

CP322895

# AutoCAD Tips and Tricks to Improve Efficiency

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## Learning Objectives

- Manipulate settings in the options menu and keyboard shortcuts
- Modify dimension style settings
- Utilize various layering commands to display objects differently
- Organize drawings using x-refs and model space / paper space

## Description

Do you find yourself behind on design projects? Do you want to familiarize yourself with AutoCAD better to become more efficient? In this class we will discuss how to manipulate settings/options to improve speed. We will also discuss how to set up drawings using x-refs and to utilize paper space and model space. We will also look at how layer commands and object display settings can reduce clutter in your drawings to make it easier to manipulate components in your drawing. This class will be a great class to help you become more familiar with some of the options in AutoCAD and make you take that next step in your career using the many tools AutoCAD has to offer.

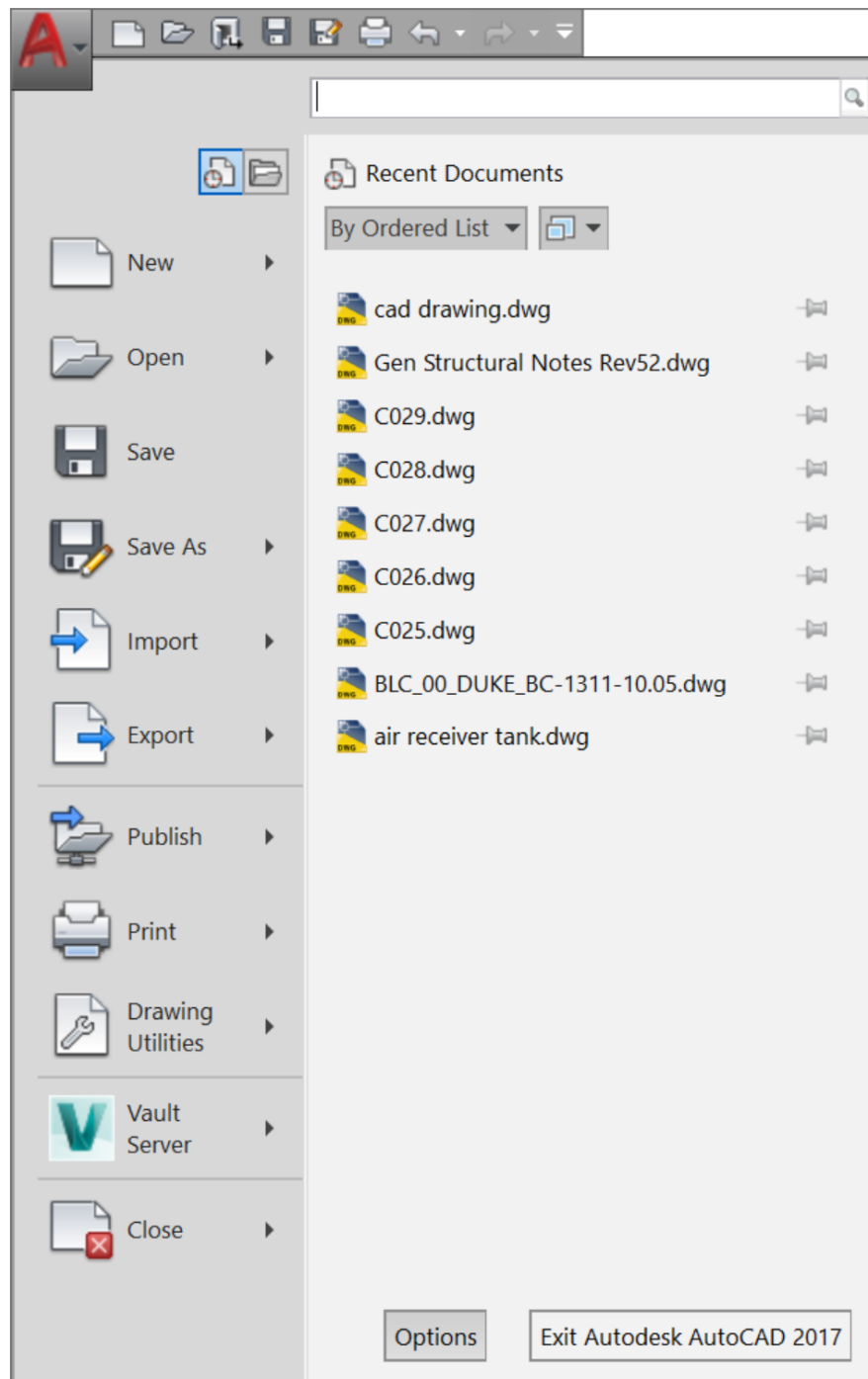
## Speaker(s)

Josh Watson is a Senior Structural Designer for Mesa Associates, Inc., a multi-discipline engineering firm with 16 offices and over 800 employees throughout the United States. Josh works in the Knoxville, TN office, which has about 325 employees. He has been at Mesa Associates, Inc. for over 11 years. Josh has more than 15 years' experience working with Autodesk products and has experience working with Autodesk AutoCAD, AutoCAD Architecture, Civil 3D, Revit, ReCap, Plant 3D, Navisworks, and Vault. He is the lead technician at Mesa Associates, Inc. in their scanning to point cloud production. He has produced laser scanning training classes that have been incorporated into the corporate training database. In his free time, he enjoys spending time with his beautiful wife and four children.

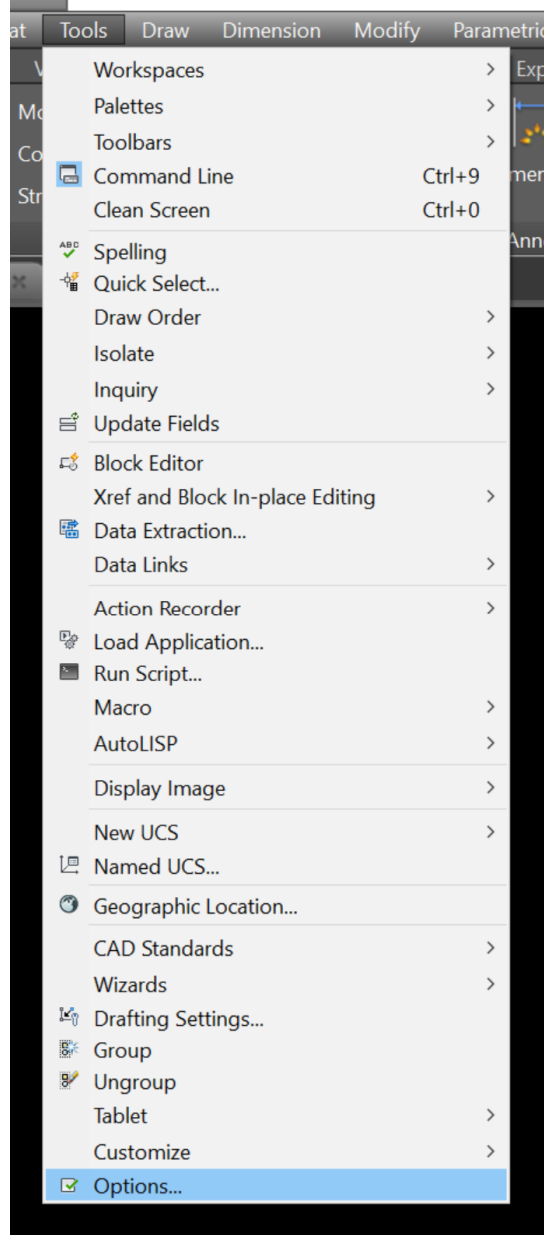
Matt DeBord is a Mechanical Engineer for Mesa Associates, Inc., a multi-discipline engineering firm with 16 offices and over 800 employees throughout the United States. Matt works in the Knoxville, TN office, which has about 325 employees. He has been at Mesa Associates, Inc. for over 11 years, where he works in the Power Generation business unit. Matt has a background in design and engineering of various piping systems, high energy piping stress analysis, finite element analysis, and hydraulics. He has experience working with Autodesk AutoCAD, Plant 3D, ReCap, Navisworks, Vault, and various other Autodesk Software's. He also has experience utilizing 3D laser scanning technology within design projects.

## Options Menu

The options menu is accommodating in customizing the display and setting various settings. The options menu can be accessed in several different ways. You can click the red A in the upper left-hand corner of AutoCAD, then click the “Options” button at the bottom.



You can also access the options menu by clicking Tools, then selecting Options at the bottom.

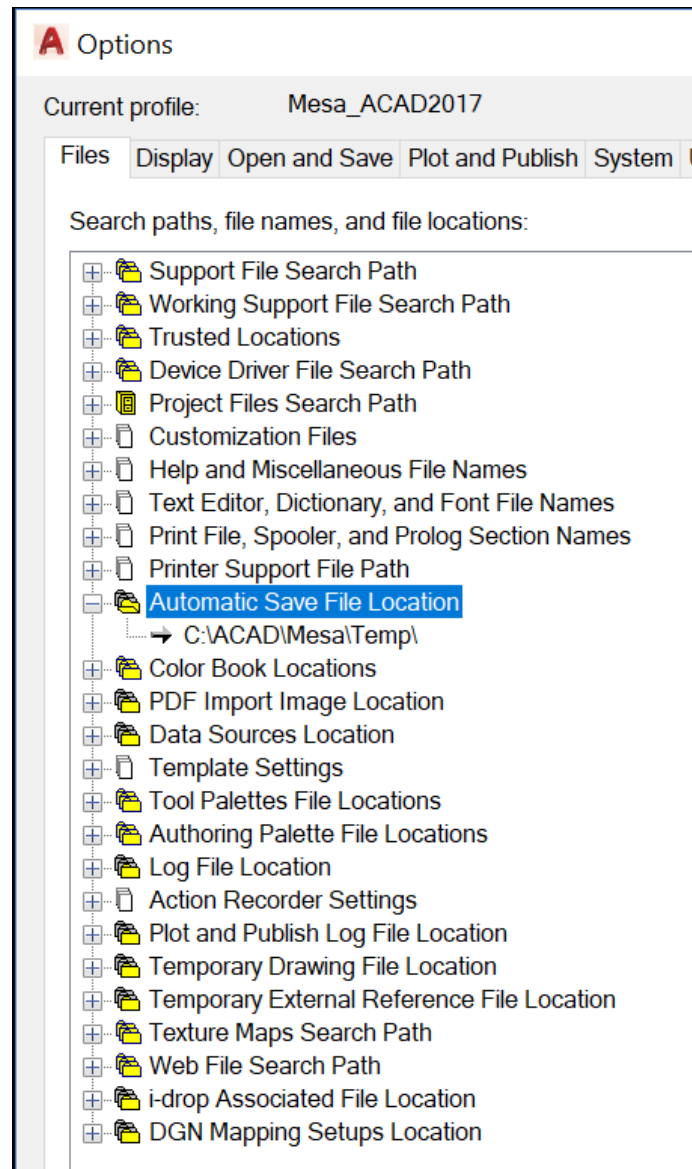


You can also simply type in "Options" or "OP" in the command line.

There are a couple of features worth pointing out that can save you some time while working with Autodesk products. The options menu will allow you to customize the way AutoCAD is displayed and hopefully make it easier for you to navigate.

## Files Tab

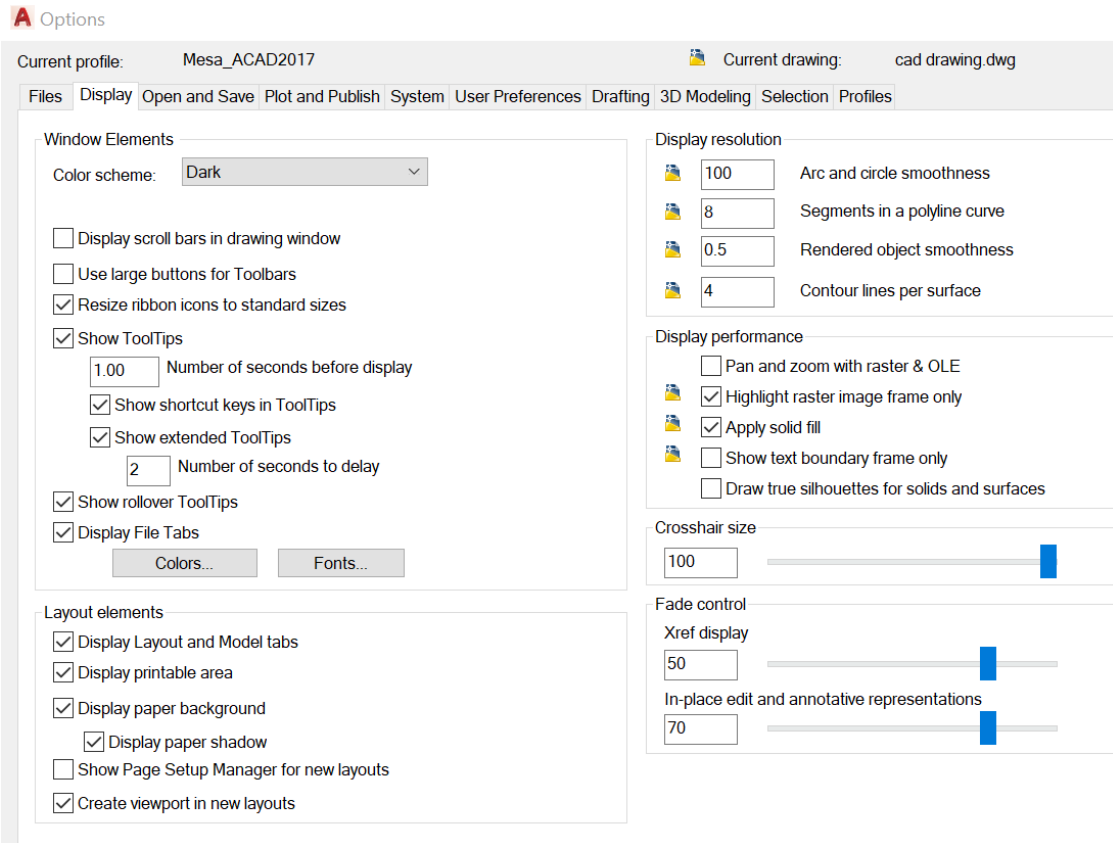
The files tab on the options menu is where you can find helpful information about where supporting files are saved on your machine. One function worth pointing out is the “Automatic Save File Location”. By clicking the + sign, you will be able to see where your autosave files are being stored. This can be a lifesaver when you get the dreaded “Fatal Error”.



## Display Tab

The Display tab allows you to customize how AutoCAD appears. Under the “Layout Elements” section, there is a toggle button for “Display Layout and Model Tabs”. If you ever accidentally close the model and layout tabs below the command line, this is a place to turn them back on to make them visible. The other item to draw attention to is the

“Crosshair Size” sliding bar. This will allow you to change the size of your crosshairs in your drawing window.



## Open and Save Tab

The Open and Save tab will allow you to set how you want the drawings to be saved and opened. Under the “File Save” section, there is a drop-down menu with the label “Save As”. The dropdown will allow you to set what version you want the drawing to saved back to. If you are working with other firms who do not have the latest version of AutoCAD installed, then you can save your drawings back to allow them to open the files. This can save much time going back and forth if you already have this setting saved to an older version. You can also set how often AutoCAD will automatically save a backup copy of your drawing. Under the “File Safety Precautions” heading, there is a checkbox for “Automatic Save”. You can also set the time between automatic saves. Be sure to look at this setting anytime you upgrade or get a new machine with a new install of AutoCAD. You should always have this setting checked and adjust the time to what best suits you.

