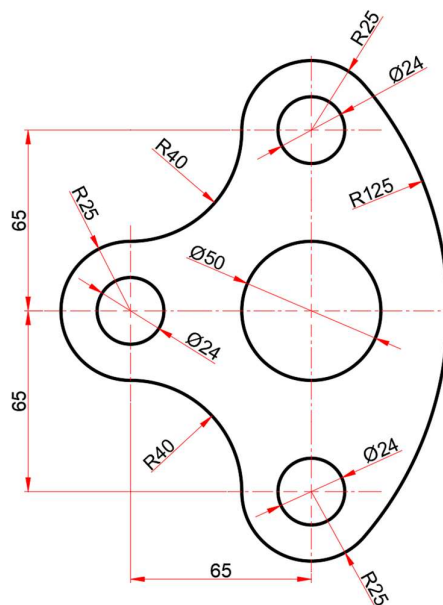
Exercise (13)



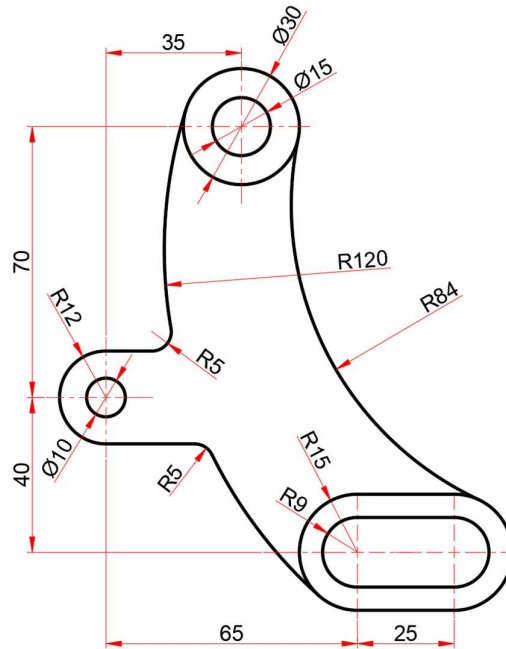
Exercise (14)



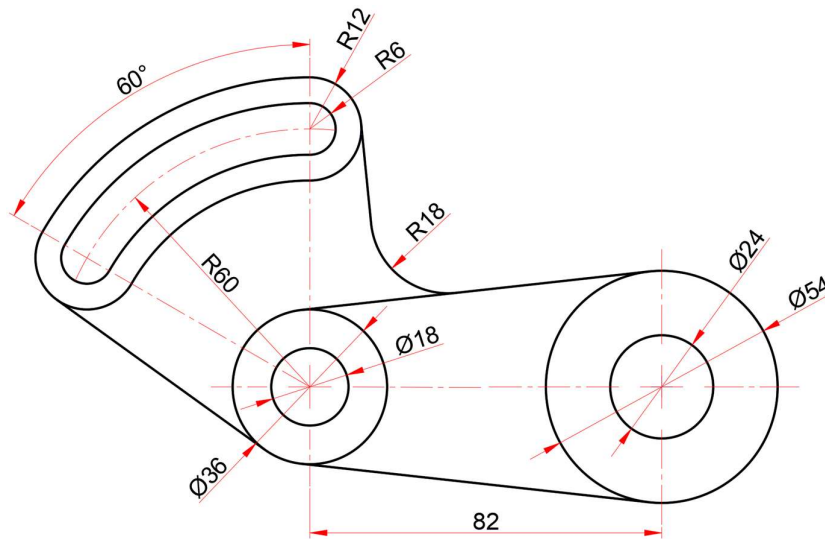
Exercise (15)



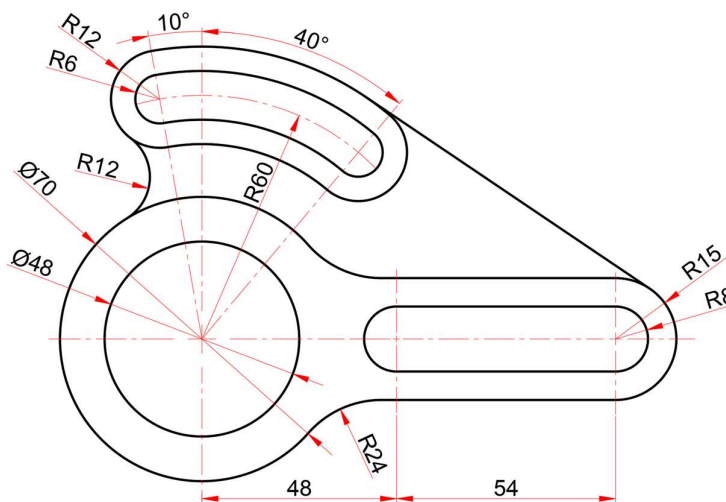
Exercise (16)



