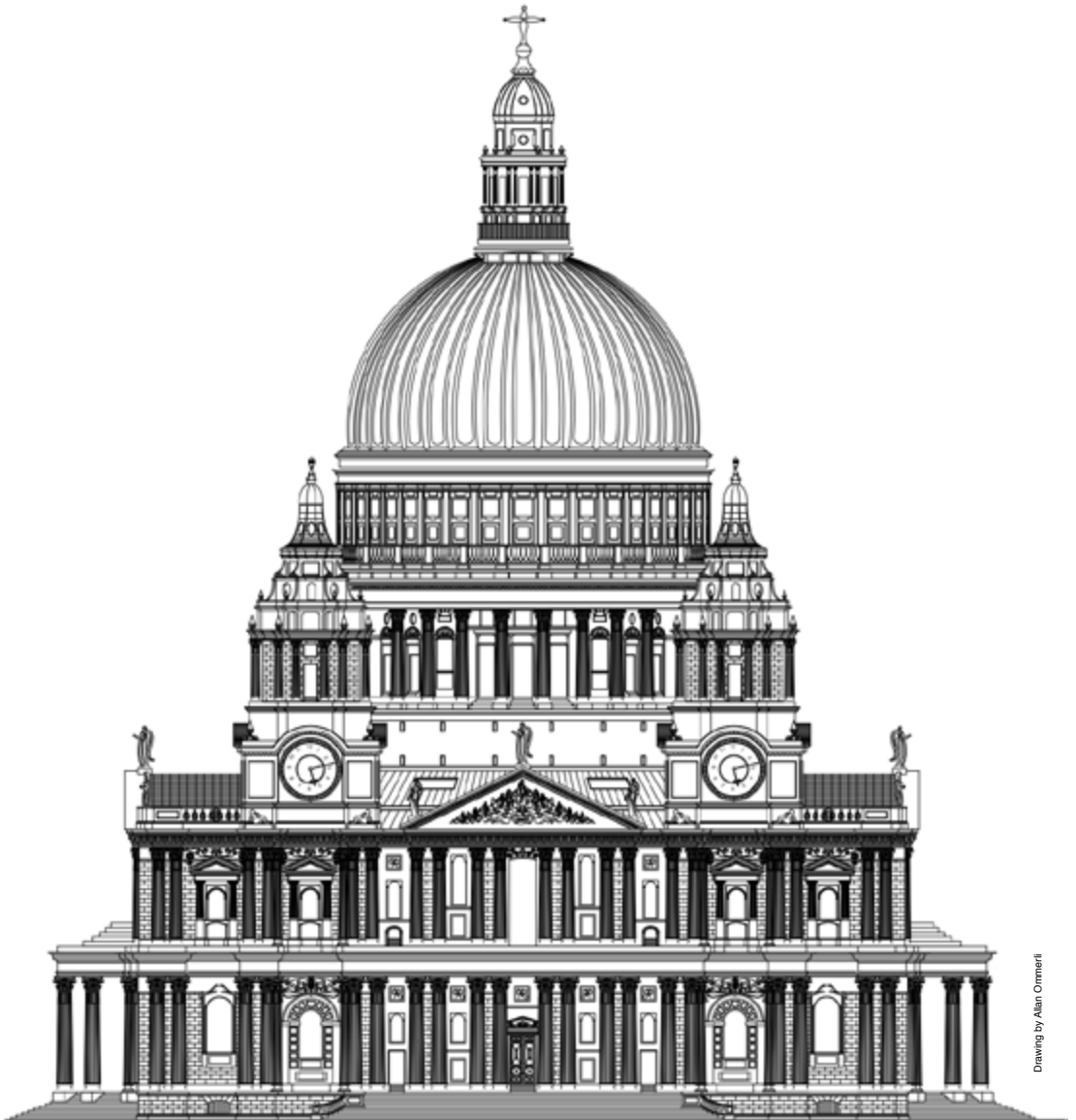


You can draw the world

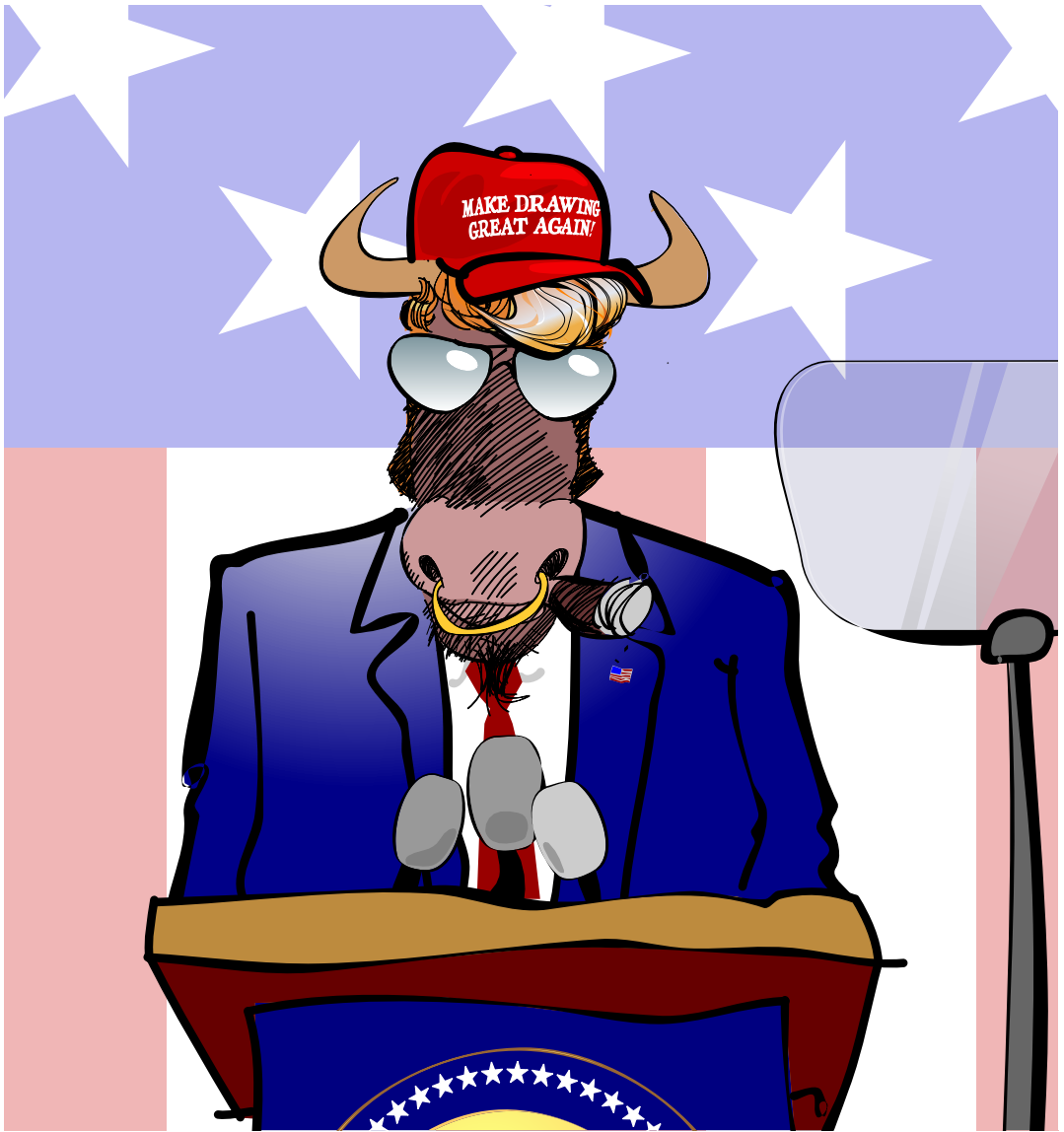
PowerCADD™



Drawing by Allan Ormerik

5/10/2018

Version 9.5



Alas, Whip, the PowerCADD Plugger has gone political on us. SAD!

How to Read This Document

This manual is an interactive PDF that acts in some ways like a book and also like a program, with links that will take you from one page to another.

To operate properly within PowerCADD, the document name must be:

PowerCADD™ Documentation.pdf

And it must be installed in the PowerCADD Documentation directory, specifically:

Applications/PowerCADD 9/Documentation/PowerCADD™ Documentation.pdf

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This manual can be downloaded to your iPad from

<http://www.seqair.com/WildTools/Documentation/Docs.html>

Symbols



The following symbols are used in this manual to highlight special information:

NOTE: Indicates a special note.



TIP: Indicates a tip.



CAUTION: Alerts you to give this special caution.



Tells where to find more details about a particular subject.



And in this wonderful new world of social media, we've included a few things to alert you to user favorites, much loved features and other emotional reactions.

Anything in red is probably obsolete and needs checking.

Anything in green needs a hyperlink on the page.

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Tools

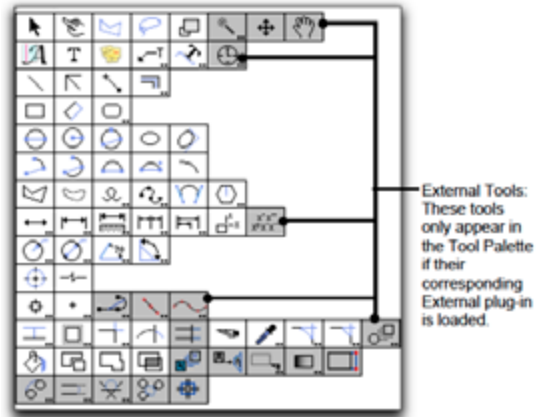
Tools

This chapter is organized around five major headings:

- ❑ Tool palette
- ❑ Drawing tools
- ❑ Dimensioning tools
- ❑ Symbol tools
- ❑ Modifier tools
- ❑ Notation tools

Thus, if you want to find information on a tool, refer to the Tool palette section of this chapter. Similarly, the rest of the tools are organized in this chapter as they appear in the Tool palette.