**Options**

Current profile: Mesa\_ACAD2017      Current drawing: cad drawing.dwg

Files   Display   Open and Save   **Plot and Publish**   System   User Preferences   Drafting   3D Modeling   Selection   Profiles

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**File Save**

Save as:

AutoCAD 2004/LT2004 Drawing (\*.dwg) ▼

☒ Maintain visual fidelity for annotative objects

☒ Maintain drawing size compatibility

Thumbnail Preview Settings...

50      Incremental save percentage

**File Open**

9      Number of recently-used files

☐ Display full path in title

---

**File Safety Precautions**

☒ Automatic save

10      Minutes between saves

☒ Create backup copy with each save

☐ Full-time CRC validation

☐ Maintain a log file

ac\$      File extension for temporary files

Digital Signatures...

☒ Display digital signature information

**Application Menu**

9      Number of recently-used files

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**External References (Xrefs)**

Demand load Xrefs:

Enabled with copy ▼

☒ Retain changes to Xref layers

☒ Allow other users to Redefit current drawing

**ObjectARX Applications**

Demand load ObjectARX apps:

Object detect and command invoke ▼

Proxy images for custom objects:

Show proxy graphics ▼

☒ Show Proxy Information dialog box

## User Preferences Tab

The User Preferences tab is beneficial by allowing the user to make settings that once again will help them become more efficient. Under the “Window Standard Behavior” there is a button labeled “Right-Click Customization”. When you click this button, a window appears that will allow you to customize the right-click on your mouse. Each user should look at this setting in AutoCAD to see which settings will work best. I will typically have the “Time-Sensitive Right-Click” activated and the “Edit Mode” set to “Shortcut Menu”. Under this tab you can also see what the line weight settings are set to. This will allow you to change the default line weight thickness in your drawing.

## Options

Current profile: Mesa\_ACAD2017 Current drawing: cad drawing.dwg

Files Display Open and Save Plot and Publish System User Preferences Drafting 3D Modeling Selection Profiles

**Windows Standard Behavior**

☒ Double click editing

☒ Shortcut menus in drawing area

Right-click Customization...

**Insertion scale**

Default settings when units are set to unitless:

Source content units:

Inches

Target drawing units:

Inches

**Hyperlink**

☒ Display hyperlink cursor, tooltip, and shortcut menu

**Fields**

☒ Display background of fields

Field Update Settings...

**Priority for Coordinate Data Entry**

☐ Running object snap

☐ Keyboard entry

☒ Keyboard entry except scripts

**Associative Dimensioning**

☐ Make new dimensions associative

**Undo/Redo**

☒ Combine zoom and pan commands

☒ Combine layer property change

Block Editor Settings...

Lineweight Settings...

Default Scale List...

## Right-Click Customization

☒ Turn on time-sensitive right-click:

Quick click for ENTER

Longer click to display Shortcut Menu

Longer click duration: 250 milliseconds

**Default Mode**

If no objects are selected, right-click means

☐ Repeat Last Command

☒ Shortcut Menu

**Edit Mode**

If one or more objects are selected, right-click means

☐ Repeat Last Command

☒ Shortcut Menu

**Command Mode**

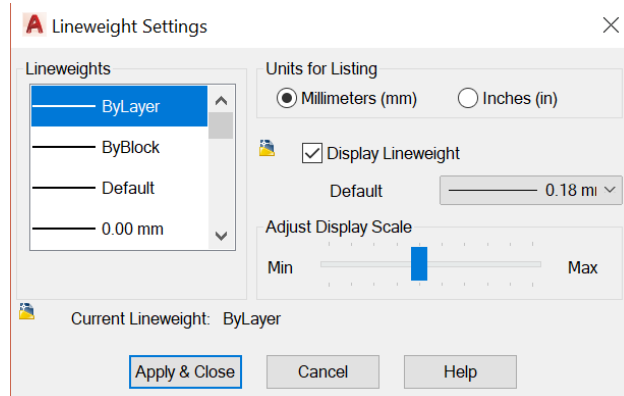
If a command is in progress, right-click means

☐ ENTER

☐ Shortcut Menu: always enabled

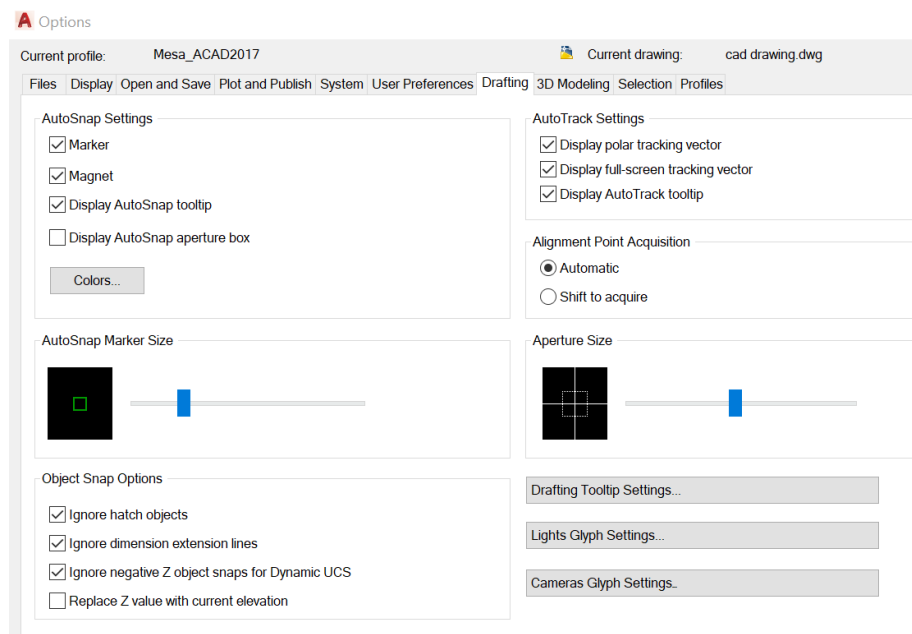
☒ Shortcut Menu: enabled when command options are present

Apply & Close Cancel Help

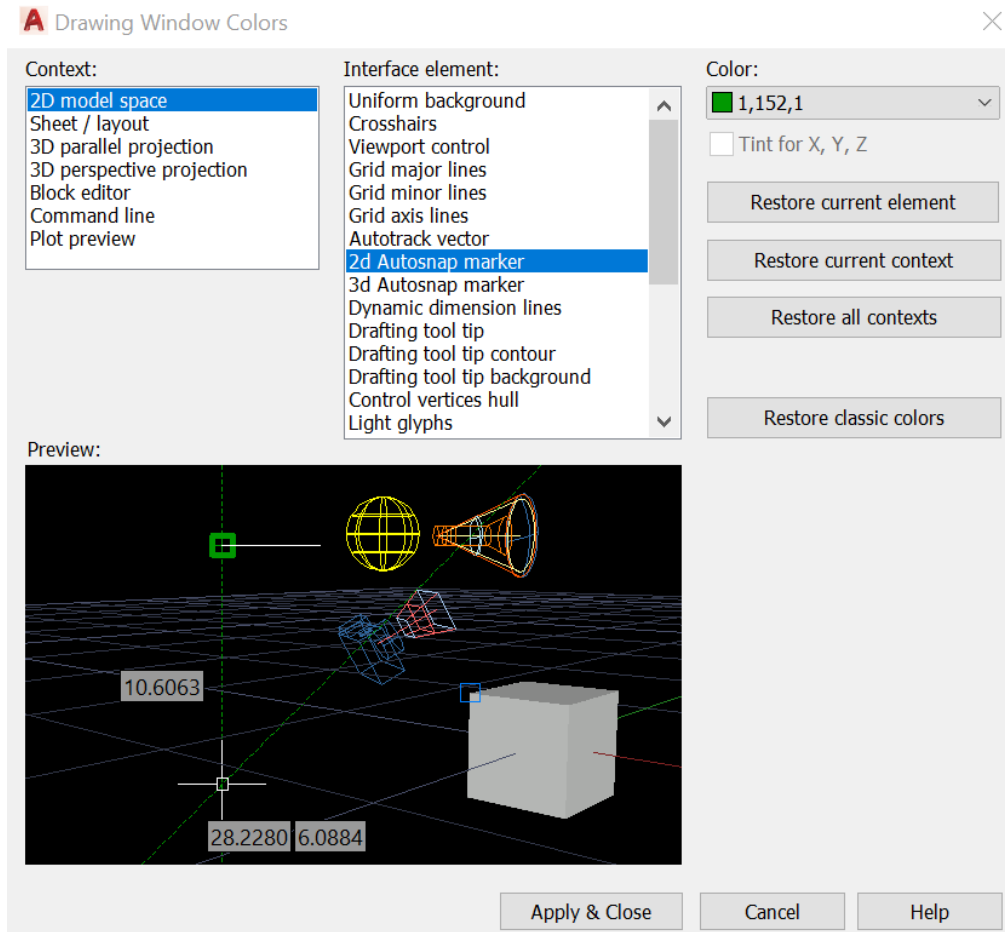


## Drafting Tab

The Drafting tab will allow the user to make changes to AutoSnap and AutoTrack. On this tab you can change the size of the “AutoSnap Marker Size” and the “Aperture Size”. These should be set on your preference for how you want these to display. The user can also set the colors for helpful tools in AutoCAD. By clicking the “Colors...” button under the “AutoSnap Settings” section a window will appear. Within this window, you can select the “Context” of the drawing and then the “Interface Element”. If the user prefers a background color in model space, then select “2D Model Space” under the “Context” section and then select “Uniform Background” under the “Interface Element” section. Once you have those selections you can change the color to whatever makes it easier for you to view your drawing file under the “Color” section. If the user prefers a different color in the layout space, then you can select that heading under the “Context” section and change the color accordingly.







There are many other features associated with the options menu that I would encourage users to look through and make changes to improve efficiency. The options settings can be conducive if set up correctly for each user.

## Keyboard Shortcuts

The keyboard shortcuts are keyboard combinations used to launch AutoCAD commands. For instance, you can press Ctrl+P to bring up the plot dialog box. This is the same effect as going to the big red A in the upper left-hand corner and clicking print. You can also directly type the corresponding keys. For instance, you can type “E” in the command line to erase an object. This is the same effect as clicking the erase button on the modify toolbar in the ribbon. Click the link below to access the Autodesk website that gives more information on keyboard shortcuts.

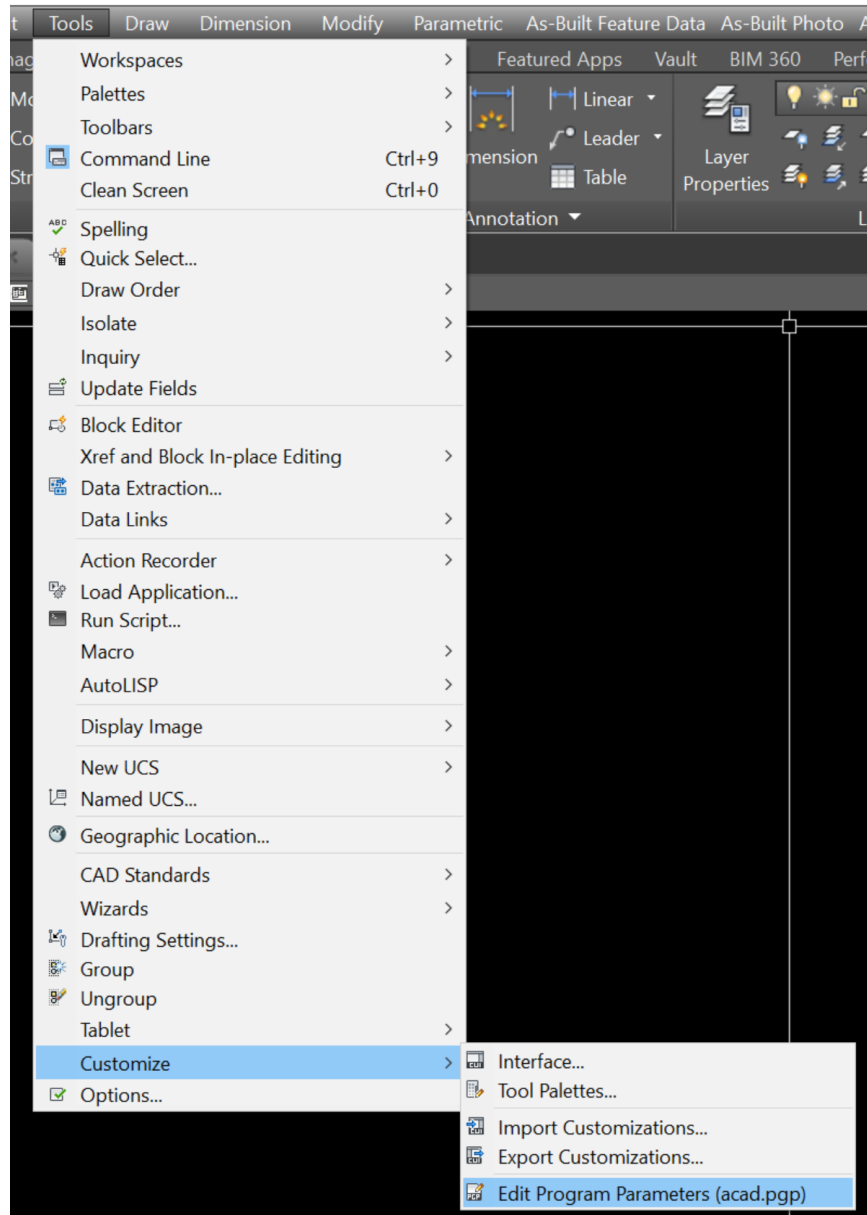
<https://www.autodesk.com/shortcuts/autocad>

Below you will find the AutoCAD one-key shortcuts. These are the default settings.