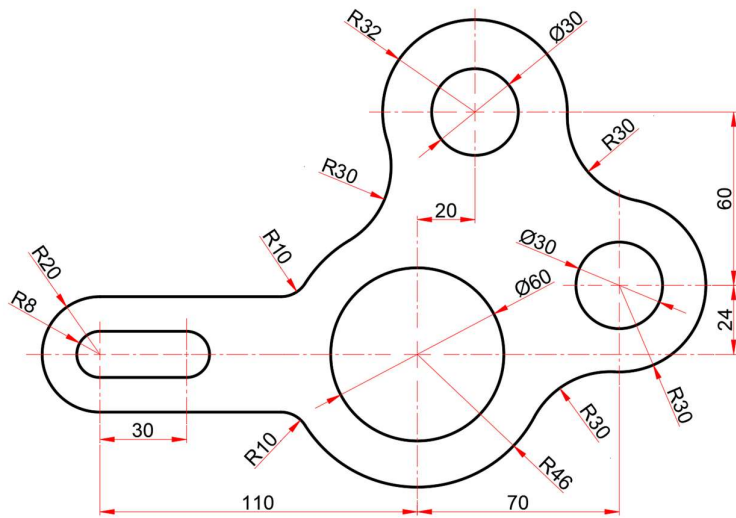
Exercise (17)



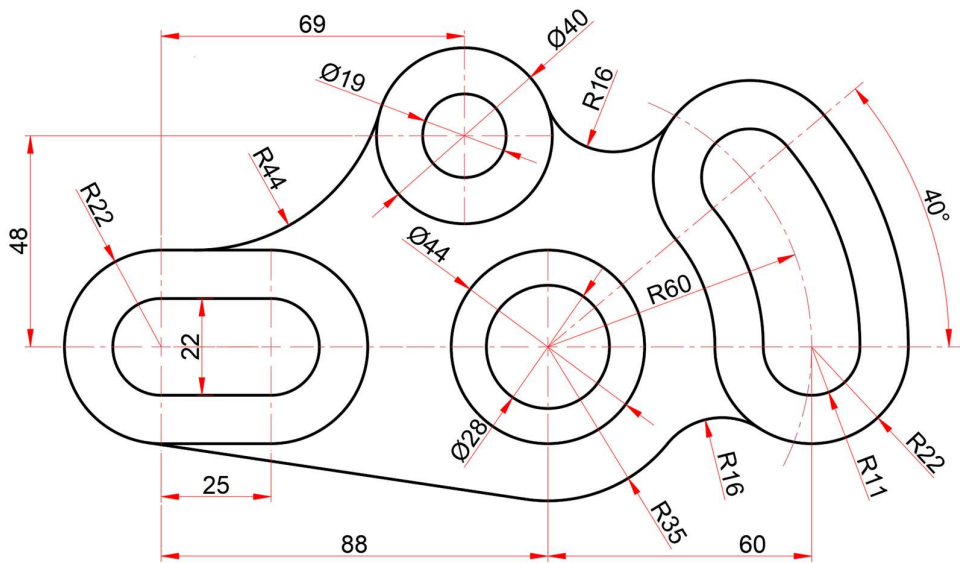
Exercise (18)



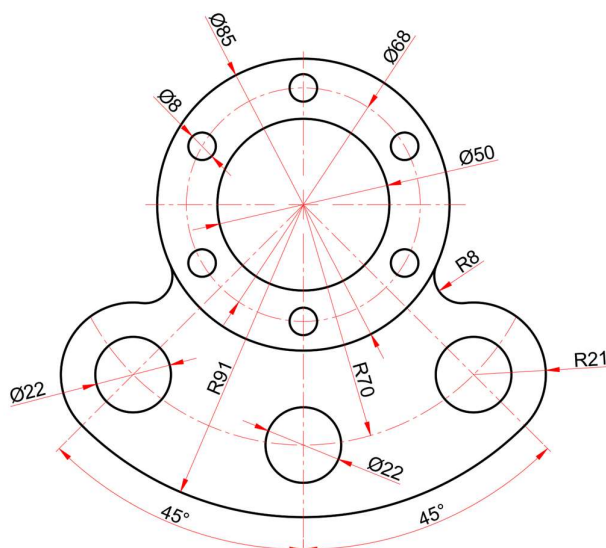
Exercise (19)



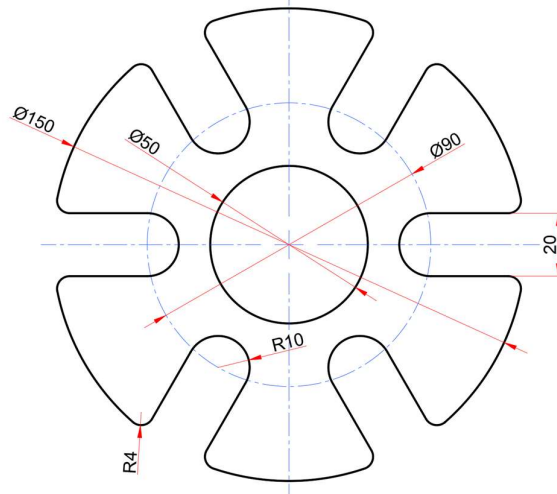
Exercise (20)



