GISCAD 2018 V1.1 Installation Notes

For Level 3 ASPs

Prepared by Network Data and Performance

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Alexis Bowman
Asset Information Systems Trainer
Network Data and Performance

Endeavour Energy

CONTENTS

Endeavour Energy

1. GISCAD Support

GISCAD SUPPORT

For help with GISCAD
Please contact the
Endeavour Energy Help Desk on

02 9853-6888

The Help Desk will log your request and A2K Technologies (GISCAD Vendor) will contact you with a solution

2. Introduction

This document has been created to provide Endeavour Energy Designers with relevant information and instructions to support updates and enhancements and provide revision notes about the installation of the GISCAD file.

3. Obtaining the Installation Files

- install files can be downloaded here:

  MS Access Database Engine:

- https://www.dropbox.com/s/gti9id522y6yeog/AccessDatabaseEngine_x64%20-%202010.exe?dl=0

  GISCAD 2018 (64-bit) *[Version 1.1]*:

- https://www.dropbox.com/s/p6kv5f1spz2vw7e/GISCAD%202018%20x64.zip?dl=0

4. System Requirements and Permissions for 64-Bit Systems

To best utilise the GISCAD Tools, the following operating systems and software are recommended:

- 64-Bit hardware running Windows 10 operating system

NOTE:
The Endeavour Energy Help Desk will only support the GISCAD tools if they are installed in AutoCAD 2018.

- Microsoft Office 64-Bit
- If you do not have Microsoft Office installed, you will need to install the 64-Bit Microsoft Access Driver: AccessDatabaseEngine_x64.exe. It can be downloaded from here: https://www.dropbox.com/s/gti9id522y6yeog/AccessDatabaseEngine_x64%20-%202010.exe?dl=0
- Read/Write/Execute permissions to the C:\ACAD2018_GISCAD folder
- AutodeskDesignRevSetup.exe – Design Review Software – to enable use of the SOPS Viewer - Can be found at http://usa.autodesk.com/design-review/.
- AutoCAD 2018 (64-bit) / AutoCAD Map 2018 (64-bit)

5. Setting Up Your User DSN and System DSN

5.1 DSN – Data Source Name

Provides connectivity to a database through an ODBC driver. The DSN contains the database name, directory, database driver, User ID, password, and other information. Once a DSN has been created for a particular database, it can be used in an application to call information from the database.

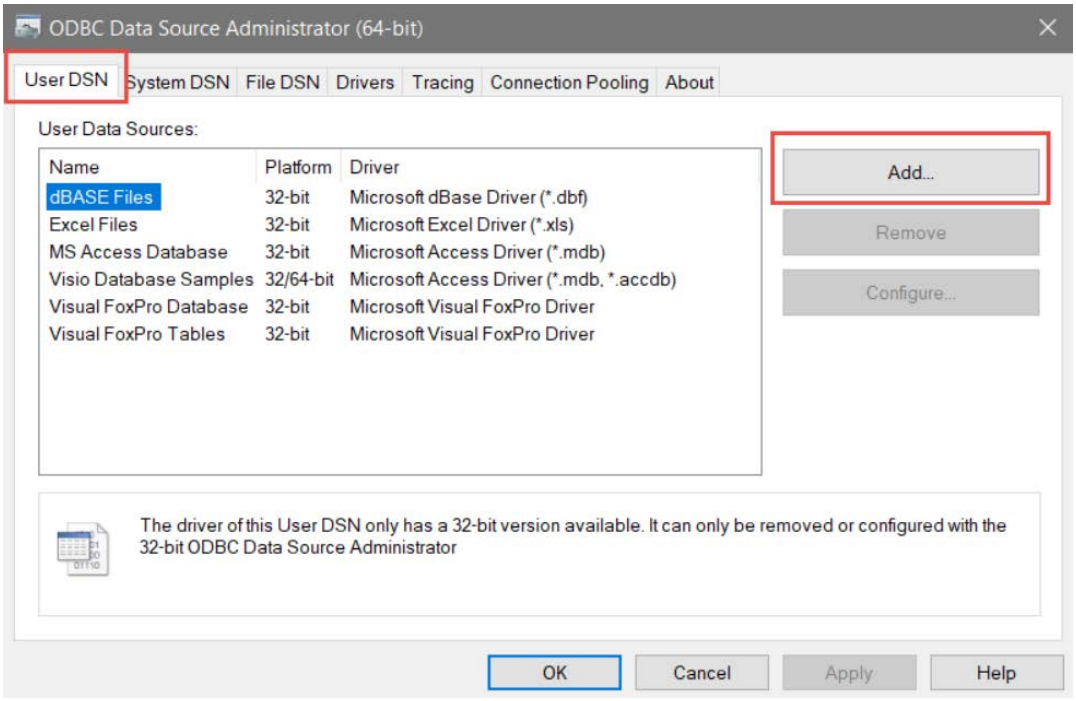
User DSN - Created for a specific user. Information is stored in the registry

System DSN - Can be used by anyone who has access to the machine. The information is also stored in the directory.

5.2 Setting Up Your User DSN

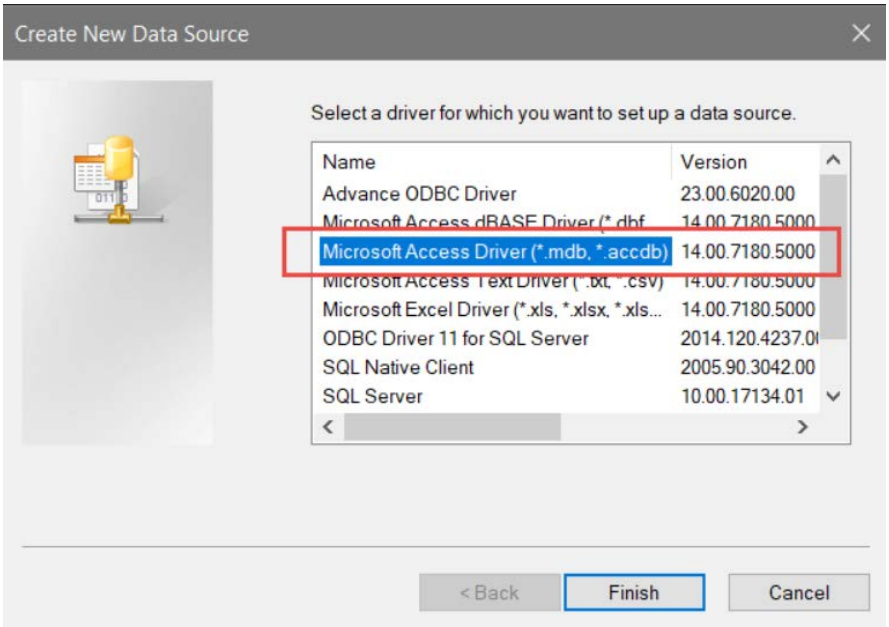
i. Via the Control Panel→Administrative Tools, select "ODBC Data Sources (64-bit)"

ii. The following screen will display;

