

LAORT CADD LECTURE NOTES (LAYOUTS 2)

WORKING WITH LAYOUT VIEWPORTS

READING

CHAPTER 15 ‘Creating A Layout To Plot’ in helpfiles

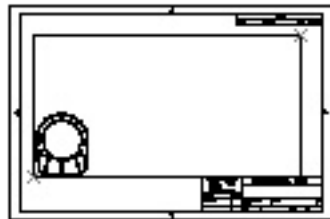
OVERVIEW

When you are composing a layout, you can consider viewports as objects with a view into model space that you can move and resize.

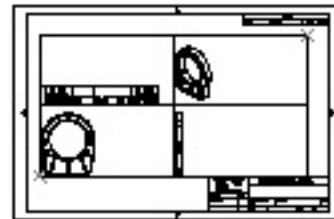
Viewports can be created in layouts in several ways. These include using the VPORTS or MVIEW commands, and using saved viewport configurations from MODEL space.

All viewports created in LAYOUT mode are FLOATING viewports. Even when using a saved viewport configuration of a MODEL space TILED view layout the views created in the layout are actually FLOATING viewports.

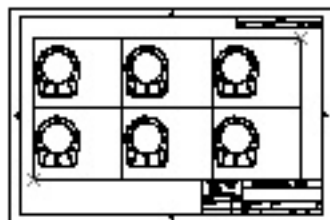
Floating viewports can be overlapping or separated from one another. Because floating viewports are AutoCAD objects, you cannot edit the model while arranging the layout in paper space.



single viewport



standard engineering viewports



arrayed viewports

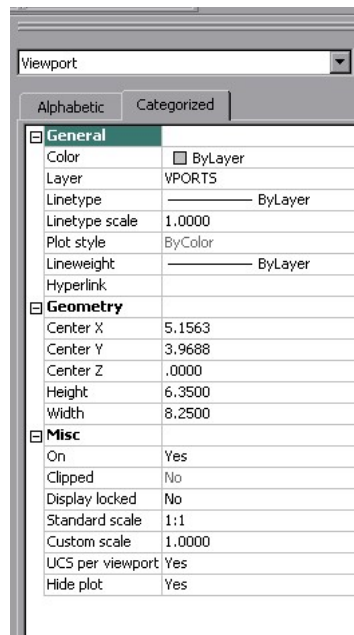
One benefit to using floating viewports is that you can selectively freeze layers in each viewport. By freezing layers, you can view different geometry in each floating viewport. You can also specify different views to display by panning and zooming in the viewport.

Once the layout viewports have been created working with them generally falls into to categories.

- **CONTROLLING VISIBILITY (How Model Space is Seen Through That Viewport)**
- **SETTING/MODIFYING THE APPEARANCE OF THE VIEWPORT ITSELF**

These tasks are accomplished using several commands as well as modifying the Layout Viewport object's properties. The properties of a Layout Viewport can only be accessed from LAYOUT (PAPERSPACE) mode. The status bar should always indicate PAPER when attempting to access a viewport's properties.

CONTROLLING VISIBILITY IN LAYOUT VIEWPORTS



You can control the visibility of model space objects in floating viewports using several methods. These methods are useful for limiting screen regeneration and for emphasizing or hiding different elements of your drawing. You can

- **Set The Scale The Objects In The Viewport Uniformly**
- **Freeze or Thaw Layers In Active Viewport**
- **Hide Lines In Active Viewport**

SCALING VIEWS RELATIVE TO PAPER SPACE

Scaling or stretching the floating viewport border does not change the scale of the view within the viewport. Scaling views relative to paper space establishes a consistent scale for each displayed view. To accurately scale the plotted drawing, you must scale each view relative to paper space.

When you are working in a paper space layout, the scale factor represents a ratio between the actual size of the model displayed in the viewports and the size of the layout. Usually the layout is plotted at a 1:1 scale. 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. You can change the plot scale of the viewport using the Properties window or the Viewports toolbar.

VIEWPORT SCALE LOCKING

Typically, you set the plot scale at 1:1 for a layout. However, as you create floating viewports, you may want to apply other scales to viewports to display different levels of detail. Once you've set the viewport scales, if you zoom into the paper space geometry in the viewport, you are changing the viewport scale at the same time. By locking the viewport scale first, you can zoom in to view different levels of detail in your viewport without altering the viewport scale. The zoom is performed in paper space, not in the viewport.