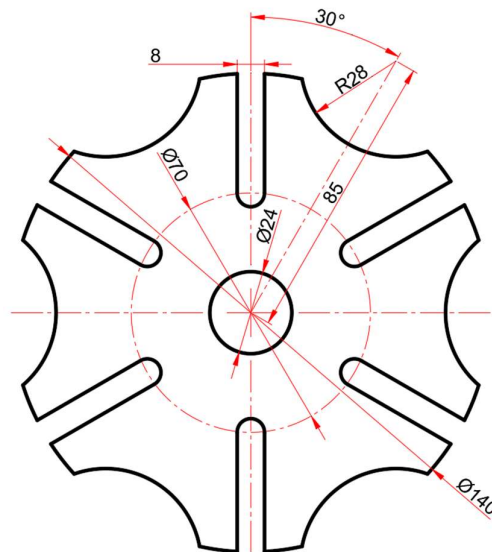
Exercise (21)



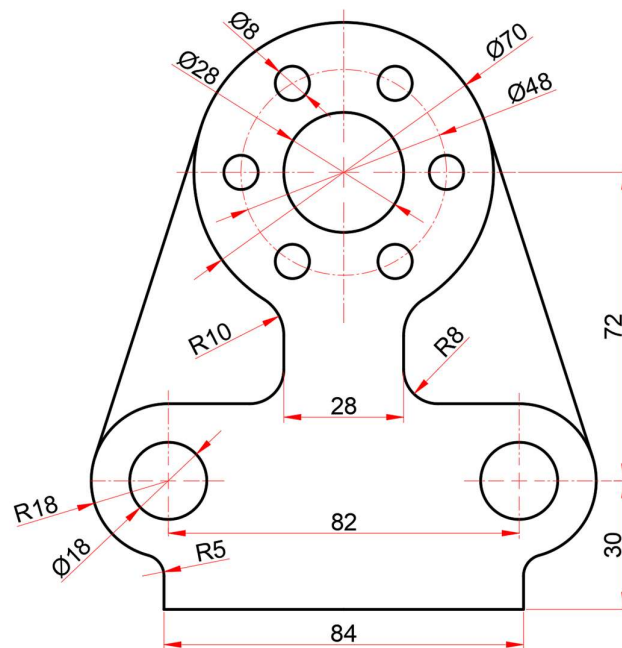
Exercise (22)



Exercise (23)



Exercise (24)



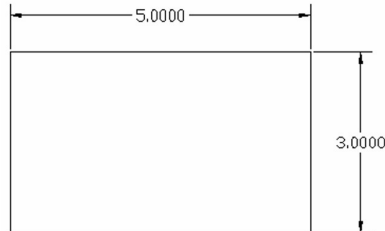
10 Adding Dimensions

10.1 Linear Dimensions

- Choose Dimension, Linear.
- or
- Click the Linear Dimension command from the toolbar.



- or
- Type DIM at the command prompt. Command: DIM
Dim: HOR or VER

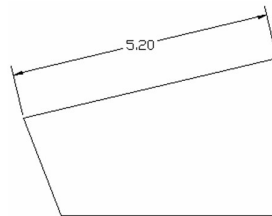


10.2 Aligned Dimensions

- Choose Dimension, Aligned
- or
- Click the Aligned Dimension command from the toolbar.



- or
- Type DIM at the command prompt. Command: DIM
Dim: ALIGNED

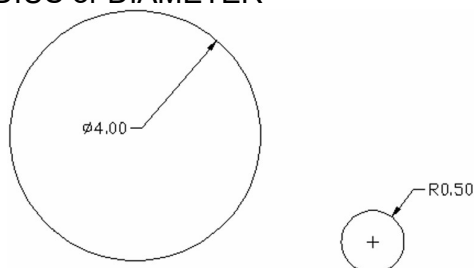


10.3 Radial Dimensions

- Choose Dimension, Radius or Diameter.
- or
- Click the Radial Dimensions command from the toolbar.



- or
- Type DIM at the command prompt. Command: DIM
Dim: RADIUS or DIAMETER

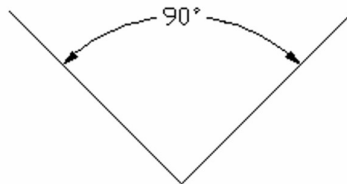


10.4 Angular Dimensions

- Choose Dimension, Angular.
or
- Click the Angular Dimensions command from the toolbar.



- Type DIM at the command prompt. Command: DIM
Dim: ANGULAR

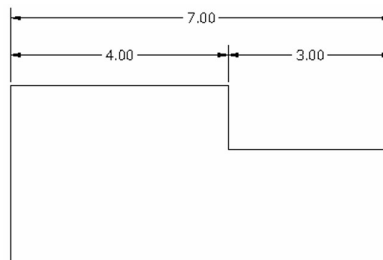


10.5 Continued and Baseline Dimensions

- Choose Dimension, Continue or Baseline.
or
- Click the Continue or Baseline Dimensions command from the toolbar.



- Type DIM at the command prompt. Command: DIM
Dim: CONTINUE or BASELINE

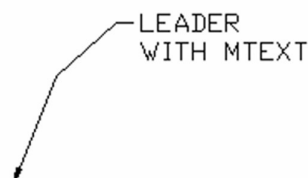


10.6 Leaders

- Choose Dimension, Leader
or
- Click the Leader icon from the Dimension toolbar.