	Display Help	Toggle text screen	Toggle object snap mode	Toggle 3DOsnap	Toggle Isoplane	Toggle Dynamic UCS	Toggle grid mode	Toggle ortho mode	Toggle snap mode	Toggle polar mode	Toggle object snap tracking	Toggle dynamic input mode			
Esc	F1	F2	F3	F4	F5	F6	F7	F8	F9	F10	F11	F12	PrtScn SysRq	ScrLK	Pause Break
~	1	2	3	4	5	6	7	8	9	0	-	=	Backspace	Home	End
Tab	Q QSAVE	W WBLOCK	E ERASE	R REDRAW	T MTEXT	Y	U	I INSERT	O OFFSET	P PAN	{	}		Insert	Page Up
Caps Lock	A ARC	S STRETCH	D DIMSTYLE	F FILLET	G GROUP	H HATCH	J JOIN	K	L LINE	:	"	,	Enter	Delete	Page Down
Shift TOGGLE ORTHO MODE	Z ZOOM	X EXPLODE	C CIRCLE	V VIEW	B BLOCK	N NEW	M MOVE	<	>	?	/	Shift TOGGLE ORTHO MODE			
Ctrl	Start	Alt							Alt			Ctrl			

Q QSAVE / Saves the current drawing.	C CIRCLE / Creates a circle.	H HATCH / Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.
A ARC / Creates an arc.	R REDRAW / Refreshes the display in the current viewport.	J JOIN / Joins similar objects to form a single, unbroken object.
Z ZOOM / Increases or decreases the magnification of the view in the current viewport.	F FILLET / Rounds and fillets the edges of objects.	M MOVE / Moves objects a specified distance in a specified direction.
W WBLOCK / Writes objects or a block to a new drawing file.	V VIEW / Saves and restores named views, camera views, layout views, and preset views.	I INSERT / Inserts a block or drawing into the current drawing.
S STRETCH / Stretches objects crossed by a selection window or polygon.	T MTEXT / Creates a multiline text object.	O OFFSET / Creates concentric circles, parallel lines, and parallel curves.
X EXPLODE / Breaks a compound object into its component objects.	G GROUP / Creates and manages saved sets of objects called groups.	L LINE / Creates straight line segments.
E ERASE / Removes objects from a drawing.	B BLOCK / Creates a block definition from selected objects.	P PAN / Adds a parameter with grips to a dynamic block definition.
D DIMSTYLE / Creates and modifies dimension styles.	N NEW / Create a new drawing	

You can also customize the key-in commands. The way I typically change the keyboard shortcuts is by accessing the acad.pgp file saved on your local machine. You can access this by clicking the “Tools” menu, then selecting “Customize”, then selecting “Edit Program Parameters (acad.pgp)”.



After you click the acad.pgp file will open in Windows Notepad. This file can be modified to set the keyboard shortcuts you prefer. I typically try to set commands on the left side of the keyboard to allow me to have one hand on the keyboard and one hand on the mouse while working. For example, I will set “C” equal to “Copy”, then “CC” equal to “Chamfer”. This method can be used for several other commands. Each user should decide if it is faster to enter keyboard commands are searching for the commands in the ribbon, menu, or toolbars. Below is an exert from the acad.pgp file.

```

3A,          *3DARRAY
3DMIRROR,    *MIRROR3D
3DNavigate,  *3DWALK
3DO,         *3DORBIT
3DP,         *3DPRINT
3DPLOT,      *3DPRINT
3DW,         *3DWALK
3F,          *3DFACE
3M,          *3DMOVE
3P,          *3DPOLY
3R,          *3DROTATE
3S,          *3DSCALE
A,           *ARC
AC,          *BACTION
ADC,         *ADCENTER
AECTOACAD,   *-ExportToAutoCAD
AA,          *AREA
AL,          *ALIGN
3AL,         *3DALIGN
AP,          *APLOAD
APLAY,       *ALLPLAY
AR,          *ARRAY
-AR,         *-ARRAY
ARR,         *ACTRECORD
ARM,         *ACTUSERMESSAGE
-ARM,        *-ACTUSERMESSAGE
ARU,         *ACTUSERINPUT
ARS,         *ACTSTOP
-ARS,        *-ACTSTOP
ATI,         *ATTIPEDIT
ATT,         *ATTDEF
-ATT,        *-ATTDEF
ATE,         *ATTEDIT
-ATE,        *-ATTEDIT
ATTE,        *-ATTEDIT

```

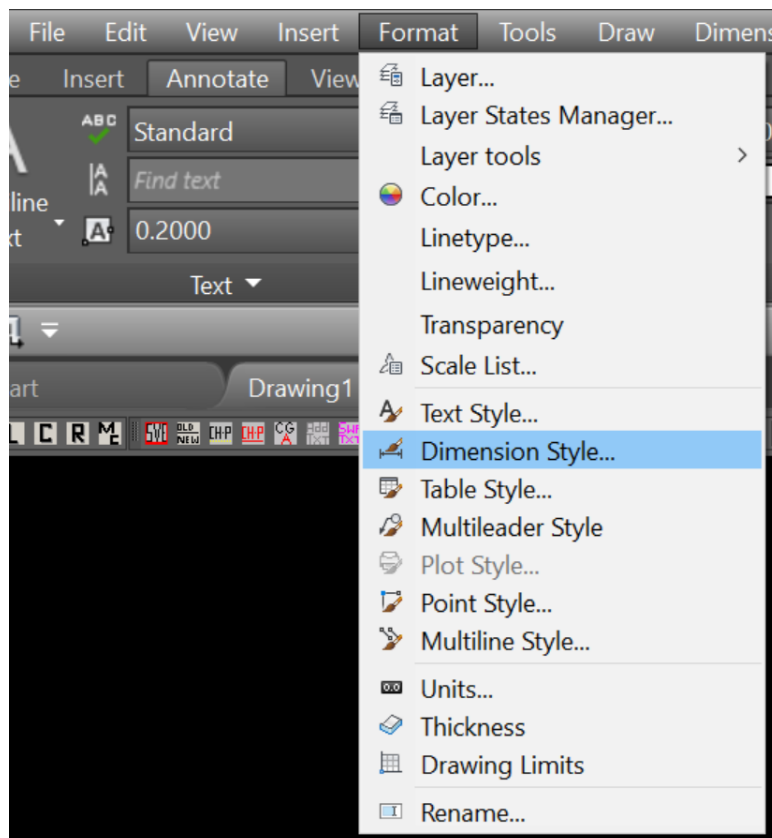
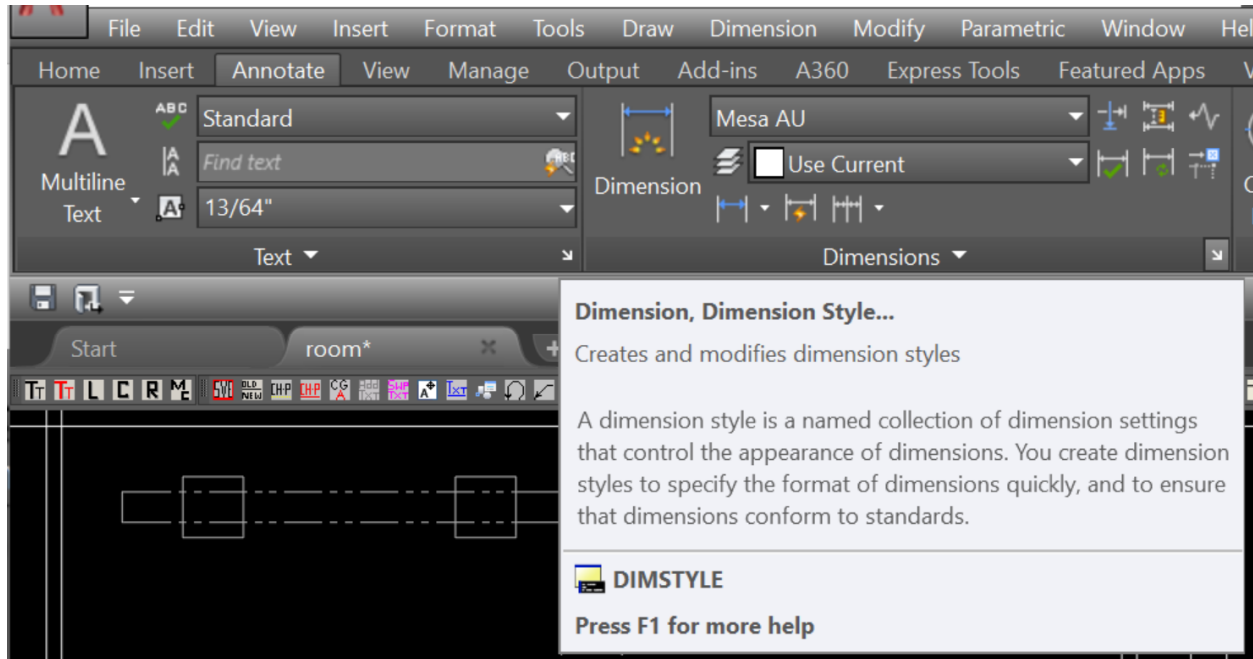
You may also access the “Customize User Interface (CUI)” editor to access keyboard shortcuts. You can access this by clicking the “Manage” tab in the ribbon, then in the “Customization” pane, select the User Interface Please find the below

<https://knowledge.autodesk.com/support/autocad-lt/learn-explore/caas/CloudHelp/cloudhelp/2019/ENU/AutoCAD-LT/files/GUID-9970E91B-A4F5-405C-99C1-46AD804A8858-htm.html>

## Dimension Styles

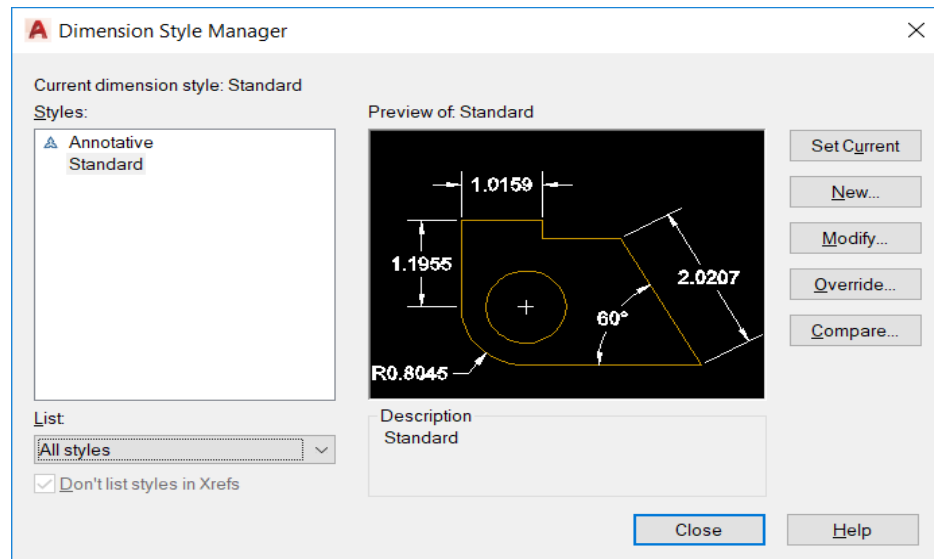
Dimensions styles are a great way to ensure your dimensional annotations are presented per client standards, correct information, presentation, and accurately reflect the drawing component. Dimension style setup and modifications are completed through the “Dimension Style Manager”. You can access the “Dimension Style Manager” by typing “dimsty” in the command line, by

clicking the arrow in the bottom right-hand corner of the dimensions panel under the “Annotate” heading, or by clicking “Format”, then selecting “Dimension Style”.



## Dimension Style Manager

The Dimension Style Manager is comprised of several components that will be expanded upon in the following to improve overall understanding and efficiency.



One must first be familiar with all components and terminology associated with components of a dimension before moving forward with Creating and/or Modifying a dimension style. For this course and a general understanding of dimension components, the user will need to be familiar with the following terms:

**Dimension Line:** Line which dimension is attached and connects the extension lines

**Extension Line:** Line which connects the dimension line and the object which is being dimensioned

**Arrowhead:** Visual object which indicates the termination point of the dimension line with the extension line as well as the termination point of leaders.

### Dimension Style Manager: Styles

The “Styles” portion of the dimension style manager lists the dimension styles in the current drawing. It also indicates which of the styles listed is set current by highlighting it.

**Note:** Styles will also indicate dimension styles, which are external references as well if the option is selected.

### Dimension Style Manager: List

List dropdown controls in which styles are presented. For example, all styles will show all styles created in the drawing while styles in use will only show the dimension styles which are currently being utilized in the drawing.

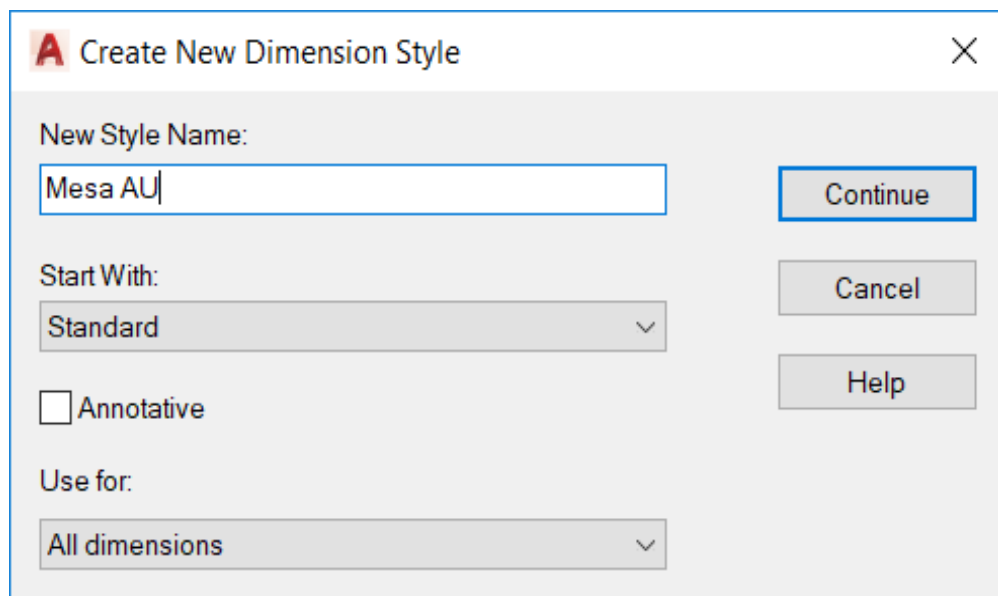
### Dimension Style Manager: Set Current

Sets the selected style from “Styles” as the current dimension style. This will affect only dimensions that are created after the change is made.

### Dimension Style Manager: New

This selection initiates the Create New Dimension Style Dialog Box, which allows you to create a new dimension style that better matches your company, client, project, or drawing needs.

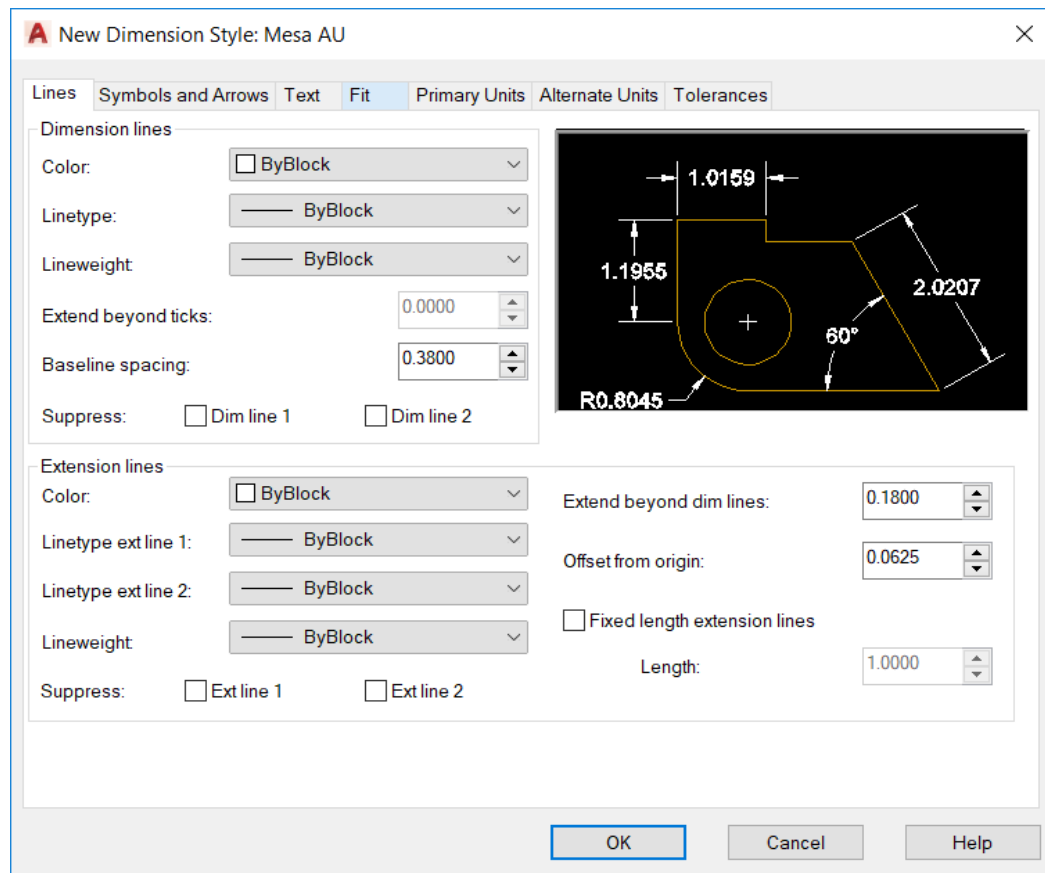
The first step to create a dimension style is to define the style name, which will be listed under in the dimension style manager. As well as defining which existing style you would like to use as a baseline to define your new style.



