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AutoCAD 2022 Essential  
Training Document



## AutoCAD:

- AutoCAD is a **commercial computer-aided design** (CAD) and drafting software application. AutoCAD is used in industry, by architects, project managers, engineers, graphic designers, city planners, and other professionals.
- AutoCAD's native file formats are denoted either by a **(.dwg, .dwt, .dws, or .dxf )** filename extension.
- The primary file format for **2D** and **3D** drawing files created with AutoCAD is **.dwg**. While other third-party CAD software applications can create .dwg files, AutoCAD uniquely creates Real DWG files.

## AutoCAD integration:

Autodesk has also developed a few vertical programs for discipline-specific enhancements such as:

- Advance Steel
- AutoCAD Architecture
- AutoCAD Electrical
- AutoCAD Map 3D
- AutoCAD Mechanical
- AutoCAD MEP
- AutoCAD Plant 3D
- Autodesk Civil 3D

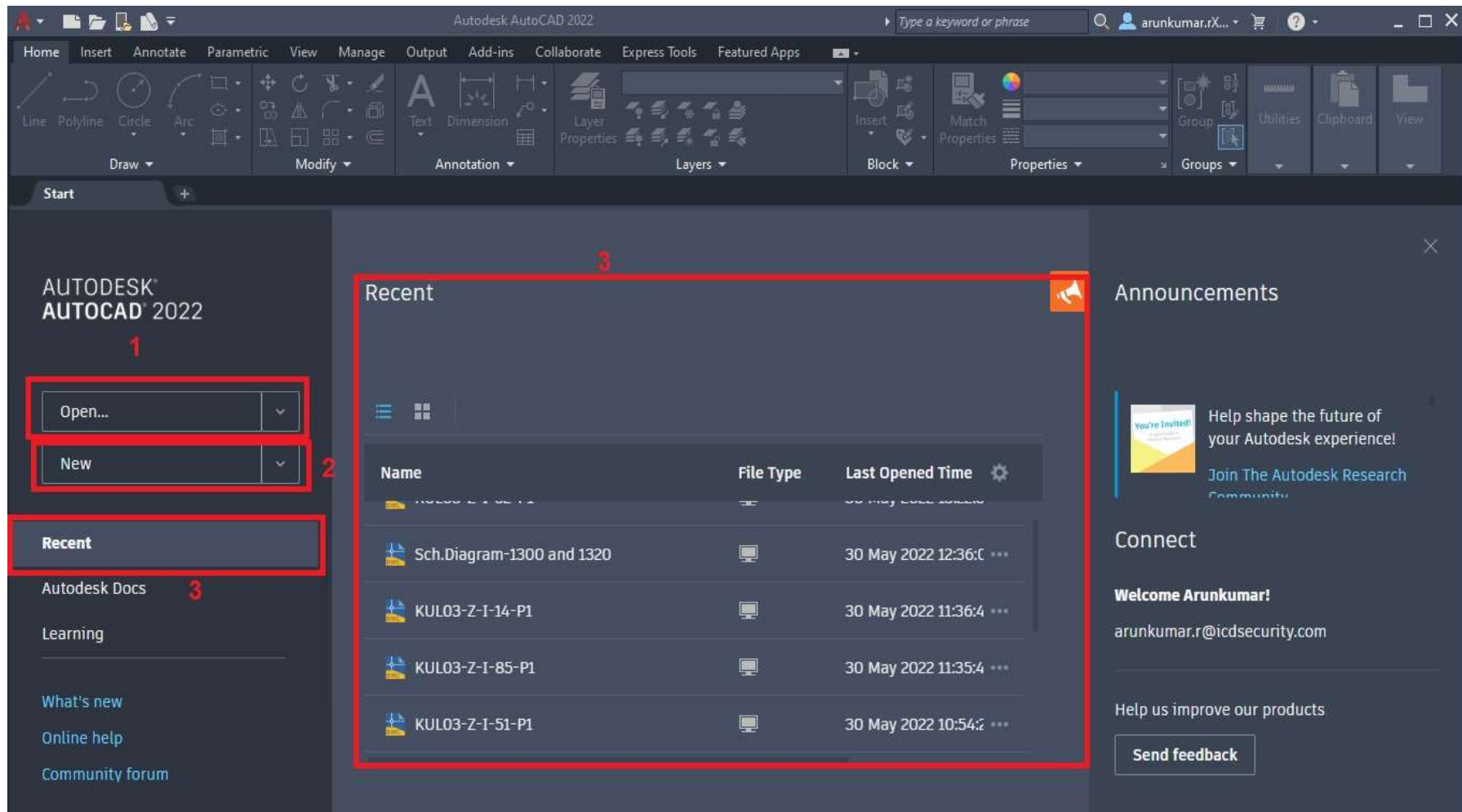
## AutoCAD Logo:





**AutoCAD Startup Page:**

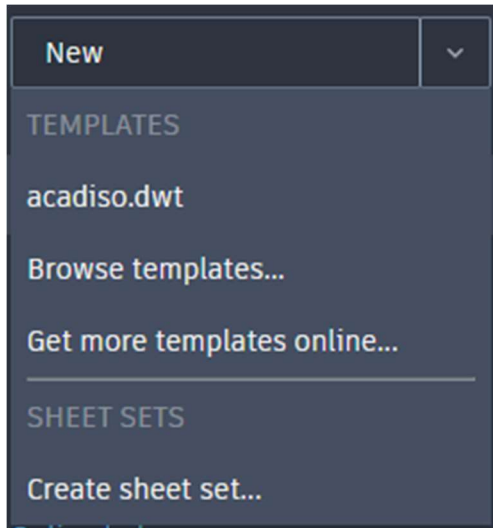




## 1. OPEN:

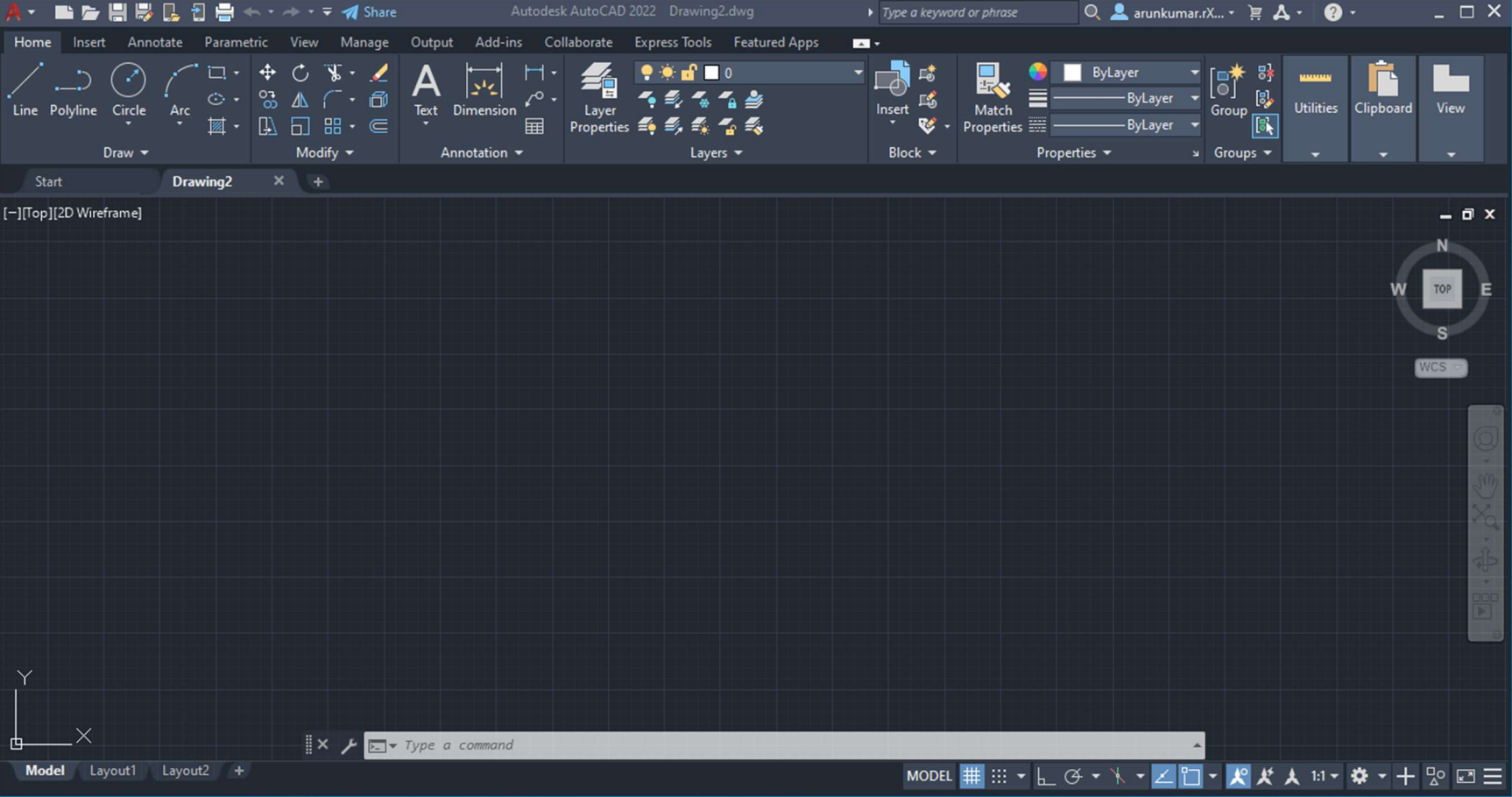
1. Click File > Open > Browse.
2. Next to the File name box, click the All Visio Files dropdown, and then select AutoCAD Drawing.
3. Find the .dwg or .dxf file on your computer, and then double-click it to open it. The AutoCAD file will appear as a new Visio drawing.

## 2. NEW:



1. Click File > New > Dropdown Box>Select options.
2. Acadiso.Dwt → New Model space will open.
3. Browse templates → Saved templates file will open → Select anyone.
4. Get More Templates online → Link: [AutoCAD 2022 Templates | AutoCAD 2022 | Autodesk Knowledge Network](#) → AutoCAD templates.

# Acadiso.Dwt:(New Model Space)



## AutoCAD 2022 windows and toolbar:

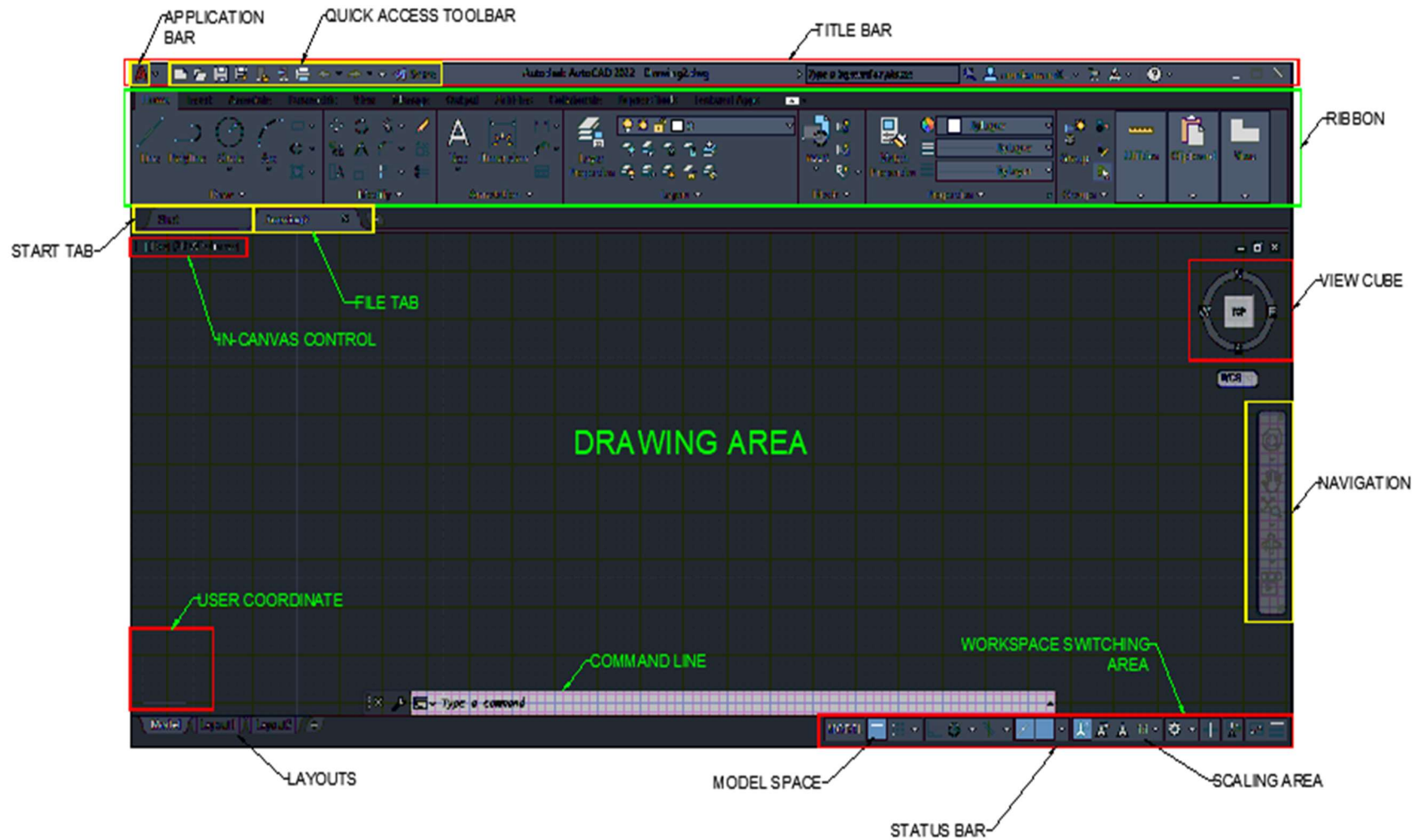


Fig.1.1-Windows and Toolbar.

## 1. APPLICATION BAR:

To access the application menu, **click on the upper-left corner of the workspace**. The application menu remains open as long as the cursor is located inside the application menu window. Ref-Image-Fig.1.2





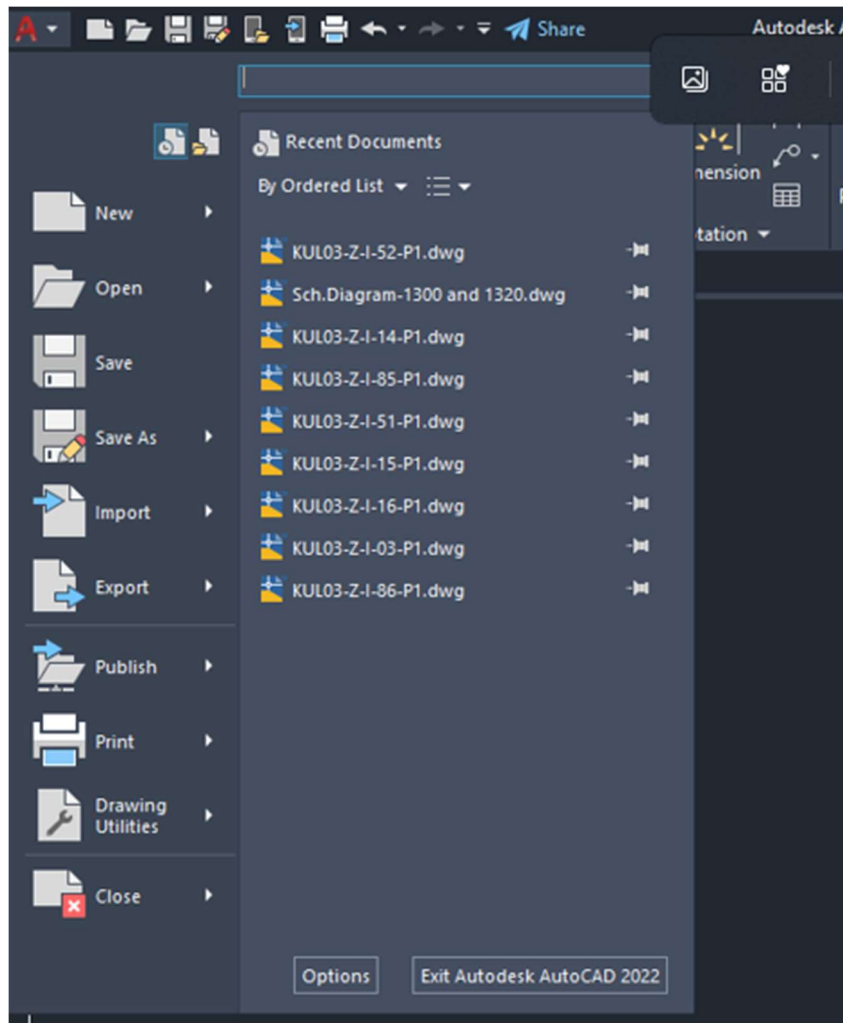


Fig.1.2- Application Menu.

Access common tools to start or publish a file in the Application menu.



Click the Application button to do the following:

- Create, open, or save a file
- Audit, recover and purge a file
- Print or publish a file
- Access the Options dialog box
- Close the application

Note: You can also close the application by double-clicking the Application button.

## 2. QUICK ACCESS TOOLBAR:

Display frequently used tools with the Quick Access toolbar.



### View Undo and Redo History

As with most programs, the Quick Access toolbar displays options to undo and redo changes to your work. To undo or redo a less recent change, click the drop-down button to the right of either the Undo or the Redo button.

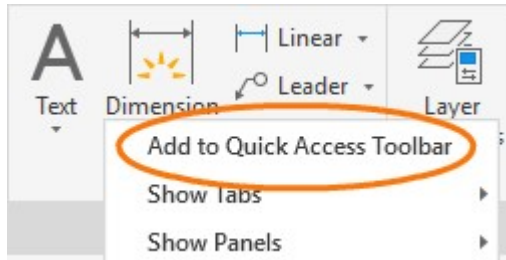


## Add Commands and Controls

You can easily add commonly used tools to the Quick Access toolbar by clicking the indicated drop-down button and clicking your choices from the drop-down menu.



Tip: For a fast way to add a ribbon button to the Quick Access toolbar, right-click any button on the ribbon, and then click Add to Quick Access Toolbar.



Buttons are added to the right of the default commands on the Quick Access toolbar. To remove one of these commands, use the CUI editor and open the [+] Quick Access Toolbars > [+] Quick Access Toolbar 1. From there, you can click and press Delete. You can also drag the elements to change their order on the toolbar.

## 3. TITLE BAR:

The title bar is analogous to the title bar in any Windows program. It contains the program name (AutoCAD or AutoCAD LT) and the title of the current drawing with its path, as long as any drawing other than the default Drawing. The Help button is a direct link to the AutoCAD help system

## 4. RIBBON:

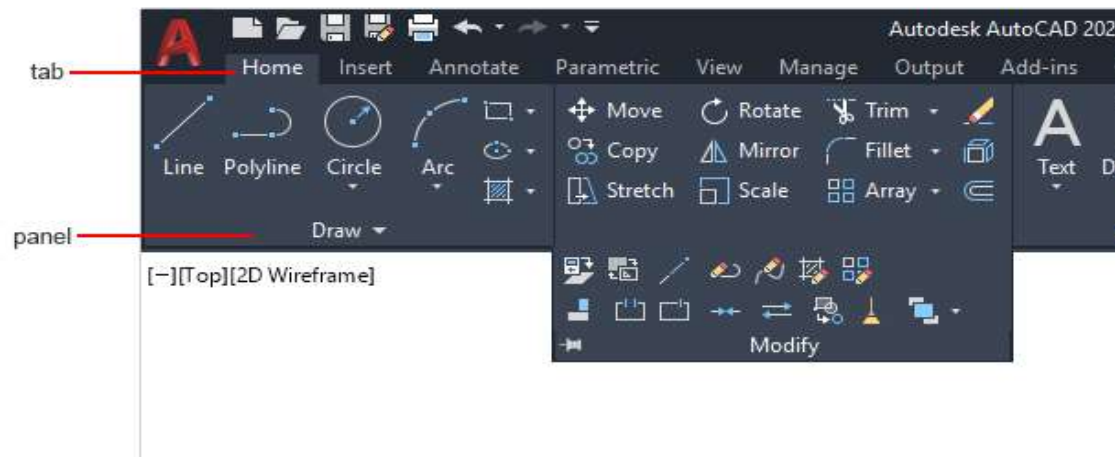
The ribbon organizes tools into logical groupings.

The ribbon provides a compact palette of all of the tools necessary to create or modify your drawing. It can be Laplace the following places:

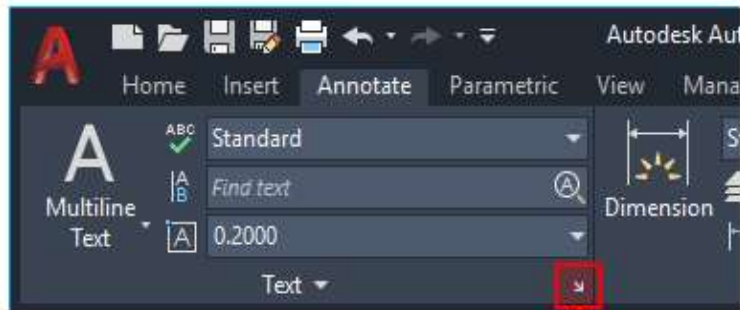
- Docked horizontally at the top of the drawing area (default)
- Docked vertically along the right or left edge of the drawing area
- Undocked, or floating within the drawing area or on a second monitor

### Ribbon Tabs and Panels

The ribbon is composed of a series of tabs, which are organized into panels that contain many of the tools and controls available in toolbars.



Some ribbon panels provide access to a dialog box related to that panel. To display the related dialog box, click the dialog box launcher is denoted by an arrow icon, in the lower-right corner of the panel.