Tool Palette



NOTE: Pressing the mouse button while on any tool will display the entire tool palette. If Tool Tips are enabled, holding your mouse over a tool will display its name. Control- or right-click anywhere on the tool palette and select the Change Palette Orientation menu item to switch between vertical or horizontal palette orientations.



NOTE: Select the [Commands](#) tab in the PowerCADD Preferences dialog to assign a custom key command to any tool in the palette.

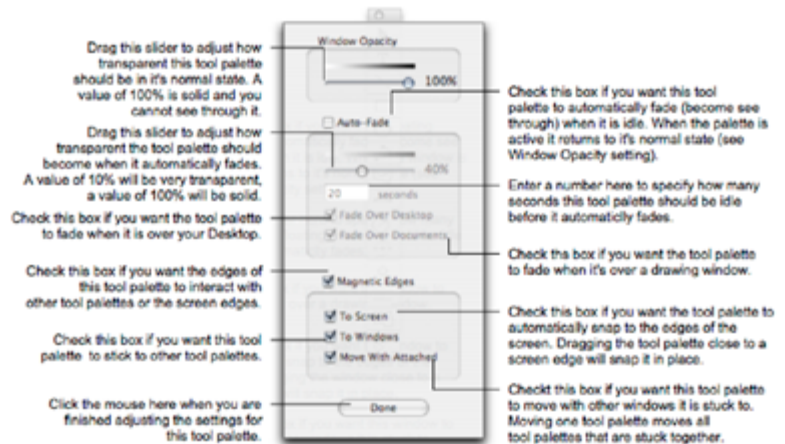


Also, see [Preferences](#) for more information on Enabling Tool Tips and [Tool Preferences](#) for personalizing your drawing environment.

The Tool palette is the portion of the PowerCADD drawing environment that holds the various drawing tools available to you. These tools permit you to draw objects such as lines, arcs, ellipses, circles, rectangles, polygons, as well as freehand shapes and dimensions. The tools are represented by icons symbolizing their function and purpose.

To customize the settings of a tool palette:

- ☐ Press the Control key and click anywhere in the palette.
- ☐ Choose Palette Settings from the Contextual menu. The following dialog will appear:



- ☐ Make the desired setting changes and click Done.

The palette settings for each tool palette may be adjusted independently.

Being able to independently adjust the settings for each tool palette provides you a great deal of flexibility in customizing your workspace.

Given the frequency with which tool palettes are accessed during a typical drawing session, auto-fade has been turned off as a default setting. To avoid disrupting your work-flow, we recommend leaving auto-fade turned off until you become familiar with PowerCADD and with the auto-fade feature for tools and floating windows.



NOTE: The normal state for a tool palette is determined by the Window Opacity setting. A tool palette that has faded on screen will automatically return to its normal state when you move the mouse over the palette.

The ability to turn Auto-Fade on or off for a tool palette is controlled by the Window Fading Preference settings. If the Auto-Fade settings are dimmed as shown on the left:

- ☐ Choose Preferences from the PowerCADD menu.
- ☐ Select the General tab.
- ☐ Select the Window Fading box to turn it on.
- ☐ Click OK.
- ☐ Try to set the Auto-Fade settings for the floating window again as described earlier.

External Tools

PowerCADD's modular design permits loading of External plug-in tools which may be appended to the main PowerCADD tool palette or appear as a stand-alone tool palette. If your tool palettes do not appear the same as the images shown in this manual or if you do not see a tool or tool palette mentioned here, it is possible you have not loaded the correct External.



Also, see [Loader Preferences](#) for more information on loading and managing External plug-ins.

With experience, you will quickly become adept at selecting the appropriate tool to use for the task you wish to accomplish. Tools are activated by clicking on them, and stay active until another tool is selected. The Return or Enter key will select the last tool used.

About the Palette Tools

Once a palette tool has been selected, it remains active, or selected, until it is deselected.

Once you select a tool from the tool palette and draw an object, you *do not* have to select the tool again in order to draw more of the same objects. This feature gives you greater efficiency compared to other drawing programs, which force you to return to the palette when you wish to use the same tool repeatedly.



Also, see [Tool Preferences](#) for more information on customizing the behavior of tool fallout and general tool behavior.

Each palette tool has its own set of default attributes, such as line weight, fill pattern, and color. These default attributes are independently assigned to each palette tool and remain in effect until you change them.



The default attributes for tools also include Transparency for Pen and Fill settings. See [Attributes](#) in the Menu section for more information on working with the Attributes floating window.



PowerCADD includes a Preference setting allowing you to assign Global Tool attributes. Please see [Global Attributes](#) for additional information.

Deep Zoom Zoom on the Fly

Deep zooming and zoom on the fly while drawing is available when using the preset key commands and any drawing tool.