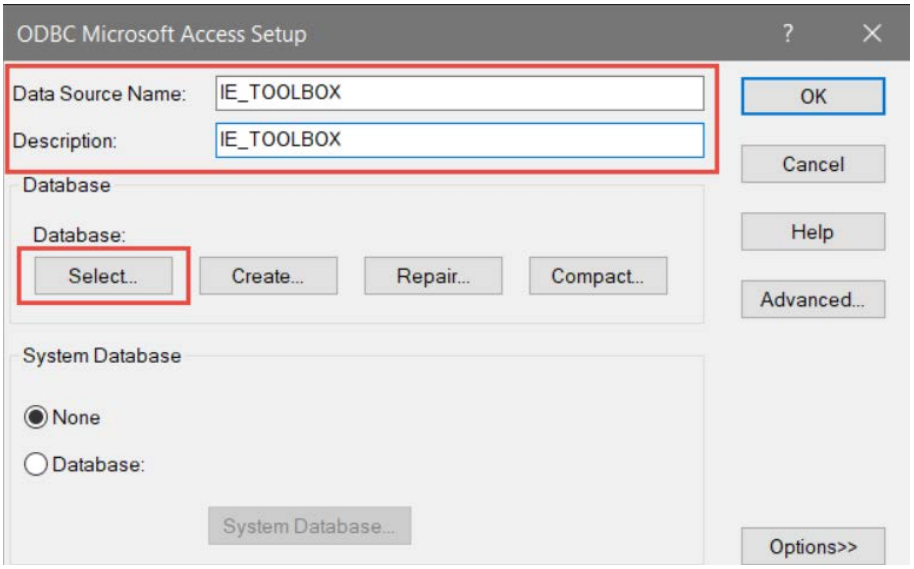


iii. Select the User DSN tab and the Add button;

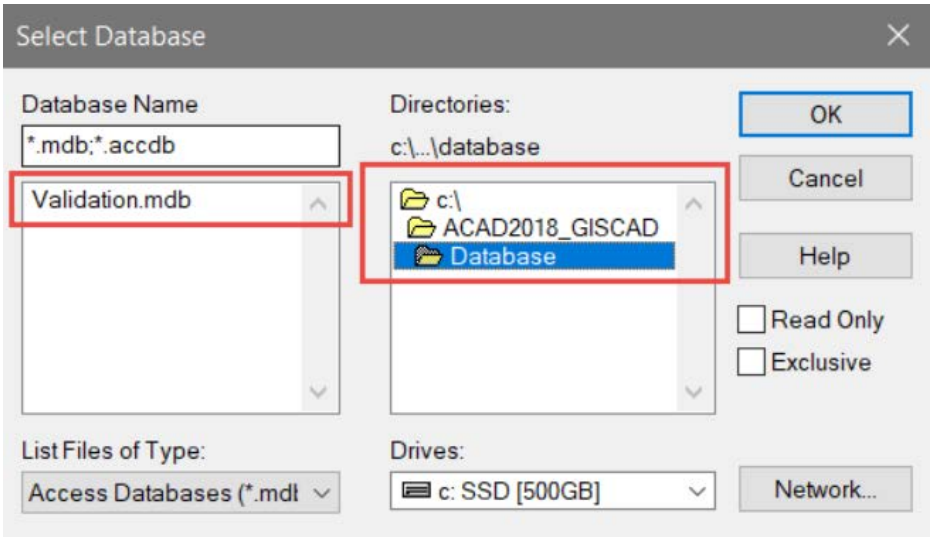
iv. Select one of the MS Access drivers and the Finish button;



v. Enter Data Source Name 'IE_Toolbox', and a description and then the 'Select' button;



i. Select C:\ACAD2018_GISCAD\Database\validation.mdb



ii. User DSN is now setup

5.3 Setting Up Your System DSN

A 64-bit System DSN is required.

i. Follow the above steps to set up the (64-bit) System DSN with the following changes.
   a. At step 2 select the ‘System DSN’ tab.
   b. At step 4 Data Source Name = ‘GISCAD’
   c. Complete steps 4 & 5
   d. (64-bit) System DSN is now setup.

6. Installation Method

- **Ensure all applications on your PC are closed before initiating the install process**
- Select ***Setup.exe*** from your downloaded files and follow on screen instructions
- If you need to install the ***AccessDatabaseDriver_x64.exe***, select this file once you have completed the GISCAD tools installation.

NOTE:

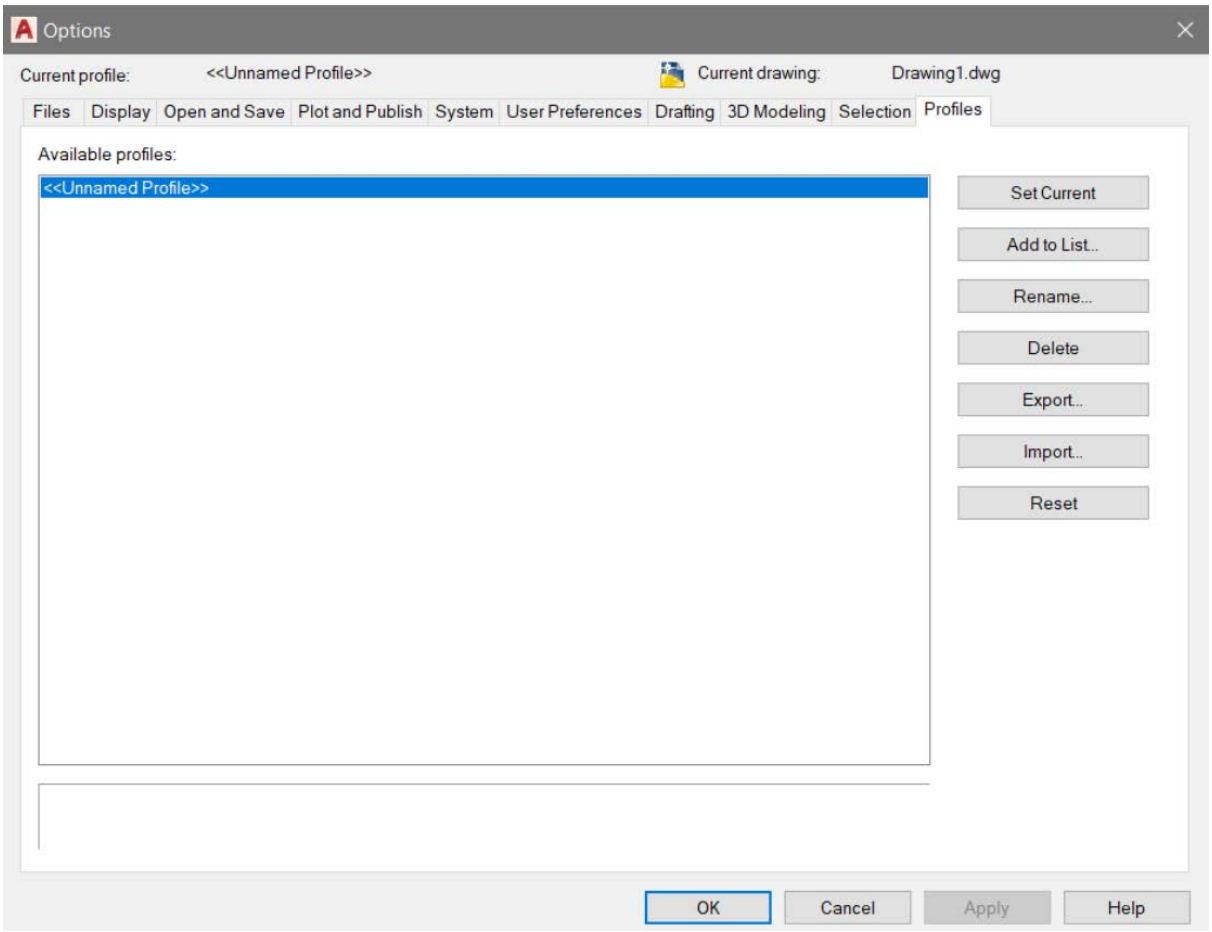
- The Setup.exe will execute the “ENDEAVOUR ENERGY - GISCAD 2018 V1.1” MSI File, the installation of the GISCAD Tools will create the following folder structure:
  C:\ACAD2018_GISCAD
- You must have the correct Read/Write/Execute permissions set up on your PC to enable the creation of the ACAD2018_GISCAD folder in C:\
- After installation of the GISCAD tools, follow the instructions for ‘Importing GISCAD Profile’ and ‘To Confirm the Correct Access and Privileges have been Granted’ (later in this guide) to ensure you have the correct Read/Write/Execute permissions for the creation of the ACAD2018_GISCAD folder in C:\.
- If you have problems with the required C:\ drive permissions, speak with your Manager and your IT Help Desk.