- Type QLEADER at the command prompt.
Command: QLEADER



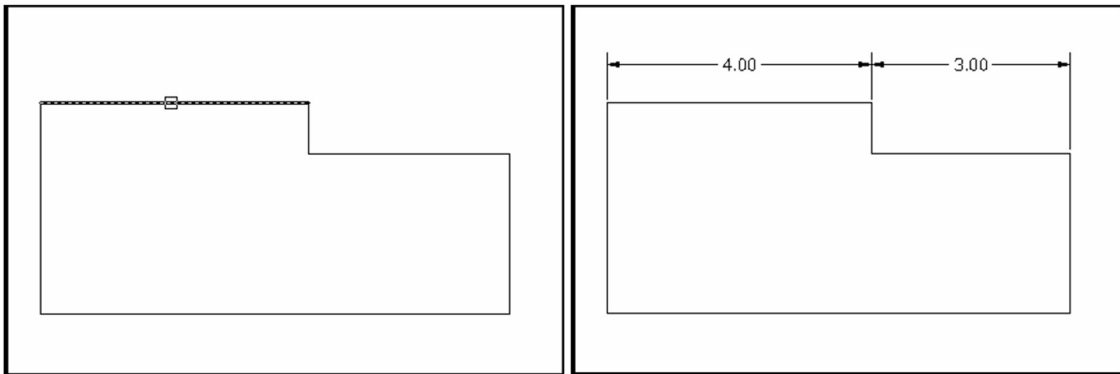
10.7 Quick Dimensions

Quickly creates dimension arrangements from the geometry you select.

- Choose Dimension, QDIM.
or
- Click the Quick Dimension icon from the Dimensions
toolbar.



- Type QDIM at the command prompt.
Command: QDIM
- Pick the objects to dimension.




11 Text, Hatching Objects and Layers

11.1 Text

11.1.1 Access Command

Creates a single-line text object

1. Type TEXT at the command prompt
Command: TEXT
or
2. Pick the Single Line Text icon from the Text Toolbar. 
3. Pick A start point
Justify/Style/<Start Point>: (point)
or
4. Type J to change the justification or S to change the text style.
5. Type A text height
Height <default>: (type value or pick two points)
6. Type A rotation angle
Rotation angle <default>: (angle or point)
7. Type A text string
Text: (type text string)
8. Press enter to exit the Text: prompt.

11.1.2 DTEXT (Dynamic Text)

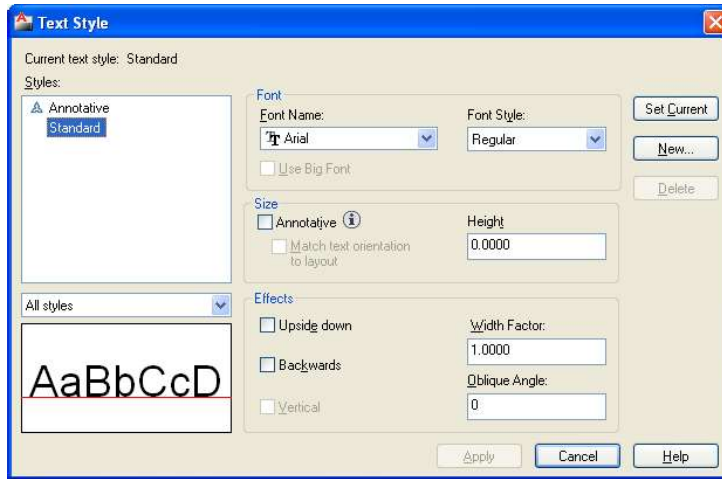
Creates a single-line text object, showing the text dynamically on the screen as it is entered.

- Choose Draw, Text, Single Line Text.
or
- Type DTEXT at the command prompt
Command : DTEXT
- Follow the steps 3-8 from above.

11.1.3 Text Styles

Style Command

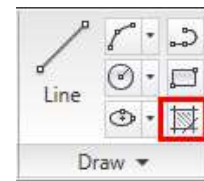
1. Choose Format, Text Style...
or
2. Type STYLE at the command prompt.
Command: STYLE
3. Choose a style from the menu or create a NEW style.
4. Choose a font file.
5. Type a height for the text (set to zero to vary heights)
6. Type a width factor for each character. Width factor
<1>: (enter)
7. Type an obliquing (slant) angle.
Obliquing angle <0>: (angle or enter)
8. Type Yes or No to place characters
backwards. Backwards? (Y or N)
9. Type Yes or No to draw characters upside
down. Upside down? (Y or N)
10. Type Yes or No to draw characters vertically