To use Deep Zooming and Zoom on the Fly while drawing:

- ☐ Select any tool in the tool palette (e.g. the Selection Arrow tool).
- ☐ Press the 1 key to zoom in.
- ☐ Press the 2 key to zoom out.
- ☐ Press the 3 key to return to a previous zoom level.

The current mouse location is the center of the zoom-in or zoom-out operation.

To zoom on the fly while drawing:

- ☐ Select any tool in the tool palette (e.g. the Line tool)
- ☐ Drag to start the drawing operation.
- ☐ Press the 1 key

The drawing will zoom in; the current mouse location is the center of the zoom operation.

- ☐ Press the 2 key to zoom out.

You may cycle back and forth between zooming in and out.

- ☐ Release the mouse when the desired zoom is reached to complete the drawing operation.

Zooming in and out on the fly while drawing works with click-drag, click-click, hybrid click-click tool settings as specified in the [Tool Preferences](#) Tab of the PowerCADD menu.

While deep zoom percentages can exceed 200,000,000, most objects will become illegible before that zoom level is reached. At approximately 20,000 times zoom WYSIWYG (What You See is What You Get) line weights are disabled and all objects are rendered to screen at one point.

Their true WYSIWYG line weight is displayed again when the zoom level is less than approximately 20,000 times. If the drawing is set up to show rulers, tick marks in the rulers are not displayed when the zoom percentage exceeds approximately 6400 times (ruler tick marks are displayed again automatically when zooming out to less than approximately 6400 times).



Also, Scroll Wheel zooming with your mouse (or touch pad on a laptop) is also available on the fly while drawing. See [Tool Preferences Mouse Wheel Options](#) for additional information.

Change a Tool's Default Attributes

A tool's default attributes are changed by selecting the tool in the palette and then choosing the desired line type, fill pattern, and colors from the appropriate menus.

If an object is selected before a default attribute is changed, then only that object's attributes are changed. If only the tool is selected, then the selected tool takes on the new attributes.

All new objects drawn with a palette tool retain the current tool settings and defaults. The settings are saved as defaults for the palette tools in the drawing. You can create a drawing template of your settings and defaults by using PowerCADD's Stationery feature explained later in the Menu chapter.



NOTE: Previously drawn objects are not affected by palette default changes and maintain their original default characteristics until selected and changed.

To change the default attributes for an individual palette tool:

- ☐ Double-click a palette tool.

This will deselect all objects *and* select the tool.

Tool defaults cannot be changed if an object is selected in the drawing area.

- ☐ Control-click on the desired tool in the palette.
- ☐ Choose the desired default settings from the contextual menu.

The selected attributes will remain active until changed by following this set of procedures again.

To change the default attributes for all palette tools:

- ☐ Double-click a palette tool.
- ☐ Control-click on the desired tool in the palette.
- ☐ After the contextual menu appears, release the Control key and hold the Shift key. Choose the desired default settings from the contextual menu.

The selected attributes will remain active until changed by following this set of procedures again.

Change an Object's Attributes Only

To change an object's attributes without changing the default attributes of the palette tool:

- ☐ Select the object(s) in the drawing.
- ☐ Control-click on a selected objects and choose the desired option changes from the contextual menu.

The selected options will apply only to the object selected and not to the palette tool.

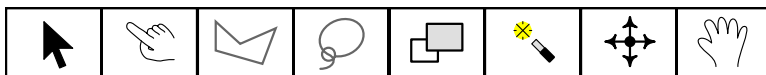


NOTE: Using a contextual menu to change the attributes of an object in the drawing, the default attributes, or the default attributes of all palette tools is only one method. You can also change attributes by choosing items from the Attr menu or by choosing items in the Attributes floating window.



NOTE: To set text default sizes and fonts for the Text and Leader Line tools, double-click the Text or Leader Line tool, then Control- or right-click and choose the desired font attributes from the contextual menu. To set text default sizes and fonts for Dimension tools, double-click any dimension tool, then Control- or right-click and pick the desired font attributes from the contextual menu.

Selection Tools



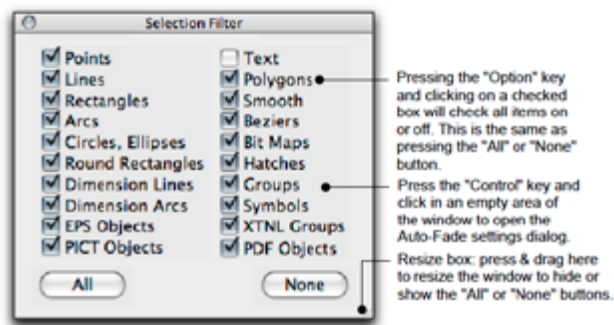
Being able to quickly and easily select objects and navigate in your drawing is critical to efficient drawing production, management and editing. The first row of the tool palette contains tools for selecting objects and navigating in the drawing window. This section provides an overview of the tools and important techniques to speed up your workflow.