Ensure all other applications are closed prior to installation, to avoid possible installation problems and data loss.

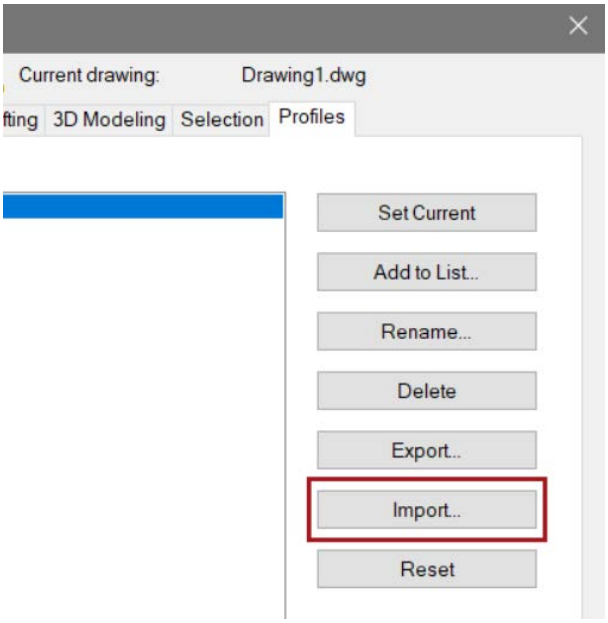
7. Importing GISCAD Profile

Once installation is complete, you will need to import the relevant GISCAD Profile, so that the Endeavour Energy Ribbons, Toolbars and Tool Palettes appear in your AutoCAD Window. To import the relevant profile:

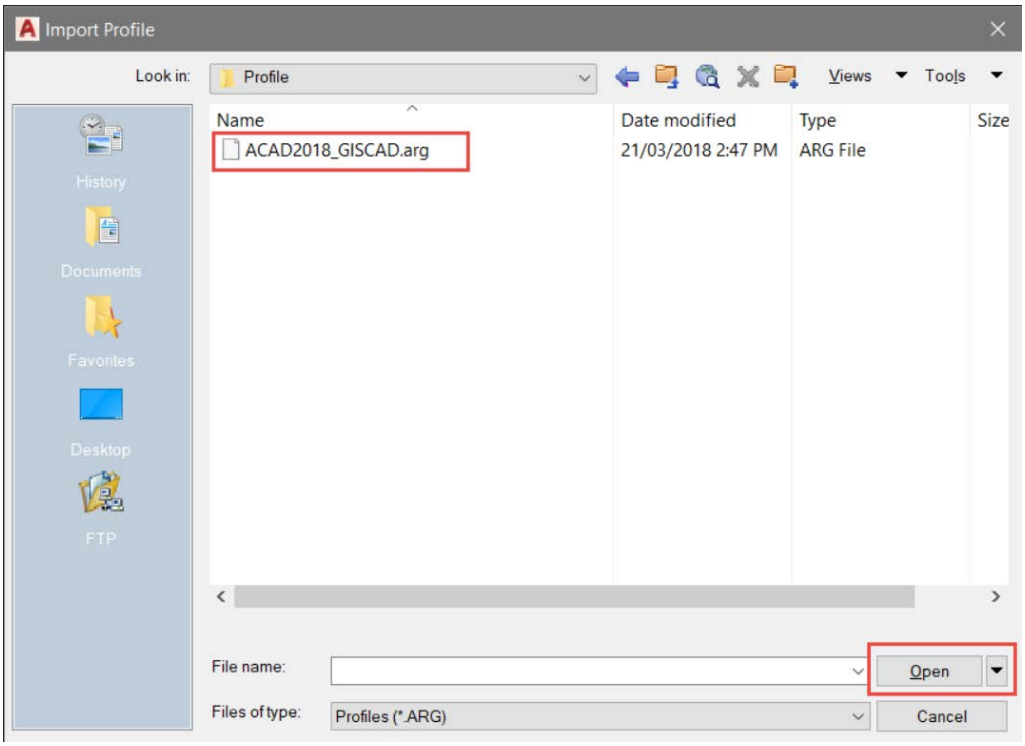
i. Open AutoCAD
ii. Open the Options Window (type options and press enter)



iii. The default AutoCAD profile is titled <<Unnamed Profile>> and provides the window settings for the standard AutoCAD window.
iv. Click the import button

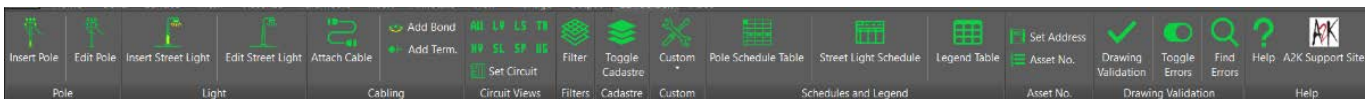


v. The ACAD2018_GISCAD profile is located in C:\ACAD2018_GISCAD\Profile. Navigate to this location.



vi. Select the relevant Profile from this location: And then click Set Current

vii. Your AutoCAD window will now show the Endeavour Energy Toolbars, Ribbons and Tool Palettes.