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, are no longer functional in that viewport.

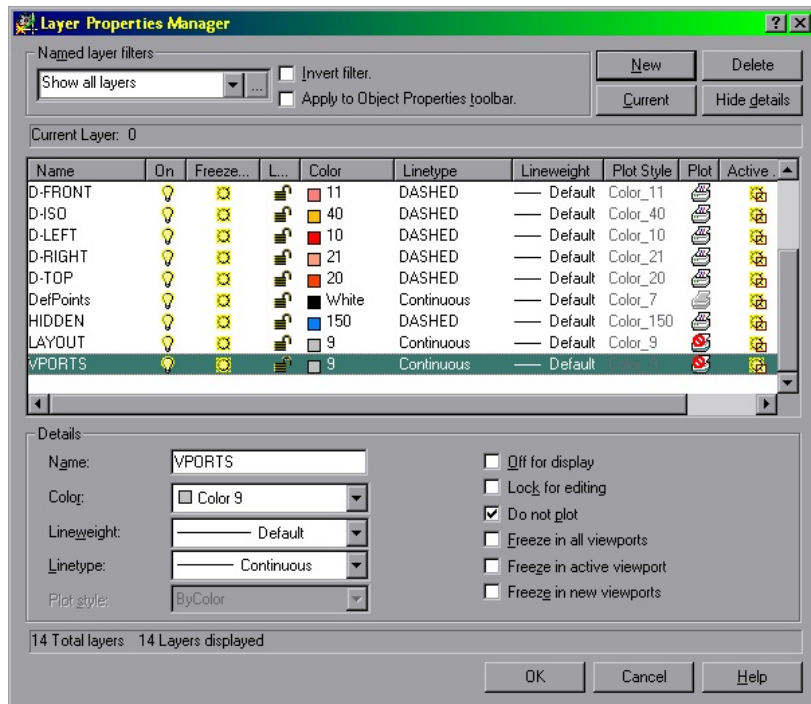
FREEZING OR THAWING LAYERS IN THE ACTIVE VIEWPORT

You can freeze or thaw layers in the active floating viewport without affecting other viewports. (Floating viewports are displayed in layouts; tiled viewports are displayed in model space.) Frozen layers are invisible. They are not regenerated or plotted. This feature is useful, for example, if you want to create an annotation layer that is visible only in a particular viewport. Thawing restores the layer's visibility (and regenerates the drawing).

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

To freeze or thaw layers in the active floating viewport

- 1 From the Format menu, choose Layer.
- 2 In the Layer Properties Manager, select the layers to freeze or thaw and choose Show Details.
- 3 Select or clear Freeze in Active Viewport.
- 4 Choose OK.



Related VPLAYER freezes and thaws layers in floating viewports.

HIDING LINES IN LAYOUT VIEWPORTS

If your drawing contains 3D faces, meshes, extruded objects, surfaces, or solids, you can direct AutoCAD to remove hidden lines in selected viewports when you plot. The Hideplot property of a viewport object affects only the plotted output, not the screen display. When you are plotting a layout, selecting Hide Objects in the Page Setup dialog box hides only the paper space geometry, not the geometry in a viewport.

To hide lines in plotted views

- 1 Select the viewport in which you want to apply hidden lines.
- 2 From the Tools menu, choose Properties.
- 3 In the Properties window, turn Hide Plot on.

To show hidden lines in the current viewport, turn Hide Plot off.

3D lines in model or paper space are hidden when you plot.

Command line -VPOR TS Hideplot

Shortcut menu Select the viewport, right-click in the drawing area, and then choose Hide Plot.

To hide lines on solid objects, AutoCAD creates a mesh image of the object. If the DISPSILH system variable is set to on (1), AutoCAD suppresses drawing the mesh and only draws silhouette lines of the solid.

When the current view is shaded, hiding lines on solid objects also suppresses drawing the mesh image and instead displays the objects using wireframe representation with the back lines hidden. This is the same as choosing the Hidden option of the SHADEMODE command or choosing Hidden from the View Shade menu.

When the current view is rendered, hiding lines on solid objects creates the mesh image.