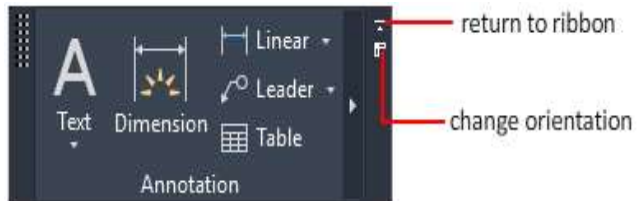


dialog box launcher

**Note:** You can control which ribbon tabs and panels are displayed. Right-click the ribbon and click or clear the names of tabs or panels listed on the shortcut menu.

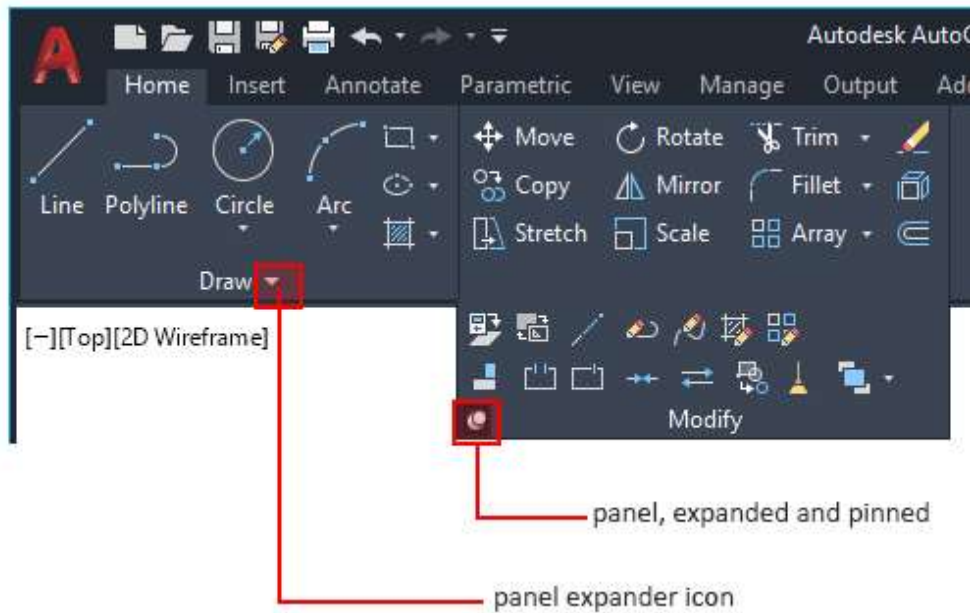
## Floating Panels

You can pull a panel off a ribbon tab and into the drawing area or onto another monitor. The floating panel remains open until you return it to the ribbon, even if you switch ribbon tabs.



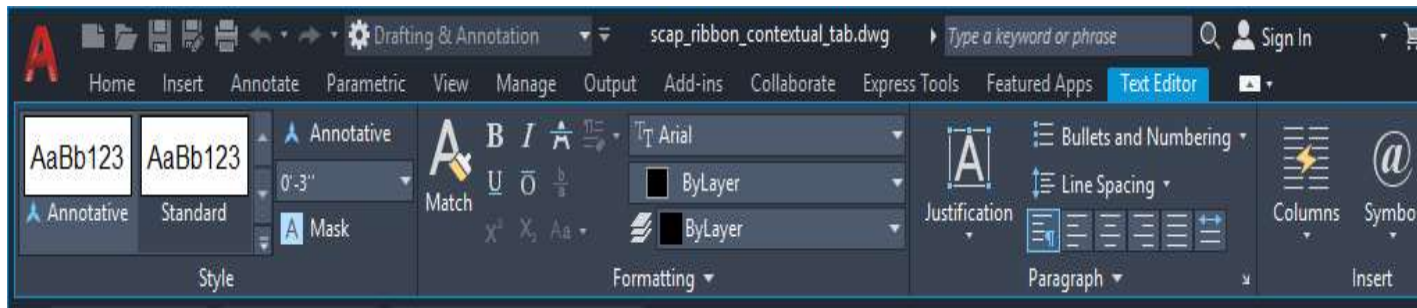
## Slide-out Panels

If you click the arrow in the middle of a panel title, the panel will expand to display additional tools and controls. By default, slide-out panels automatically close when you click another panel. To keep a panel expanded, click the push pin, in the bottom-left corner of the slide-out panel.



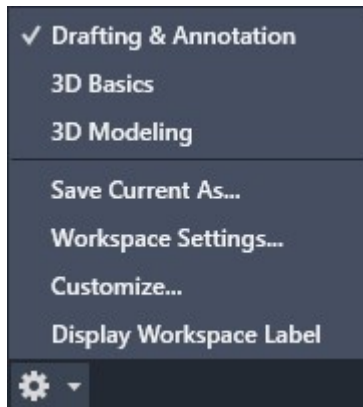
## Contextual Ribbon Tabs

When you select a certain type of object or start certain commands, a contextual ribbon tab is displayed instead of a toolbar or dialog box. The contextual tab closes when you end the command.



## Workspaces and the Ribbon

For products that run on the Windows operating system, a workspace is a set of ribbon tabs and panels, menus, toolbars, and palettes that provide you with a custom, task-oriented drawing environment. You can change to a different ribbon by changing the workspace. On the status bar, click Workspace Switching and select the workspace you want to use. For example, here are the initial workspaces available in AutoCAD.



### 5. START TAB:

The Start tab is displayed by default at startup, providing easy access to a variety of initial actions, including access to drawing template files, recently opened drawings and sheet sets, and online and learning options.

The Start tab contains the following pages:

#### ***Create Page***

The following sections are displayed:

#### ***Get Started***

Access common methods to start a file.



### **Start New Drawing**

- Creates a new drawing from the default drawing template file. You specify the default drawing template file from the Options dialog box, Files tab, Templates, and Default Template File Name the for QNEW setting.
- When a default drawing template file is set to None or is not specified, the new drawing is based on the most recently used drawing template file.

### **Templates**

- Lists all the available drawing template files.

### **Open Files**

- The Select File dialog box is displayed.

### **Open a Sheet Set**

- The Open Sheet Set dialog box is displayed.

### **Get More Templates Online**

- Download additional drawing template files when available.

### **Explore Sample Drawings**

- Access the installed sample files.

### ***Recent Documents***

View the most recently used files. You can keep a file listed by clicking the push pin button. A pinned document is displayed at the top of the list until you turn off the push pin button. You can choose between image, image, and text or text only an of the display options.

### ***Notifications***

Displays all notifications with regards to product updates, hardware acceleration, trial period, and the help file information. A notification badge is displayed at the bottom of the page when there are two or more new notifications.

### ***Connect***

Sign in to your Autodesk account to access online services.

Feedback: Access the online form to provide feedback and any improvements you would like to see.

### ***Learn Page***

The Learn page provides access to learning resources such as videos, tips, and other relevant online content or services if available. For every new content update, a notification badge is displayed at the bottom of the page.





Note: If no internet connection is available, the Learn page is not displayed.

## 6. STATUS BAR:

- The status bar displays the cursor location, drawing tools, and tools that affect your drawing environment.
- The status bar provides quick access to some of the most commonly used drawing tools.
- You can toggle settings such as grid, snap, polar tracking, and object snap. You can also access additional settings for some of these tools by clicking their drop down-arrows. Ref.Image.Fig.1.3

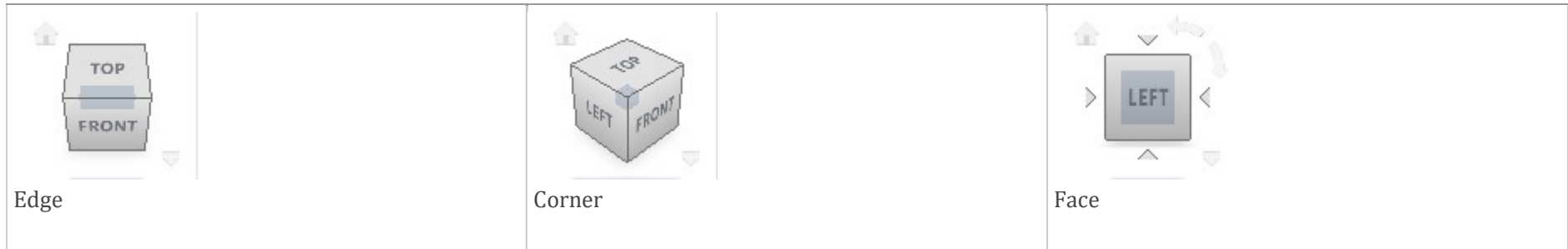


Fig.1.3- Status Bar.

## 7. VIEWCUBE

The View Cube is a navigation tool that is displayed when you are working in a 2D model space or 3D visual style. With ViewCube, you can switch between standard and isometric views.

The View Cube is a persistent, clickable, and draggable interface that you use to switch between standard and isometric views of your model. When you display the View Cube, it is shown in one of the corners of the drawing area over the model in an inactive state. The View Cube tool provides visual feedback about the current viewpoint of the model as view changes occur. When the cursor is positioned over the View Cube tool, it becomes active. You can drag or click the View Cube, switch to one of the available preset views, roll the current view, or change to the Home view of the model.



### ***Control the Appearance of View Cube***

The View Cube is displayed in one of two states: inactive and active. When inactive, it appears partially transparent by default so that it does not obscure the view of the model. When active, it is opaque and may obscure the view of the objects in the current view of the model.

In addition to controlling the inactive opacity level of the View Cube, you can also control the following properties for the ViewCube:

- Size
- Position
- Display the UCS menu
- Default orientation
- Compass display

### ***Using the Compass***

The compass is displayed below the View Cube and indicates which direction North is defined for the model. You can click a cardinal direction letter on the compass to rotate the model, or you can click and drag the compass ring to interactively rotate the model around the pivot point.



Fig.1.4- View Cube.

## 8. NAVIGATION TOOLS.

3D navigation tools allow you to view objects in a drawing from different angles, heights, and distances.

Use the following 3D tools to orbit, swivel, adjust distance, zoom, and pan in a 3D view.

- **3D Orbit.** Moves around a target. The target of the view stays stationary while the camera location, or point of view, moves. The target point displays temporarily as a small, dark sphere. The method used for locating the target point is determined by the ORBITAUTOTARGET system variable.
- **Constrained Orbit.** Constrains 3D Orbit along the *XY* plane or the *Z-axis*.
- **Free Orbit.** Orbits in any direction without reference to the planes. The point of view is not constrained along the *XY* plane of the *Z-axis*.
- **Continuous Orbit.** Orbits continuously. Click and drag in the direction you want the continuous orbit to move, and then release the mouse button. The orbit continues to move in that direction.
- **Adjust Distance.** Changes the distance of objects as you move the cursor vertically. You can make objects appear larger or smaller, and you can adjust the distance.
- **Swivel.** Simulates panning with a camera in the direction that you drag. The target of the view changes. You can swivel the view along the *XY* plane or the *Z axis*. Simulates moving the camera closer to an object or farther away. Zooming in magnifies the image.
- **Pan.** Starts the interactive 3D view and enables you to drag the view horizontally and vertically.

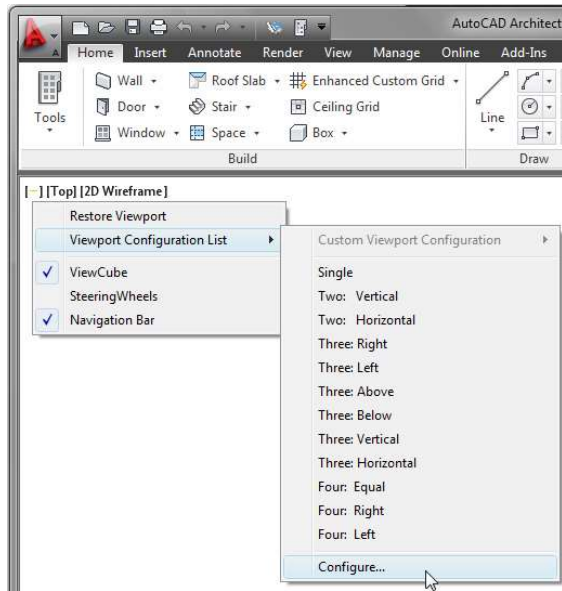


Fig.1.4- Navigation Tool.

## 9. In-Canvas View (or) View Port Controls.



1. Ref. Image.Fig.1.5 → Appearing at the top left corner of the drawing area on the Model "tab" (TILEMODE 1) and consuming a minimum of screen real estate, there are three text strings enclosed in square brackets; a left-click on one of the strings gives access to popup menus related to that particular view control. The first string - either a "-" (one viewport displayed) or a "+" (multiple viewports displayed - allows you to control the visibility of the View Cube, Steering Wheels, and Navigation Bar as well as access to Viewport configurations, including any custom configurations you may have created).



**Fig.1.5- 1-View Port.**

2. Ref. Image.Fig.1.6 → The second string allows you to quickly select a Model View to restore, either preset or custom, as well as access to the View Manager. The name of the current View direction or Model Space View (if current) is displayed in the second string, for easy reference while you work.

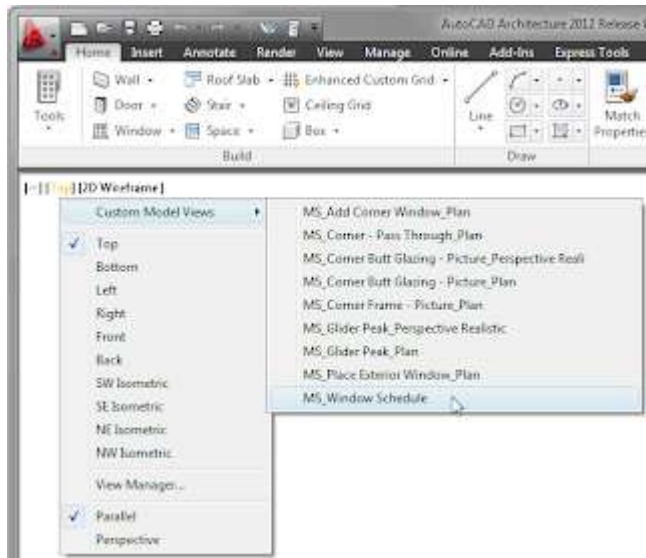


Fig.1.6- 1-View Port.

3. **Ref. Image.Fig.1.7** → The third-string displays the name of the current Visual Style, and the menu allows you to set a different Visual Style current (predefined or custom) or to open the Visual Styles Manager.

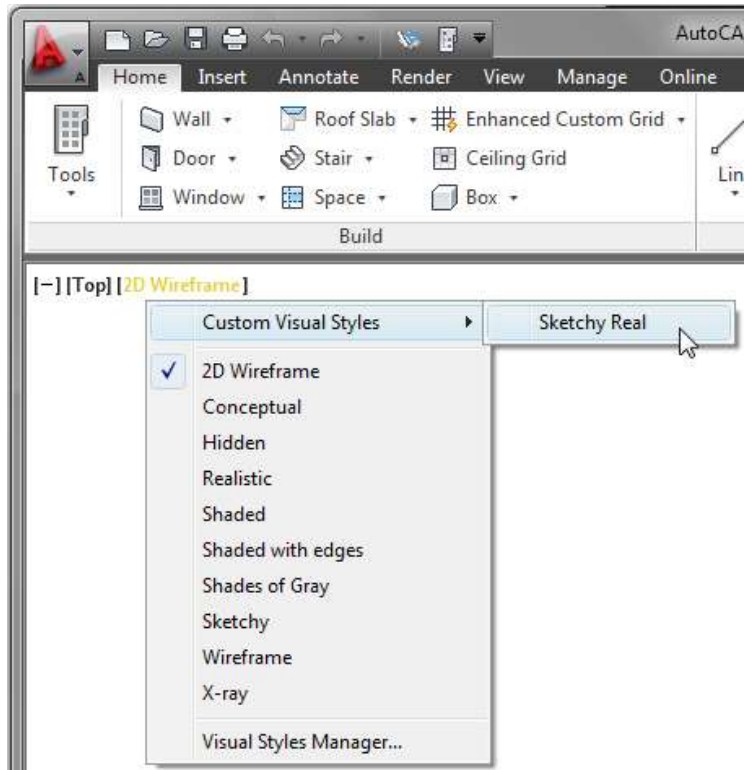


Fig.1.7- 1-View Port.

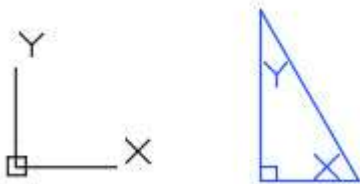
## 10. User coordinate system. (Command→ucs)

The user coordinate system (UCS) establishes the location and orientation of a movable Cartesian coordinate system. The UCS is an essential tool for many precision operations.

The UCS defines

- The XY plane, also called the work plane or construction plane, on which objects are created and modified
- The horizontal and vertical directions used for features like Ortho mode, polar tracking, and object snap tracking
- The alignment and angle of the grid, hatch patterns, text, and dimension objects
- The origin and orientation for coordinate entry and absolute reference angles
- For 3D operations, the orientation of work planes, projection planes, and the Z-axis for vertical direction and axis of rotation

By default, the UCS icon appears in the lower-left corner of the drawing area for the current model viewport. The UCS in each paper space layout is displayed as a drawing triangle.

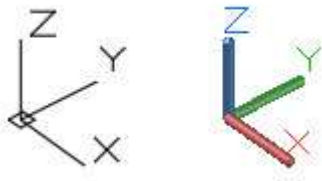


**Note:** A variety of colour, size, and style settings for the UCS are available from the UCSICON command. You can also turn it off when not needed



## Understand the UCS in 3D

When you create or modify objects in a 3D environment, you can move and reorient the UCS anywhere in 3D space to simplify your work. The UCS is useful for entering coordinates, creating 3D objects on 2D work planes, and rotating objects in 3D. The UCS icon changes automatically depending on the current visual style. The colorful 3D icon on the right displays when a 3D visual style is current.



**Note:** The UCS icon follows the traditional right-hand rule in determining positive axis directions and rotation directions.

## 11. LAYOUTS

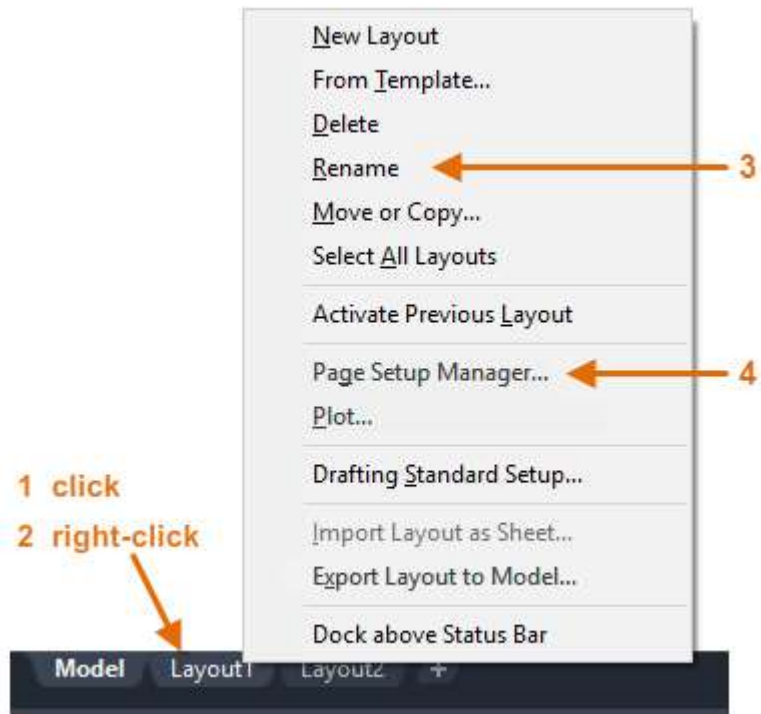
Display one or more scaled views of your design on a standard-size drawing sheet called a layout.

After you finish creating a model at full size, you can switch to a *paper space layout* to create scaled views of the model and to add notes, labels, and dimensions. You can also specify different line types and line widths for display in paper space.

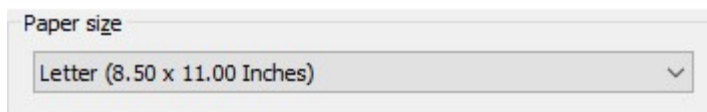
### Specifying the Paper Size of a Layout

The first thing that you should do when you access a layout tab (1) is to right-click the tab (2) and rename it (3) to something more specific than Layout 1. Names that reference a specific ANSI or ISO size might be a good choice.





Next, open the Page Setup Manager (4) to change the paper size displayed in the layout tab. There are a lot of controls here, but you only need to change a few. The first is to specify the size of your sheet.