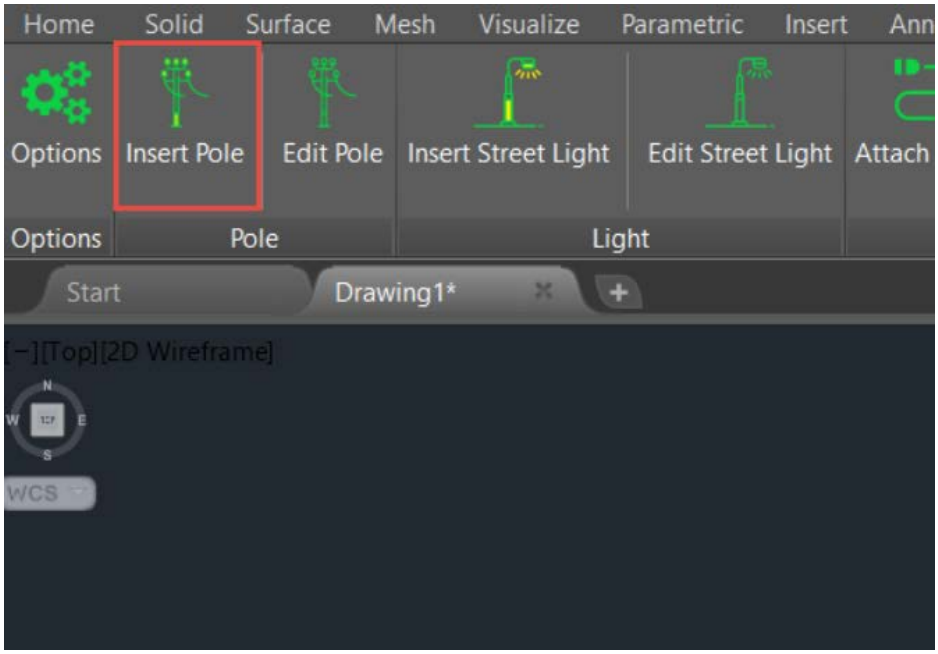


8. To Confirm the Correct Access and Privileges have been Granted

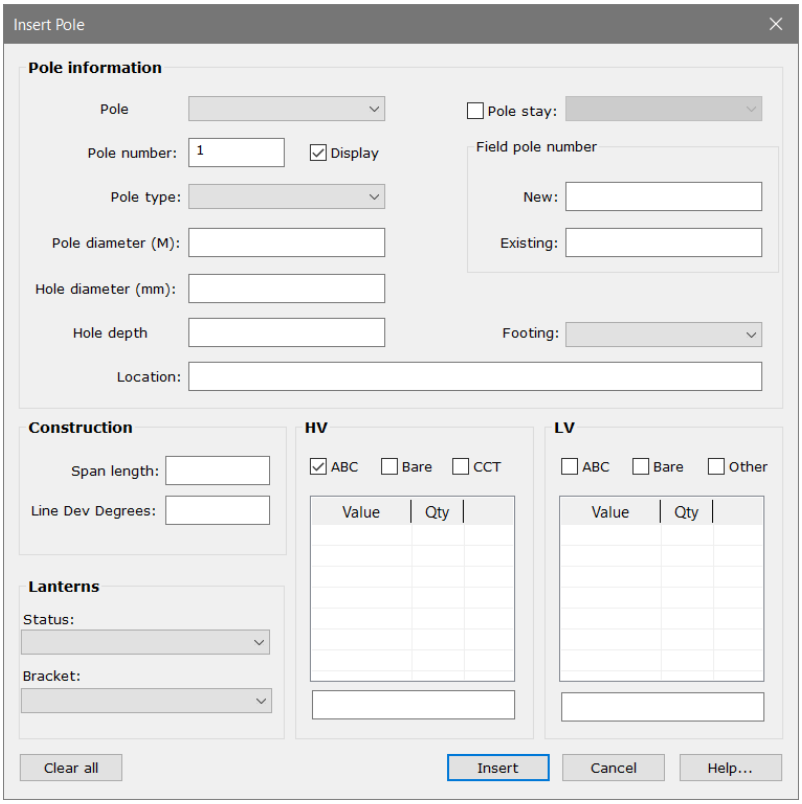
To ensure you have been granted the correct Access and Privileges to C:\Program Files\GISCAD2016, the easiest way is to insert a new pole and run a validation:

i. Ensure you have a Blank / New drawing open

ii. Select the 'Insert Pole' from the toolbar



iii. The Insert Pole dialog window will appear



iv. If this dialog window appears, it is confirmed that you have the correct permissions set up on your PC. Select 'Cancel' to close the dialog

9. IMPORTANT INFORMATION – Viewports

Endeavour Energy has noticed that a number of designers are not using viewports within AutoCAD. It is important that the dwg file you submit must have viewports.

Viewports are a key component in the ability to plot model space geometry from a layout. Each viewport acts as a display portal from the paper layout to the geometry in model space. By creating multiple viewports in a single layout, you can display different aspects of the model geometry at different scales on the same page.

The following pages contain an excerpt of the Autodesk AutoCAD User Guide with information and instructions for creating and using viewports.

1 Display Multiple Views in Model Space

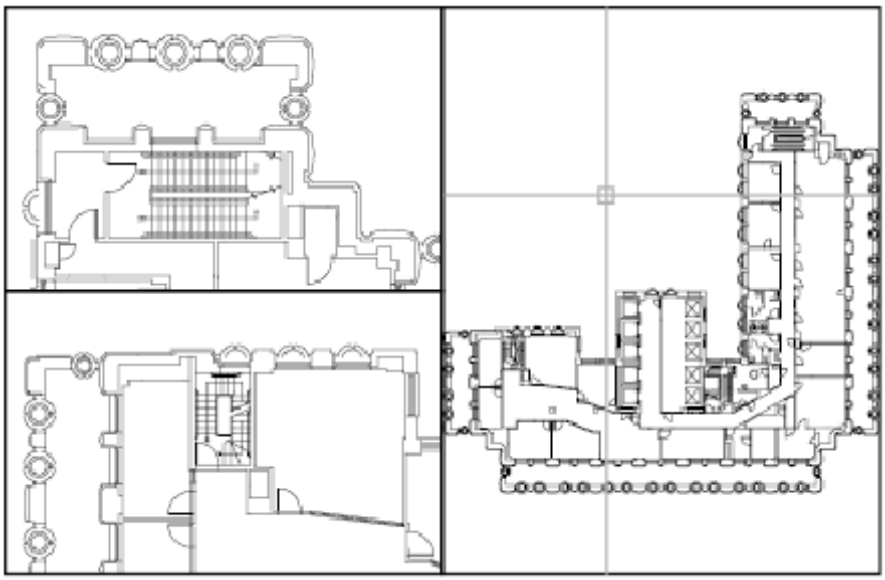
To see several views at the same time, you can split the drawing area of the Model layout into separate viewing areas called *model space viewports*. You can save arrangements of model space viewports for reuse at any time.

2 Set Model Space Viewports

On the Model layout, you can split the drawing area into one or more adjacent rectangular views known as *model space viewports*.

Viewports are areas that display different views of your model. As you work on the Model layout, you can split the drawing area into one or more adjacent rectangular views known as *model space viewports*. In large or complex drawings, displaying different views reduces the time needed to zoom or pan in a single view. Also, errors you might miss in one view may be apparent in the others.

Viewports created on the Model layout completely fill the drawing area and do not overlap. As you make changes in one viewport, the others are updated simultaneously. Three model space viewports are shown in the illustration.



You can also create viewports on a named (paper space) layout. You use those viewports, called *layout viewports,* to arrange the views of your drawing on a sheet. You can move and resize layout viewports. By using layout viewports, you have more control over the display; for example, you can freeze certain layers in one layout viewport without affecting the others.

3 Use Model Space Viewports

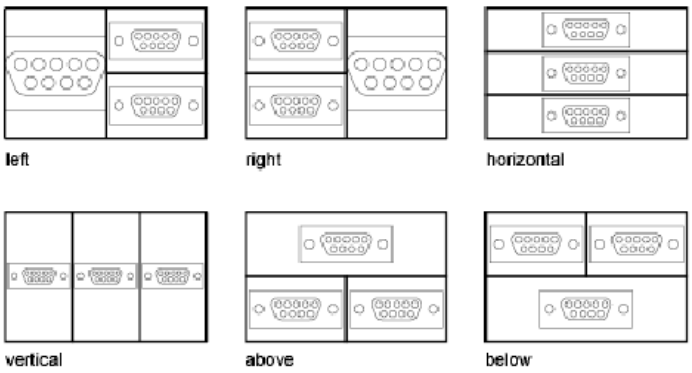
With model space viewports, you can do the following:

- Pan; zoom; set Snap, Grid, and UCS icon modes; and restore named views.
- Save user coordinate system orientations with individual viewports.
- Draw from one viewport to another when executing a command.
- Name a viewport arrangement so that you can reuse it on the Model layout or insert it on a named layout.

Setting up different coordinate systems in individual viewports is useful if you typically work on 3D models.

4 Split and Join Model Space Viewports

The following illustrations show several default model space viewport configurations.