Before we discuss the specific selection tools, let's review fundamental aspects of selecting objects in PowerCADD that affect all tools.

Selection Filter Floating Window

The Selection Filter floating window allows you to specify the type of objects to be selected. This becomes particularly important when working in drawings containing large numbers of objects or when working in a busy drawing area.

To open the Selection Filter Window choose Selection Filter from the Window menu or select Selection Filter in the Anchor Window.



The checkboxes allow you to specify the objects to be selected. By choosing only those objects you want to select, you may work with ease in busy areas of the drawing without fear of moving or deleting other objects.

- ☐ Check the boxes corresponding to the objects you wish to select. Unchecked object types cannot be selected.
- ☐ The objects checked on in the Selection Filter window will be the only items selected regardless of the method of selection.

Selection Filter settings have unlimited potential to save significant amounts of time. For example, if it became necessary to change all the arcs to a fill pattern in a large drawing, it would be extremely tedious to visually locate and manually select each arc. By setting the Selection Filter to Arcs only,

you may perform a Select All and change every arc in a matter of seconds rather than minutes.



CAUTION: Selection Filter settings do not see inside a Group or Symbol when performing a Select All operation. For example, setting the Selection Filter to Arcs only and performing a Select All would only select Arcs which are NOT inside a Group.



Selection Filter settings also apply when using the [Select Special...](#) menu item, the [Magic Wand Tool](#), and [Edit All Layers](#). See Menu chapter for additional information on Edit All Layers.



CAUTION: Only Vertex and Align snapping will work for unchecked objects in the Selection Filter window.

Forms of Selection Available

PowerCADD offers many different forms of object selection. You may select single objects, multiple objects, objects in a specific area, random selection, and Select All.

To select any object, simply place the cursor on the object and click. If the object is filled, click anywhere inside the perimeter of the object. If the object is not filled, select the perimeter of the object.

The Shift key allows you to add or subtract from the set of currently selected objects regardless of the method of selection used.

- ☐ Press the Shift key.
- ☐ Click on each object to be selected.
- ☐ To deselect a selected object, select it again.

Various methods of selecting objects exist including tools, menu items and combinations thereof and no single method will address every drawing situation. While we'll cover the basic operation of the selections tools here, it is worth experimenting with how all the selection features described earlier integrate with how the selection tools operate.



Selection Arrow Tool

The Selection Arrow tool allows you to use cursors to select objects in the drawing area in order to change or move them, to choose items from the menu, and to select tools from the drawing palette.

The Selection Arrow tool is one of the most frequently used tools from the Tool palette and must be understood thoroughly in order to master PowerCADD.

The Selection Arrow cursor is primarily used to select tools from the Tool palette, to choose commands and options from the Menu bar, and to use the scroll bars.

In addition, when the Selection Arrow cursor is moved to the drawing area, it becomes a Selection cursor for selecting an object or group of objects from the drawing for further editing, changes, or movement. The Selection cursor will change states to reflect the current operation as described below.


Selection Arrow Cursor


Selection Cursor


Selection Marquee Cursor


Duplication Cursor


Drawing Cursor


Move Cursor


Target Cursor



In the palette area, or when choosing menu items, the Selection cursor is an arrow.

When the Selection Arrow cursor is moved to the drawing area, it becomes a large crosshair cursor, or simply a Selection cursor.

When the Selection Arrow cursor is moved to the drawing area and you perform a click and drag operation, the cursor changes to a Selection Marquee cursor.

When the Option key is pressed when dragging an object the duplication cursor appears.

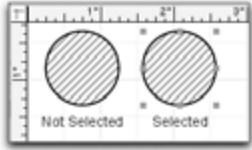
When drawing tools are selected, the Selection Arrow cursor converts to a Drawing cursor when moved into the drawing area.

The Move cursor appears when dragging or moving objects.

The Target cursor is used to place objects, establish a rotation point, or place a dynamic radius. Generally the Target cursor is used when you need to show PowerCADD where something should be placed.

NOTE: Autoscroll feature: When you attempt to move the Selection cursor beyond the screen but where *paper* still exists, PowerCADD will

automatically scroll the screen to accommodate your action. This Autoscroll feature makes it easy for you to draw objects larger than the screen without the inconvenience of having to stop your drawing, scroll manually to another part of your paper, select the object, and then resize it. By moving the cursor farther off the page, you will increase the speed at which you autoscroll. This will allow you to move rapidly across a large drawing.



Any time an object is selected or not selected, visual confirmation is provided as illustrated here.

Notice the small squares (handles) at each of the corners and mid points of the circle. The handles are the limits of the objects bounding box and may be grabbed using the Selection cursor to resize the selected object(s). A bounding box is the smallest rectangular area that will completely enclose an object's geometry. The sides of the bounding box are always oriented parallel to the horizontal and vertical axes.