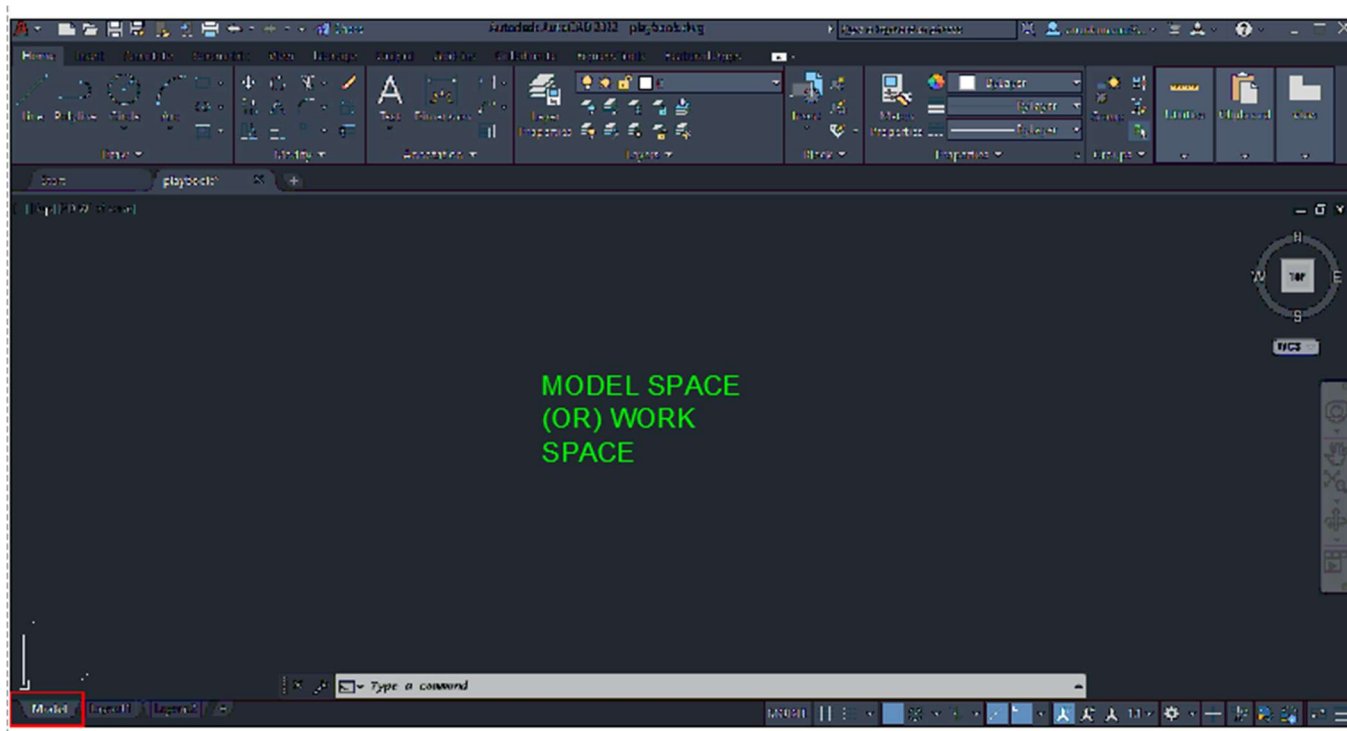


**Note:** You might be wondering why there are two entries in the list for every sheet size. This is because some printers and plotters do not recognize the drawing orientation setting.



## Model Space and Paper Space

As you know, you create the geometry of your model in *model space*. **Ref. Image-** Fig.1.8-1-MODEL SPACE.



**Fig.1.8- 1-MODEL SPACE.**

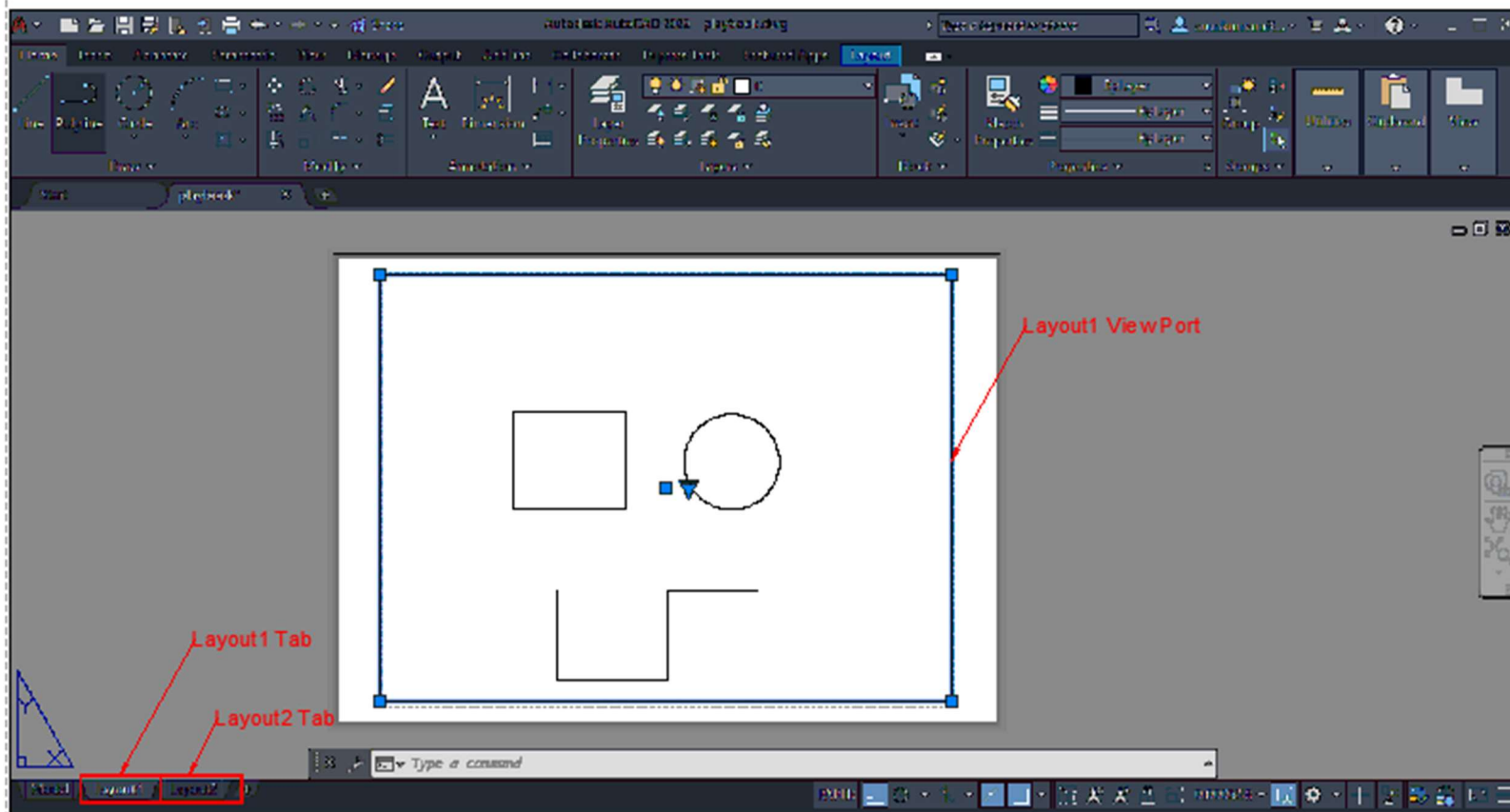


Fig.1.9- 2-Layouts.

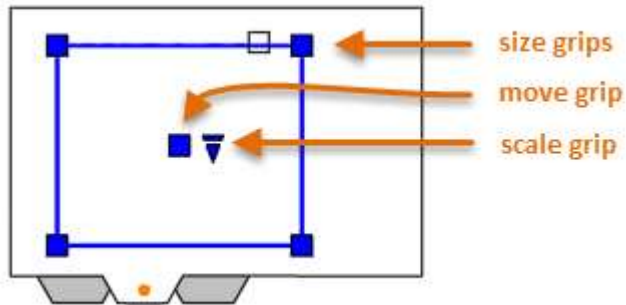
Originally, this was the only space available in AutoCAD. All notes, labels, dimensions, and the drawing border and title block were also created and scaled in model space. For some applications, this method is still entirely sufficient.

With the *paper* space feature, you can click a layout tab designed specifically for displaying multiple views, automatic scaling, and electronic or printing output.

For example, a layout tab is selected in the following illustration. There are currently two objects in paper space: a block reference for the title and drawing border, and a single layout viewport that displays a scaled view of model space.

## Layout Viewports

A layout viewport is an object that's created on a layout tab to display a scaled view of model space. You can think of it as a closed-circuit TV monitor that displays part of the model space. You can select a layout viewport as you would any other object. When you select it, several grips display provides vide a way to adjust the size of the viewport, move the viewport, and specify the scale of the view that's contained in it.



Several editing commands such as Move, Erase, and Copy can be used on layout viewports. When you select a layout viewport, you can use the Properties palette to provide a complete list of options and settings.

**Tip:** When you're ready to create an electronic output or print a layout, you will want to hide the viewport borders. To accomplish this, create all viewport objects on a separate layer and then turn that layer off.

## 12. COMMAND LINE.

The command line is one of the most unique and recognizable components of the interface. It has been there since the very earliest releases of AutoCAD when the program ran under the DOS operating system. The command line is where you will see instructions and options when using commands.

**For example,** on the Home ribbon, in the Draw panel, click the Line tool. On the command line, the program prompts you to specify the first point of the line. Once you click to pick that point, the prompt changes. **Ref. Image-** Fig.1.10- Command-Line.

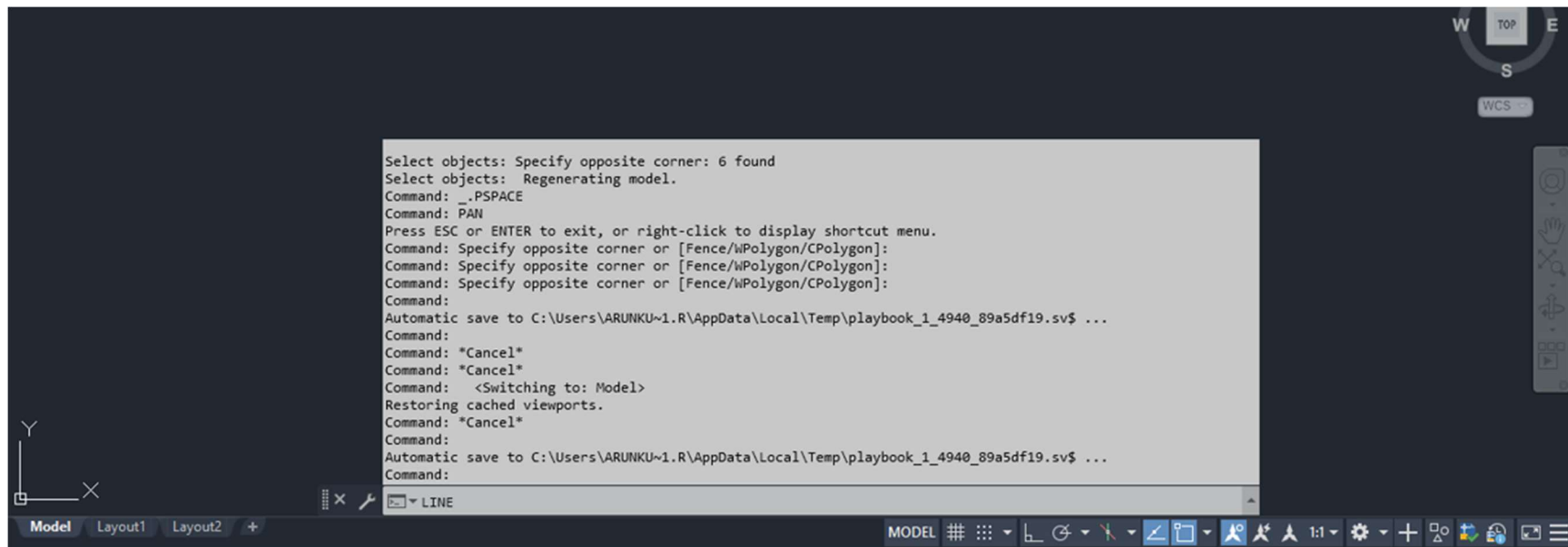
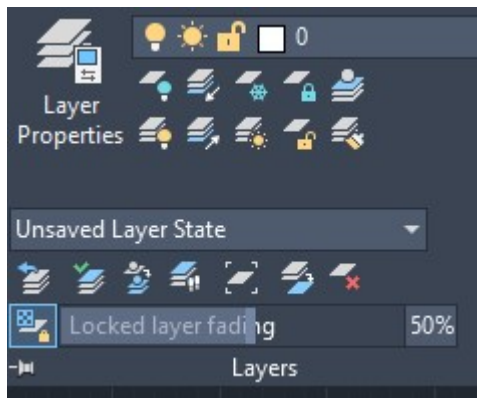


Fig.1.10- Command-Line.

## AutoCAD Layer:

- The layer command is used to control and manage the drawings in AutoCAD for different purposes.
- It increases the display performance of the AutoCAD by hiding the portion of our drawing when needed. It also improves the visual complexity of the drawing.
- We are required to create a set of layers having different properties. For example, in a floor plan or house plan, we can create separate layers for doors, walls, etc.
- We can create many layers by specifying the name for the corresponding layer. We can also control the order of the layers.
- The shortcut command of the layer is 'LA'.

The Layer on the ribbon panel looks like the below image:

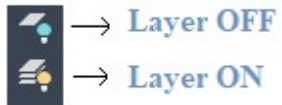


Let's discuss the layer properties in detail.

### Layer ON/OFF:

It is used to turn **ON** or turn **OFF** the layer of the selected object.

The layer is represented as:



The shortcut command for the Layer ON and Layer OFF is **LayON** and **LayOFF**.

### **Layer Isolate/Unisolate:**

The Isolate is used to hide or lock the layers. The Isolated layers are defined as visible layers. The layers are locked except for the selected objects.

The layer is represented as:



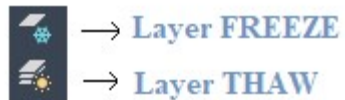
The UNISOLATE is used to unlock the layers. It is the reverse of the LAYER ISOLATE command.

### **Layer Freeze/Thaw:**

The FREEZE is used to freeze the layer of the selected object, while the THAW is used to remove the freeze from the layer.

The LAYER THAW unfroze the layers.

The layer is represented as:



The objects become invisible after applying the LAYER FREEZE command.

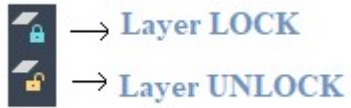
### **Layer LOCK/UNLOCK:**

The **LOCK** is used to lock the layer of the selected object, while **UNLOCK** removes the lock from the layer.

The layer is represented as:







The LAYER LOCK command prevents the object from actually being modified.

### **Current layer:**

The Current layer sets the selected layer as the current layer on the viewport. The objects will be created based on the selected current layer.

The layer is represented as:



### **Match layer:**

The **MATCH LAYER** is used to change the current layer to match the destination layer. The current layer is the layer of the selected objects.

The layer is represented as:



The other icons are listed below:




 → PREVIOUS Layer

 → CURRENT Layer

 → COPY TO Layer

 → WALK Layer

 → VPI Layer

 → MERGE Layer

 → Delete Layer

**Previous layer:**

It is used to **undo** the set of changes made to the settings of layers.

**Current layer:**

It sets the selected layer as the current layer in a drawing. The properties of the selected objects are changed to the current layer.

**Copy to layer:**

It is used to copy one or more objects to the other layer.

**Walk layer:**

The Walk Layer command is used to display the object of the selected layer while hiding the objects of the other layer.

**VPI layer:**

The layers in all layout viewports are frozen, except the current layer.



**Merge layer:**

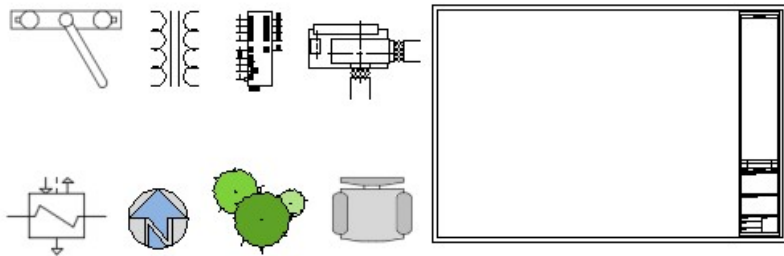
It removes the previous layers from the drawing and merges the selected layers into the target layer.

**Delete layer:**

It is used to delete the selected layer from the layer settings or the list of layers. It also removes all the objects on that layer.

**Blocks:**

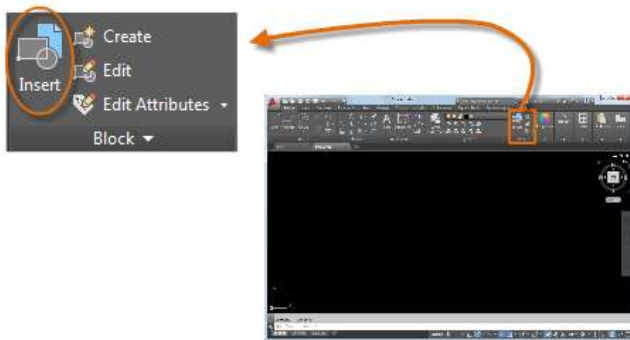
In AutoCAD, a *block* is a collection of objects that are combined into a single named object. The following are some sample blocks at various scales.



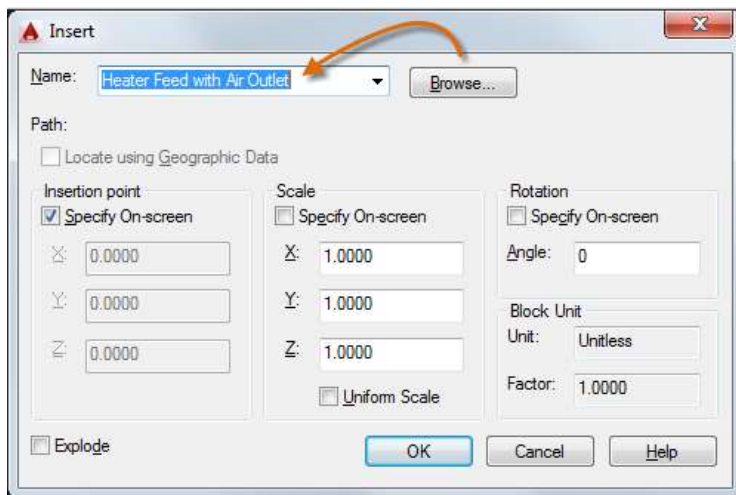
Some of these blocks are realistic representations of objects, some are symbols, and one of them is an architectural title block for a D-size drawing.

**Insert a Block**

Typically, each of these blocks is an individual drawing file, perhaps saved in a folder with similar drawing files. When you need to insert one into your current drawing file, you use the INSERT command (or enter I in the Command window).



The first time you insert the drawing as a block, you need to click Browse to locate the drawing file. Make sure you organize your blocks into easy-to-find folders.

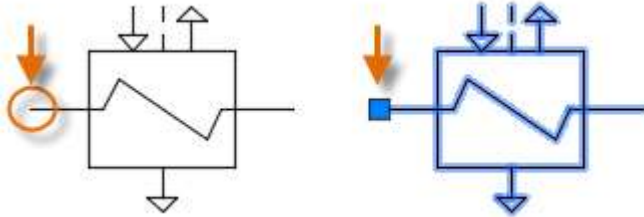


Once inserted, the block definition is stored in your current drawing. From then on, you can choose it from the Name drop-down list without needing to click the Browse button.

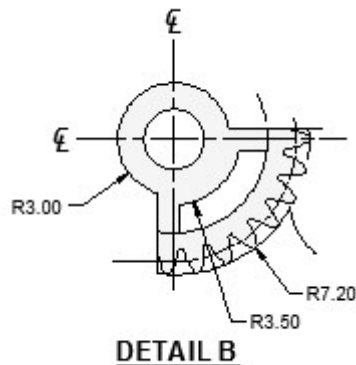
**Tip:** The default settings in the Insert dialog box are usually acceptable. After you choose the block name, click OK, and then specify its location in your drawing. You can rotate it later, if necessary.

Notice that when you insert a block, it is attached to your cursor at the point indicated. This location is called the insertion point. By default, the insertion point is the origin point (0,0) of the original drawing.

After inserting the block, you can select it and a grip appears. You can easily move and rotate this block using this grip.



In the following example, a drawing file is inserted into the current drawing to provide a standard detail view.



**Note:** Inserting a drawing file as a block provides a static reference to the specified drawing. For a reference that automatically updates, you can attach the drawing with the External References palette (XREF command) instead.

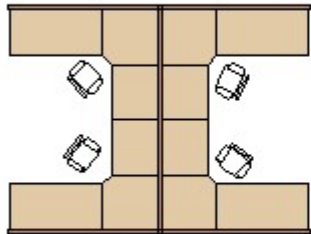
### Create a Block Definition:

Instead of creating a drawing file to be inserted as a block, you might want to create a block definition directly in your current drawing. Use this method if you do not plan to insert the block into any other drawing. In that case, use the BLOCK command to create the block definition.

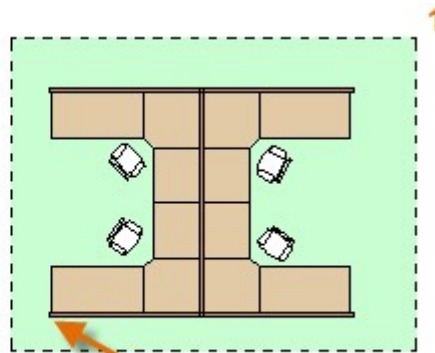


**For example**, this is how you could create a module for a cubicle design.

1. Create the objects for the block.
2. Start the BLOCK command.
3. Enter a name for the block, in this case, Quad-Cube.
4. Select the objects that you created for the block (click 1 and 2).
5. Specify the block insertion point.

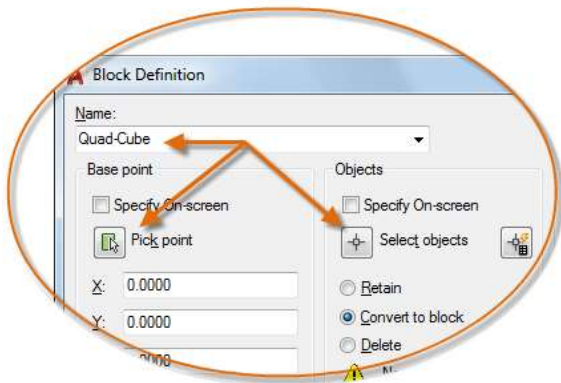


create the objects

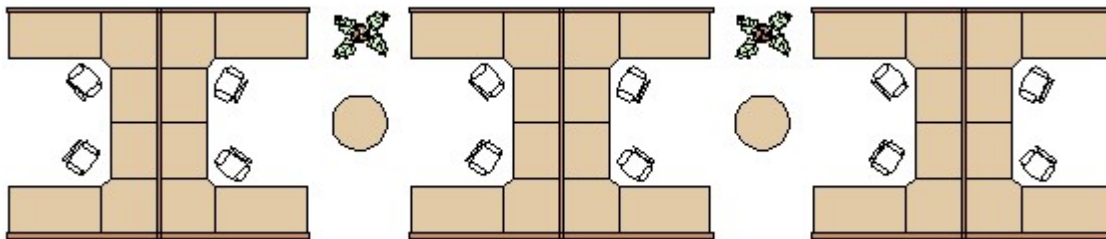


insertion point

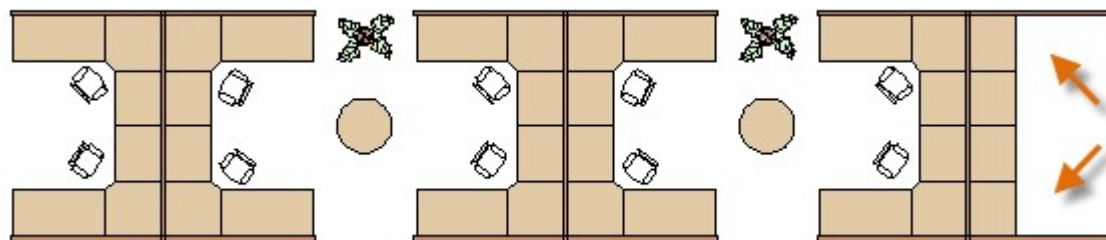
You can enter the information for steps 3, 4, and 5 into the Block Definition dialog box in any order.