You can easily modify model space viewports by splitting and joining them. If you want to join two viewports, they must share a common edge of the same length.

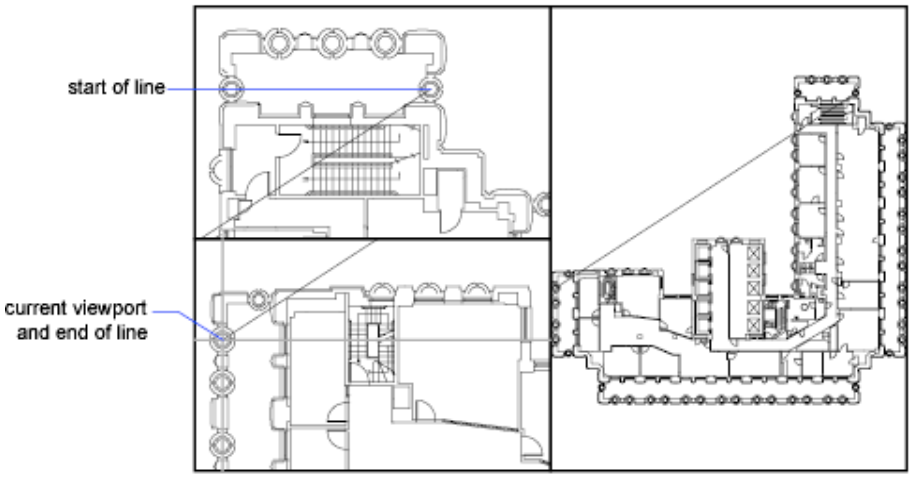
5 Select and Use the Current Viewport

When you use multiple viewports, one of them is the current viewport, which accepts cursor input and view commands.

When a viewport is current, the cursor is displayed as crosshairs rather than an arrow, and the viewport boundary is highlighted. You can change the current viewport at any time except when a View command is in progress.

To make a viewport the current viewport, you click inside it or press Ctrl-R to cycle through the existing viewports.

To draw a line using two model space viewports, you start the line in the current viewport, make another viewport current by clicking within it, and then specify the endpoint of the line in the second viewport. In a large drawing, you can use this method to draw a line from a detail in one corner to a detail in a distant corner.



6 Save and Restore Model Layout Viewport Arrangements

Arrangements of model viewports can be saved and restored by name.

You do not have to set up viewports and views every time you need them. With VPORTS, viewport arrangements can be saved and later restored by name. Settings that are saved with viewport arrangements include

- The number and position of viewports
- The views that the viewports contain
- The grid and snap settings for each viewport
- The UCS icon display setting for each viewport

You can list, restore, and delete the available viewport arrangements. A viewport arrangement saved on the Model layout can be inserted on a named layout.

7 Create Single-View Drawings (Model Space)

To create a two dimensional drawing that has one view, you can create the drawing and its annotation entirely in model space. This is the traditional method for creating drawings with AutoCAD.

With this method, you create the building, mechanical part, or geographic area that you want to represent at full scale (1:1), but you create the text, dimensions, and the title block of the drawing at a scale to match the intended plot scale.

8 Quick Start for Model Space Drafting

The process of creating and plotting a drawing file in model space is very different from the process used in manual drafting.

In AutoCAD, there are two distinct working environments that are represented by Model and named layouts.

If you are going to create a two-dimensional drawing that has one view, you can create both the model and its annotation entirely in model space, not using a layout. This is the traditional method for creating drawings with AutoCAD. This method is simple but has several limitations, including

- It is suitable for 2D drawings only
- It does not support multiple views and view-dependent layer settings
- Scaling the annotation and title block requires computation unless you use objects.

With this method, you always draw geometric objects at full scale (1:1) and text, dimensions, and other annotation at a scale that will appear at the correct size when you output the drawing.

9 Draw, Scale, and Annotate in Model Space

If you draw and plot from model space, you must determine and apply a scale factor to annotate objects before you plot.

You can draw and plot entirely from model space. This method is useful primarily for two-dimensional drawings that have a single view. With this method, you use the following process:

- Determine the unit of measurement (drawing units) for the drawing.
- Specify the display style for the drawing unit.
- Calculate and set the scale for dimensions, annotations, and blocks.
- Draw at full scale (1:1) in model space.
- Create the annotation and insert the blocks in model space.
- Print the drawing at the predetermined scale.

You can also use objects if you want to scale annotations automatically.

9.1 Determine the Unit of Measurement

Before you begin drawing in model space, you determine the unit of measurement (drawing units) that you plan to use. You decide what each unit on the screen represents, such as an inch, a millimetre, a kilometre, or some other unit of measurement. For example, if you are drawing a motor part, you might decide that one drawing unit equals a millimetre. If you are drawing a map, you might decide that one unit equals a kilometre.

9.2 Specify the Display Style of Drawing Units

Once you have determined a drawing unit for the drawing, you need to specify the style for displaying the drawing unit, which includes the unit type and precision. For example, a value of 14.5 can be displayed as 14.500, 14-1/2, or 1'2-1/2".

Specify the display style of drawing units with the UNITS command. The default drawing unit type is decimal.

9.3 Set the Scale for Annotations and Blocks

Before you draw, you should set the scale for dimensions, annotations, and blocks in your drawings. Scaling these elements beforehand ensures that they are at the correct size when you plot the final drawing.

You should enter the scale for the following objects:

- Text Set the text height as you create text or by setting a fixed text height in the text style (STYLE).
- Dimensions Set the dimension scale in a dimension style (DIMSTYLE) or with the DIMSCALE system variable.
- Linetypes Set the scale for non-continuous linetypes with the CELTSCALE and LTSCALE system variables.
- Hatch patterns Set the scale for hatch patterns while creating the hatch (HATCH), before creating the hatch object with the HPSCALE system variable, or edit the hatch after it has been created.
- Blocks Specify the insertion scale for blocks either as you insert them, or set an insertion scale in the Insert Block dialog box (INSERT).

The system variables used for inserting blocks are INSUNITS, INSUNITSDEFSOURCE, and INSUNITSDEFTARGET. This also applies to the border and title block of the drawing.

You can also use objects if you want to scale annotations automatically.

9.4 Determine the Scale Factor for Plotting

To plot your drawing from the Model layout, you calculate the exact scale factor by converting the drawing scale to a ratio of 1:n. This ratio compares plotted units to drawing units that represent the actual size of the objects you are drawing.

For example, if you plan to plot at a scale of 1/4 inch = 1 foot, you would calculate the scale factor 48 as follows:
1/4" = 12"
1 = 12 x 4
1 (plotted unit) = 48 (drawing units)

Using the same calculation, the scale factor for 1 centimeter = 1 meter is 100, and the scale factor for 1 inch = 20 feet is 240.

9.5 Sample Scale Ratios

The sample architectural scale ratios in the table can be used to calculate text sizes in model space.

Scale	Scale factor	To plot text size at	Set drawing text size to
1 cm = 1 m	100	3 mm	30 cm
1/8" = 1'-0"	96	1/8"	12"
3/16" = 1'-0"	64	1/8"	8"
1/4" = 1'-0"	48	1/8"	6"
3/8" = 1'-0"	32	1/8"	4"
1/2" = 1'-0"	24	1/8"	3"
3/4" = 1'-0"	16	1/8"	2"
1" = 1'-0"	12	1/8"	1.5"
1 1/2" = 1'-0"	8	1/8"	1.0"

If you are working in metric units, you might have a sheet size of 210 x 297 mm (A4 size) and a scale factor of 20. You calculate grid limits as follows:

210 x 20 = 4200 mm
297 x 20 = 5900 mm

10 Create Multiple-View Drawing Layouts (Paper Space)

Paper space is a sheet layout environment where you can specify the size of your sheet, add a title block, display multiple views of your model, and create dimensions and notes for your drawing.