Once active, the object can then be changed. Resizing, stretching, shrinking, or filling an object with a pattern are examples of a change.



In addition to displaying handles, you can also have selected objects highlight in any color you choose. See [View Preferences](#) on customizing how selected objects are displayed on screen.

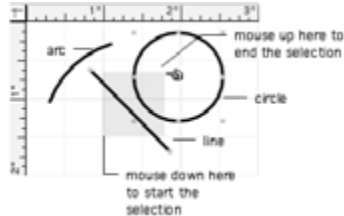


You can adjust the size of the handles. See View Preferences in the Menu chapter for additional information on adjusting the [size of the handles](#).

The Selection cursor allows you to select objects by clicking on items as previously discussed, or by performing a click and drag to define a selection marquee.

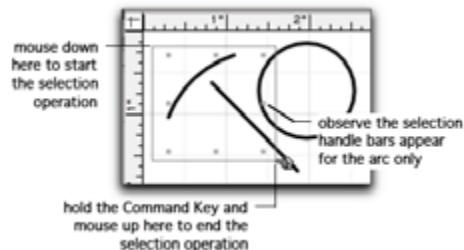
When picking objects using a selection marquee the following rules apply:

- ❑ All objects touching the marquee are selected. In this example only the line and circle are selected because the gray selection marquee touched only those objects. Notice there are no handles shown for the arc, only for the circle and line.



- ❑ Press the Command key and only objects completely inside the marquee are selected.

The marquee becomes transparent instead of a gray fill when holding the Command key. In this example, only the arc is selected because it is entirely inside the selection marquee.



See [Tool Preferences](#) for information on how to reverse the behavior of the Select by Touching versus Select Enclosing process.



NOTE: You cannot change an object without first selecting it using any of the selection tools, selection menu items, or any combination thereof.

How to Select an Object

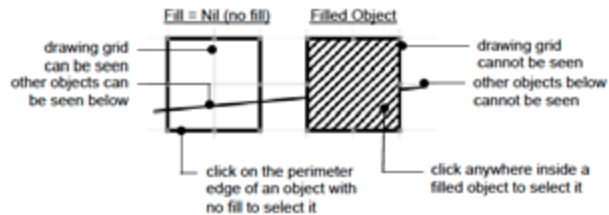
To select a line or an arc:

- ☐ Select the Selection Arrow tool from the Tool palette.
- ☐ Move the cursor to the object you want to select.

As you move the cursor from the palette area to the drawing area, the Selection Arrow cursor changes into a Selection cursor as the following shows.

Click on the perimeter (edge) of an object with no fill to select it.

Click anywhere inside or on a filled object to select it.



Move Cursor

NOTE: Release the mouse button when you select an object. Otherwise the selection cursor will turn into a Move cursor. If this happens, don't panic—just release the mouse button. If you accidentally move the object simply choose Undo from the Edit menu to return the object to its previous position. The Move cursor is shown to the left.

How to Select all Objects in a Drawing

To select all objects in the current layer of the drawing:

- ☐ Choose Select All from the Edit menu.

You will notice each of the drawing's objects have their handles activated.



CAUTION: Be aware when you use the Select All command; you are selecting all objects in the currently selected layer whether the objects can be seen in the present window or not. Any commands issued subsequent to Select All will affect each object on the active layer of the drawing.



CAUTION: If you have Edit All Layers turned On, you are selecting all objects on all visible layers.



See [Select All](#) for more details.

Deselect All Objects

To deselect all objects in the current layer of a drawing, that is, to select no objects at all:

- ☐ With the Selection cursor, click in a blank, empty space in the drawing.
- ☐ Double-click on a palette tool.
- or—
- ☐ Choose Deselect All from the Edit menu.

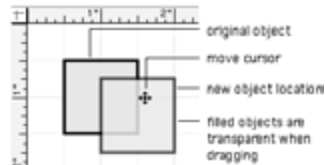
How to Move an Object



Move Cursor

To move an object:

- ☐ Use the Selection cursor to select the object you wish to move, and continue to hold the mouse button. It is important not to release the mouse button at this time.
- ☐ As you hold the mouse button on an object, the Selection cursor turns into a Move cursor.
- ☐ While continuing to hold the mouse button, drag the object to the desired location.



If you move the object past the boundaries of the current window, the Autoscroll feature will move the document for you within the limits of the current page size.

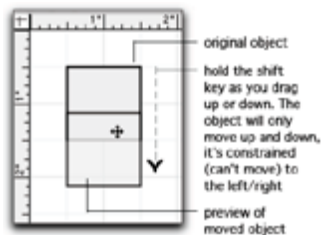


NOTE: Reference the [Move Object Tool](#) discussed later in this section for additional information on selecting and moving an object.