After creating the block definition, you can insert, copy, and rotate the block as needed.



Using the **EXPLODE** command, you can explode a block back to its component objects if you need to make changes. In the illustration below, the cubicle on the right is exploded and modified.



In this example, you would probably create a new block definition from the objects in the exploded block.

Note: You can create block definitions that include one or more attributes that store and display information. The command that you would use is **ATTDEF**. Typically, attributes include data such as part number, name, cost, and date. You can export block attribute information to a table or to an external file.

### **Recommendations:**

There are several different schemes for saving and retrieving block definitions.

- You can create an individual drawing file for each block that you intend to use. You save these drawing files in folders, each of which would contain a family of related drawing files.
- You can include the block definitions for title blocks and common symbols in your drawing template files to make them available immediately when starting a new drawing.
- You can create several drawing files, which are sometimes called ***block library drawings***. Each of these drawings contains a family of related block definitions. When you insert a block library drawing into your current drawing, all the blocks that are defined in that drawing become available.
- 

**Tip:** With online access, you can download AutoCAD drawing files from the websites of commercial vendors and suppliers. This can save you time, but always check to make sure that they are drawn correctly and to scale.

## **Create a Dynamic Block:**

Dynamic blocks contain rules and restrictions that control the appearance and behavior of a block when it is inserted into a drawing or when it's later modified.

You can add these rules and controls to any existing block as well as using them when you create new blocks. The controls are limited to 2D operations only.





## Examples of Using Dynamic Blocks

A variety of controls and behaviors are available for adding flexibility and efficiency of working with blocks. Some examples of how you could use dynamic blocks include the following:

- Automatically align a block to the geometric objects that represent a wall or a pipe
- Create additional movement grips on a block
- Add multiple insertion points so you can cycle through them by pressing Ctrl when inserting a block
- Choose whether a block is flipped or mirrored when inserted
- Control the size or shape of a block to standard increments such as the length of a fastener or width of a door
- Display a family of parts or similar styles in a tabular format to choose different versions of the same block

If you notice an action that you commonly repeat when you work with blocks, you may be able to reduce the amount of trivial work and achieve substantial gains in efficiency.

## The Block Editor

The Block Editor is a dedicated block authoring environment that lets you create and modify block geometry, attributes, and parameters, actions, and constraints. It includes the following components:

- A drawing area with a gray background
- A contextual ribbon tab that contains operations such as saving and testing the block, and exiting the editor
- The Block Authoring palette with several tabs that contain tools to add dynamic capabilities to block definitions
- A temporary area to test a dynamic block design

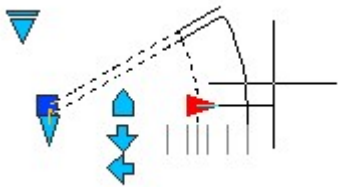
You can access the Block Editor by using the BEDIT command, which is available on the Home tab, Block panel on the ribbon.

Note: The third tab on the Block Authoring palette set, Parameter Sets, simply contains predefined parameters and actions that have already been combined for your convenience.



## Applying Parameter Sets

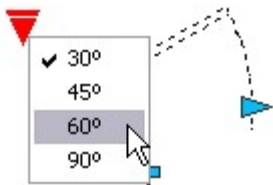
With dynamic blocks you can insert a single block that can change its shape, size, or display instead of inserting one of many static block definitions. For example, instead of creating multiple interior door blocks of different sizes, you can create a single dynamic door block that can be resized to a doorway opening.



In the previous illustration, there are several visible *parameters* that appear as specialized grips or controls, each of which is associated with an *action*. The highlighted custom grip is a *linear parameter* that's associated with a *stretch action*. Paired together, these are termed a *parameter set*.

**Tip:** The order in which you specify key points when creating a parameter determines the location of a grip or control, and may have additional significance.

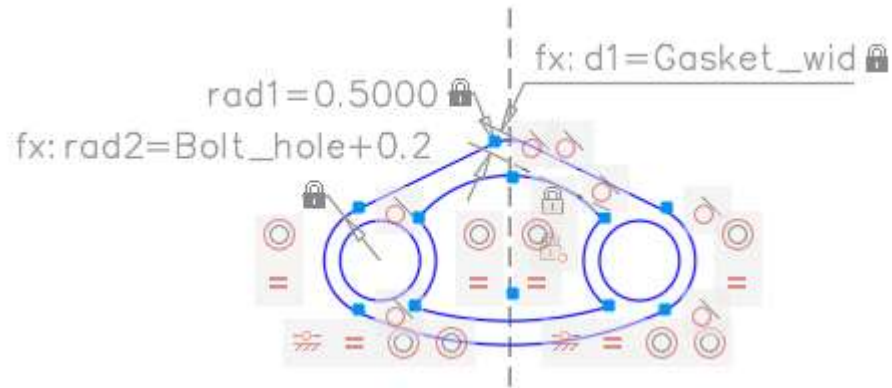
Here's an example of a lookup list that provides a choice between several options.



In this case, the highlighted control is a *lookup parameter* that's associated with a *lookup action*. This *parameter set* was designed to display a list of options to control the displayed swing angle of the door.






## Applying Constraints

The parametric drawing feature set, which includes geometric *constraints*, *dimensional constraints*, and *parametric formulas*, is available for use in blocks.





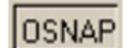


The ability to define constraints is not available in AutoCAD LT, but blocks that are defined in another AutoCAD-based product can be employed in AutoCAD LT without restriction.







## Introduction to AutoCAD Commands





Command	Keystroke	Icon	Menu	Result
Line	<b>Line / L</b>		<u>D</u> raw > <u>L</u> ine	Draw a straight line segment from one point to the next
Circle	<b>Circle / C</b>		<u>D</u> raw > <u>C</u> ircle > Center, <u>R</u> adius	Draws a circle based on a center point and radius.
Erase	<b>Erase / E</b>		<u>M</u> odify > <u>E</u> rase	Erases an object.
Print	<b>Print / Plot Cntl+P</b>		<u>F</u> ile > <u>P</u> rint	Enables the Print/Plot Configuration Dialog Box
Undo	<b>U</b> (Don't use 'Undo' for now)		<u>E</u> dit > <u>U</u> ndo	Undoes the last command.



Rectangle	<b>RECTANGLE / REC</b>		<u>D</u> raw > R <u>e</u> ctangle	Draws a rectangle after you enter one corner and then the second.
Multi Lines	<b>MLINE / ML</b>	No Icon	<u>D</u> raw > <u>M</u> ultiline	Draw parallel lines based on the parameters you define.
Trim	<b>TRIM / TR</b>		<u>M</u> odify > <u>T</u> rim	Trims objects to a selected cutting edge.
Extend	<b>EXTEND / EX</b>		<u>M</u> odify > Ext <u>e</u> nd	Extends objects to a selected boundary edge.
Offset	<b>OFFSET / O</b>		<u>M</u> odify > Off <u>s</u> et	Offsets an object (parallel) by a set distance.
Object Snaps	<b>OSNAP / OS / F3</b>	CLICK 	<u>T</u> ools > Object S <u>n</u> ap Settings	Brings up the OSNAP dialog box.





Command	Keystroke	Icon	Menu	Result
Move	Move / M		<u>M</u> odify > M <u>o</u> ve	Moves an object or objects
Copy	Copy / CP		<u>M</u> odify > C <u>o</u> py	Copies object(s) once or multiple times
Stretch	Stretch / S		<u>M</u> odify > S <u>t</u> retch	Stretches an object after you have selected a portion of it
Mirror	Mirror / MI		<u>M</u> odify > M <u>i</u> rror	Creates a mirror image of an object or selection set
Rotate	Rotate / RO		<u>M</u> odify > R <u>o</u> tate	Rotates objects to a certain angle
Fillet	Fillet / F		<u>M</u> odify > F <u>i</u> llet	Creates a round corner between two lines




Chamfer	<b>Chamfer / CHA</b>		<u>M</u> odify > <u>C</u> hamfer	Creates an angled corner between two lines
Array	<b>Array / AR</b>		<u>M</u> odify > <u>A</u> rray	Creates a repeating pattern of the selected objects
Layer	<b>Layer / LA</b>		<u>F</u> ormat > <u>L</u> ayer	Starts the Layer and Linetype property dialog box
Text	<b>Text</b>	No Icon	<u>D</u> raw > Single Line <u>T</u> ext	Creates a single line of text
Dimension	<b>Dim</b>	Many	<u>D</u> imension > ( <i>pick one</i> )	Dimensions previously drawn objects
Scale	<b>Scale / SC</b>		<u>M</u> odify > <u>S</u> cale	Proportionately resizes (or scales) objects






RIGHT TO LEFT	CROSSING SELECTION	GREEN (with a dotted outline)	SELECTIONS ANY OBJECT THAT EITHER CROSSES THE BOUNDARY OR IS INSIDE IT
LEFT TO RIGHT	WINDOW SELECTION	BLUE	SELECTIONS ON OBJECTS THAT ARE COMPLETELY WITHIN THE BOX

COMMAND OPTION	ICON	DESCRIPTION
Zoom <u>A</u> ll		This option causes AutoCAD to display the whole drawing as far as its drawing <u>limits</u> or drawing <u>extents</u> (whichever is the greater of the two).
Zoom <u>C</u> enter		This option requires two things: a point that is to be the <u>center</u> of the new display and a value to be its <u>new height in drawing units</u> . The existing height is the default for the new height to allow for panning across the drawing. If the new height value is followed by "X" (eg. 2x), then it is taken as a magnification factor relative to the current height. If followed by "XP", then it is taken as a scale factor relative to paper space and can be used for scaling the contents of paper space viewports.



Zoom <u>E</u> xtents		This option will display all the graphics that are contained in the drawing (referred to as the <i>drawing extents</i> ) with the largest image possible.
Zoom <u>P</u> revious		This option restores the displayed view prior to the current one. For the purpose of this option, up to 10 views are saved so that the last ten views can be recalled. This option includes every time you use the scroll bar, which is one reason to <b>avoid the scroll bars</b> for panning a lot in your drawing.
Zoom <u>D</u> ynamic		This is a very useful ZOOM option once it is understood. It permits very quick movement around the drawing. Once selected, this option redraws the graphics area of the screen and displays two rectangles. The larger box shows the extents of the current drawing. The smaller box shows the current view with an "X" in the middle. This moves with the mouse. This view box should be positioned so that its lower left corner is at the lower left corner of the view required. By pressing the left button on the mouse, the "X" is replaced by an ">" pointing to the right side of the view box. This allows you to change the magnification. As the mouse is moved, the view box shrinks and expands so that the size of the required view can be set. The left mouse button toggles between PAN "X" and ZOOM ">" mode so that fine adjustments can be achieved. When the view required has been selected, press <ENTER> or right click to cause AutoCAD to display it.



<p><u>Z</u>oom <u>S</u>cale</p>		<p>This is a 'hidden' default option. You do <u>not</u> have to type "S" to choose this option. It simply requires the entry of a number that represents a magnification factor. Note that the factor is applied to the entire drawing (as defined by the drawing's limits). Numbers less than 1 will reduce the displayed size of the drawing, while numbers greater than 1 will enlarge it. If "X" is inserted after the number (eg. 0.8x) then the factor is applied to <i>the current view</i>. If "XP" is inserted after the scale factor, then the view is scaled relative to paper space. This is useful for zooming a view within a paper space viewport to a specific scale, for example, "1/48XP" will produce a view of model space at a scale of 1/4" = 1' relative to paper space.</p>
<p><u>Z</u>oom <u>E</u>xtents</p>		<p>This option will display all the graphics that are contained in the drawing (referred to as the <i>drawing extents</i>) with the largest image possible.</p>
<p><u>Z</u>oom <u>P</u>revious</p>		<p>This option restores the displayed view prior to the current one. For the purpose of this option, up to 10 views are saved so that the last ten views can be recalled. This option includes every time you use the scroll bar, which is one reason to <b>avoid the scroll bars</b> for panning a lot in your drawing.</p>







Zoom  
Realtime




Zoom Realtime provides interactive zooming capability. Pressing <ENTER> (after entering zoom) on the command line automatically places you in Realtime mode. Hold the left mouse button down at the midpoint of the drawing and move the cursor vertically to the top (positive direction) of the window to zoom in up to 100% (2x magnification). Hold the left mouse button down at the midpoint of the drawing and move the cursor vertically to the bottom (negative direction) of the window to zoom out to 100% (.5x magnification). *You cannot zoom out beyond the extents of the current view.*



When you release the pick button, zooming stops. You can release the pick button, move the cursor to another location in the drawing, and then press the pick button again and continue zooming from that location. To exit Realtime Zoom mode, press <ENTER> or (ESC).

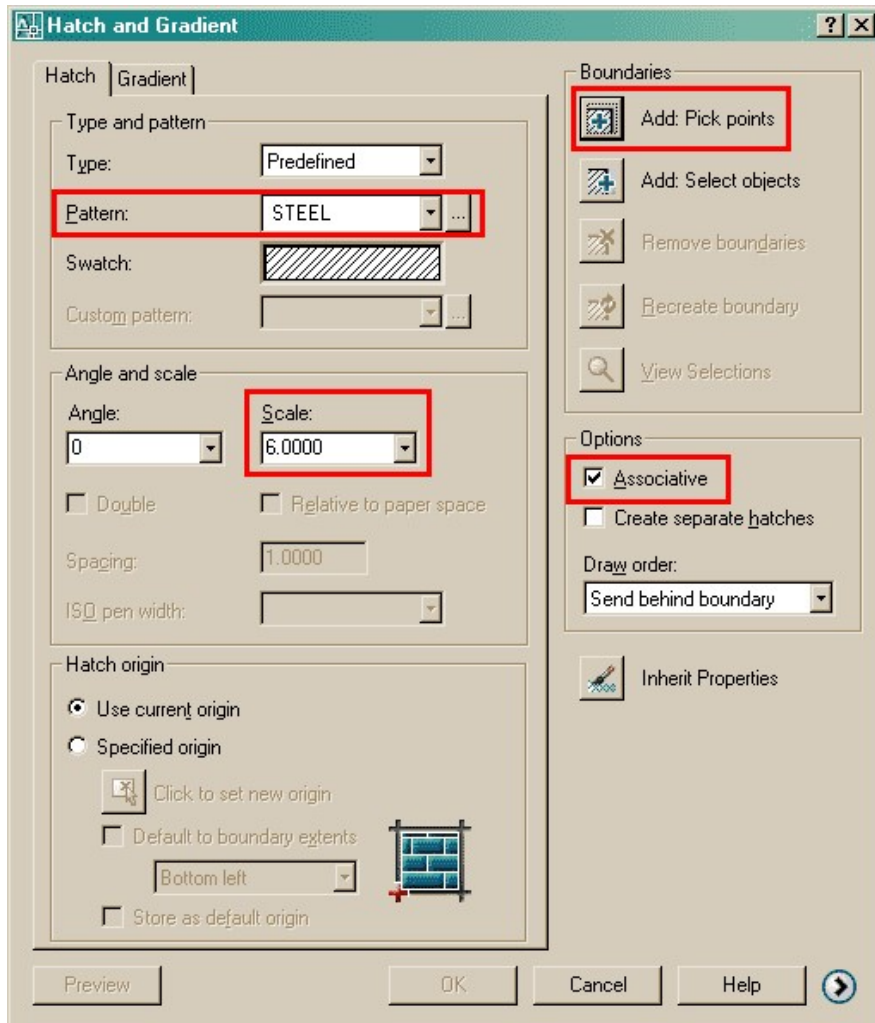
<p>Aerial View command: <b>DSVIEWER</b></p>	<p>None</p>	<p>Aerial View is a zooming tool that displays a view of the drawing in a separate window so that you can quickly move to that area. If you keep the Aerial View window open as you work, you can zoom and pan without choosing a menu option or entering a command. You can change the view by creating a new view box in the Aerial View window. To zoom in to the drawing, make the view box smaller by left clicking a rectangle. To zoom out of the drawing, make the view box larger. As you zoom in or out of the drawing, a real-time view of the current zoom location is displayed in the graphics area. The screenshot shows how the view box looks. Right click in the box and you can move the box to where you want to zoom to.</p>
<p><b>Z</b>oom <b>O</b>bject</p>		<p>This option asks you to select an object or objects, then press &lt;ENTER&gt; and the screen will zoom to those objects only. This is great for when you want to work on object.</p>
<p><b>Z</b>oom <b>I</b>n</p>		<p>Clicking this icon will zoom in to the drawing by about 50%. This option is only available as an icon and cannot be invoked by the command line.</p>
<p><b>Z</b>oom <b>O</b>ut</p>		<p>Similar to 'Zoom In' - this icon will zoom out of your drawing and allow you to see about 50% more of your drawing space.</p>
<p><b>Z</b>oom <b>W</b>indow</p>		<p>This option (also a 'hidden' default) prompts the user to pick two corners of a box on the existing view in order to enlarge that area to fill the display.</p>



















<b>Mouse Scroll</b>	-	If you have a scrolling wheel on your mouse, you can use it to zoom in and out of your drawing. Scroll towards you to zoom out and away from you to zoom in. You have the option to change the amount of zoom per wheel click with the Zoomfactor <b>system variable</b> . Keep in mind that you will zoom in and out using your mouse location as a 'centre point'.
<b>PAN</b>		Panning allows you to quickly move around the drawing area at the same magnification you currently have set. Type in <b>PAN</b> (or <b>P</b> ) <ENTER> and a hand will appear on the screen. Left click and hold to move around your drawing.




## Hatch:

Command	Keystroke	Icon	Menu	Result
Boundary Hatch	<b>Bhatch / H</b>		Draw > Hatch	Covers an area with a predefined pattern
Hatch Edit	<b>HatchEdit / HE</b>		Modify > Object > Hatch...	Edits an existing Hatch



ICON	SETTING		ICON	SETTING
	Endpoint			Perpendicular
	Midpoint			Tangent
	Center			Nearest
	Node			Apparent Intersection
	Quadrant			Parallel
	Intersection			None
	Extension			Osnap Setting
	Insertion Point			Temporary Tracking Point



	Insertion Point		Temporary Tracking Point
			Snap From

### 13. General Commands

In this massive list of AutoCAD commands, I have tried to include some of the most useful commands which every AutoCAD user must know. Some of the commands in this list are very basic commands which are used very frequently and also there are commands which are not often used despite their great features.