11 Quick Start for Layouts

There are two distinct working environments, or “spaces,” in which you can create objects in a drawing. Typically, a model composed of geometric objects is created in a three-dimensional space called *model space*. A final layout of specific views and annotations of this model is created in a two-dimensional space called *paper space*.

Working in model space, you draw a model of your subject at 1:1 scale. Working on a named layout, you create one or more *layout viewports*, dimensions, notes, and a title block to represent a drawing sheet.

Each layout viewport is like a picture frame containing a “photograph” of the model in model space. Each layout viewport contains a view that displays the model at the scale and orientation that you specify. You can also specify which layers are visible in each layout viewport.

After you finish arranging the layout, you turn off the layer that contains the layout viewport objects. The views are still visible, and you can plot the layout without displaying the viewport boundaries.

12 Understand the Layout Process

When you use a named layout to prepare your drawing for output, you follow a series of steps in a process.

You design the subject of your drawing in model space and prepare it for output on a named layout in paper space. A drawing always has at least one named layout.

Before you can use a layout, it must be initialized. A layout does not contain any page setup information before it is initialized. Once initialized, layouts can be drawn upon and output.

12.1 Process Summary

When you prepare a layout, you typically step through the following process:

- Create a model of your subject in model space.
- Initialize a named layout.
- Specify layout page settings such as output device, paper size, drawing area, output scale, and drawing orientation.
- Insert a title block into the layout (unless you have started with a drawing template that already has a title block).
- Create a new layer to be used for layout viewports.
- Create layout viewports and position them on the layout.
- Set the orientation, scale, and layer visibility of the view in each layout viewport.
- Add dimensions and annotate in the layout as needed.
- Turn off the layer containing the layout viewports.
- Output your layout.

You can also use annotative objects if you want to annotate your drawing in model space and scale the annotations automatically.

13 Work with Model Space and Paper Space

There are several benefits to switching between model space and paper space to perform certain tasks. Use model space for creating and editing your model. Use paper space for composing your drawing sheet and defining views.

14 Work in Model Space

By default, you start working in a limitless drawing area called *model space*. In model space, you draw, view, and edit your model.

You first decide whether one unit represents one millimetre, one centimetre, one inch, one foot, or whatever unit is most convenient or customary in your business. You then create your model at 1:1 scale.

In model space, you can view and edit model space objects. The crosshairs cursor is active over the entire drawing area.

In model space, you can also define named views that you display in layout viewports on a layout.

15 Work on a Named Layout

Named layouts access an area called *paper space*. In paper space, you place your title block, create layout viewports to display views, dimension your drawing, and add notes.

In paper space, one unit represents the actual distance on a sheet of paper. The units will be in either millimetres or inches, depending on how you configure your page setup.

On a named layout, you can view and edit paper space objects, such as layout viewports and title blocks. You can also move an object (such as a leader or a title block) from model space to paper space (or vice versa). The crosshairs cursor is active over the entire layout area.

15.1 Create Additional Named Layouts

By default, a new drawing starts with two named layouts, named Layout1 and Layout2. If you use a drawing template or open an existing drawing, the layouts in your drawing may be named differently.

You can create a new layout using one of the following methods:

- Add a new layout with no settings and then specify the settings in the Page Setup Manager.
- Copy a layout and its settings from the current drawing file.
- Import a layout from an existing drawing template (DWT) file or drawing (DWG) file.

16 Access Model Space from a Layout Viewport

You can access model space from a layout viewport to edit objects, to freeze and thaw layers, and to adjust the view.

After creating viewport objects, you can access model space from a layout viewport to perform the following tasks:

- Create and modify objects in model space inside the layout viewport.
- Pan the view inside the layout viewport and change layer visibility.

The method you use to access model space depends on what you plan to do.

16.1 Create and Modify Objects in a Layout Viewport

If you plan to create or modify objects, use the Maximize Viewport button on the status bar to make the layout viewport fill the application window. The centre point and the layer visibility settings of the layout viewport are retained, and the surrounding objects are displayed.

You can pan and zoom while you are working in model space, but when you restore the viewport to return to paper space, the position and scale of the objects in the layout viewport are restored.

NOTE If you use PLOT while a viewport is maximized, the layout is restored before the Print dialog box is displayed. If you save and close the drawing while a viewport is maximized, the drawing opens with the named layout restored.

If you choose to switch to the default model space to make changes, the layer visibility settings are the settings for the drawing as a whole, not the settings for that particular layout viewport. Also, the view is not centred or magnified the same way it is in the layout viewport.

16.2 Adjust the View in a Layout Viewport

If you plan to pan the view and change the visibility of layers, double-click within a layout viewport to access model space. The viewport border becomes thicker, and the crosshairs cursor is visible in the current viewport only. All active viewports in the layout remain visible while you work. You can freeze and thaw layers in the current viewport in the Layers palette, and you can pan the view. To return to paper space, double-click an empty area on the layout outside a viewport. The changes you made are displayed in the viewport.

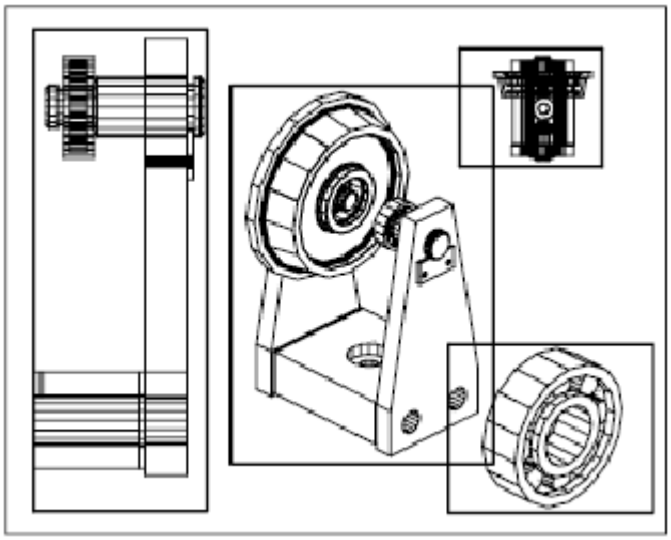
If you set the scale in the layout viewport before you access model space, you can lock the scale to prevent changes. When the scale is locked, you cannot use ZOOM while you work in model space.

17 Create and Modify Layout Viewports

You can create a single layout viewport that fits the entire layout or create multiple layout viewports in the layout. Once you create the viewports, you can change their size, their properties, and also scale and move them as needed.

NOTE
It is important to create layout viewports on their own layer. When you are ready to output your drawing, you can turn off the layer and output the layout without the boundaries of the layout viewports.

With MVIEW, you have several options for creating one or more layout viewports. You can also use COPY and ARRAY to create multiple layout viewports.



17.1 Create Nonrectangular Layout Viewports

You can create a new viewport with nonrectangular boundaries by converting an object drawn in paper space into a layout viewport.

You can use the MVIEW command to create nonrectangular viewports.

- With the Object option, you can select a closed object, such as a circle or closed polyline created in paper space, to convert into a layout viewport. The object that defines the viewport boundary is associated with the viewport after the viewport is created
- With the Polygonal option, you can create a nonrectangular layout viewport by specifying points. The prompts are the same as the prompts for creating a polyline

NOTE When you want to suppress the display of the boundary of a layout viewport, you should turn off the layer of the nonrectangular viewport instead of freezing it. If the layer of a nonrectangular layout viewport is frozen, the viewport is not clipped correctly.