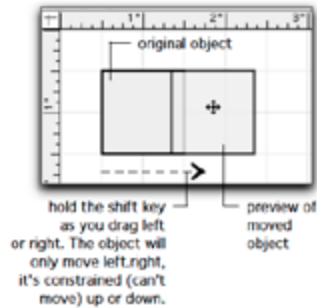


NOTE: Dynamic Snapping is also in effect at both the start and end of a move by drag operation. The Shift key may also be used to constrain the move by drag operation to either the X or Y axis.

To move an object (or all selected objects) but constrain the move in only the vertical orientation, do this:



To move an object (or all selected objects) but constrain the move in only the horizontal orientation, do this:



You can use snap commands to grab any object from a specific point and snap it into place to finish the move operation. Snap commands can be applied to anything you can see on screen, including the object you've selected for moving. It takes a bit of practice to get the co-ordination just right, but it is easy once you learn how.



NOTE: You can duplicate an object as part of a move operation by holding the Option key while you drag the objects with the mouse.

Duplicate an Object While You Move It



To duplicate the object while you move it:

- ☐ Start the move by dragging operation as previously described.
- ☐ Press the Option key. Observe, the duplication cursor appears.
- ☐ Continue to hold the Option key and drag to where you want the object to be duplicated.
- ☐ Release the mouse.

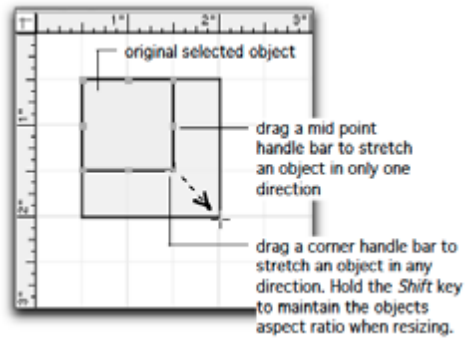
The duplicate is created in the specified location.

NOTE: The distance and angle you specified when dragging the original object is remembered. You can duplicate another object the same distance and angle by Command-D.

How to Resize an Object

To resize an object:

- ☐ Select the Selection Arrow tool from the Tool palette.
- ☐ Move the cursor to the drawing area.
- ☐ Select the object you wish to resize.
- ☐ The selected object's handles have become activated.
- ☐ Place the cursor on top of one of the handles.
- ☐ Press the mouse button and drag the edit handle in the desired direction.



NOTE: Remember, snap commands are available during a reshape/resize operation.

How to Delete an Object

To delete an object:

- ☐ Select the Selection Arrow tool from the Tool palette.
- ☐ Move the cursor to the drawing area.
- ☐ Select the object you wish to delete.

Notice the selected object's handles appear.

- ☐ Press the Delete or Backspace key.
- or—*
- ☐ Choose Clear from the Edit menu.



CAUTION: If all objects in the current layer of the drawing are selected, pressing the Delete or Backspace key will remove them entirely from the PowerCADD document. Failure to Undo such a mistake could be disastrous (see Undo in the Edit menu). Remember to save your drawing at regular intervals see [Save](#) in the File menu.



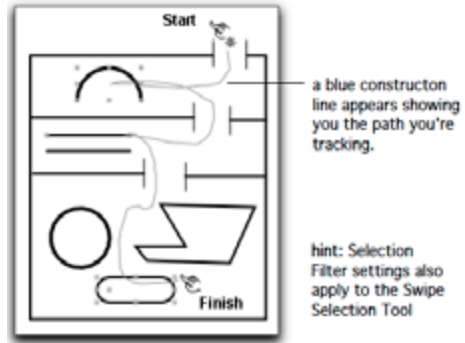
Swipe Selection Tool

This tool selects all objects the cursor touches.

It provides a quick way to select one item or multiple items by simply drawing a freehand shape across any part of the object(s).

To use the Swipe Selection tool:

- ☐ Select the Swipe Selection tool from the first row of the Tool palette.
- ☐ Drag the cursor over the object(s) to be selected.
- ☐ Press the Shift key to add or subtract items from the set of selected objects.



TIP: Press the Z key to automatically activate the Swipe Selection tool when any other tool is active. Release the Z and the Selection Arrow tool is automatically activated.



Polygon Select Tool

This tool selects all objects completely enclosed by a polygon fence you draw.

The tool provides tremendous selection control as a selection fence may be drawn in any shape needed. Thus, you can draw the sides of the polygon precisely where you need them. A polygon corner can even be snapped to an object.

For an object to be selected, all of its endpoints or vertices must lie within the polygon.

To use the Polygon Select tool:

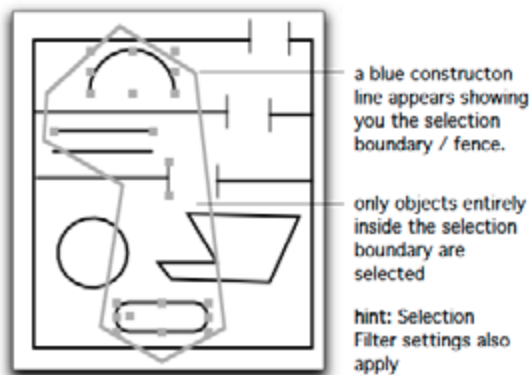
- ☐ Select the Polygon Select Tool from the Tool palette.
- ☐ Move the cursor to the location in your drawing area you wish to begin and press the mouse button.
- ☐ Drag the cursor to the end of the first side and release the mouse button.