General Commands		
1	BCOUNT	To count number of blocks
2	TXTEXP	To explode text
3	XLINE(XL)	To make infinite construction lines
4	POINT(PO)	For making points



## General Commands

5	REVCLOUD	For makirevisionsion cloud
6	SKETCH	For making freehand sketch
7	MULTIPLE	To repeat a command multiple times without pressing enter
8	NCOPY	To copy nested objects from Blocks and xRef
9	'CAL	To make calculations on the command line
10	BURST	Explodes attributed block keeping attribute value retained
11	SP	Spelling check-in AutoCAD
12	ARCTEXT	To make arc aligned text
13	OOPS	Restore the last deleted object in drawing
14	CHSPACE	To change space from model space to paper space and vice versa

## General Commands

15	DIVIDE	To divide an object into an equal number of segments
16	RENAME	To rename named objects in AutoCAD
17	BREAK	To break an object into two or more segments
18	TIME	Time-related information on the drawing
19	QSELECT	For making intelligent selection sets from your drawing
20	DI	To find the distance between two points
21	COPYBASE	To copy an object from a base point
22	PASTECLIP	To paste the object from the clipboard to your drawing
23	LIST	To list most of the properties of a selected object
24	DIM	Single command to do most of the dimensioning tasks

## General Commands

25	TOLERANCE	Command for entering feature control frame in the drawing
26	SCALETEXT	To change the scale of existing Text or Mtext
27	BASE	To change or assign a base point in the drawing
28	DWGPREFIX	To find the location of the saved drawing
29	TJUST	To change the justification of Text and Mtext
30	UNITS	To set the display units of the drawing
31	ML	For making multiline in AutoCAD
32	TINSERT	To insert a block or drawing in an AutoCAD table
33	MIRRTXT	To make a mirrored copy of the f Text
34	AREA	To find the area of a closed space

### General Commands

35 TORIENT To orient selected text entities in the best readable position

36 ID To find the coordinates of a point

37 MINSERT Insert blocks as a rectangular array

38 BOUNDARY Extract boundary from closed regions of the drawing

39 DIMROTATED For making a dimension line inclined to a particular angle

40 BREAKLINE For making a breakline symbol on the line

41 TXT2MTXT Convert text to Mtext

42 SAVEALL To save all open drawings

### Performance-based commands

43 PURGE To remove all unused objects from drawing

## General Commands

44	LAYDEL	To delete stubborn layers
45	FILLMODE	To toggle visibility of filled areas like a hatch and solid fill
46	APERTURE	To change the influence region of the object snap
47	OVERKILL	To remove redundant objects from drawings
48	ISAVEPERCENT	Change the time taken by AutoCAD to save the drawing
49	SELECTIONPREVIEW	Show or hide highlight preview when hovering the cursor over object
50	HIGHLIGHT	Show or hide highlight when the object is selected
51	RECOVERALL	Repair a drawing and it's attached xRef's
52	LAYMCH	To change the layer of one object with reference to other
53	PICKBOX	Change the size of the picked box which appears on the cursor

## General Commands

54	STARTMODE	To show or hide the start tab
55	CLOSEALL	To close all open instances of the drawing
56	SELECTIONCYCLING	To cycle your selection between different overlapping objects
57	AUDIT	To remove errors from the drawing
58	RECOVER	To repair a drawing
59	MA	To match properties of one object with other
60	CLOSEALLOTHER	To close all open tabs except the current one
61	TEXTFIT	To fit the text within the selected region
62	WIPEOUT	To create a wipeout object that cleans the underlying area
63	TEXTMASK	To create automatic masking behind the text to make text legible

## General Commands

64	LAYMRG	To merge multiple layers into one
65	DS	To open the drafting settings window
66	LAYWALK	To create a layer walkthrough in your drawing
67	CENTERMARK	To add a center mark on a circle or arc
68	CENTERLINE	To add a centerline between parallel or intersecting lines
69	TABLET	To use a tablet as a pointing device instead of a mouse

## Drawing Management

70	TEXTSCR	To open the history of commands in a text window
71	BATTMAN	To modify attribute information of blocks
72	3DCONFIG	Used to change graphics related properties

## Drawing Management

73	SEEK	To open an online CAD and BIM object library
74	FS	Making selections with reference to the connected objects
75	TEXTTOFRONT	To bring text, Dimensions, and Leaderlines on the top of every underlying object like hatches
76	ISOLATE	To hide all objects from drawing except the selected one
77	HIDEOBJECTS	To hide only selected objects
78	GROUP	To make a group of many dissimilar objects
79	ADCENTER	To open the design center palette
80	IMPORT	To import drawings of different cad formats into AutoCAD
81	FIELD	To add Fields in the drawing
82	WBLOCK	To export certain parts of the drawing as an external file



## Drawing Management

### 3D Commands

83	THICKEN	To change a surface into a solid by adding thickness
84	CONVTOSOLID	To change a watertight surface or 3D mesh to solid
85	DELOBJ	Control the behavior of 2D curves when 3D tools are used
86	VOLUME	To find the volume and other solid related information
87	REGEN3	To remove graphics related anomalies from drawing
88	SECTIONPLANE	To create a section plane on a 3D
89	FLATSHOT	To create 2D views from 3D solid
90	INTERFERE	To check the interfering parts among mating solids
91	PROJECTGEOMETRY	To project a 2D curve on a 3D surface

## Drawing Management

92	Shell (SOLIDEDIT)	To create a shell or hollow shape
93	SURFPATCH	To create a closed patch on the open part of the solid
94	HELIX	To create a helix
95	Moveface (SOLIDEDIT)	To move faces like a hole or fillet
96	SURFTRIM	To trim surfaces
97	SURFOFFSET	To create offset surfaces
98	IMPRINT	To imprint 2D geometry on planar 3D objects
99	SOLIDHIST	To record the history of solids on which boolean operations are performed.
100	3DALIGN	To align a 3D object with respect to another 3D object

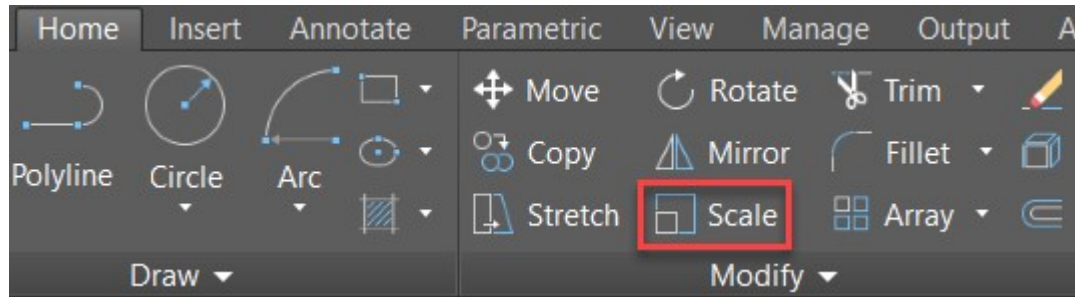
## SCALE

### Simple Scale command

Using the scale command you can change the size of an object in AutoCAD. To explore a scale command, I will use this simple window block which has the size of units unit along the X-axis and 15 units altogether along the Y-axis.



Select the scale tool from Modify panel of the Home tab or use its command SC.



Now select the window drawing and press enter. Now you need to select the base point. The base point will remain fixed when the object is scaled, in this case, I am using the lower-left corner of the window as the base point.

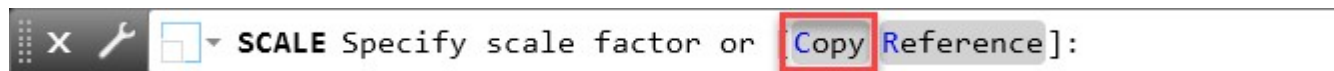
The command line will now prompt you to specify the scale factor. This scale factor determines the final size of your window.

A scale factor of 0 to 1 will decrease the size of your object (in this case window). For example, a scale factor of 0.5 will reduce the overall size of your drawing to half of its original dimensions and a scale factor of 0.25 will decrease the size of the object to one-fourth of the original size.

Similarly, a scale factor greater than 1 will increase the size of the drawing. For example, a scale factor of 2 will increase the size to twice its original value and a scale factor of 3 will increase the size three times.

Enter the scale factor and press enter. In this case, I am using a scale factor of 2, the final size of the object will become 20 units along the X-axis and 30 units along the Y-axis.

**Additional tip:** If you want to keep the original object even after changing the scale then click on the “Copy” option from the command line when prompted for the scale factor.



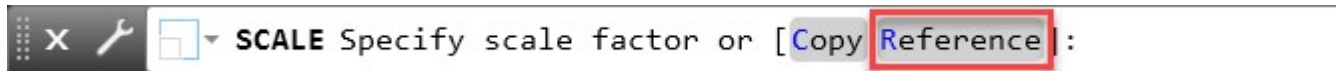
### Scale using reference

Consider this scenario, we have the window used in the previous example with a length along the X-axis of 10 units and a height of 15 units. Now we want to change the size of this window so that the length along the X-axis now becomes 1 unit and the height should change proportionally.

Now, in this case, you can use the “Reference” option of the scale command to change the size of the object.

Once again start the scale command and then select the window and press enter. Now specify the lower-left corner of the window as the base point, you can use any other base point as well if you want.

Now the command line will prompt you for scale factor, on this prompt click on the Reference option from the command line.



Now the command line will prompt you to specify the reference length. Type 10 and press enter, this is the current length of the window along the X-axis.

Now command line will prompt you to specify the new length which you want for the 10-unit line of the window. In this case type 17 and press enter again.

Now the base of the window will become 17 units in length and the height of the window will automatically change proportionally.

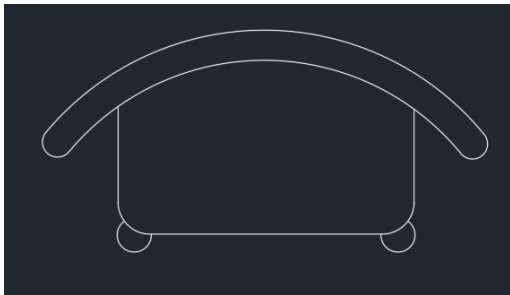
As you can see, this command is quite useful if you want to change the size of an object with reference to other existing objects or any reference length and in this case, you obviously don't need to specify the scale factor.

### Scaling Blocks

The scale commands explained above will work for block references too and you don't need to use any special property to scale blocks. However, you can scale your blocks with different scale factors along the X, Y, and Z-axis which I will explain here.

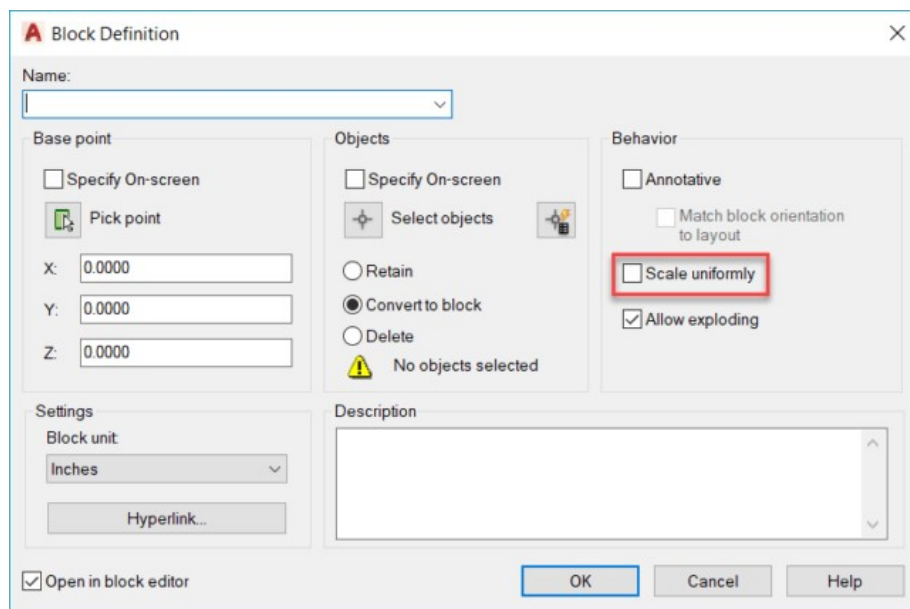
I will use this drawing of a simple chair to explain this.





First, we need to convert this simple object into a block and for that, you can use the B command. This article explains more about [making a block in AutoCAD](#).

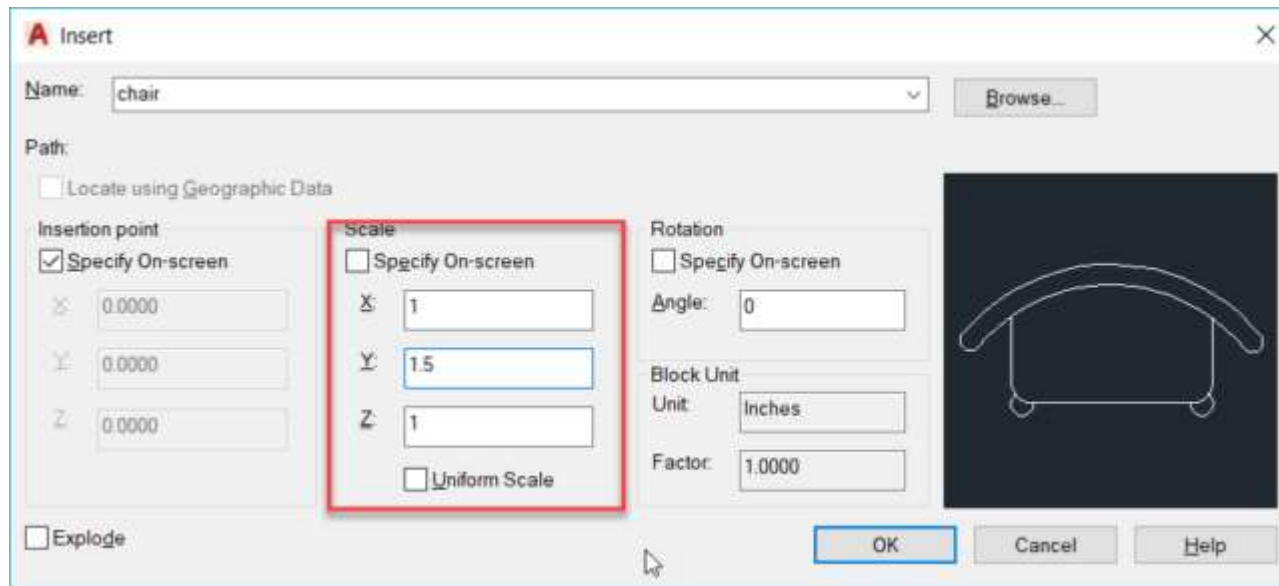
Just make sure that while creating the block you keep the “Scale Uniformly” box **unchecked** as shown in the image here.



Once you have your block prepared you can change the scales and then insert them into your drawing. To insert the block the e I command.

Type I and press enter, and the Insert window will show up. In this window, you can assign the scale of the block the on X, Y, and Z-axis separately. If you keep the value of the scale the actor samthe e along the X, Y, and Z-axis then the overall size of the block will change.

Let us use the scale factor of 1 along the X-axis and 1.5 along the Y-axis, I will keep the scale along Z-axisaxis as 1.



Now click OK and click at a point to insert the block. The block will look like image B shown below where image A is the original block at the overall scale of 1.



As you can see the block, in this case, will be scaled to a factor of 1.5 only in the Y direction and not in any other direction.

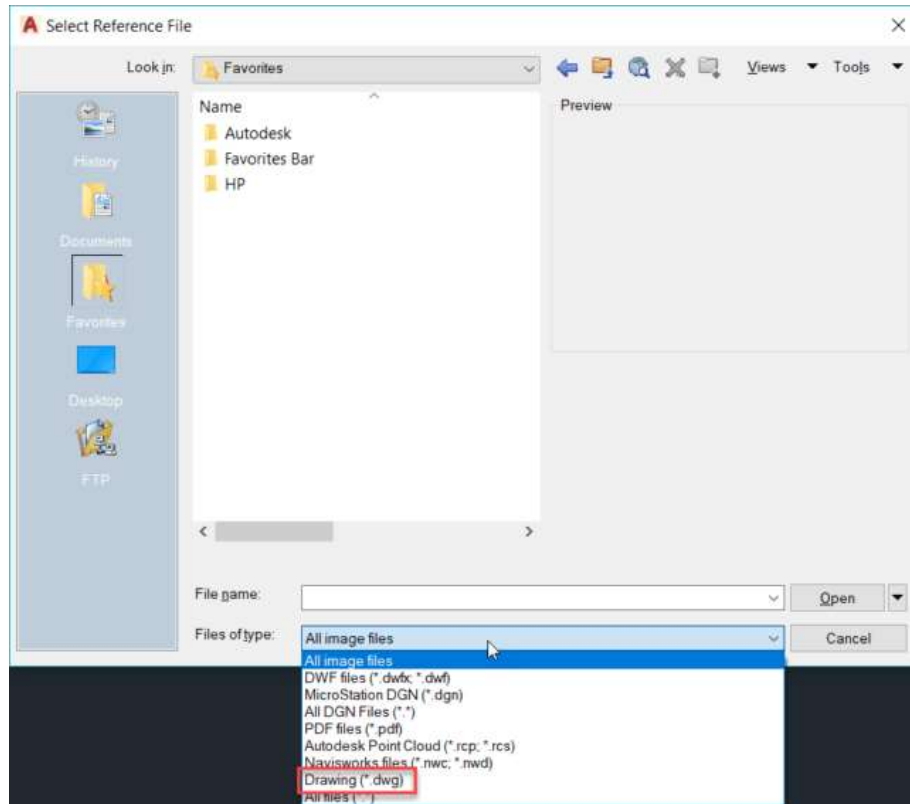
#### Scaling Xrefs and images

Xrefs or external references are entities like DWG files, PDFs, or images that can be inserted in a drawing as a reference. You can scale your Xrefs while inserting them or you can change their size even after inserting them into the drawing.

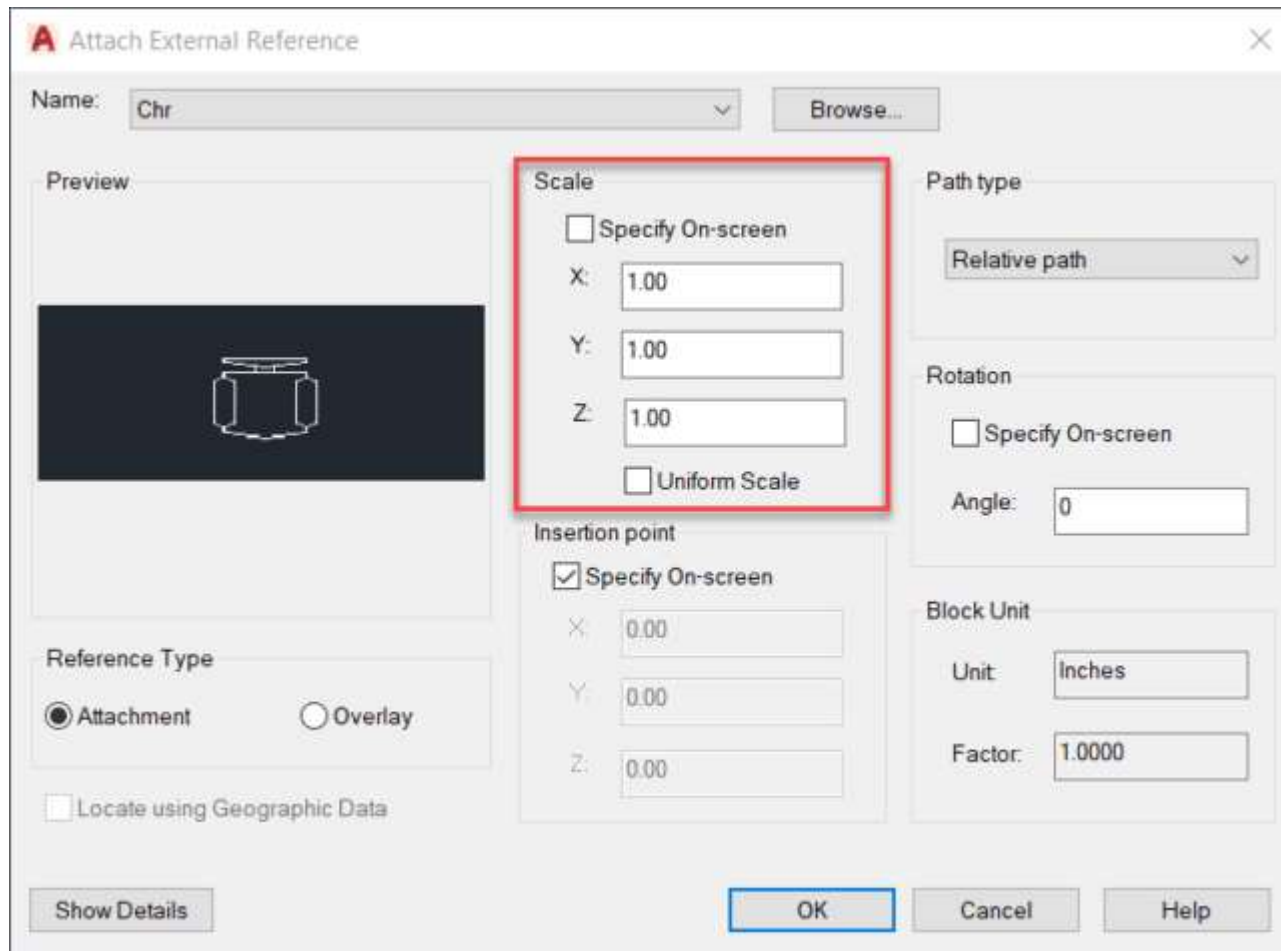
For this example, I will use a DWG file, but you can use the same method for PDF files and Images.

Type ATTACH and press enter, the attach window will open up. Select DWG from the format dropdown as shown in the image below. Now select the location where your file is saved then select the drawing and click on open.

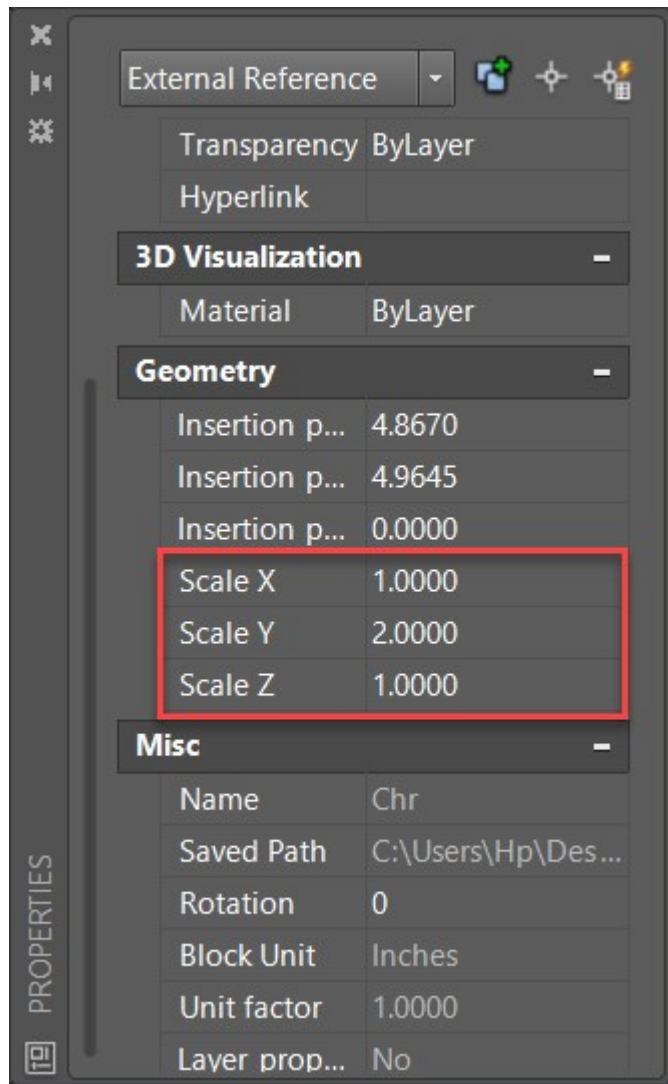




The Xref window will open up where you can assign different scales for different axes and you can even keep it at 1 for all the axes to keep the original size of the attached Xref.



Even after adding the Xref, you can change its scale using the property palette. Just select the Xref from the drawing area then type PR and press enter. The property palette will open up, scroll down in the palette and change the scale from the geometry tab of the property palette.



You can also use the scale command to change the overall scale of the attached Xref.

Similarly, you can change the scale of attached images and PDF files. Here I will attach an image of the floor plan to explain it.