17.2 Redefine Layout Viewport Boundaries

You can redefine the boundary of a layout viewport by using the VPCLIP command. You can either select an existing object to designate as the new boundary, or specify the points of a new boundary. The new boundary does not clip the old boundary, it redefines it.

A nonrectangular viewport consists of two objects: the viewport itself and the clipping boundary. You can make changes to the viewport, the clipping boundary, or both.

NOTE In the Properties Inspector, the default selection for a nonrectangular viewport is Viewport. This is because you are more likely to change the properties of the viewport than of the clipping boundary.

17.3 Resize Layout Viewports

If you want to change the shape or size of a layout viewport, you can use grips to edit the vertices just as you edit any object with grips.

18 Control Views in Layout Viewports

When you create a layout, you can add layout viewports that act as windows into model space. In each layout viewport, you can control the view that is displayed.

19 Scale Views in Layout Viewports

To scale each displayed view in output accurately, set the scale of each view relative to paper space.

You can change the view scale of the viewport using

- The Properties Inspector
- The XP option of the ZOOM command
- The Viewports Scale on the status bar

NOTE You can modify the list of scales that are displayed in all view and print scale lists with SCALELISTEDIT. After you add a new scale to the default scale list, you can use the Reset button in the Edit Drawing Scales dialog box to add the new scale to your drawing.

When you work in a layout, the scale factor of a view in a layout viewport represents a ratio between the actual size of the model displayed in the viewport and the size of the layout. The ratio is determined by dividing the paper space units by the model space units. For example, for a quarter-scale drawing, the ratio would be a scale factor of one paper space unit to four model space units, or 1:4.

Scaling or stretching the layout viewport border does not change the scale of the view within the viewport.

When creating a new drawing based on a template, the scales in the template are used in the new drawing. The scales in the user profile are not imported.

19.1 Lock the Scale of Layout Viewports

Once you set the viewport scale, you cannot zoom within a viewport without changing the viewport scale. By locking the viewport scale first, you can zoom in to view different levels of detail in your viewport without altering the viewport scale.

Scale locking locks the scale that you set for the selected viewport. Once the scale is locked, you can continue to modify the geometry in the viewport without affecting the viewport scale. If you turn a viewport's scale locking on, most of the viewing commands, such as VPOINT, DVIEW, 3DORBIT, PLAN, and VIEW, no longer function in that viewport.

NOTE Viewport scale locking is also available for nonrectangular viewports. To lock a nonrectangular viewport, you must perform an extra step in the Properties Inspector to select the viewport object rather than the viewport clipping boundary

19.2 Annotative Objects and Scaling

Annotative objects are defined at a paper height instead of a model size and assigned one or more scales. These objects are scaled based on the current annotation scale setting and automatically displayed at the correct size in the layout or when plotted. The annotation scale controls the size of the annotative objects relative to the model geometry in the drawing.

You can specify the default list of scales available for layout viewports, page layouts, and printing in Default Scale List dialog box.

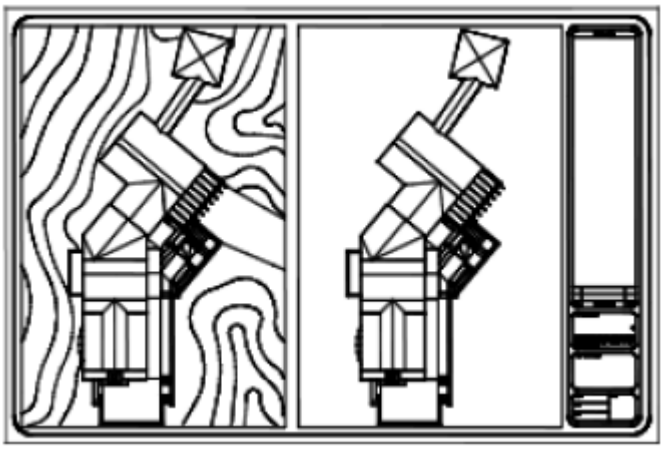
20 Control Visibility in Layout Viewports

You can control the visibility of objects in layout viewports using several methods. These methods are useful for emphasizing or hiding different elements of a drawing, and for reducing screen regeneration time.

21 Freeze Specified Layers in a Layout Viewport

A major benefit to using layout viewports is that you can selectively freeze layers in each layout viewport. You can also specify default visibility settings for new viewports and for new layers. As a result, you can view different objects in each layout viewport.

You can freeze or thaw layers in current and future layout viewports without affecting other viewports. Frozen layers are invisible. They are not regenerated or plotted. In the illustration, the layer showing terrain has been frozen in one viewport.



Thawing the layer restores visibility. The easiest way to freeze or thaw layers in the current viewport is to use the Layers palette.

In the Layers palette, on the right side, use the column labelled VP Freeze to freeze one or more layers in the current layout viewport. To display the VP Freeze column, you must be on a layout. Specify the current layout viewport by double-clicking anywhere within its borders.

21.1 Freeze or Thaw Layers Automatically in New Layout Viewports

You can set visibility defaults for specific layers in all new layout viewports. For example, you can restrict the display of dimensions by freezing the DIMENSIONS layer in all new viewports. If you create a viewport that requires dimensions, you can override the default setting by changing the setting in the current viewport. Changing the default for new viewports does not affect existing viewports.

21.2 Create New Layers That Are Frozen in All Layout Viewports

You can create new layers that are frozen in all existing and new layout viewports. Then you can thaw the layers in the viewports you specify. This is a shortcut for creating a new layer that is visible only in a single viewport.

22 Screen Objects in Layout Viewports

Screening refers to applying less ink to an object when it is plotted. The object appears dimmer on the screen and output to paper. Screening can be used to help differentiate objects in a drawing without changing the objects' colour properties.

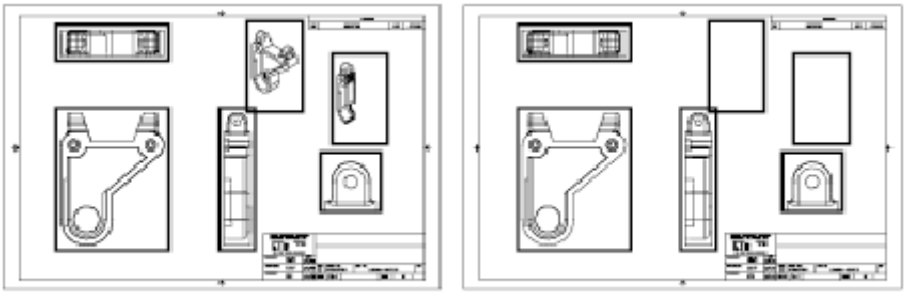
To assign a screening value to an object, you must assign a plot style to the object, and then define the screening value in that plot style.

You can assign a screening value from 0 to 100. The default setting, 100, means no screening is applied, and the object is displayed with normal ink intensity. A screening value of 0 means the object contains no ink and is thus invisible in that viewport.

23 Turn Layout Viewports On or Off

You can save time by turning some layout viewports off or by limiting the number of active viewports.

Displaying a large number of active layout viewports can affect your system's performance as the content of each layout viewport regenerates. You can save time by turning some layout viewports off or by limiting the number of active viewports. The following illustration shows the effects of turning off two layout viewports.



New layout viewports are turned on by default. If you turn off the layout viewports you aren't using, you can copy layout viewports without waiting for each one to regenerate.