11.2 Hatching Objects

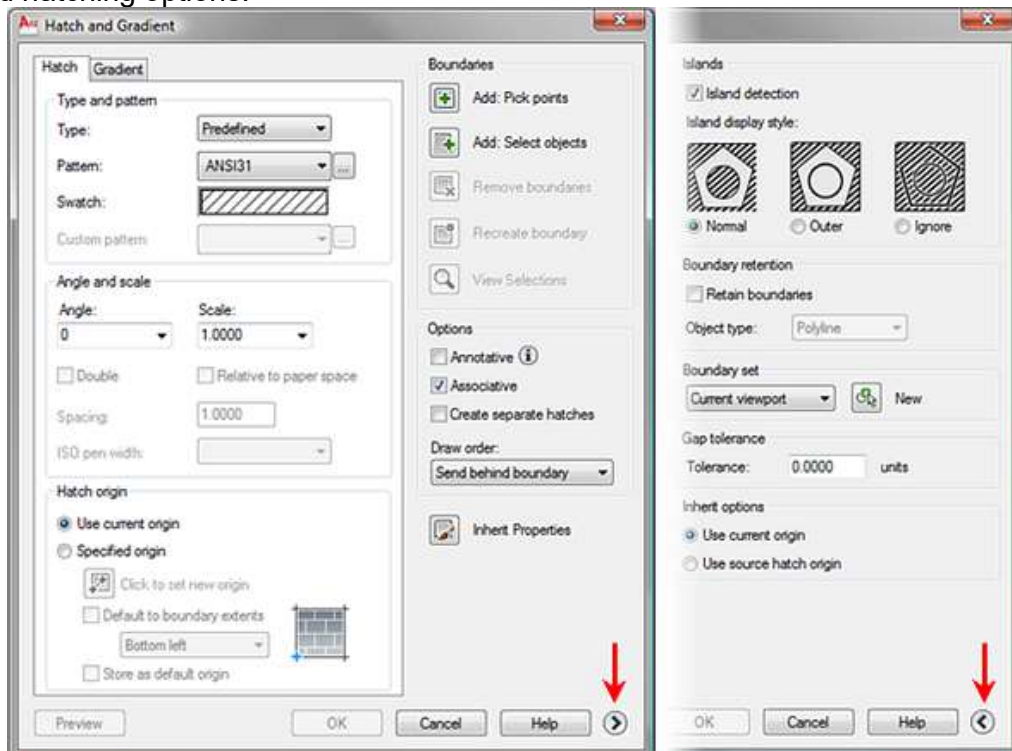
Hatch patterns are graphical elements that are often used to represent materials, special regions, or textures in a drawing. In addition to using a pattern to define an area, you can use gradient or solid fill.

Command Line: HATCH, H
Ribbon: Home tab > Draw panel > Hatch



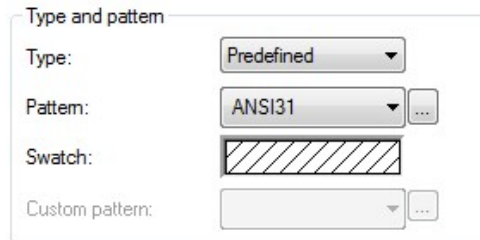
Hatch and Gradient Dialog Box: Hatch Tab

Click the arrow at the bottom-right of the Hatch and Gradient dialog boxes to access the advanced hatching options.

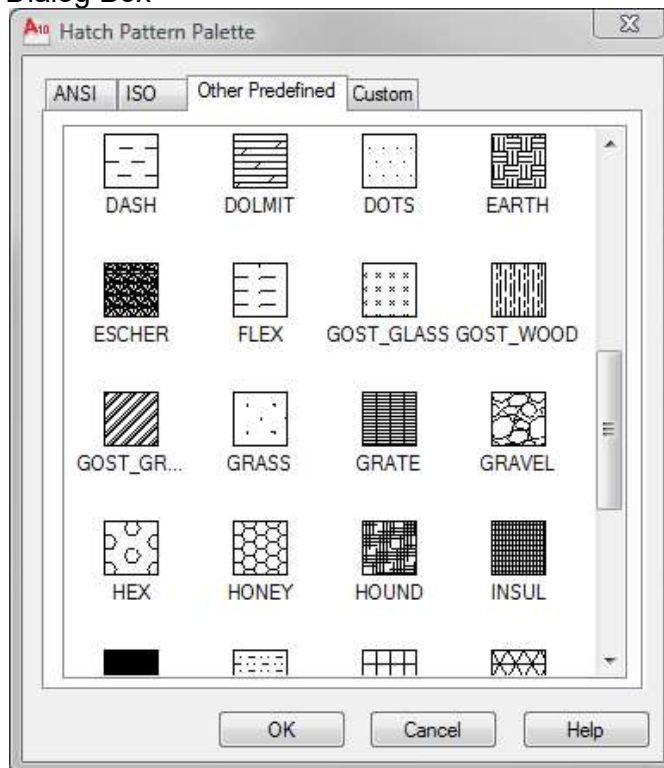


11.2.1 Type and Pattern

You use the Type and Pattern area of the Hatch and Gradient dialog box to select the hatch pattern. You can select patterns from the Pattern list or click the browse button or Swatch area to open the Hatch Pattern Palette dialog box. Using this dialog box, you can select patterns based on a visual swatch.