Type ATTACH and press enter then select an image and click OK. Specify a point in the drawing to insert it and press enter to accept the default scale of 1.

After inserting the image into my drawing, it looked like this.

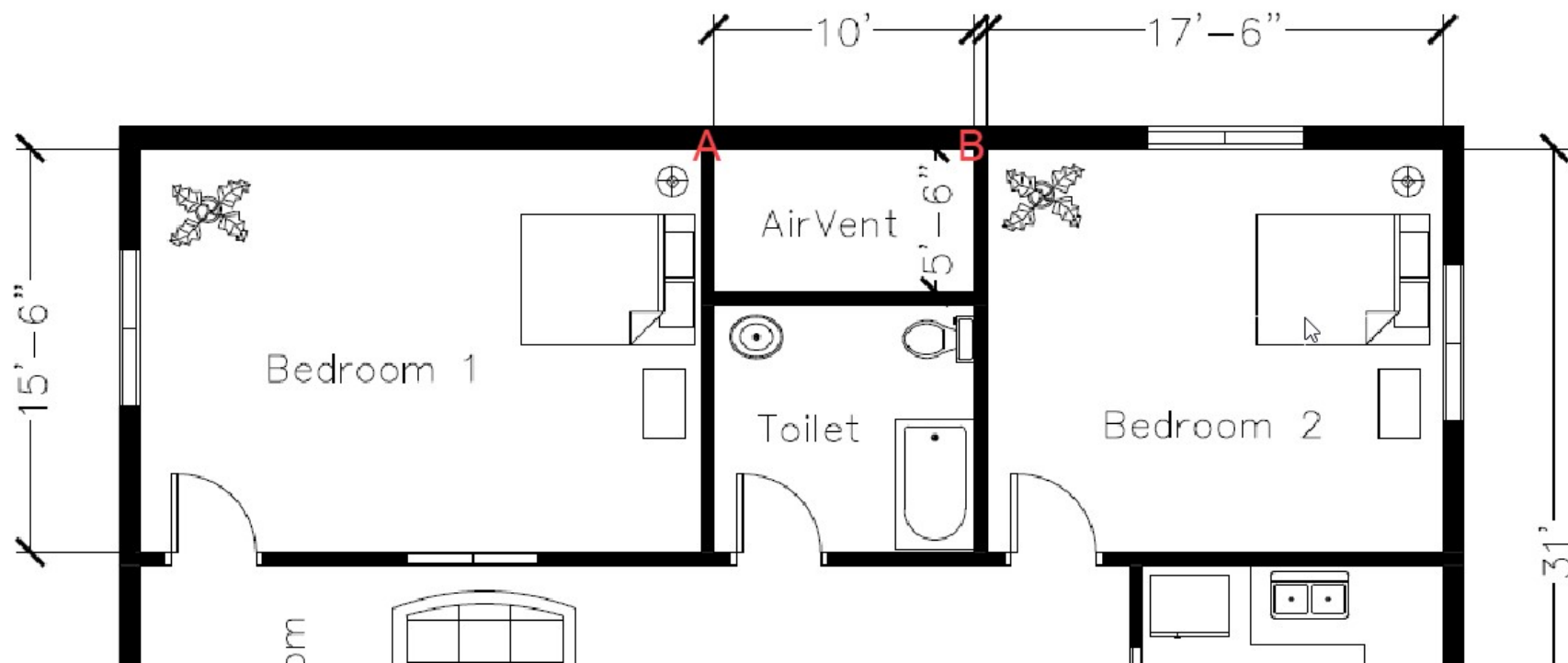


This image is not properly scaled i.e. the 10' lengths on the image are not actually 10' on the drawing. To fix this we need to scale this image again with reference to any existing dimension in the floor plan image.

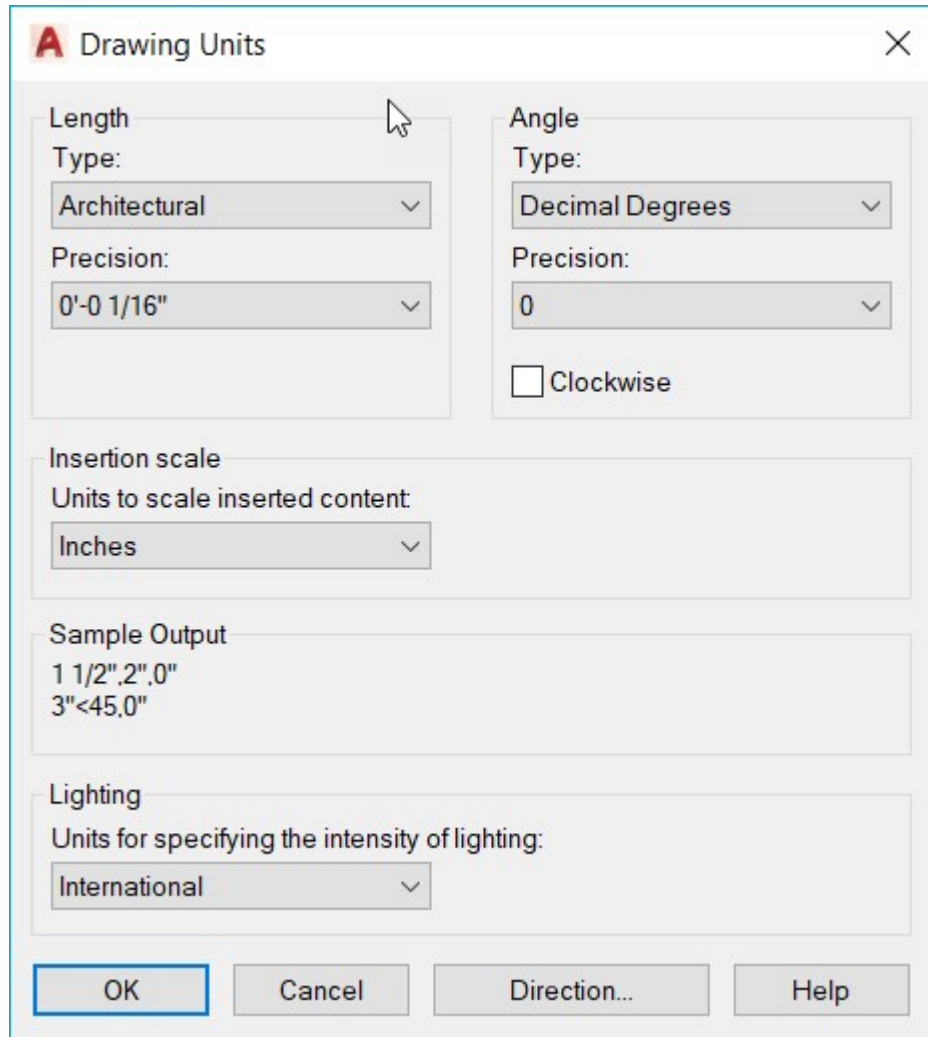
Let's take the reference of 10' length which is on the top of this floor plan.

Type SC and press enter to start the scale command. Now select the floor plan image and press enter again. Click on the top left corner of the image as the base point and then select the reference option from the command line.

Now click at point A than point B (refer to the image below) to specify the reference and type 10' and press enter again. Now the distance AB on the image will change to 10' and other distances will adjust accordingly.



In this case, I have used Architectural units so make sure you change the unit to architectural before using the scale option in this example. To change it to Architectural type UN press enter and then select Architectural the rom Length drop-down as shown in the image here.

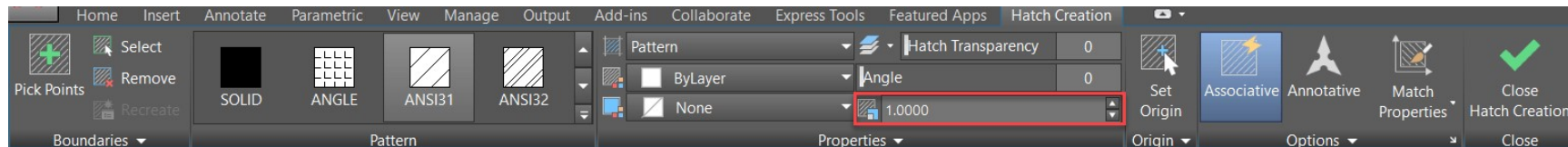


### Scaling Hatches

When you add hatches in any closed area it may appear like a solid pattern, or it may not appear at all. Often this happens because of the improper scale of hatches which can be easily adjusted from the hatch creation panel to make them look right.

Start hatch command from the Draw panel or use its command H. A temporary hatch creation tab will show up. Select the hatch pattern and then specify the area where you want to add the hatch.

If the size of the hatch pattern is very small or very large, then change its scale from the scale field of the properties panel as shown in the red box of the image below.

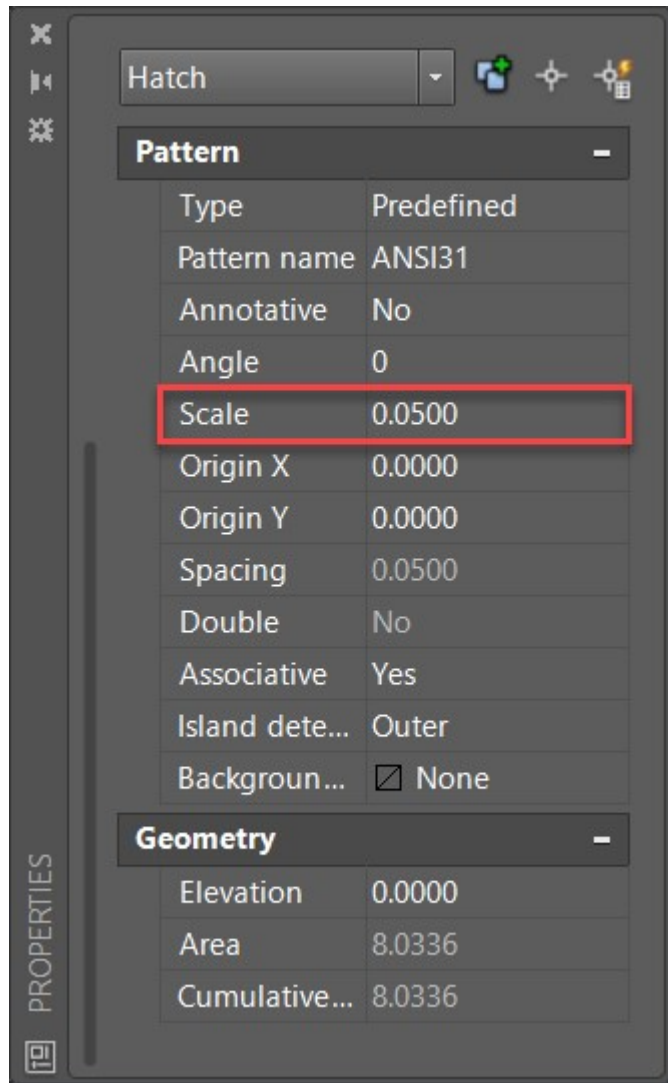


If the pattern is very small, then you need to increase the value in the scale field and if the pattern is widely spaced then you need to decrease the scale value. Here is how my ANSI 31 hatch pattern looks on a scale of 1, 0.05, and 50.



Even after adding the hatches if you select them again, it will open the hatch creation tab from where you can again modify the hatch scale or other properties.

You can also change the hatch scale from the property palette, just select the hatch pattern, then type PR and press enter. This will open the property palette, scroll down to the Pattern section and change the scale as shown in the image below.

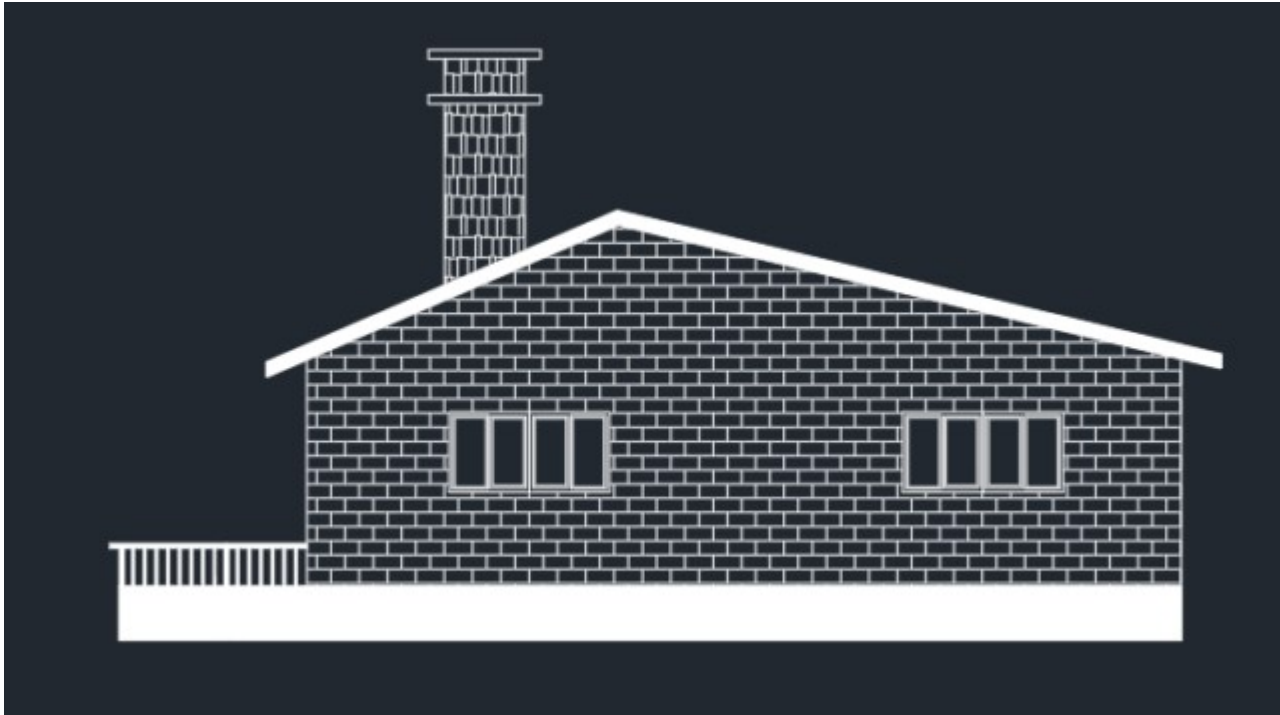




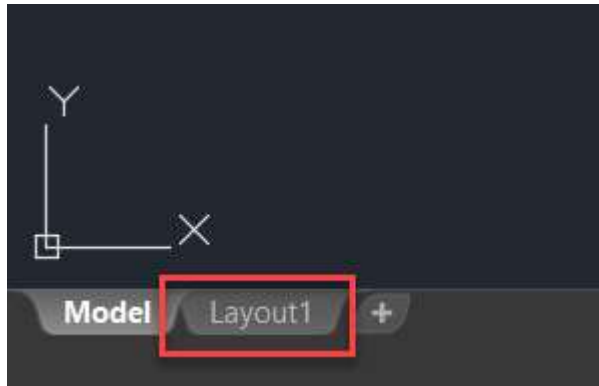
## Scaling Viewports

Drawings are often made in model space but text, title blocks, and other required information related to the drawing are generally added to the layout or paper space. In this section, I will explain methods of creating viewports in paper space and changing their scale to fit the drawing.

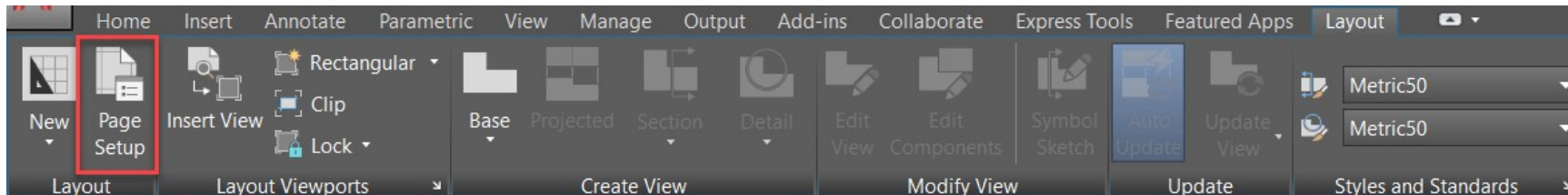
I am using this simple elevation drawing to explain this feature.



This drawing is in model space but we will insert this one in viewports of different scales in the layout. Start by selecting Layout 1 from the bottom left tabs, you can also hit the + icon to make a new layout tab.



The layout space will open up with a default viewport and your model space drawing will be automatically added to it. Delete the existing viewport and then click on the Page Setup option from the Layout tab as shown in the image here.



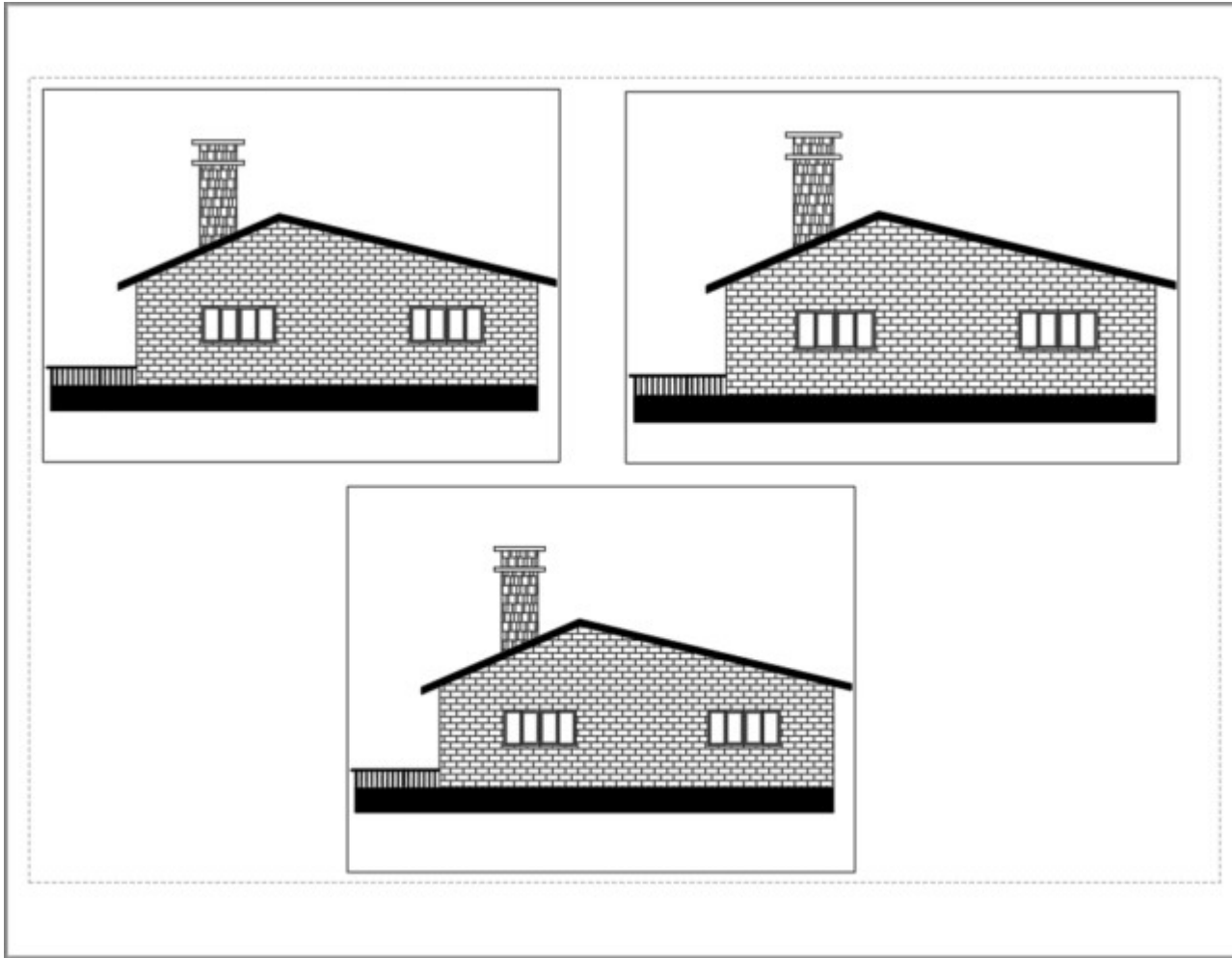
Click on Modify button in the page setup manager window and change the settings of the layout like printer/plotter and page size. I have selected DWG to PDF.pc3 in the plotter and ANSI A as the paper size.

After making these changes click OK to exit all open windows. Now we are all set to make our viewports.