If you don't want to plot a layout viewport, you can turn the layout viewport off.

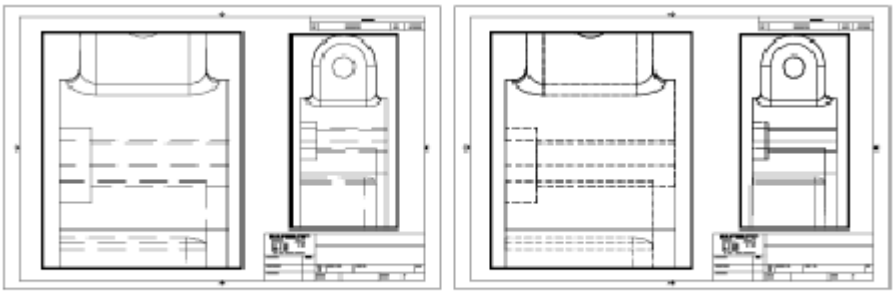
24 Scale Linetypes in Layout Viewports

You can scale linetypes in paper space either based on the drawing units of the space in which the object was created or based on the paper space units.

You can set the PSLTSCALE system variable to maintain the same linetype scaling for objects displayed at different zoom factors in a layout and in a layout viewport. For example, with PSLTSCALE set to 1 (default), set the current linetype to dashed, and then draw a line in a paper space layout.

In the layout, create a viewport with a zoom factor of 1x, make that layout viewport current, and then draw a line using the same dashed linetype. The dashed lines should appear to be the same. If you change the viewport zoom factor to 2x, the linetype scaling for the dashed line in the layout and the dashed line in the layout viewport will be the same, regardless of the difference in the zoom factor.

With PSLTSCALE turned on, you can still control the dash lengths with LTSCALE and CELTSCALE. In the following illustration, the pattern of the linetypes in the drawing on the left has been scaled to be the same regardless of the scale of the view. In the drawing on the right, the scale of the linetypes matches the scale of each view.



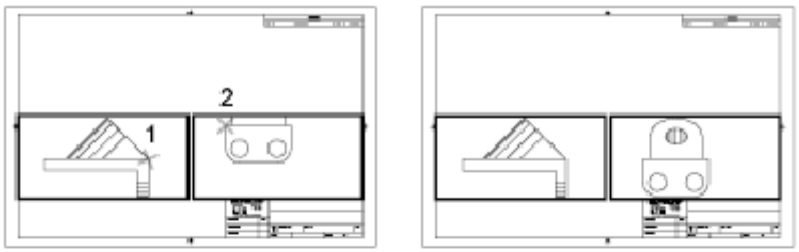
PSLTSCALE=1, dashes scaled to paper space

PSLTSCALE=0, dashes scaled to space where they were created

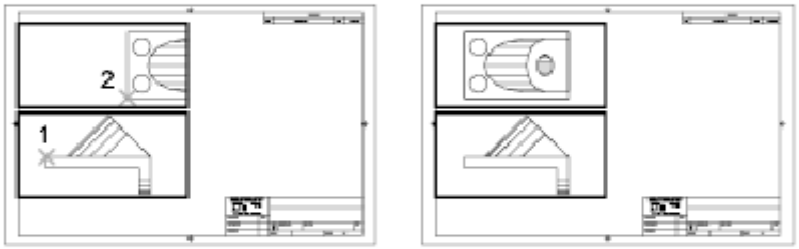
25 Align Views in Layout Viewports

You can arrange the elements of your drawing by aligning the view in one layout viewport with the view in another viewport.

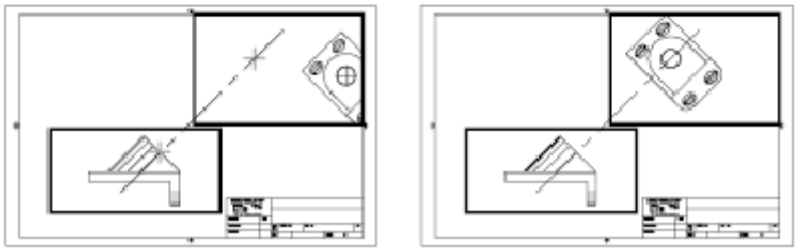
For angled, horizontal, and vertical alignments, you can move each layout viewport relative to distances defined by the model-space geometry displayed.



Horizontal alignments



Vertical alignments



Angled alignments

To adjust the views on a layout with precision, you can create construction geometry, use object snaps on the model space objects displayed in layout viewports, or use one of the drafting aids on the status bar.

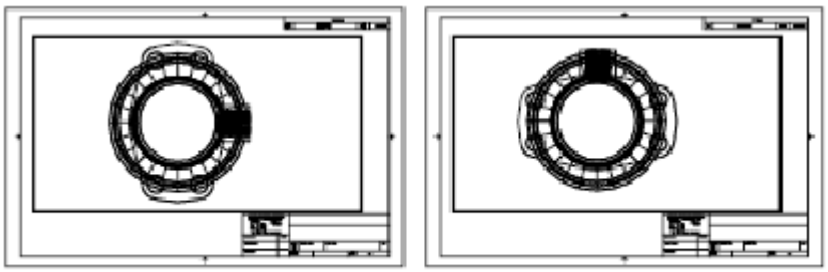
26 Rotate Views in Layout Viewports

You can rotate an entire view within a layout viewport with the VPROTATEASSOC system variable.

When VPROTATEASSOC is set to 1, the view within a viewport is rotated with the viewport. When VPROTATEASSOC is set to 0, the view remains when the viewport is rotated.

You can also rotate an entire view within a layout viewport by changing the UCS and using the PLAN command.

With the UCS command, you can rotate the *XY* plane at any angle around the Z axis. When you enter the PLAN command, the view rotates to match the orientation of the *XY* plane.



Another way is to use the Align and then Rotate View options in the MVSETUP command.

NOTE The ROTATE command rotates individual objects only and should not be used to try to rotate a view.

27 Reuse Layouts and Layout Settings

When you create a layout, you can choose to apply the information from an existing template.

A layout template is a layout imported from a DWG or DWT file. When you create a layout, you can choose to apply the information from an existing template. The program has sample layout templates to use when you design a new layout environment. The paper space objects and page setup in the existing template are used in the new layout. Thus, the layout objects, including any viewport objects, are displayed in paper space. You can keep any of the existing objects from the template you import, or you can delete the objects. No model space objects are imported.

The layout templates are identified with a *.dwt* file extension. However, a layout template or layout from any drawing or drawing template can be imported into the current drawing.

27.1 Save a Layout Template

Any drawing can be saved as a drawing template (DWT file), including all of the objects and layout settings. You can save a layout to a new DWT file by choosing the Save As option of the LAYOUT command. The template file is saved in the drawing template file folder as defined in the Application tab (Application Preferences dialog box). The layout template has a *.dwt* or *.dwg* extension like a drawing template or drawing file, but it contains little information not essential to the layout.

When you create a new layout template, any named items, such as blocks, layers, and dimension styles, that are used in the layout are saved with the template. These definition table items are imported as part of the layout settings if you import this template into a new layout. It is recommended that you use the Save As option of the LAYOUT command to create a new layout template. When you use the Save As option, unused definition table items are not saved with the file; they are not added to the new layout into which you import the template.

If you insert a layout from a drawing or template that was not created using the Save As option of the LAYOUT command, definition table items that are used in the drawing but not in the layout are inserted with the layout. To eliminate unnecessary definition table items, use the PURGE command.