The image shows a screenshot of the 'Create New Dimension Style' dialog box. The dialog has a title bar with the Autodesk 'A' logo and the text 'Create New Dimension Style'. Inside the dialog, there are four main sections: 'New Style Name:' with a text input field containing 'Mesa AU'; 'Start With:' with a dropdown menu showing 'Standard'; an 'Annotative' checkbox which is currently unchecked; and 'Use for:' with a dropdown menu showing 'All dimensions'. On the right side of the dialog, there are three buttons: 'Continue' (highlighted with a blue border), 'Cancel', and 'Help'. A close button (X) is in the top right corner of the title bar.

Once “Continue” is selected the style will be created and the definition options will be displayed as follows:

## Dimension Style New – Lines Tab

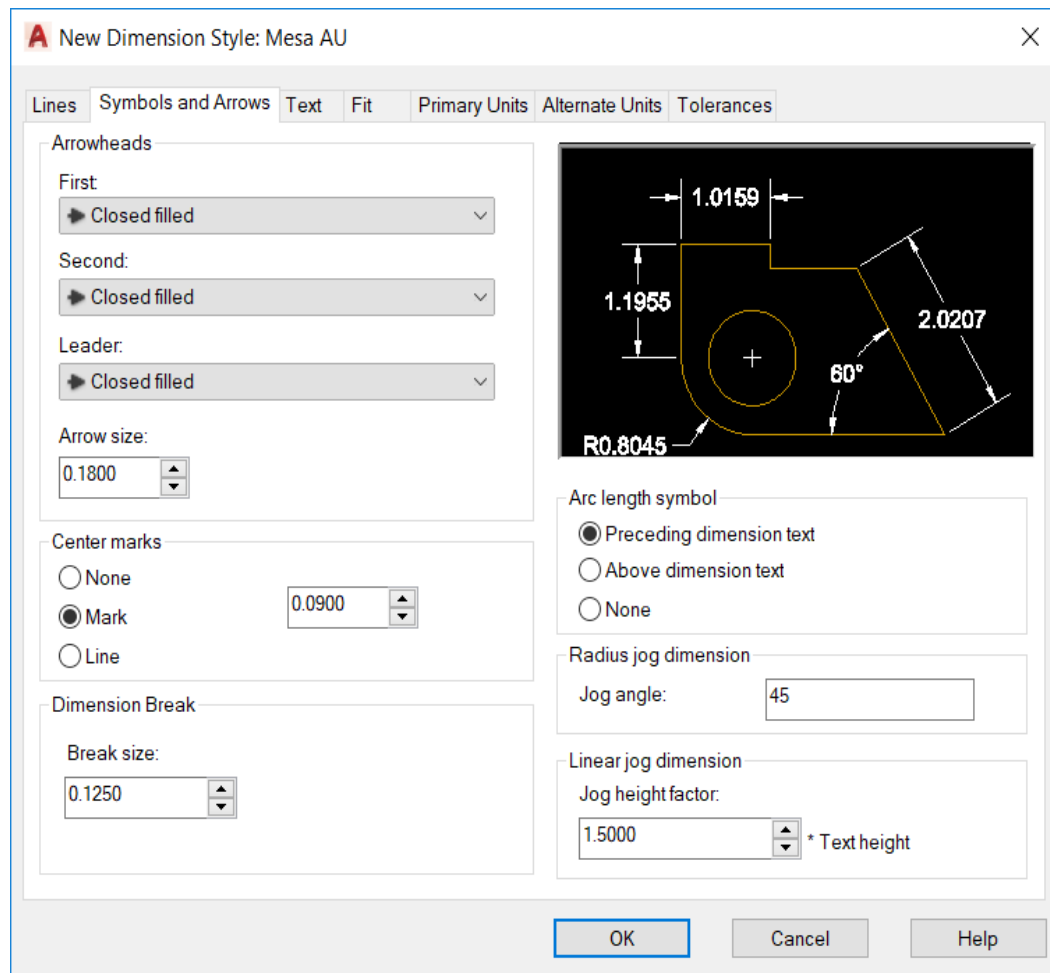


The “Lines” tab is utilized to define all properties of lines associated with the new dimension style. “Dimension Lines” color, line type, and line weight can be defined to match required drafting standards. These can be set to “By Block” or “By Layer” and will inherit the properties of the layer which the dimension is placed. “Extend Beyond Ticks” controls how far the dimension line extends beyond the extension lines. The “Baseline Spacing” controls the default spacing between baseline dimensions. The option to suppress (display or not display) either side of the dimension line is also provided.

Extension lines have similar options with the addition of a few. “Offset from Origin” sets the distance in which the extension line is offset from the location which the dimension is pulled. The user can also set a fixed extension line length by toggling the checkbox next to the “Fixed Length Extension Lines” call out.

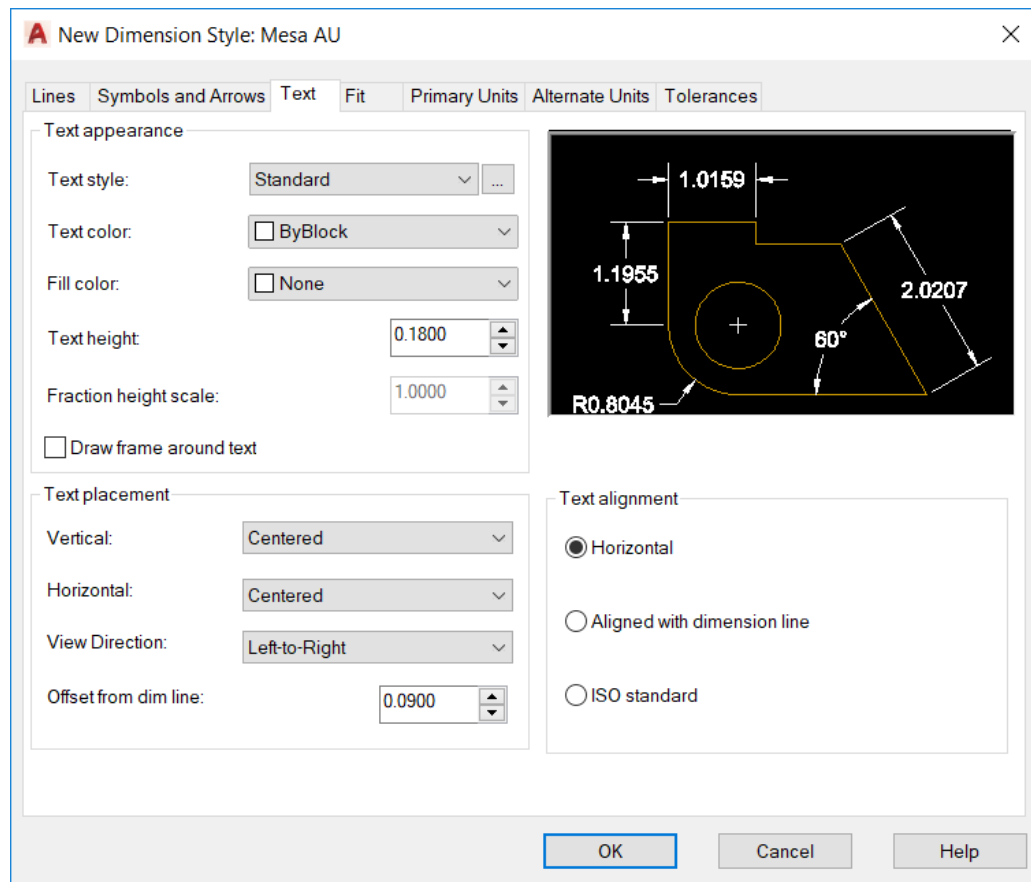


## Dimension Style New – Symbols and Arrows Tab



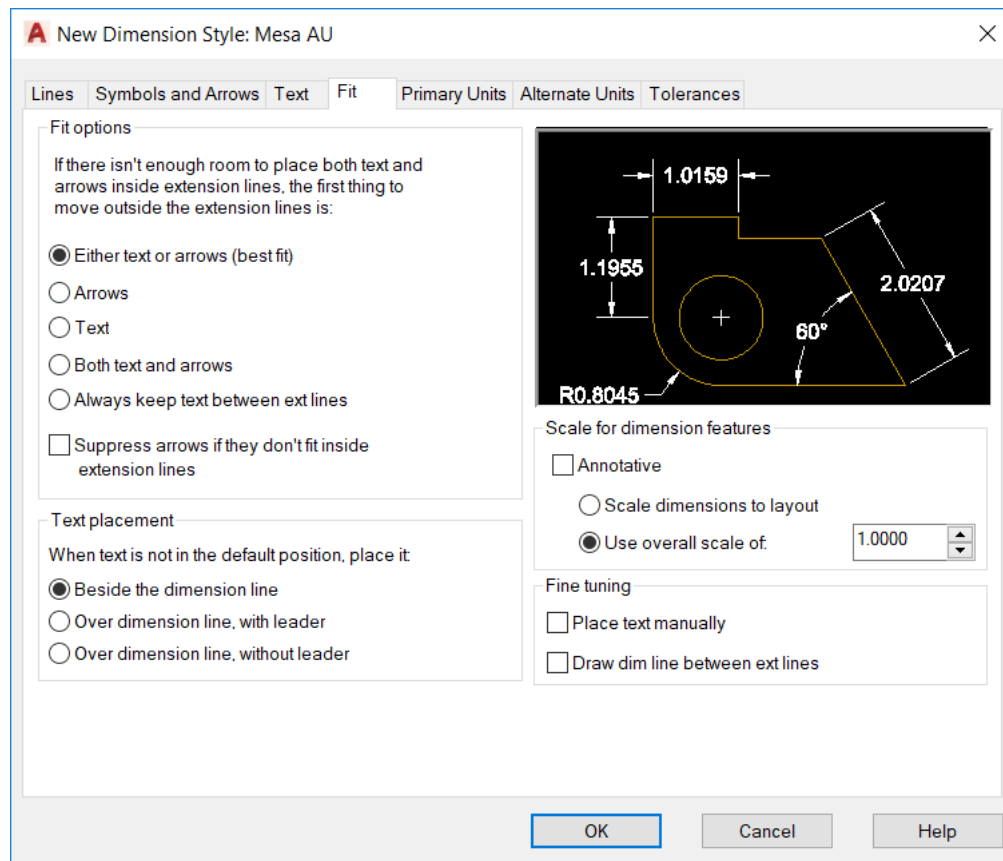
The “Symbols and Arrows” tab controls the presentation of all symbols associated with dimensions. “Arrowheads” allows the user to select the type and size of the arrowhead, which is used. The “Center Mark”, “Dimension Break”, and “Linear Jog Height Factor” allow the size and/or scale of the indicated component to be adjusted as required by the user. “Arc Length Symbol” indicates position for the symbol to be placed if one is required. “Radius Jog Dimension” allows the jog angle to be changed.

## Dimension Style New – Text Tab



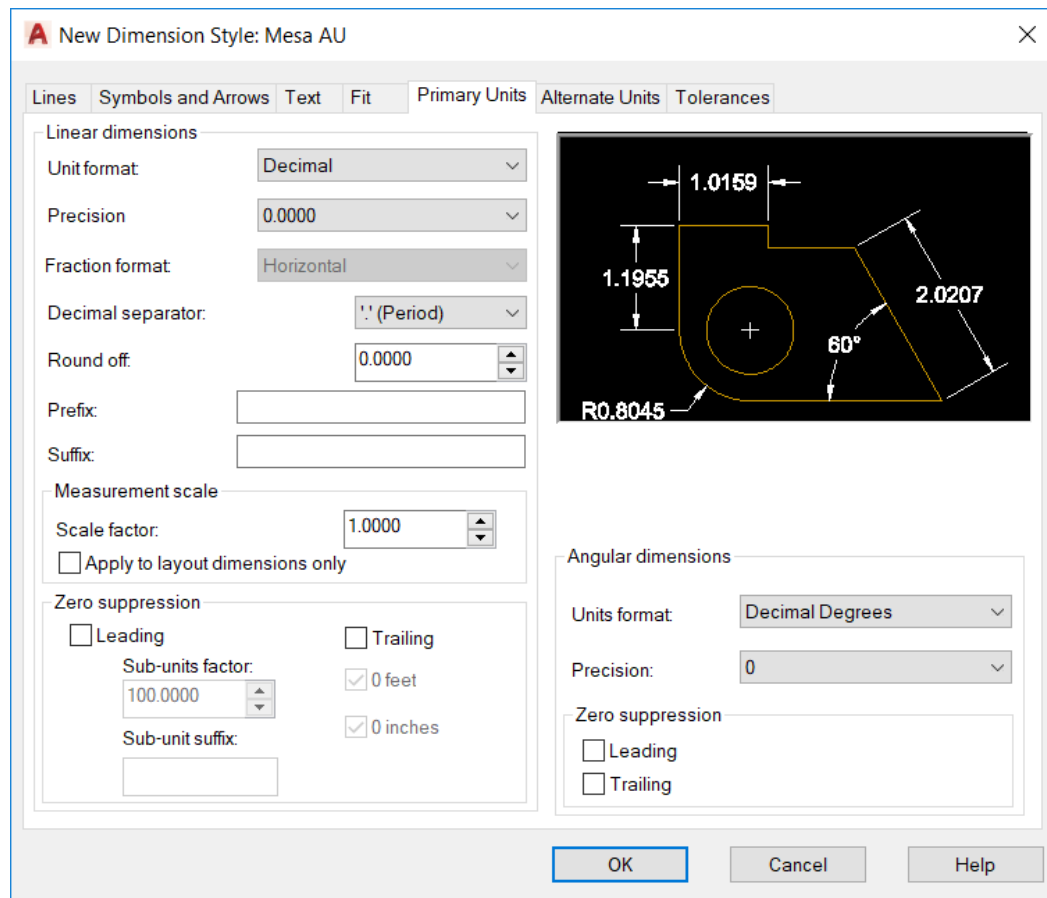
The “Text” tab controls the appearance and placement of dimension text. “Text Appearance” options allow the user to select an existing text style for the dimension text to follow. Also, the ellipsis will bring up the “Text Style Manager”. This area also allows for the text height and fraction scale to be set. “Text Placement” controls the position of the dimension concerning the dimension line as well as its spacing from the dimension line. “Text Alignment” sets the angular position between the dimension and dimension line.