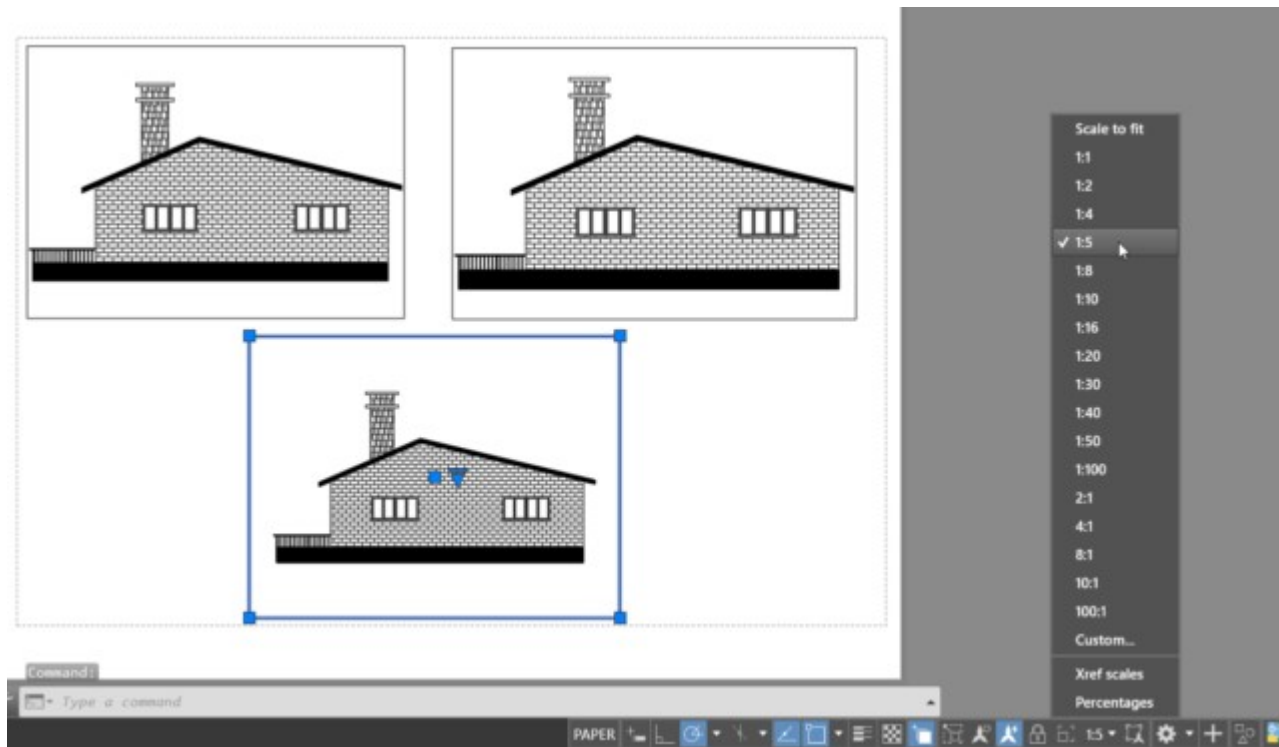
Type `-VPOR`Ts and press enter (don't forget to add the dash before vports). Now click at a point close to the top left of the layout well within the plotter margin and then click at another point to make a rectangular viewport. Repeat the command and make two more rectangular viewports in the same layout view.

After making all the viewports your layout space should look like this.



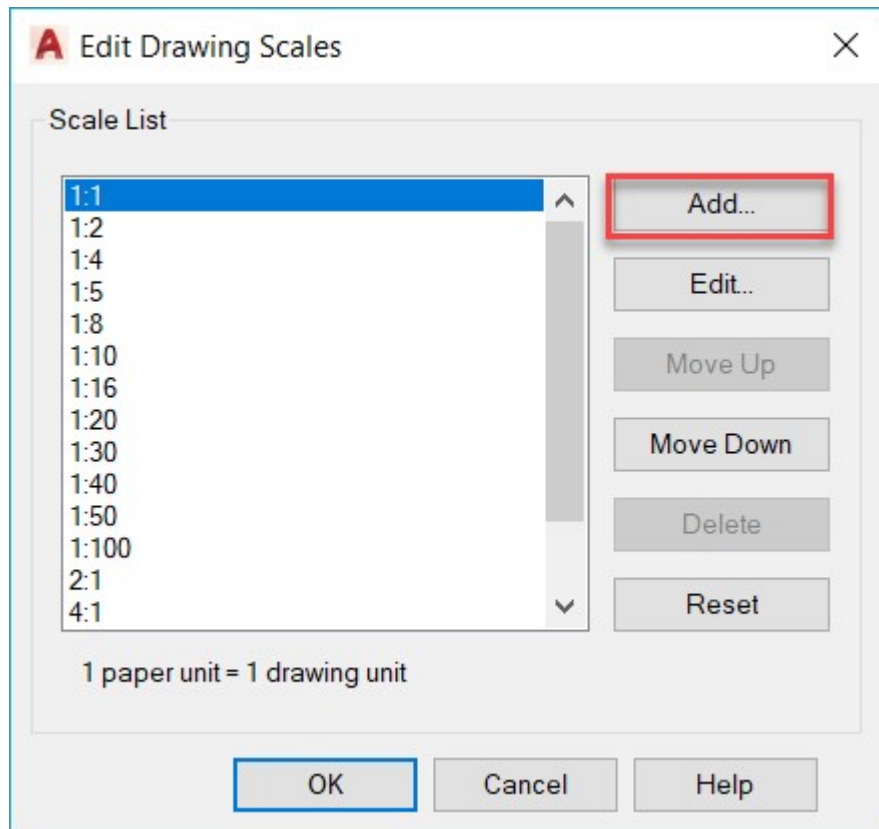


Currently, the scale of drawing inside the viewport is automatically set by AutoCAD but you can change it manually. To change the scale select the viewport boundary then change the scale from the scale option of the status bar as shown in the image here.



If the scale which you want to use is not available in this list of viewport scales then you can add your custom scale too. To add the custom scale select any viewport then click on the scale option from the status bar as shown in the image above and select the Custom option from the list of scales.

The “Edit Drawing Scales” window will open up. Let’s assume that we want to add a scale of 1:6 in this list, for that click on add button.

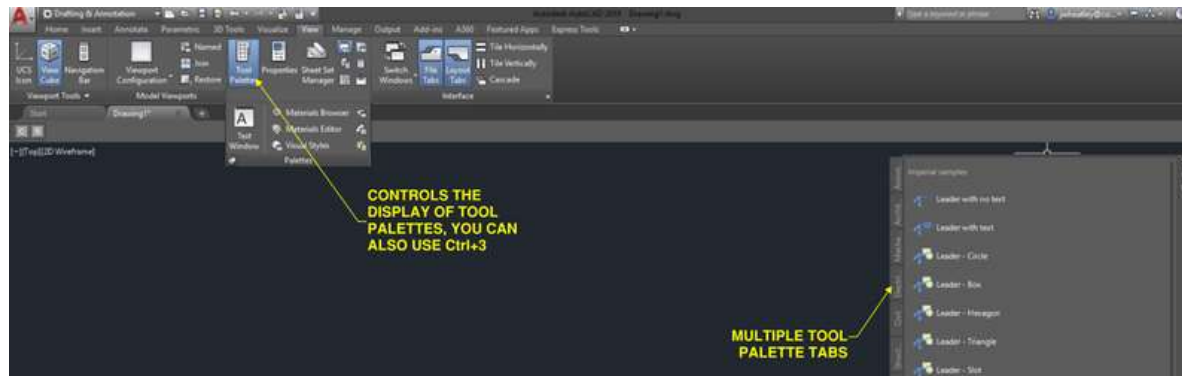


In the name field of the next window add 1:6 and then add 1 in the paper unit and 6 in the drawing unit field. Click OK to accept the changes and close all open windows.

You will notice that your new scale is now added to the list and you can now use it in your viewports. You can also watch the video shown below to know more about scaling drawings in paper space or layouts.

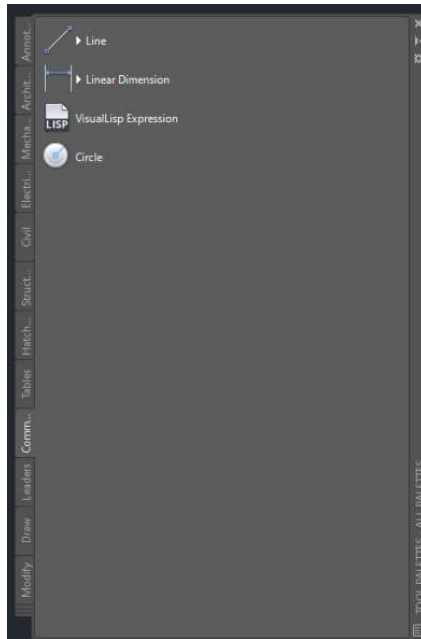
## Tool Palettes: Exploring the Features and Benefits

Tool palettes provide you with a combined method of accessing tools. Whether you prefer ribbons or toolbars, tool palettes provide the best of both worlds. Tool palettes can consist of commonly used blocks, hatch patterns, and commands. AutoCAD comes with many default tool palette categories that can be expanded or customized, allowing you to create one for your own needs. As with other palettes, you can dock the tool palettes or have them float within your workspace or even another monitor. Tool palettes can also be shared by exporting/importing to/by other users and can be grouped like directories to provide you with even more flexibility and organization.



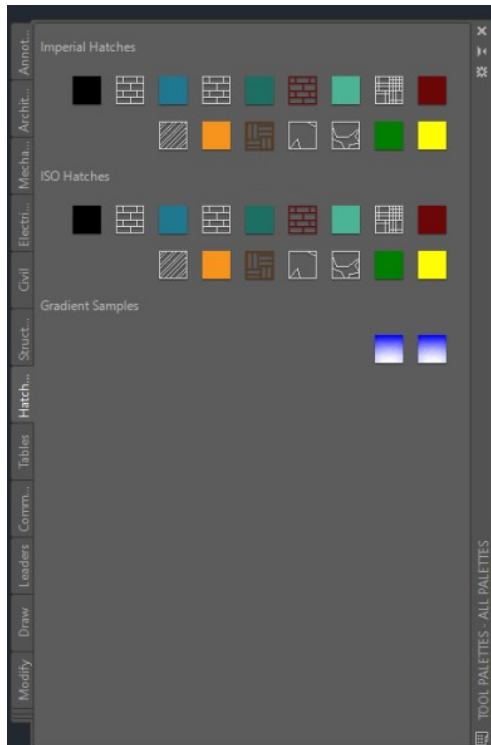
### Commands

One type of tool palette is frequently used commands. Sure, there are *many* ways and locations to access commands: You can use the Command Window, ribbons, pull-down menus, the CUI dialog box, and more. However, creating a tool palette of commands allows you to store some of your most-used commands in one location. You can even create flyout commands that combine multiple commands from one location and manipulate properties (and even images) associated with commands.



## Hatching

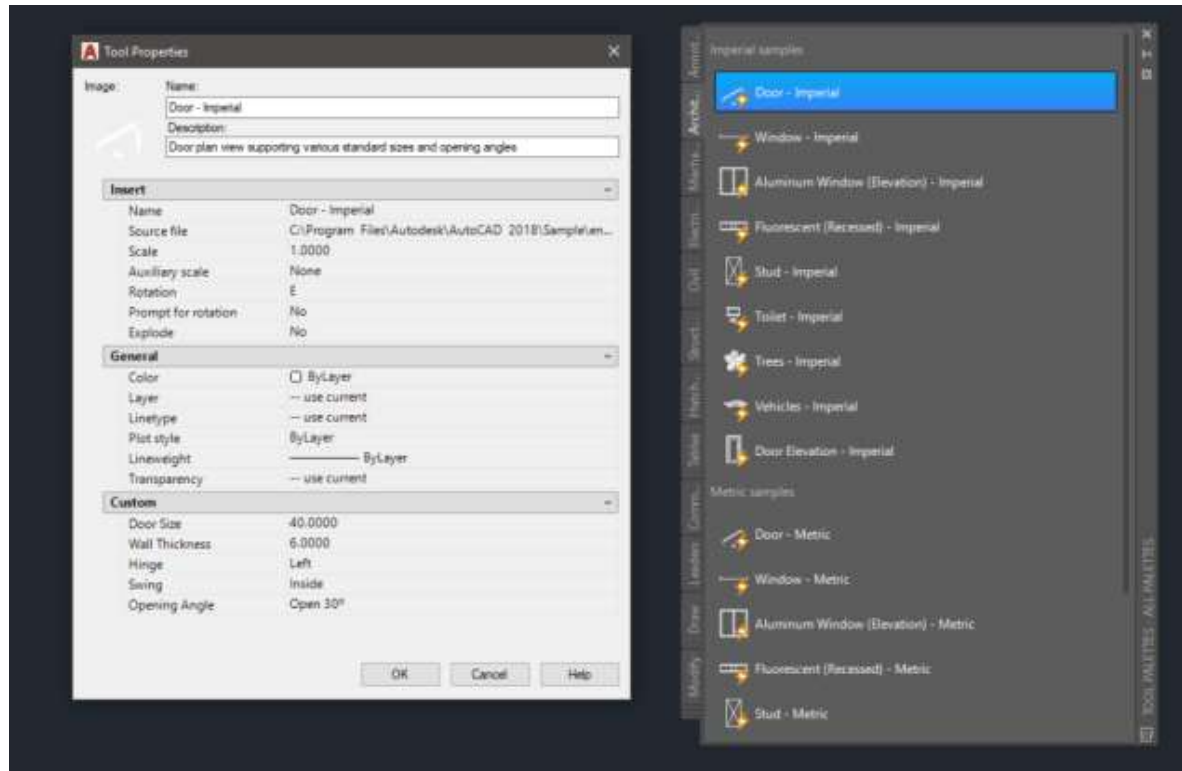
Hatch patterns can also be saved as a palette. Whether you need a predefined pattern or gradient hatch, simply select the hatch pattern you want and drag it inside the area you wish to hatch. Again, you can modify properties for each pattern. For example, if you are hatching areas within a plan view and have multiple patterns and scales, you could run the hatch command and modify your settings each time you changed a pattern, adding many additional clicks to achieve what you need. With tool palettes, create the pattern once, save it as a palette, and then drag and drop it to other areas. This cuts down on how many clicks of the mouse you have to make, making you more productive.



## Blocks

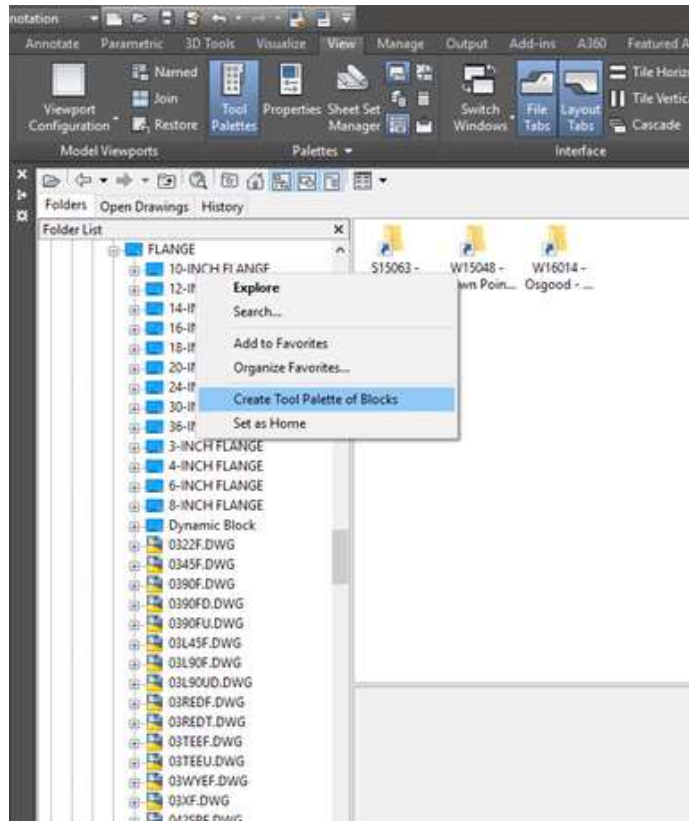
The most common use for tool palettes is blocks. Instead of searching through multiple directories or libraries, you can create palettes and palette groups for each set of blocks. The main advantage of tool palette blocks is the property options. For example, if a palette is created consisting of multiple types of pipe fittings, I can establish within the palette property to insert each fitting as a block or Xref. You can establish a set scale, prompt to rotate, explode, etc. These options are held within each block property option. However, the properties set in the tool palette do not affect the block itself. This allows you to customize blocks as you need to, but will not affect other users.





## Creation of Tool Palettes

One of the many benefits of tool palettes is how easy they are to create and organize. When you have libraries of blocks that you have created, use the Design Center to create a tool palette in one step. As shown in the image below, simply go to the directory with your blocks, right-click, and choose to create a tool palette. As you get more familiar with creating tool palettes, they can also be customized and grouped, giving you even more flexibility.



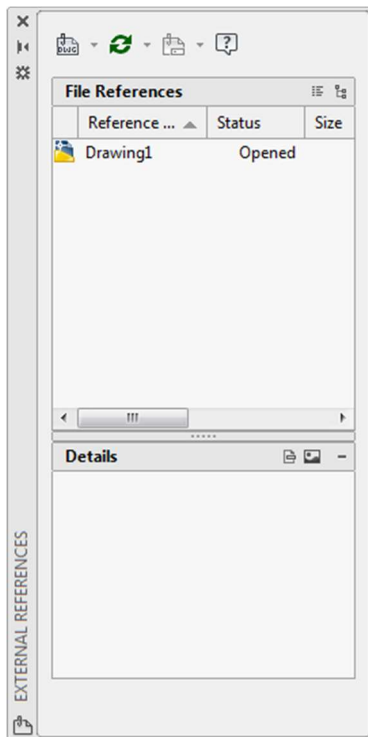
For many of us who use AutoCAD, the main objective is to produce accurate, precise drawings and models as quickly and efficiently as possible, and tool palettes are a great way to do just that. Tool palettes give you a proverbial one-stop shop for commonly used features within AutoCAD. Learning the core functions is just the beginning. See the reference links below, to gain more knowledge and understanding of why tool palettes should be used within AutoCAD.

## External Reference ("Xref")

"External Reference" ("Xref") is just fancy AutoCAD-speak for a document you attach to a drawing. When you Xref a file into your drawing, you are doing the digital equivalent of what drafters did decades ago: laying a piece of tracing paper (your drawing) over an existing site base, planting plan, or any other document you would need as a basis for your site plan.

You can Xref several document formats into a CAD drawing. Perhaps the most common format for Xref is another DWG file. (As you likely know, DWG is the file format of CAD drawings.)

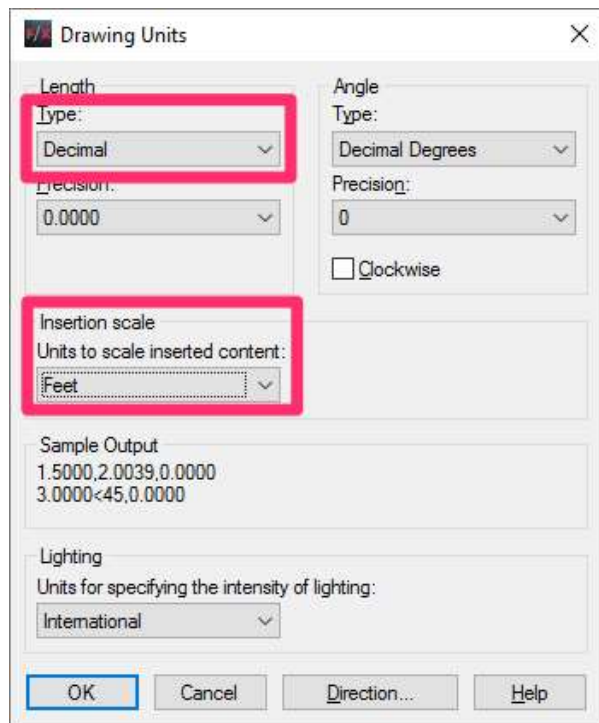
To begin working with Xrefs, type Xref in the Command line and press **Enter** to open the Xref Manager (pictured throw).



**Note: Before Attaching an Xref: Make Sure the Units are Set Correctly.**

For an Xref to place and scale correctly with your main drawing, *the Xref's Units setting* needs to match the units and system of measurement (e.g., Architectural Inches, Decimal Feet, etc.) *in which the Xref was originally drawn.*

To verify the unit setting, open the Xref. Type Units in the Command line, and press **Enter**.



In the **Drawing Units** dialog box, make sure the unit **Type** and **Insertion Scale** settings match the units and measurement system in which that file was drawn.

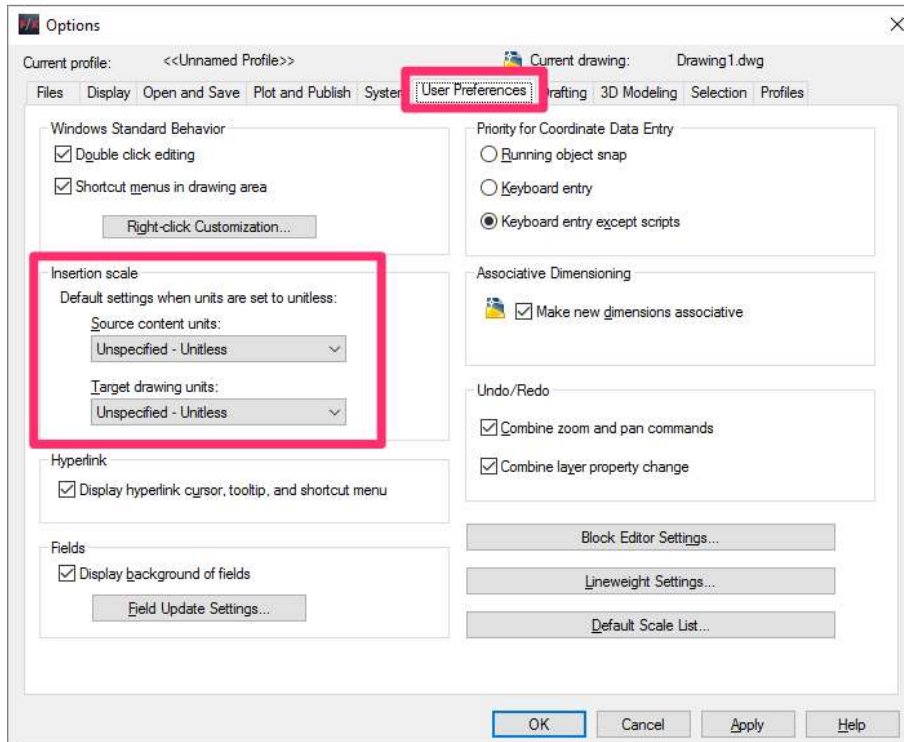
For example, if it was drawn in Imperial units in Decimal Feet, the **Type** menu should be set to **Decimal**, and the **Insertion Scale** menu should be set to **Feet**.

## The Unitless Setting (Options Dialog Box)

We highly recommend that you set both **Insertion Scale** menus to **Unspecified - Unitless** in the **User Preferences** tab of the **Options** dialog box.