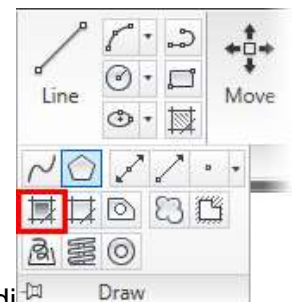


Hatch Pattern Palette Dialog Box



11.2.2 Gradient Hatch

- Command Line: GRADIENT
- Ribbon: Home tab > extended Draw panel > Gradient

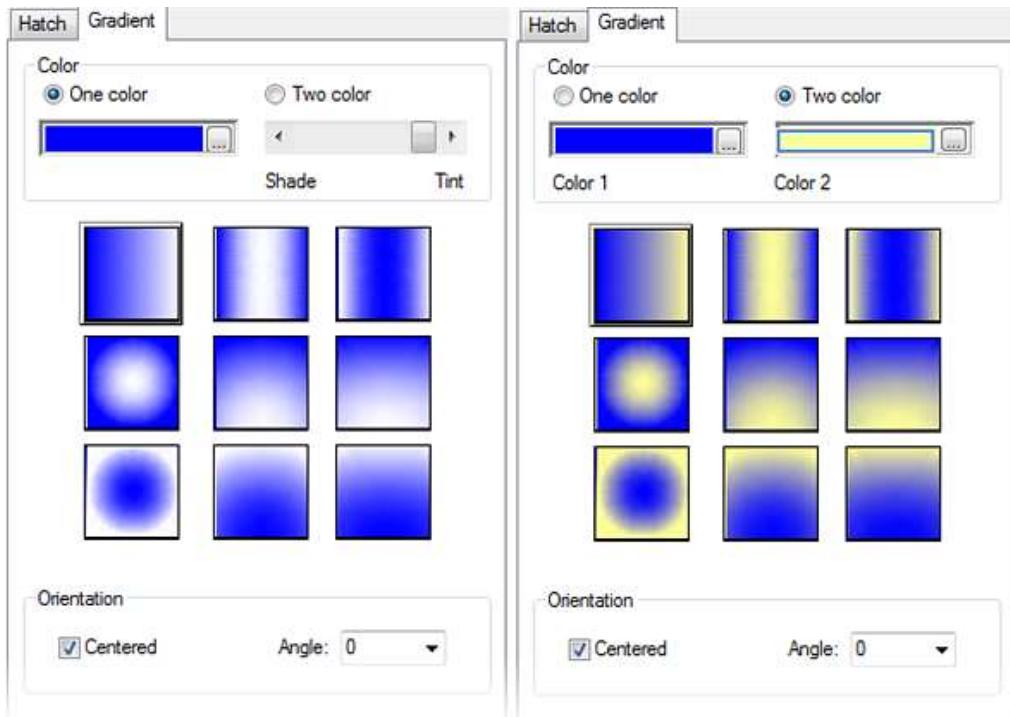


11.2.3 Hatch and Gradient Dialog Box: Gradient Tab

In the Hatch and Gradient dialog box, click the Gradient tab to create gradient drawing. You can choose between a one color gradient with adjustments to mimic shading or tint and a two color option in which you specify two colors for the gradient.

Select the gradient pattern from nine predefined choices.

In the Orientation area, you can choose to have the gradient centered within the boundary and adjust the angle of the gradient.



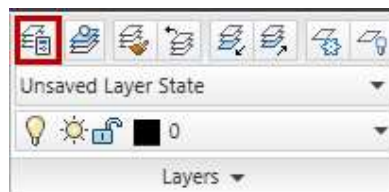
11.3 Layers

11.3.1 Access Command

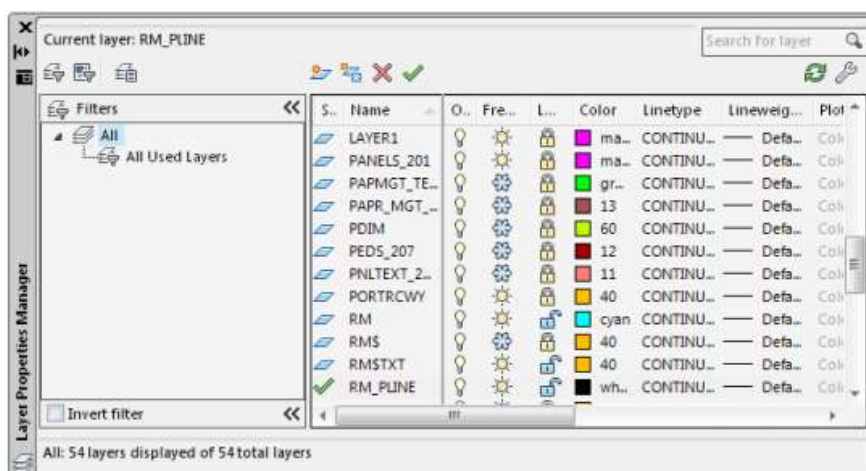
Use the Layer Properties Manager to create layers and control the color, linetype, lineweight, and other properties of each layer.