## Dimension Style New – Fit Tab



The “Fit” tab sets automatic settings for placement of dimensions components based on the allotted space for the dimension text. “Fit Options” and “Text Placement” sets which component is relocated if there is not adequate space between the extension lines and the dimension texts secondary position. “Scale of Dimension Features” is the setting that adjusts the physical appearance of the dimension components to match the drawing scale, which is also referred to “Dimscale”.

## Dimension Style New – Primary Units Tab



**New Dimension Style: Mesa AU**

Lines Symbols and Arrows Text Fit **Primary Units** Alternate Units Tolerances

**Linear dimensions**

Unit format: Decimal

Precision: 0.0000

Fraction format: Horizontal

Decimal separator: '.' (Period)

Round off: 0.0000

Prefix:

Suffix:

**Measurement scale**

Scale factor: 1.0000

☐ Apply to layout dimensions only

**Zero suppression**

☐ Leading ☐ Trailing

Sub-units factor: 100.0000

☒ 0 feet ☒ 0 inches

Sub-unit suffix:

**Angular dimensions**

Units format: Decimal Degrees

Precision: 0

**Zero suppression**

☐ Leading ☐ Trailing

OK Cancel Help

The “Primary Units” tab is utilized to define properties associated with the unit of measurement present in each dimension. This covers both “Linear Dimensions” and “Angular Dimensions”. These settings include but are not limited to, Decimal, Scientific, Architectural, etc. “Measurement Scale” is a scale factor which multiplies the measured dimension by an appropriate factor. This is primarily used for items that do not match the overall drawing scale or working in model space/paper space. Refer to model space/paper space section of this presentation for discussion on “Dimassoc” system variable. This is also where the appearance of zeros is controlled through the “Zero Suppression” area.

## Dimension Style New – Alternate Units Tab

**New Dimension Style: Mesa AU**

Lines Symbols and Arrows Text Fit Primary Units **Alternate Units** Tolerances

☒ Display alternate units

Alternate units

Unit format: Decimal

Precision: 0.00

Multiplier for alt units: 25.4000

Round distances to: 0.0000

Prefix:

Suffix:

Zero suppression

☐ Leading ☐ Trailing

Sub-units factor: 100.0000

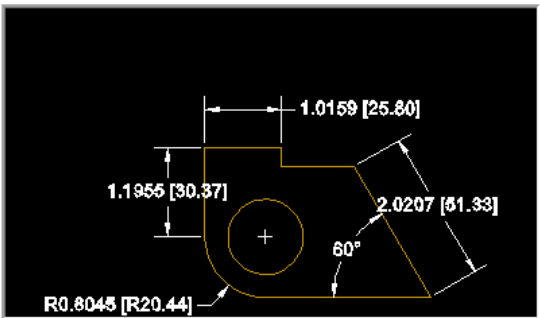
☒ 0 feet ☒ 0 inches

Sub-units suffix:

Placement

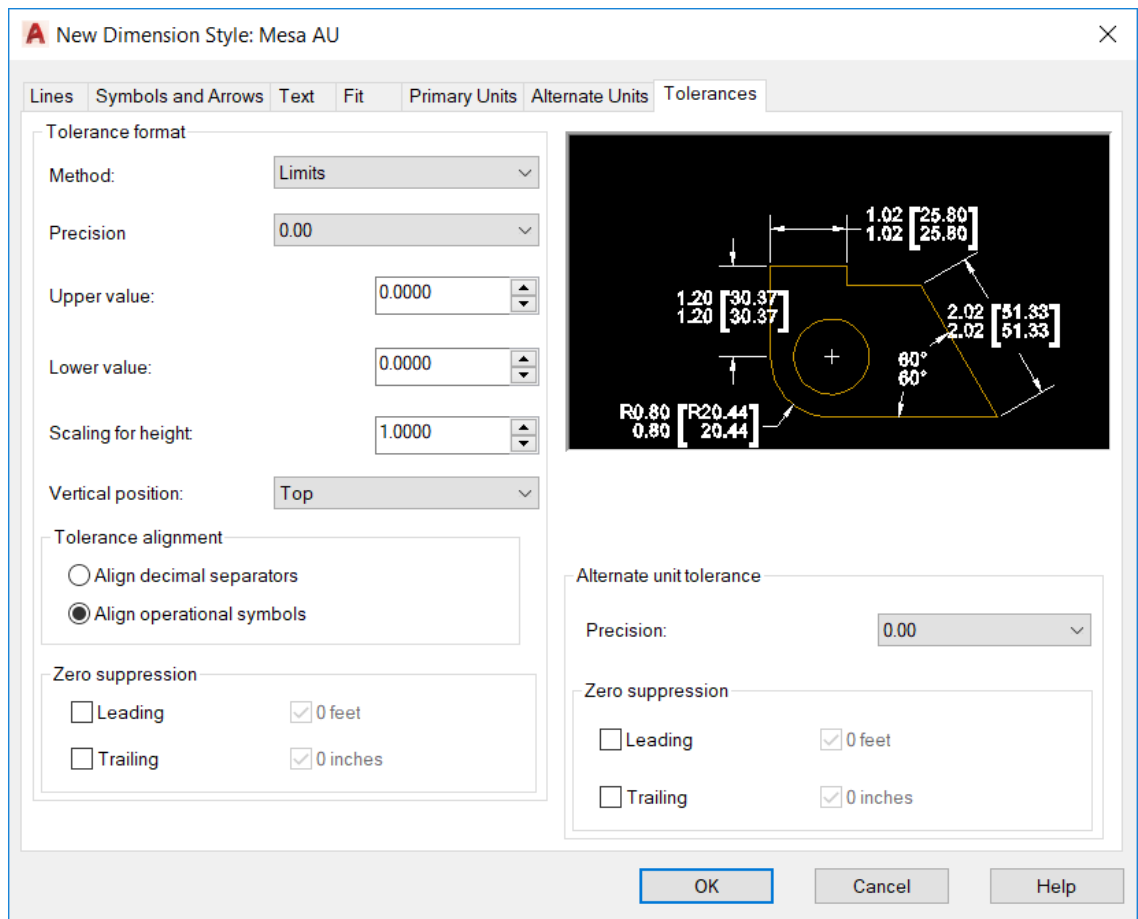
☒ After primary value ☐ Below primary value

OK Cancel Help



The “Alternate Units” tab shares the same settings as the “Primary Units” setting tab. However, sometimes, it is required to indicate both a primary and secondary unit. For example, inches and millimeters.

## Dimension Style New – Tolerances Tab



The “Tolerance” tab controls the appearance and settings associated with any tolerance required. “Tolerance Format” sets the tolerance appearance and presentation of the units set up in the primary unit tab while the “Alternate Unit Tolerance” controls the same for other units.

## Dimension Style Manager – Modify and Override

The “Modify” and “Override” options have identical settings as describe above for defining a new dimension style. The “Modify” option allows for a permanent modification to be made to a dimension style while an “Override” is temporary and is listed as an unsaved change for the dimension style.

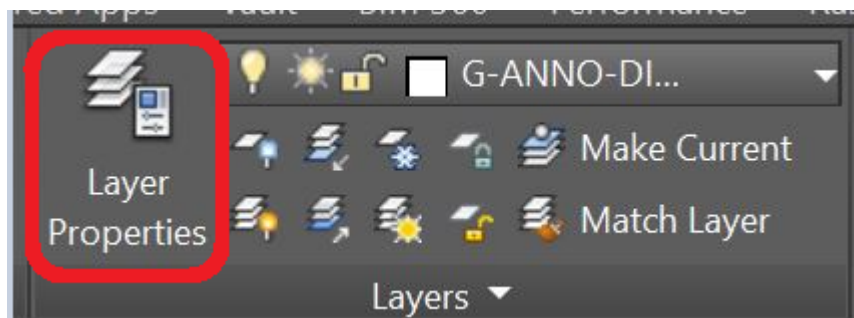
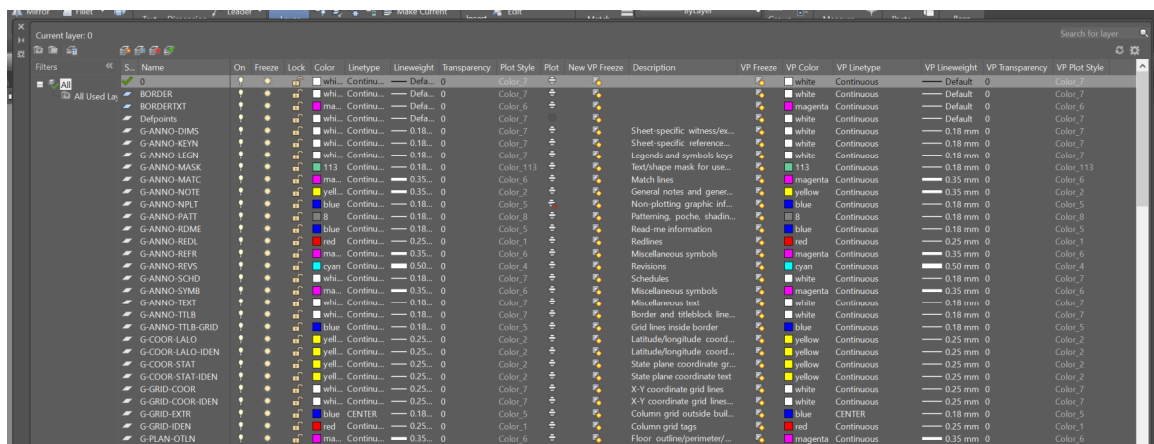
## Dimension Style Manager – Compare

The “Compare” option allows for two-dimension styles to be easily compared. This can also display the difference between the base and the override version of a dimension style. The comparison will result in a list of differences between the two and displayed for the user.

## Layers Panel

### Layer Properties

The layer properties manager allows you to add, delete, and rename layers. As well as change layer properties, set property overrides in layout viewports, and add layer descriptions. You can access the “Layer Properties Manager” from the “Layers” panel in the ribbon.

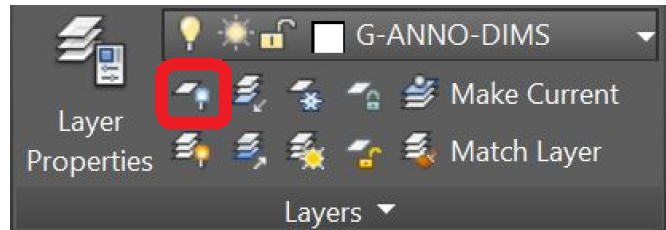
Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight	Transparency	Plot Style	Plot	New VP Freeze	Description	VP Freeze	VP Color	VP Linetype	VP Lineweight	VP Transparency	VP Plot Style
✓	0				white	Continuous	Default	0	Color 7					white	Continuous	Default	0	Color 7
	BORDER				white	Continuous	Default	0	Color 7					white	Continuous	Default	0	Color 7
	BORDERTXT				white	Continuous	Default	0	Color 7					white	Continuous	Default	0	Color 7
	Depoints				white	Continuous	Default	0	Color 7					white	Continuous	Default	0	Color 7
	G-ANNO-DIMS				white	Continuous	0.18	0	Color 7					white	Continuous	0.18 mm	0	Color 7
	G-ANNO-KEYN				white	Continuous	0.18	0	Color 7					white	Continuous	0.18 mm	0	Color 7
	G-ANNO-LEGN				white	Continuous	0.18	0	Color 7					white	Continuous	0.18 mm	0	Color 7
	G-ANNO-MASK				113	Continuous	0.18	0	Color 113					113	Continuous	0.18 mm	0	Color 113
	G-ANNO-MATC				ma...	Continuous	0.35	0	Color 6					magenta	Continuous	0.35 mm	0	Color 6
	G-ANNO-NOTE				yell...	Continuous	0.35	0	Color 2					yellow	Continuous	0.35 mm	0	Color 2
	G-ANNO-NPFT				blue	Continuous	0.18	0	Color 5					blue	Continuous	0.18 mm	0	Color 5
	G-ANNO-PATT				blue	Continuous	0.18	0	Color 8					blue	Continuous	0.18 mm	0	Color 8
	G-ANNO-ROME				blue	Continuous	0.18	0	Color 5					blue	Continuous	0.18 mm	0	Color 5
	G-ANNO-REDI				red	Continuous	0.25	0	Color 1					red	Continuous	0.25 mm	0	Color 1
	G-ANNO-REFR				ma...	Continuous	0.35	0	Color 6					magenta	Continuous	0.35 mm	0	Color 6
	G-ANNO-REVS				cyan	Continuous	0.50	0	Color 4					cyan	Continuous	0.50 mm	0	Color 4
	G-ANNO-SCHD				white	Continuous	0.18	0	Color 7					white	Continuous	0.18 mm	0	Color 7
	G-ANNO-SYMB				ma...	Continuous	0.35	0	Color 6					magenta	Continuous	0.35 mm	0	Color 6
	G-ANNO-TEXT				white	Continuous	0.18	0	Color 7					white	Continuous	0.18 mm	0	Color 7
	G-ANNO-TLB				white	Continuous	0.18	0	Color 7					white	Continuous	0.18 mm	0	Color 7
	G-ANNO-TLB-GRID				blue	Continuous	0.18	0	Color 5					blue	Continuous	0.18 mm	0	Color 5
	G-COOR-LALO				yell...	Continuous	0.25	0	Color 2					yellow	Continuous	0.25 mm	0	Color 2
	G-COOR-LALO-IDEN				yell...	Continuous	0.25	0	Color 2					yellow	Continuous	0.25 mm	0	Color 2
	G-COOR-STAT				yell...	Continuous	0.25	0	Color 2					yellow	Continuous	0.25 mm	0	Color 2
	G-COOR-STAT-IDEN				yell...	Continuous	0.25	0	Color 2					yellow	Continuous	0.25 mm	0	Color 2
	G-GRID-COOR				white	Continuous	0.25	0	Color 7					white	Continuous	0.25 mm	0	Color 7
	G-GRID-COOR-IDEN				white	Continuous	0.25	0	Color 7					white	Continuous	0.25 mm	0	Color 7
	G-GRID-EXTR				blue	CENTER	0.18	0	Color 5					blue	CENTER	0.18 mm	0	Color 5
	G-GRID-IDEN				red	Continuous	0.25	0	Color 1					red	Continuous	0.25 mm	0	Color 1
	G-PLAN-OTLN				ma...	Continuous	0.35	0	Color 6					magenta	Continuous	0.35 mm	0	Color 6

The “Layer Properties Manager” window will pull up when you click the “Layer Properties” icon from the panel. The layer properties manager will display the layers that are loaded in your drawing. For each layer, the following headings are given: Status, Name, On, Freeze, Lock, Color, Linetype, Lineweight, Transparency, Plot Style, Plot, New VP Freeze, Description, VP Freeze, VP Color, VP Linetype, VP Lineweight, VP Transparency, and VP Plot Style. If you do not want to display any of these settings, you can simply right-click on the heading and uncheck the checkbox beside the heading you do not want to display. You can also sort the heading however best suits you and the type of drawings you are working on. If you click on a heading, the column will sort ascending or descending.