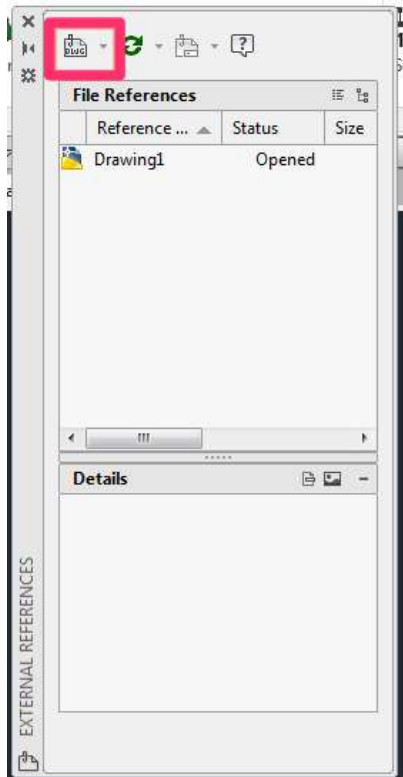
(Open the **Options** dialog box by typing Options in the Command line and pressing **Enter**.)





This option will prevent scaling issues should you need to Xref a drawing that was produced with the units set to **Unitless**

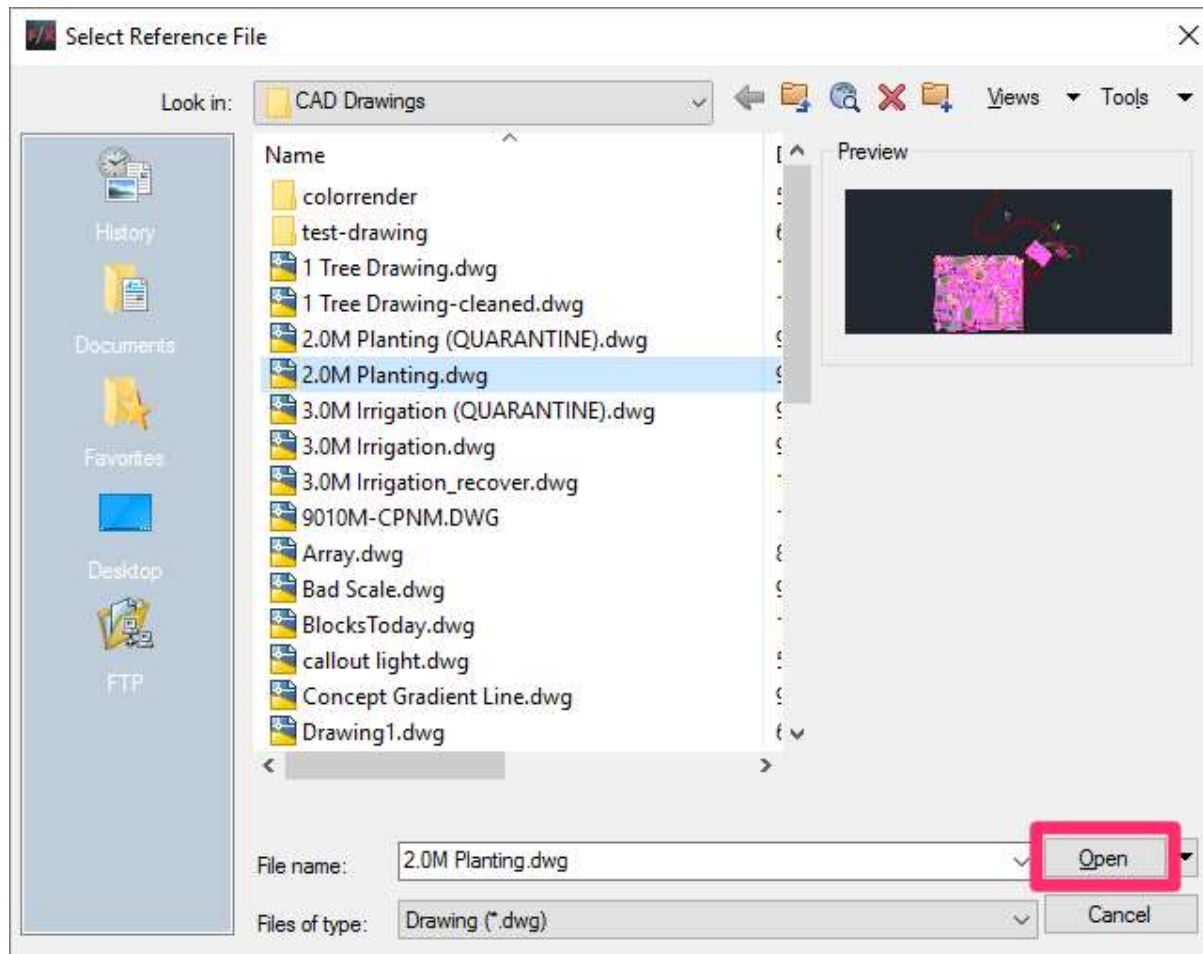
## Equally Important! Clean Your Xrefs Before Attaching Them Attaching an Xref



- To begin Xrefing a DWG file into your drawing, click the **DWG** button in the top left corner of the Xref Manager.
- Do not see a **DWG** button? Your Xref Manager might have another file format enabled.
- Click the small triangle in the top left corner of the Manager and select **Attach DWG** from the menu that opens.








- In the **Select Reference File** dialog box, navigate to the DWG you want to attach.
- Click **Open** to attach the DWG you selected.
- The **Attach External Reference** dialog box will open.
- Click **OK** to attach the DWG file.

**Attach External Reference** [Close]

Name: 2.0M Planting [Browse...]

Preview



Scale

Specify On-screen

X: 1.00

Y: 1.00

Z: 1.00

Uniform Scale

Path type

Full path

Rotation

Specify On-screen

Angle: 0

Reference Type

Attachment  Overlay

Locate using Geographic Data

Insertion point

Specify On-screen

X: 0.00

Y: 0.00

Z: 0.00

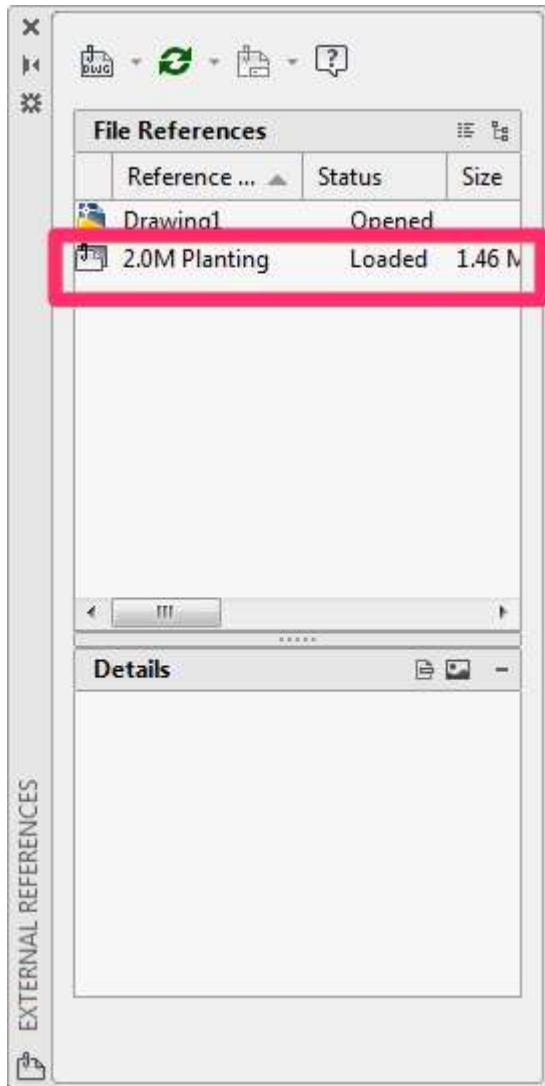
Block Unit

Unit: Meters

Factor: 39.3701

Show Details

**OK** Cancel Help



The file will be Xrefed into your drawing.  
You should now see an entry for it in the Xref Manager.

## Internet Transmission:

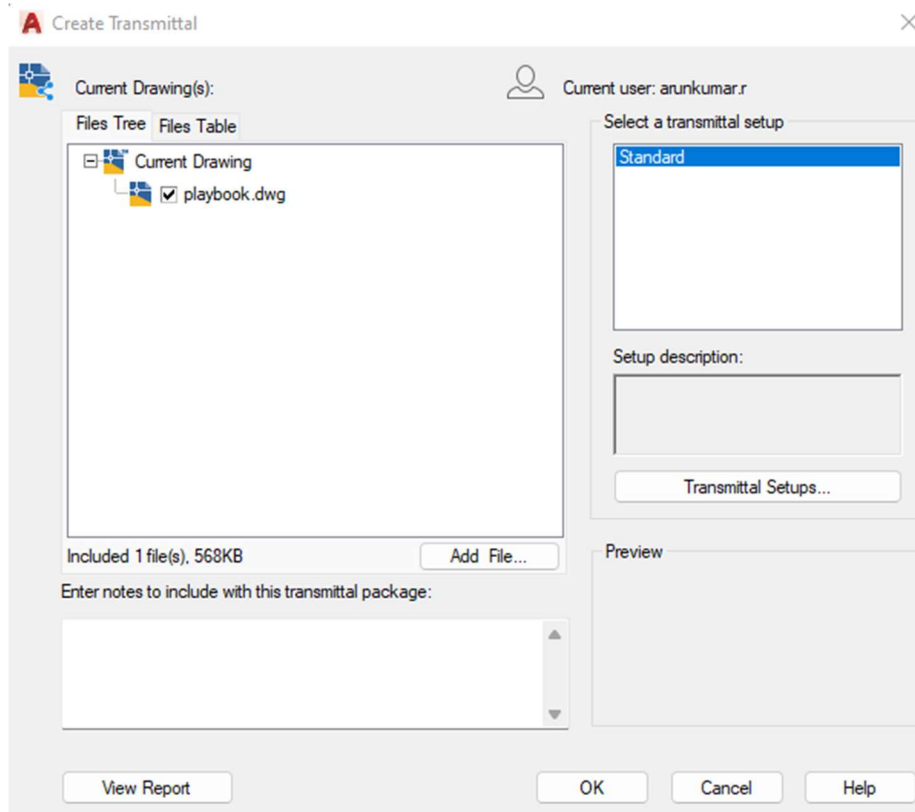


Find

The Create Transmittal dialog box is displayed.

If you enter -ETRANSMIT at the Command prompt, options are displayed.

When you select a set of drawing files in a transmittal package, it automatically includes all related dependent files such as xrefs and font files.



## Publishing Layouts:

Publishing provides a streamlined alternative to plotting multiple drawings by providing compressed representations of drawings in a file that is easy to view and distribute.

An electronic drawing set is the digital equivalent of a set of plotted drawings. You create an electronic drawing set by publishing drawings to a DWF, DWFx, or PDF file.

You can publish an entire sheet set from the Sheet Set Manager. With one click, you can create an electronic drawing set by publishing the sheet set to a single, multi-sheet DWF, DWFx, or PDF file.

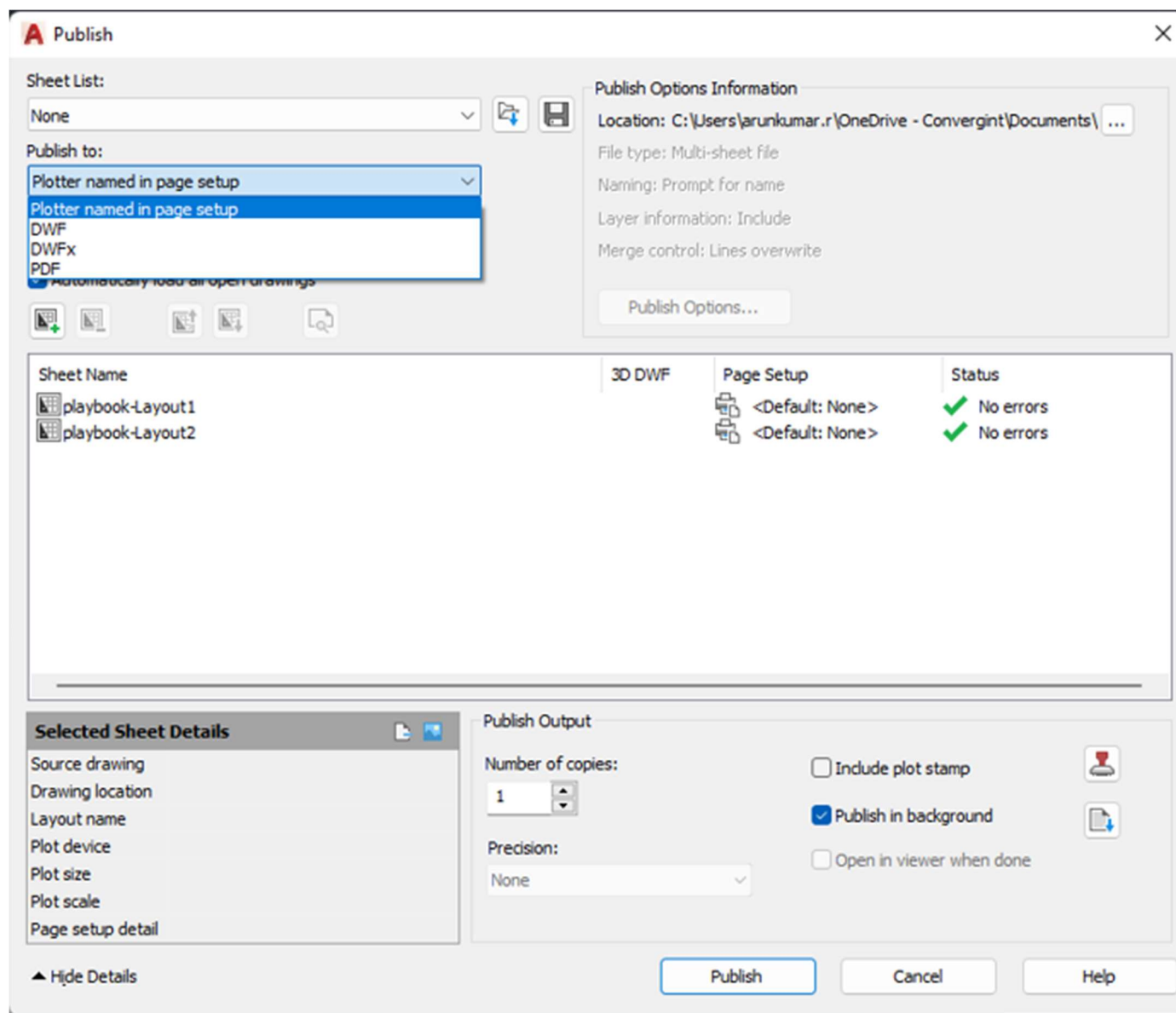
You can create a paper drawing set by publishing the sheet set to the plotter named in each sheet's page setup.

Using the Publish dialog box, you can assemble a collection of drawings to publish and save the list as a Drawing Set Descriptions (DSD) file. You can customize this collection of drawings for a specific user, and you can add and remove sheets as a project evolves. Once you've created a list of drawing sheets in the Publish dialog box, you can publish the drawings to any of the following:

- The plotter named in each sheet's page setup (including drawings that you want to plot to file)
- A single, multi-sheet DWF or DWFx file containing both 2D and 3D content
- A single, multi-sheet PDF file containing 2D content
- Multiple single-sheet DWF or DWFx files containing both 2D and 3D content
- Multiple single-sheet PDF files containing 2D content

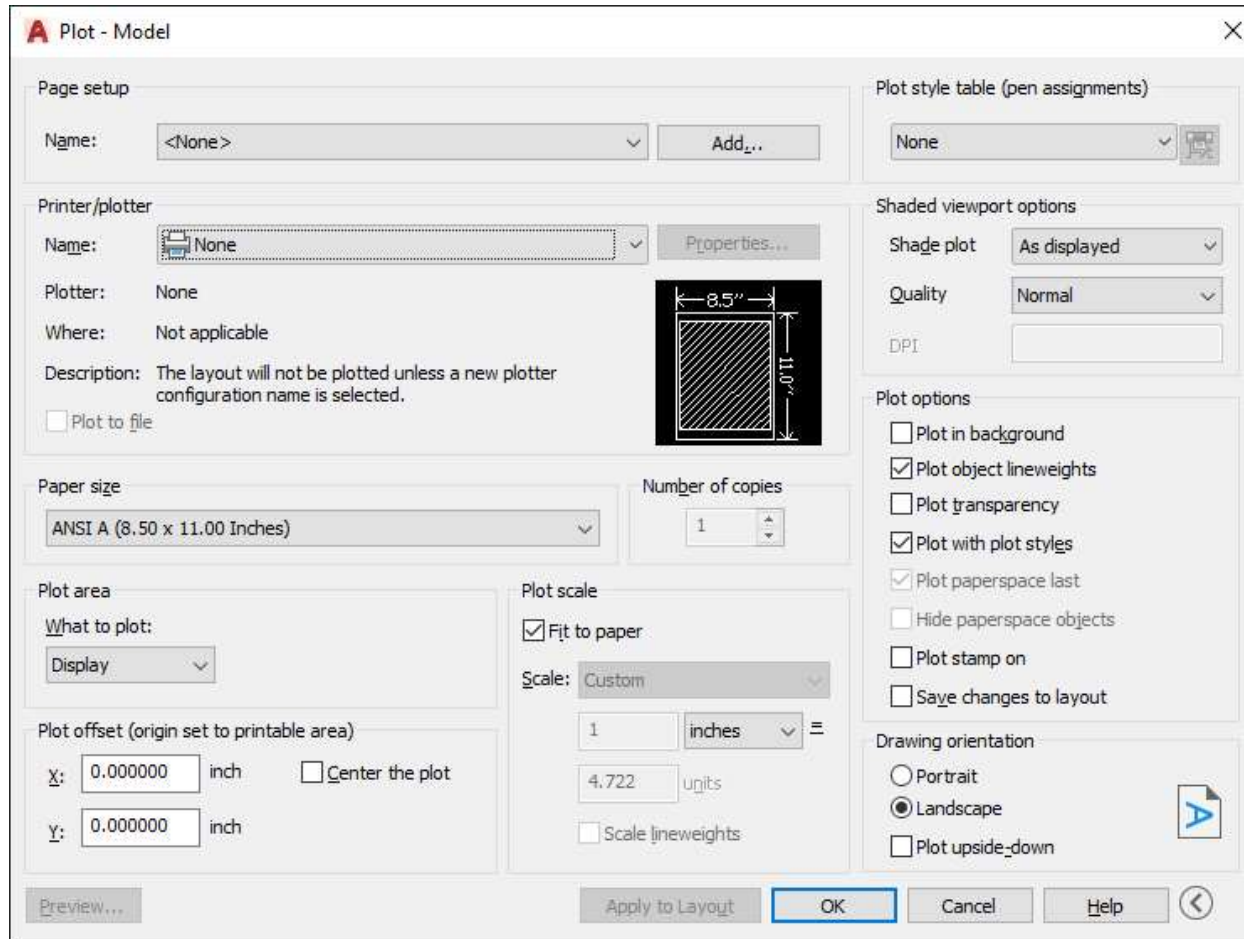
Using 3D DWF publishing, you can create and publish DWF files of your 3D models and view them with Autodesk Design Review.





# Plot (or) Print Drawing:

1. Click Output tab ► Plot panel ► Plot.  Find



2. Select a plotter.
3. Select paper size, plot area, plot scale, orientation, and other options.

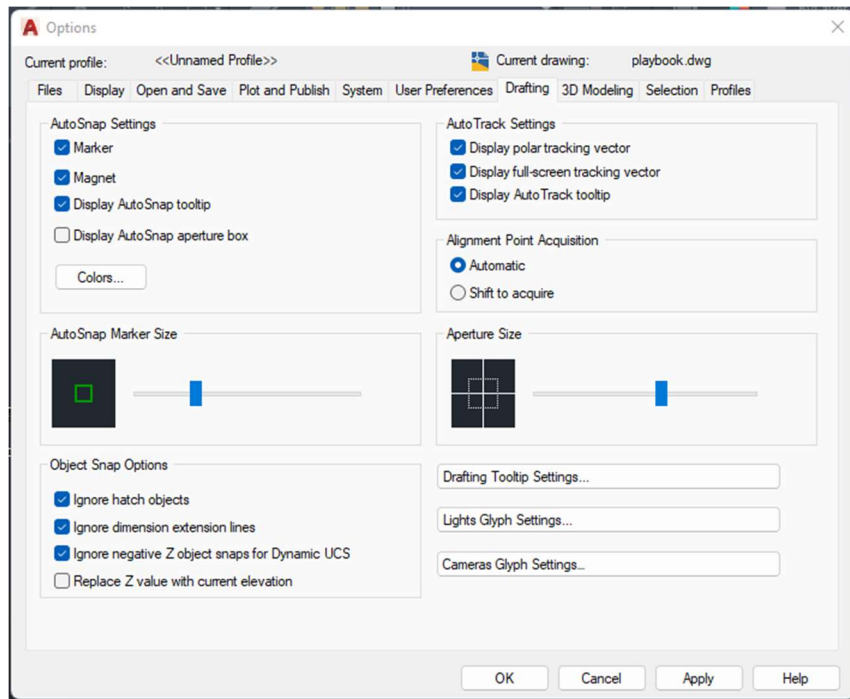


4. For additional options, click the More Options button.

## How to select options and specify preferences for general drawing settings.



1. Click the Application button > Options.
2. In the Options dialog box, click a tab and set options as desired.
3. Save the settings:
  - To save the settings and keep working in the dialog box, click Apply.
  - To save the settings and close the dialog box, click OK.



## Conclusion

The use of the import function in Auto CAD definitely has some advantages. It can help to reduce the time it takes to produce the drawings needed in the work environment. However, if the individual that has created the imported file does not receive credit for their work, problems may arise in the future. As well, by using the function people may slowly start to place a lesser value on the intellectual property of others, which will undoubtedly create future conflict. The use of the import function may also be relied on heavily to create efficiencies within the industry. This could lead to a decrease in the rate of innovation. It's obvious that the import function has a place in the building industry, but its use will have to be regulated in order to create a balance between the negative and the positive effects it creates.

**Thank  
You!**