Command Line: **LAYER, LA**




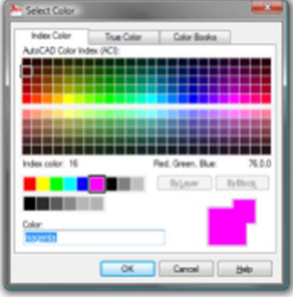
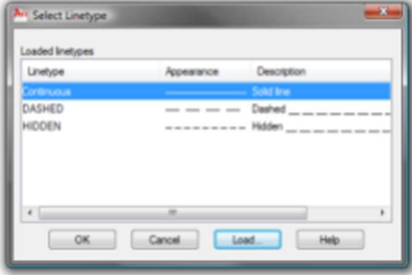
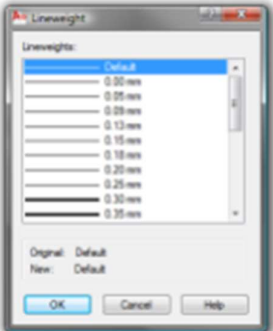
Ribbon: **Home tab > Layers panel > Layer Properties**



Your primary tool for managing layers is the Layer Properties Manager. Use this dialog box to create and manage layers.



Command Options

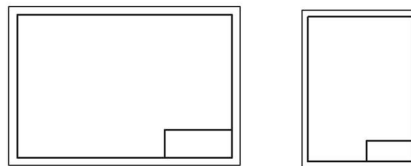
Option	Description																																													
	<p>Click to create a new layer. Enter a name for the layer in the field. Select the layer name twice to rename. Layer names can be listed in the Name column in ascending or descending order</p>																																													
	<p>Click to delete the selected layer. You cannot delete the current layer or layers containing objects. You can Undo by selecting the Undo icon from the Quick Access toolbar</p>																																													
	<p>Click to make the selected layer Current. You can also double-click the layer to make it the current layer.</p>																																													
<table border="1" data-bbox="140 678 649 888"> <thead> <tr> <th>Color</th> <th>Linetype</th> <th>Lineweight</th> <th>Plot Style</th> <th>Plot</th> </tr> </thead> <tbody> <tr> <td> magenta</td> <td>CONTINUOUS</td> <td> Default</td> <td>Color_6</td> <td></td> </tr> <tr> <td> magenta</td> <td>CONTINUOUS</td> <td> Default</td> <td>Color_6</td> <td></td> </tr> <tr> <td> green</td> <td>CONTINUOUS</td> <td> Default</td> <td>Color_3</td> <td></td> </tr> <tr> <td> 13</td> <td>CONTINUOUS</td> <td> Default</td> <td>Color_13</td> <td></td> </tr> <tr> <td> 60</td> <td>CONTINUOUS</td> <td> Default</td> <td>Color_60</td> <td></td> </tr> <tr> <td> 12</td> <td>CONTINUOUS</td> <td> Default</td> <td>Color_12</td> <td></td> </tr> <tr> <td> 11</td> <td>CONTINUOUS</td> <td> Default</td> <td>Color_11</td> <td></td> </tr> <tr> <td> 40</td> <td>CONTINUOUS</td> <td> Default</td> <td>Color_40</td> <td></td> </tr> </tbody> </table>	Color	Linetype	Lineweight	Plot Style	Plot	magenta	CONTINUOUS	Default	Color_6		magenta	CONTINUOUS	Default	Color_6		green	CONTINUOUS	Default	Color_3		13	CONTINUOUS	Default	Color_13		60	CONTINUOUS	Default	Color_60		12	CONTINUOUS	Default	Color_12		11	CONTINUOUS	Default	Color_11		40	CONTINUOUS	Default	Color_40		<p>Click the layer row and the appropriate column to assign or change the properties for that layer. Properties include Color (see below), Linetype (see below), Lineweight (see below) and whether the layer will Plot or not.</p>
Color	Linetype	Lineweight	Plot Style	Plot																																										
magenta	CONTINUOUS	Default	Color_6																																											
magenta	CONTINUOUS	Default	Color_6																																											
green	CONTINUOUS	Default	Color_3																																											
13	CONTINUOUS	Default	Color_13																																											
60	CONTINUOUS	Default	Color_60																																											
12	CONTINUOUS	Default	Color_12																																											
11	CONTINUOUS	Default	Color_11																																											
40	CONTINUOUS	Default	Color_40																																											
	<p>Color: Select a color from the AutoCAD Color Index. Note that the Color white will print black and be displayed black on a white drawing background and white on a black background.</p>																																													
	<p>Linetype: A blank drawing will contain the Continuous linetype. Select the Load button to load other linetypes.</p>																																													
	<p>Lineweight: Various lineweights will not be displayed in the drawing window unless the Show Lineweight option is selected in the status bar. Note that the default lineweight is 0.01 inches or 0.25 mm.</p>																																													

12 Setting Up a Layout

Engineering drawings are to be prepared on standard size drawing sheets. The correct shape and size of the object can be visualized from the understanding of not only the views of it but also from the various types of lines used, dimensions, notes, scale, etc. To provide the correct information about the drawings to all the people concerned.