### LAYOFF

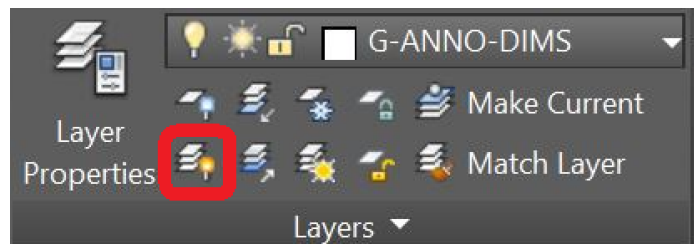
Under the “Layers” panel in the ribbon, there is a button labeled “Off” or “LAYOFF”. This button will allow you to click on an object in your drawing and it will turn everything on that

layer off. You can also type “LAYOFF” in the command line to start the same command. This command is helpful if you need to look at an object unobstructed or if you need to turn layers off when printing. (Ex. Reference lines)



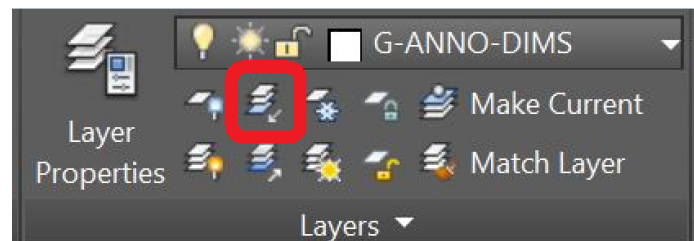
## LAYON

Directly underneath the “Off” button is a button labeled “Turn All Layers On” or “LAYON”. This button will allow you to turn all the layers in the drawing on in one click. If the layer is also frozen, it will not become visible at this time. You can also type “LAYON” in the command line to start the same command.



## LAYISO

Directly beside the “Off” button is a button labeled “Isolate” or “LAYISO”. This button will allow you to hide all layers except those of the selected layers. The layers that remain after clicking on an object are isolated. This command is beneficial when you want to view just one specific layer in a crowded drawing. You can also type “LAYISO” in the command line to start the same command.

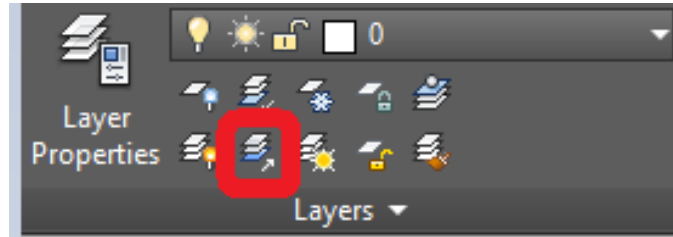


## LAYUNISO

Directly under the “LAYIOS” button is a button labeled “Unisolate” or “LAYUNISO”. This button will allow you to display all the layers that were hidden or locked by the layer isolate command. This command is beneficial when you want to quickly display all the layers that

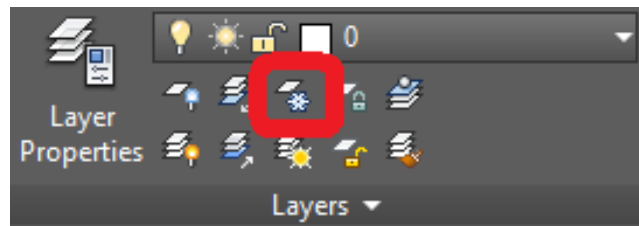


you had previously frozen. You can also type “LAYUNISO” in the command line to start the same command.



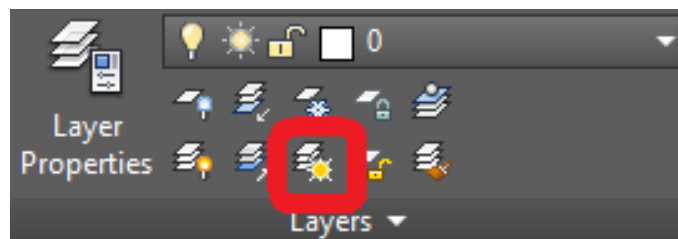
## LAYFRZ

Directly beside the “LAYIOS” button is a button labeled “Freeze” or “LAYFRZ”. This button will allow you to freeze the layer of the selected object. This command is beneficial when you want to freeze one layer at a time. This command will allow you to quickly freeze a layer that is in your drawing. You can select more than one layer at a time to freeze. You can also type “LAYFRZ” in the command line to start the same command.



## LAYTHW

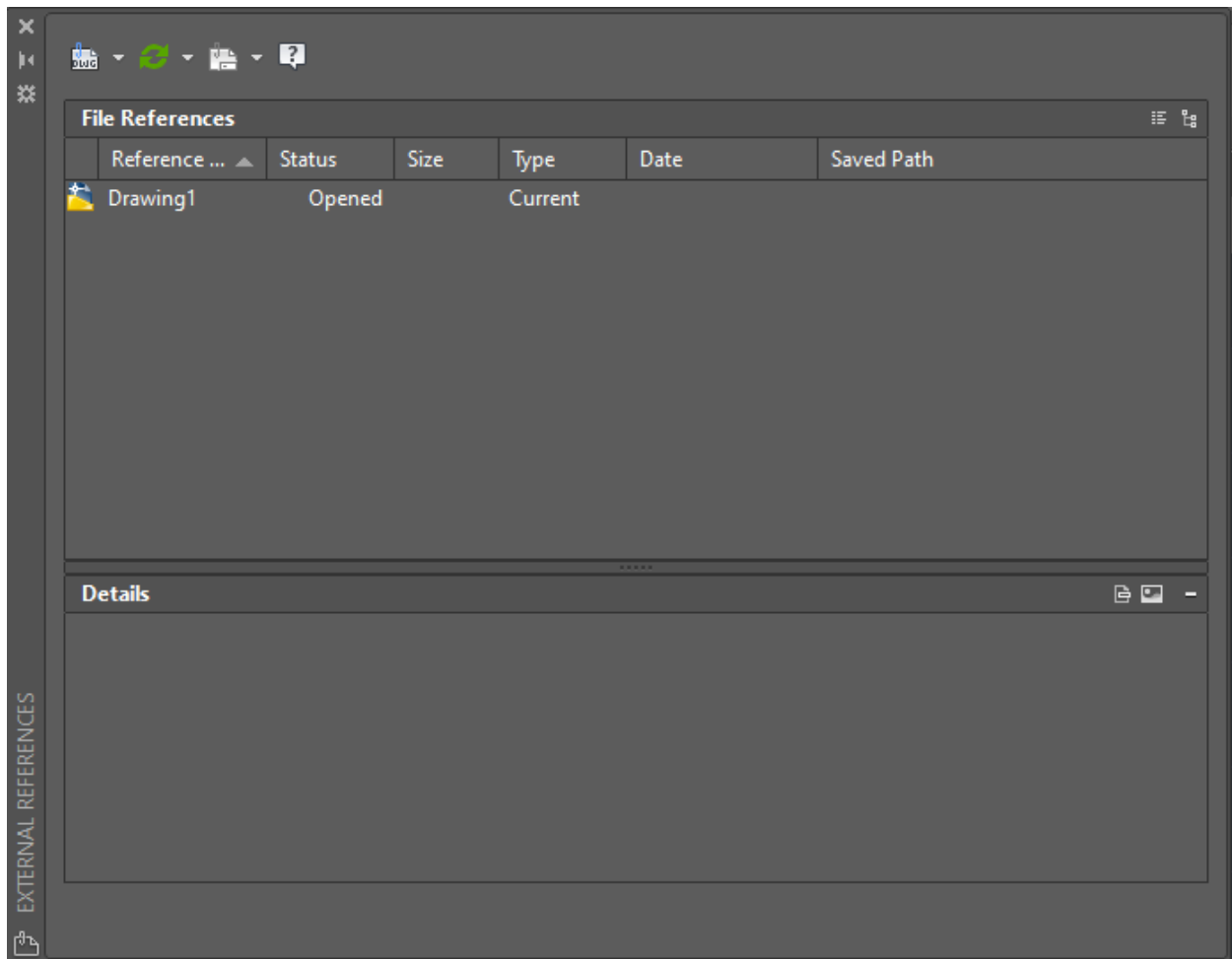
Directly under the “LAYFRZ” button is a button labeled “Thaw All Layers” or “LAYTHW”. This button will allow you to thaw all the layers that were previously frozen in the drawing. If the layer is turned off and frozen, it will still not become visible because it is turned off. If the layer is frozen in a viewport, you will still have to unfreeze the layer in the viewport to become visible. This command will allow you to quickly display all the frozen layers in your drawing. You can also type “LAYTHW” in the command line to start the same command.



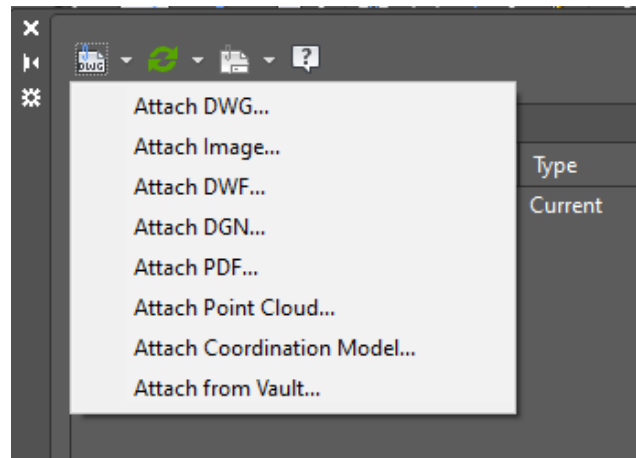
TIP: AutoCAD still sees things that are on an “OFF” layer but does not see those items if the layer is “Frozen”. Therefore, regens are faster if layers are frozen. Also, you can turn off the current layer, but you cannot freeze the current layer.

## External Reference

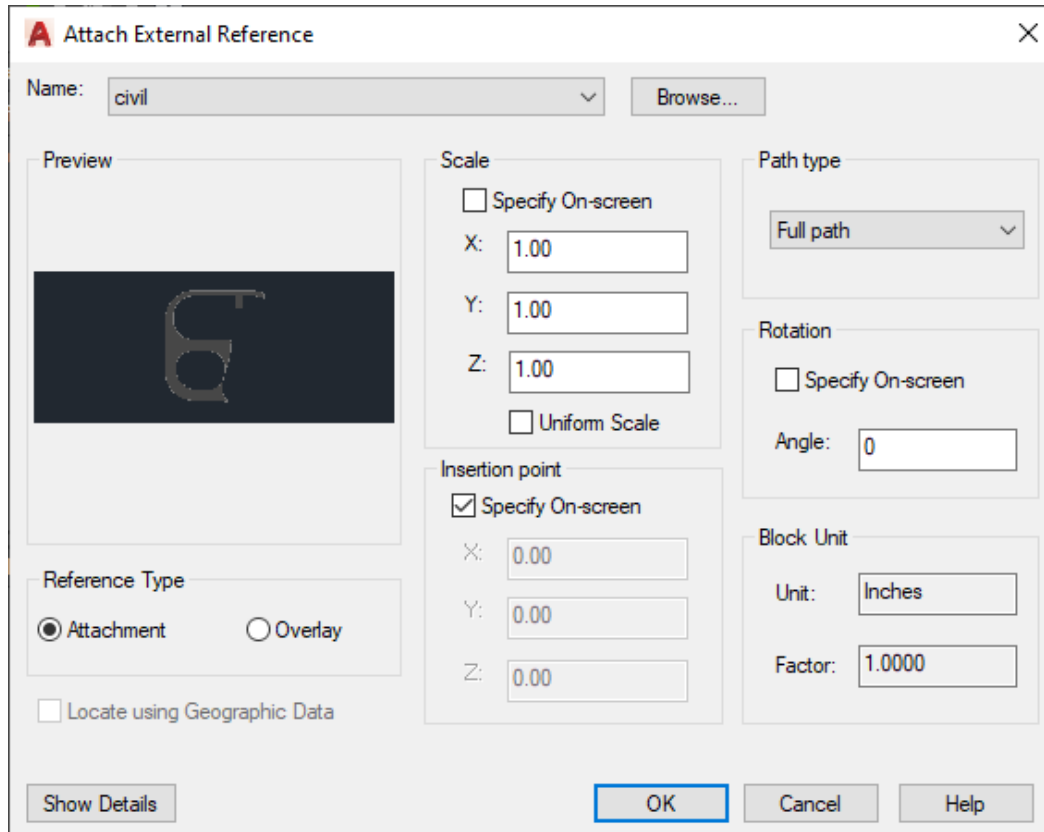
You can insert any drawing file as an external reference (x-ref) in the current drawing. You can also insert types of image files. The benefit of using an x-ref is when the referenced drawing changes, it will update the current drawing. There are several methods for attaching x-refs; you can type “EXTERNALREFERENCES” or “XATTACH” in the command window, you can also go to the “View” tab, then “Palettes” panel, then “External References” palette, or you can go to the “Tools” menu, then click “Palettes”, then click “Reference Manager”. Any of these methods will open the external reference window.



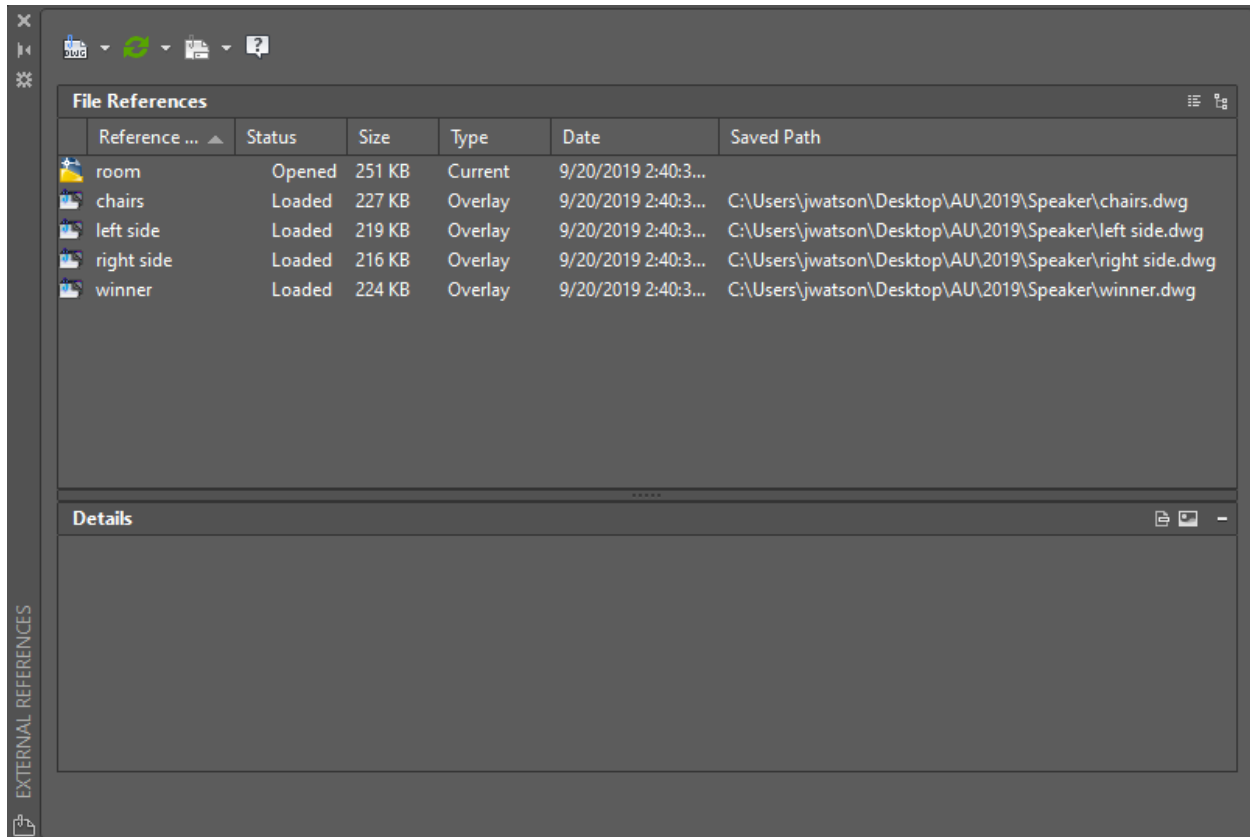
The default option for attaching an xref is a .dwg file, but if you click the drop-down arrow, you will see there are several other types of files you can attach.