SETTING/MODIFYING THE APPEARANCE OF THE VIEWPORT ITSELF

TURNING OFF THE VIEWPORT BORDERS FOR PLOTTING

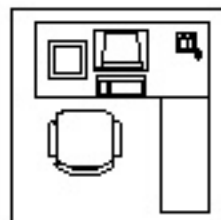
Because the layout viewports are objects, they have a LAYER property. The best way to manage the display of the viewport edges is to create a layer specifically for layout viewports. This layer should be designated as a non-plotting layer. Normally you would not want to FREEZE or turn OFF this layer because it makes modifying the viewports more difficult.

CREATING NON-RECTANGULAR VIEWPORTS

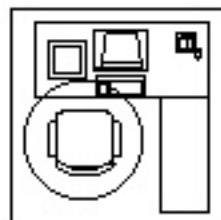
Using previous AutoCAD releases, you can create rectangular viewports in paper space to display different views of the model. With AutoCAD 2000, you can associate a boundary with a rectangular viewport object and clip the geometry in the viewport to the edge of the boundary. Thus, you can create viewports with irregular or nonrectangular boundaries.

A nonrectangular viewport is created when you associate a clipping object (polyline, circle, region, spline, or ellipse) with a normal viewport object. When you associate these two objects, they remain connected as long as the nonrectangular viewport boundary exists.

To create nonrectangular viewports, you can either create a new viewport with irregular boundaries or alter an existing viewport by redefining its boundary. The new boundary can consist of a closed polyline, circle, spline, ellipse, region, or arc segment.



model space
geometry



defining
non-rectangular
viewport



non-rectangular
viewport

NOTE if you want to use BHATCH to apply a hatch to a nonrectangular viewport, use the internal point method. If you use the select objects method and try to select the clipped viewport, the underlying rectangular viewport is added to the selection set and

is therefore hatched. To apply the hatch inside the viewport boundary, select an internal point.

MODIFYING THE VIEWPORT BOUNDARIES

The simplest way to modify the boundaries is to use GRIPS. This operation must be done in PAPER space. To completely redefine the shape of an existing viewport you must use the VPCLIP command.

VIEWPORT CLIPPING

You can redefine the boundary of a viewport by using the VPCLIP command. To clip a viewport, you can use your pointing device to either select an existing object to designate as the new boundary, or specify the points of a new boundary.

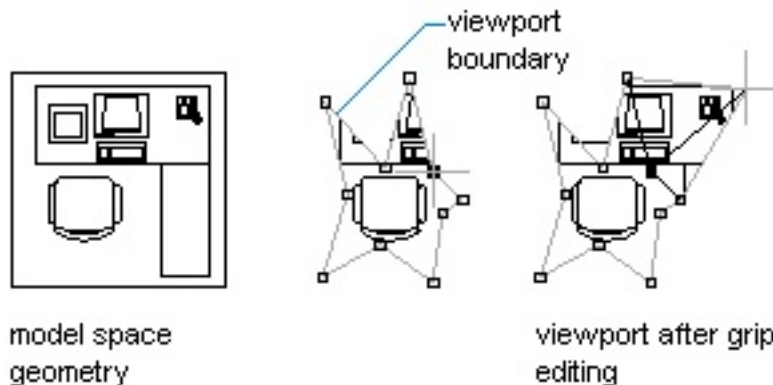
To redefine a viewport boundary

- 1 At the Command prompt, enter `vpclip`.
- 2 Select the viewport to clip.
- 3 Enter `d` (Delete) to delete the clipping boundary.
- 4 Enter `p` (Polygonal).
- 5 Specify points, or select the object, to define the new viewport boundary.

Shortcut menu Select the viewport to clip, right-click in the drawing area, and then choose Viewport Clip.

CHANGING SHAPE OF IRREGULAR SHAPED VIEWPORTS

When you create an irregularly shaped viewport, AutoCAD calculates the extents of the object selected and places a viewport object at the corner of the extents of the boundary. Depending on the shape of the boundary, the geometry may not be completely displayed in the irregularly shaped viewport. If you want to change the shape of the viewport, you can use grips to edit the vertices of an irregular boundary just as you edit any object with grips.



If the boundary layer of a nonrectangular viewport is frozen, the boundary is not displayed, and the viewport is not clipped. If the boundary layer is turned off but not frozen, the viewport remains clipped.