12.1 Title Block

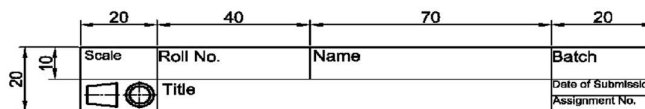
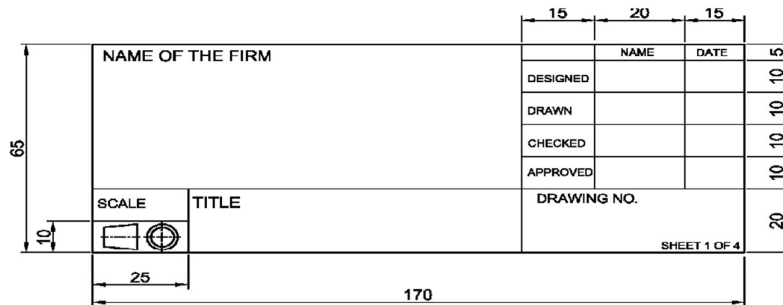
The title block should lie within the drawing space such that, the location of it, containing the identification of the drawing, is at the bottom right-hand corner. This must be followed, both for sheets positioned horizontally or vertically (below figure).



Location of title block

The direction of viewing of the title block should correspond in general with that of the drawing. The title block can have a maximum length of 170 mm. the below figure shows a typical title block, providing the following information:

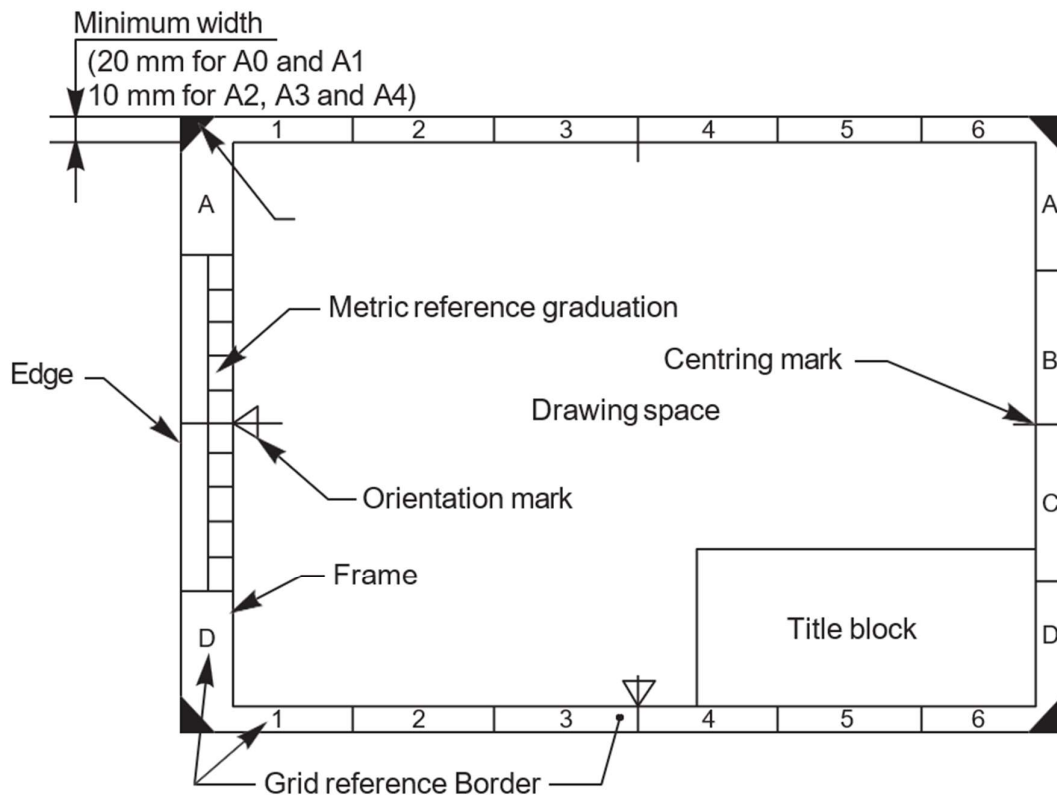
- Title of the drawing
- Sheet number
- Scale
- Symbol, denoting the method of projection
- Name of the firm
- Initials of staff drawn, checked and approved.



Details in title block

12.2 Borders and Frames

Borders enclosed by the edges of the trimmed sheet and the frame, limiting the drawing space, should be provided with all sheet sizes. It is recommended that these borders have a minimum width of 20 mm for the sizes A0 and A1 and a minimum width of 10 mm for the sizes A2, A3 and A4 . A filing margin for taking perforations, may be provided on the edge, far left of the title block.



Drawing sheet layout

12.3 Scales

Scale is the ratio of the linear dimension of an element of an object as represented in the drawing, to the real linear dimension of the same element of the object itself. Wherever possible, it is desirable to make full size drawings, so as to represent true shapes and sizes. If this is not practicable, the largest possible scale should be used. While drawing very small objects, such as watch components and other similar objects, it is advisable to use enlarging scales.

The complete designation of a scale should consist of the word Scale, followed by the indication of its ratio as:

SCALE 1 : 1 for full size.

SCALE × : 1 for enlarged scales.

SCALE 1 : × for reduced scales.

The designation of the scale used on the drawing should be shown in the title block.

The recommended scales for use on technical drawings are given in Table below. The scale and the size of the object in turn, will decide the size of the drawing.

Category	Recommended Scales		
Enlarged scales	50:1	20:1	10:1
	5:1	2:1	
Full size			1:1
Reduced scales	1:2	1:5	1:10
	1:20	1:50	1:100
	1:200	1:500	1:1000
	1:2000	1:5000	1:10000