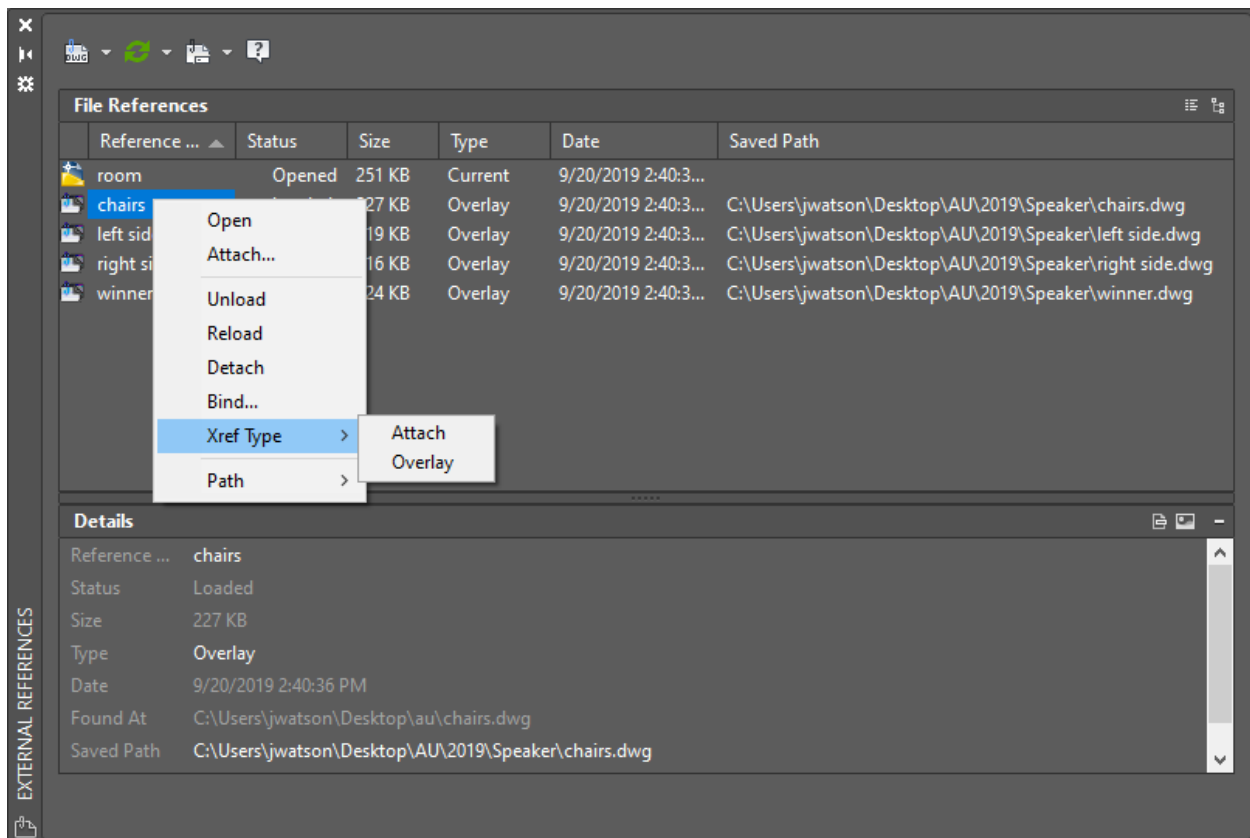
When you click on the dwg icon, the “Select Reference File” window will appear. From there you will go to the location of the file you want to attach and click open. Then the “Attach External Reference” window will appear. From here you have a few different options. You can specify the insertion scale, the insertion point, the rotation, the path type, and the reference type. The Scale, Insertion Point, and Rotation can also be specified on screen if you click the “Specify On-Screen” option. There are two options for Reference Type; “Attachment” or “Overlay”. “Overlay” is best used when you want to insert a drawing as reference. When you use the “Attachment” option then the drawing becomes part of the master drawing.



After you have made the appropriate selections, the referenced file will appear in your master drawing. The referenced file will appear in the “External Reference” window.



If you right-click on the referenced file, it will bring up a couple of options. You can “Unload” the file, and it will not appear in your drawing. It will still show up in the external reference window, but not be visible in your drawing. You can also go and “Reload” the file if you want it to reappear in your drawing. You can “Detach” the reference file, and it will no longer appear in the drawing or in the external reference window. You can “Bind” the referenced file to your drawing. Once you click bind it will bring up a dialog that gives you the options of “Bind” or “Insert”. You can also change the xref type from “Overlay” or “Attachment”. The last option you have is changing the path type in the bottom option. When you select the referenced file in the external reference window it will also display the location the file is saved at.

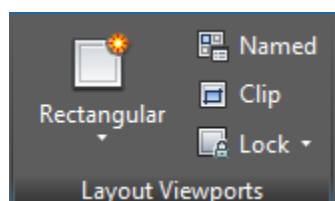


## Model Space vs. Paper Space

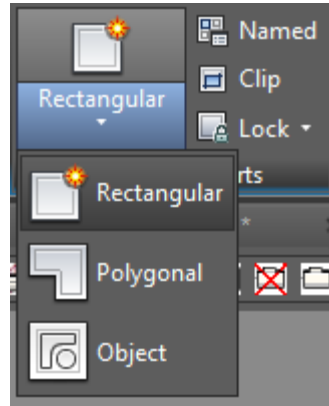
Model space/paper space can be a great way to layout drawings. Across the bottom of AutoCAD, below the command line, you will see “Model” and “Layout” tabs. The model space is where the objects of your drawing will be housed. In the layout tabs you will find the borders and annotations. A viewport is always placed in the “Layout” tab. A viewport can be explained as taking a sheet of paper and cutting a hole in it. By looking through the hole, you are primarily looking into model space. You can zoom in and out and pan around looking at the model space.

### Creating a viewport

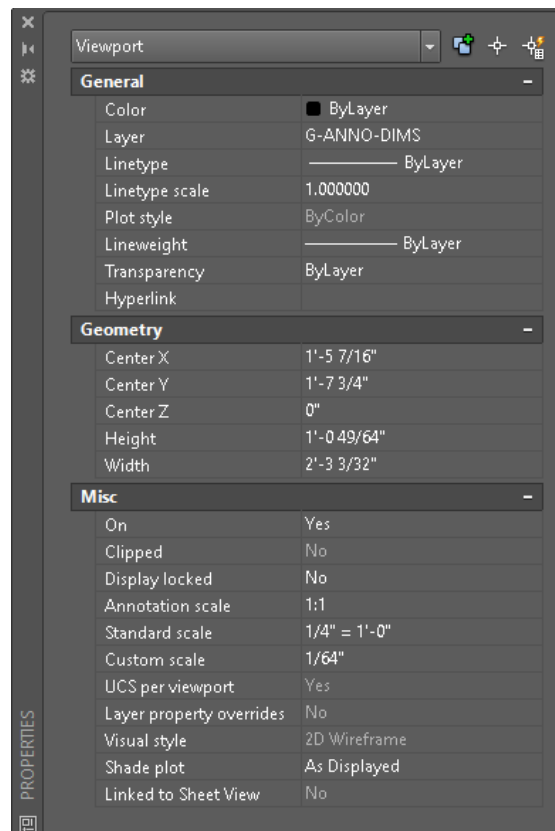
To create a viewport, you will first want to select the appropriate layout tab across the bottom. Then you will need to set the current layer to the layer that is reserved for your viewport window. (Ideally on a no print layer) Then click on the layout tab, and under Layout Viewports, you will see the options for viewports.

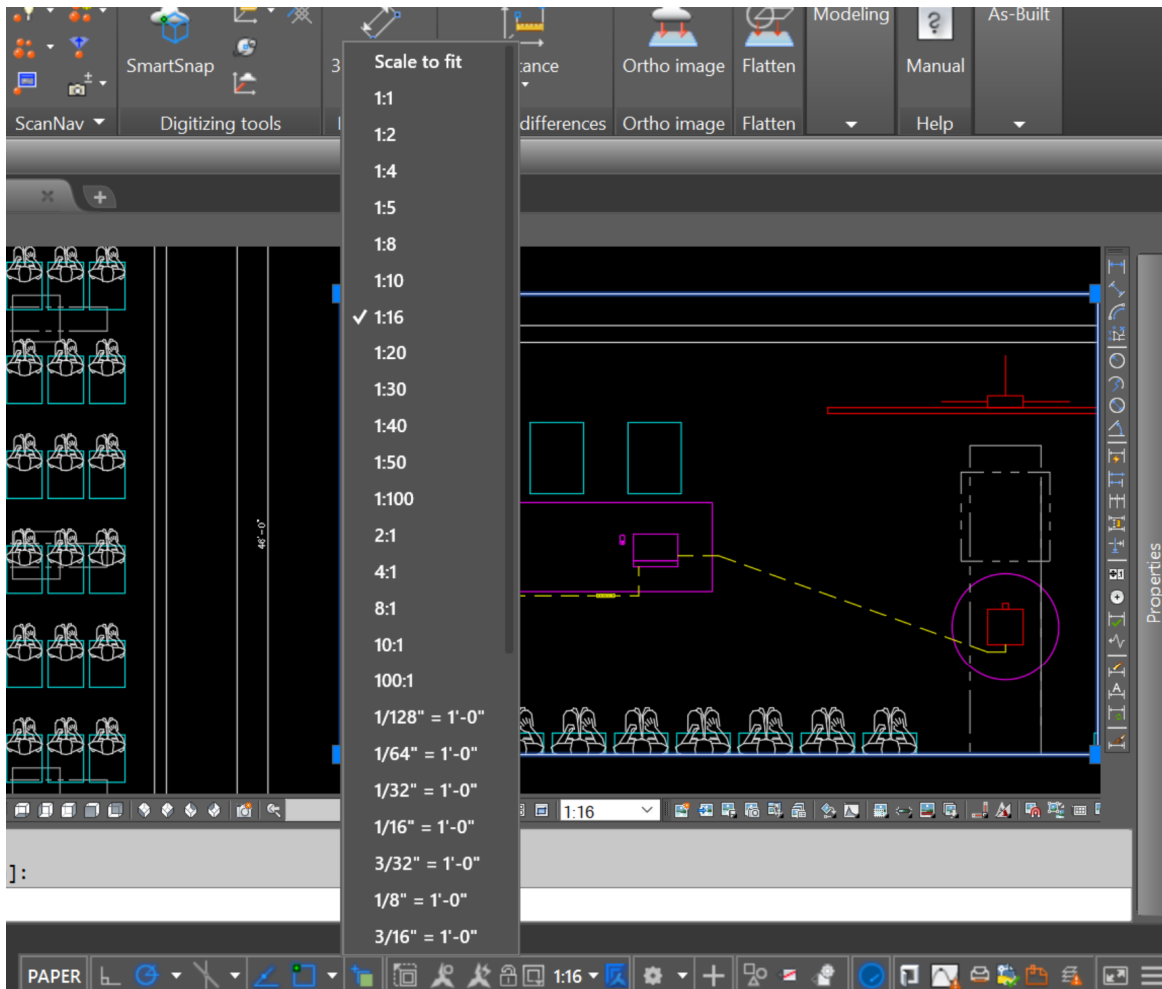


On the left side of the “Layout Viewports” panel, you will find options for inserting a viewport. A “Rectangular” viewport is the default, but you can also select a “Polygonal” viewport or “Object” viewport.

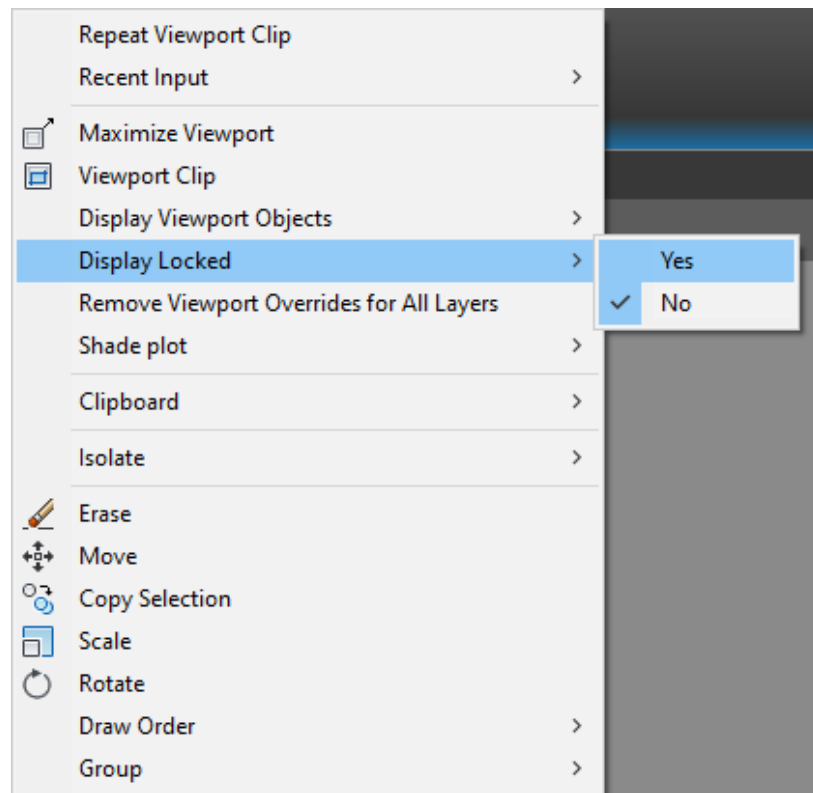


Once you have selected the desired type of viewport, it will prompt you in the command window to select the corners of the viewport. Now that the viewport has appeared, you need to select the scale. To do this, select the viewport and go to properties. Within the properties window, there is a setting for “Standard Scale”. You can find in the status bar along the bottom a setting for selecting the scale of the selected viewport. These are the ways you can set the scale for the viewport.





Now that you have the scale set, you can reposition the viewport by the move command and resize the viewport by dragging the grips once you select the viewport. As soon as possible, you need to lock the viewport. By locking the viewport, you lock the scale, so you don't accidentally change it. You can lock the viewport by selecting the viewport and either setting it to be locked in the properties window or by right-clicking, then select "Display Locked" and choose "Yes".



## Manipulating Layers in a Viewport

Now that you have your viewport set in your drawing and it is locked, you have a few options to display objects in a viewport differently. When you double click inside the viewport, (and the viewport window becomes a thicker line), you are then in model space. Once inside the viewport, open your “Layer Properties” from the “Home” menu, under the “Layers” panel.



New VP Freeze	VP Freeze	VP Color	VP Linet...	VP Linew...
		white	Continu...	— Defa...
		cyan	Continu...	— 0.18...
		magenta	Continu...	— 0.35...
		red	Continu...	— 0.35...
		white	Continu...	— 0.35...
		white	Continu...	— Defa...
		magenta	Continu...	— Defa...
		cyan	Continu...	— 0.18...
		magenta	Continu...	— 0.35...
		red	Continu...	— 0.35...
		white	Continu...	— 0.35...
		white	Continu...	— Defa...
		magenta	Continu...	— Defa...
		yellow	chairs[D...	— 0.00...
		white	Continu...	— 0.18...
		white	Continu...	— 0.18...
		red	Continu...	— 0.18...
		white	Continu...	— 0.00...
		white	Continu...	— 0.00...
		white	Continu...	— 0.50...
		white	Continu...	— 0.18...
		cyan	Continu...	— 0.18...
		white	Continu...	— 0.18...

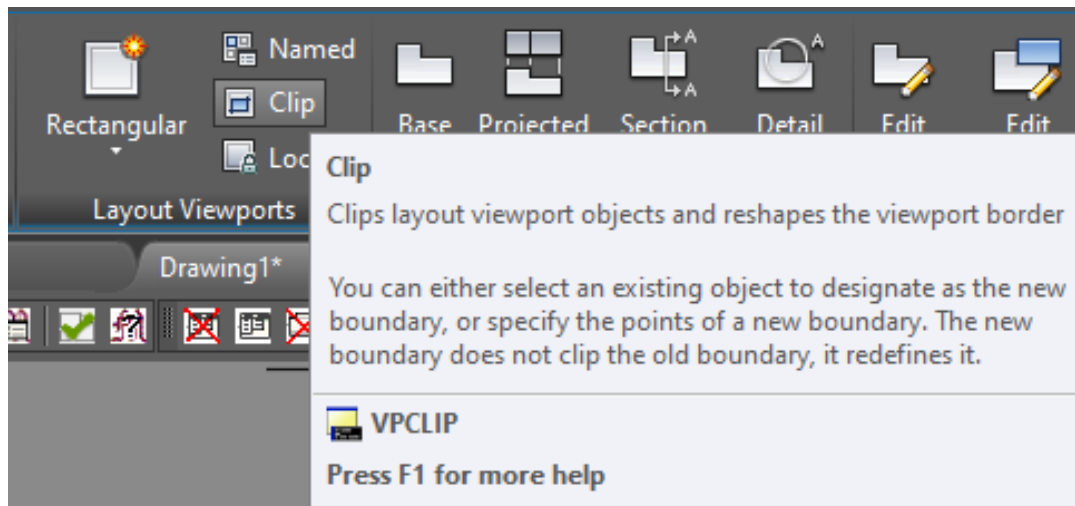
You will see that you have your standard settings for the layers, but you also have some viewport settings. From the “Layer Manager” you can change how an object shows up in the viewport. If you freeze the new viewport setting, then when you make a new viewport that layer will not display. If you have a layer showing in your viewport and you want to keep it turned on in model space but not displayed in the viewport, then you can click the icon under “VP Freeze”. The following setting will allow you change the color of an object inside the viewport but leave the color alone in model space. You can also change the line type and line weight settings for the viewport under the “VP” settings, this will also leave the settings in model space alone.

### Model Space and Paper Space on the Layout Tab

When you are on the layout tab, and you have already created your viewports, you can double click inside the viewport to enter model space. Once you are ready to go back to paper space on the layout tab you can double click outside of the viewport. One way to tell if you are in model space or paper space on the layout tab is to look and see if the viewport border is a thin or thick line. If the viewport border is thick, then you are in model space. If the viewport border is thin, you are in paper space.

## Viewport Clipping

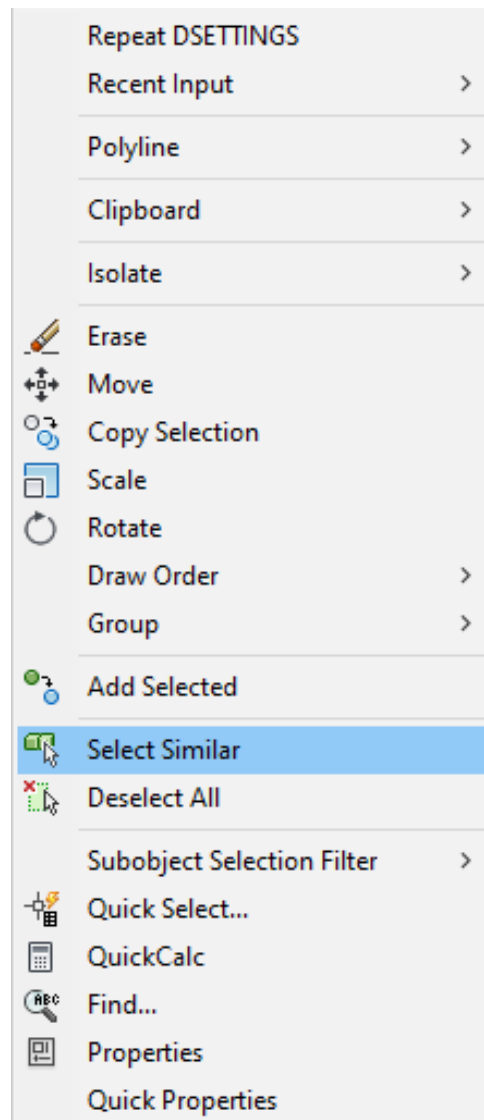
You can clip a viewport if you have inserted a viewport and need to make room to place other objects in the “Layout” tab. To clip a viewport, go to the “Layout” tab, and under “Layout Viewports” panel you will see the icon for “Clip”, or you can type “VPCLIP” in the command window. Once the command starts it will ask you to select a viewport, and then it will ask you to clip the viewport. This can be helpful if you already have your drawings setup but need to add something to your drawing, and you don’t want to have to change the scales of your viewports.



## Bonus Material

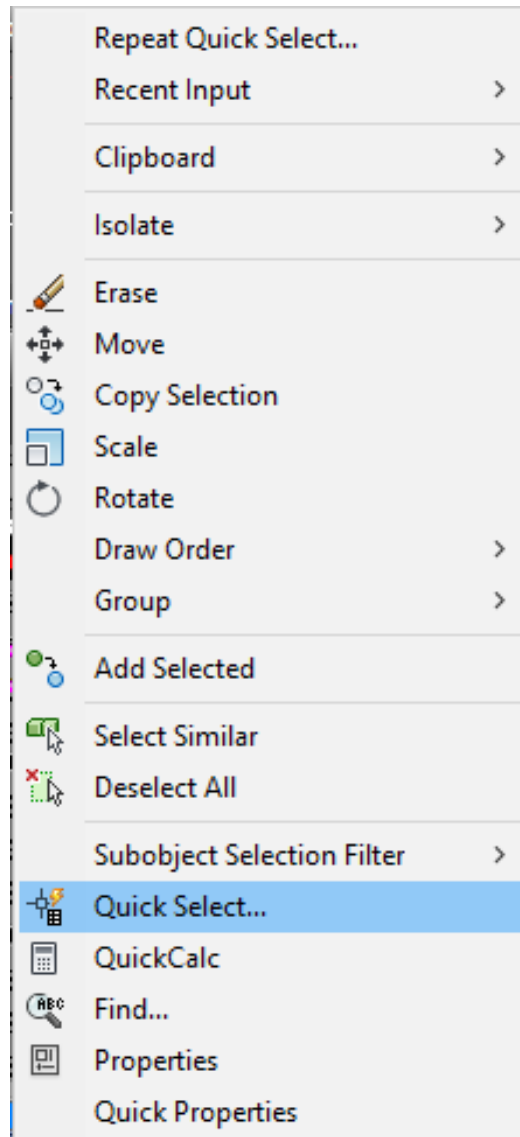
### Select Similar

The “Select Similar” command will select objects of the same type (line, polyline, rectangle, circle, etc.). To execute this command, you select an object in your drawing for the category of objects that you want to select. Select the object, right-click and choose “Select Similar”.

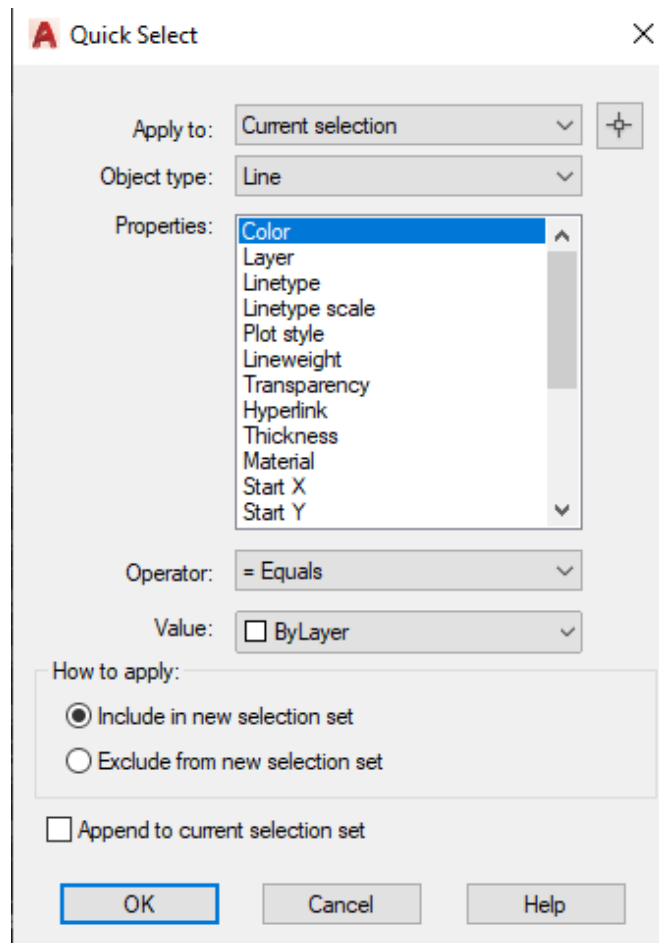


## Quick Select

The “Quick Select” command creates a selection set by filtering by object type and property. To execute this command, you right-click to bring up the “Shortcut Menu” and select “Quick Select” or type “QSELECT” in the command line.

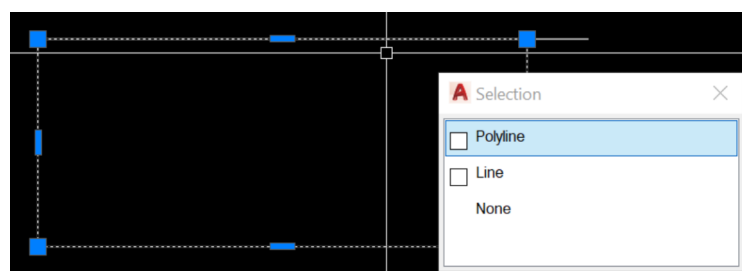
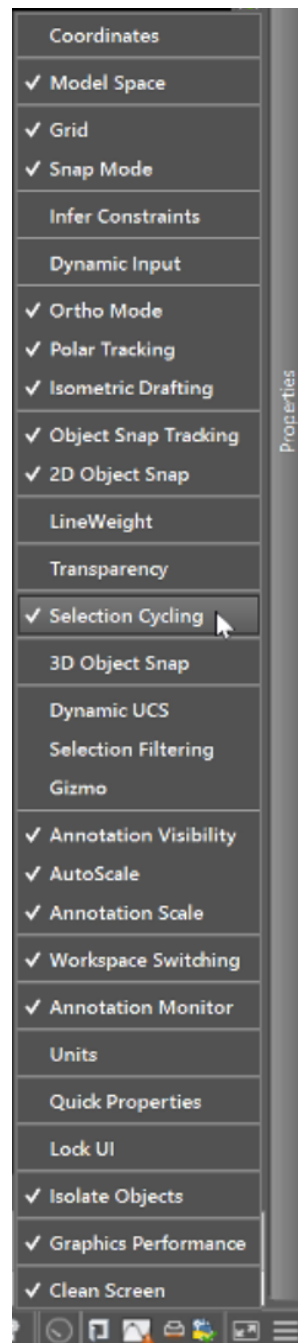


The Quick Select window will appear. From within this window, you can select if you want the selection to be for the objects selected in the drawing or the entire drawing. You can also select the objects you want the selection to apply to. Then you can select the properties that you want to filter your selection by. Once you select the property you can change what you want that value to be set to. The Quick Select command is beneficial for narrowing down your selection in a crowded drawing.



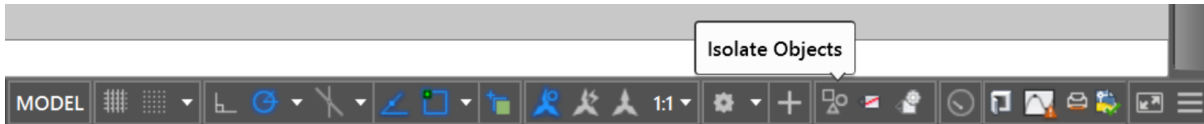
## Selection Cycling

Selection Cycling will allow you to toggle through objects that overlap each other. You can turn on selection cycling by clicking the “Customization” button in the bottom right and then pick selection cycling, or you can type “Ctrl + W” in the command line. When selection cycling is turned on, you pick on objects that are overlapping, then a window will appear that will allow you to toggle through the objects and select the one you want. This command is especially helpful when you are working with a point cloud and objects.

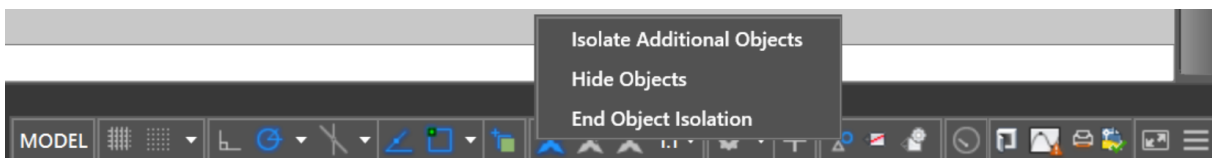


## Isolate/Hide Objects

The “Isolate Objects” command is another tool that will allow you to isolate objects not based on their layer, but simply by selecting the objects you want to display and select “Isolate Objects” from the taskbar or by typing “Isolateobjects” in the command window.



Once you have initially isolated objects, you can select the icon in the taskbar and either isolate additional objects, hide objects, or end object isolation.



You can also select objects and click the same icon in the status bar, then select “Hide Objects” or type “hideobjects” in the command line. This will hide the objects you had selected.



Once you are ready to view all the objects again, you can select “End Object Isolation”. This tool is helpful when working in a congested 3d model; especially when you are trying to put in a connection with two perpendicular beams intersecting.