The first side of the selection polygon has now been established.

- ☐ Move the cursor to the end of the second side and click.

The second side of the selection polygon has been established.

- ☐ Continue this pattern to add as many sides as you wish to enclose the objects you wish to select, with the last mouse click closing the polygon.





Selection Lasso Tool

This tool lets you select all objects enclosed by a freehand shape that you draw with the tool.

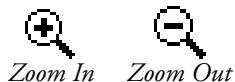
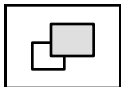
A Lasso Selection is, in principle, a cross between the Polygon Select and the Swipe Selection tools, and they complete the suite of tools.

Like the Polygon Select tool, only objects inside the selection boundary (fence) are selected. Like the Swipe Selection tool, the boundary can be an irregular shape.

To use the Selection Lasso tool:

- ☐ Select the Selection Lasso tool from the Tool palette.
- ☐ Move the cursor to the area of the drawing you wish to make a selection.
- ☐ Drag the cursor around the area of the objects you wish to select and release.

The objects enclosed in the area will be selected.



Zoom Tool

This is a fast and efficient way to magnify part of a PowerCADD drawing document. It allows you to zoom in or out of a section of the drawing by factors of 2X or dynamically to any percentage factor you specify.

There are two states for the Zoom tool's cursors. Zoom in (+) is the default when you pick the tool. To Zoom out (-) hold the Option key when the Zoom tool is active.

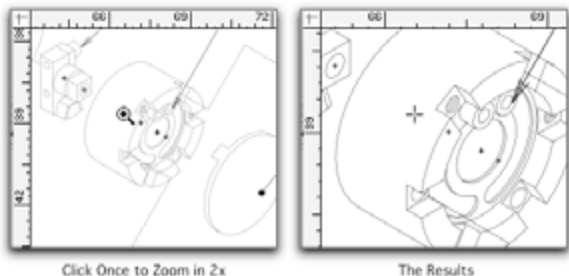
To zoom in 2X with the Zoom tool:

- ☐ Select the Zoom tool from the Tool palette.
- ☐ Move the cursor to the drawing area.

The cursor will change to the Zoom In cursor.

- ☐ Place the cursor in the area you wish to magnify and click.

You will automatically zoom in 2X with each click of the mouse button when the Zoom tool is active.



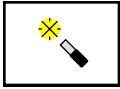
To return to the original, less magnified view, choose Previous View from the View menu.

The Zoom tool can also be used to dynamically zoom into a specific area by encompassing the area with a rectangle drawn with the Zoom tool.

To magnify a selected part of the drawing:

- ☐ Select the Zoom tool from the Tool palette.
- ☐ Move the cursor to the drawing area.
- ☐ Place the cursor in the area you wish to magnify and drag diagonally away from this point. As you drag the mouse, you will notice a selection rectangle being drawn. The rectangle should be drawn to encompass the area that you wish to enlarge.
- ☐ Release the mouse button.

The screen will now be redrawn to display the area you selected. The size of your zoom selection rectangle will govern the zoom factor.

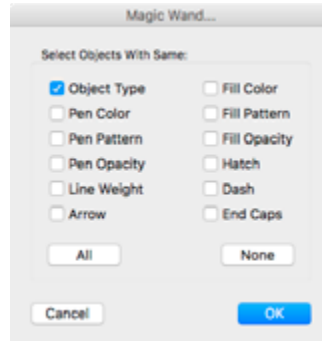


Magic Wand Tool

This tool lets you use a single click to select all objects that match your specified attributes.

To use the Magic Wand tool:

- ❑ Press the Option key and select the Magic Wand tool from the Tool palette. The following dialog will appear:



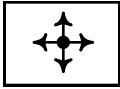
- ❑ Click on an inactive checkbox to activate the attribute to be used as the selection criteria. To deactivate an attribute click on an active checkbox.
- ❑ Click OK to return to the drawing window; the Magic Wand cursor appears.
- ❑ Click on an object to select it.

All other objects matching the attributes specified in the dialog box are selected.

Checking Object Type

The tool will select only those objects which are the same as the first object selected. For example, if a circle is the first object selected, only circles (along with the related matching attributes) are selected.

Checking Pen Color, Pen Pattern, Pen Opacity, Line Weight, Arrow, Fill Color, Fill Pattern, Fill Opacity, Hatch, Dash, and End Caps specifies those attributes by which objects are selected.



Move Object Tool

This tool uses a single click to select an object about a preset snap point and a drag operation to move it.

To use the Move Object tool:

- ☐ Select the Move Object tool from the Tool palette.
- ☐ Move the cursor into the drawing area, the Selection cursor changes to a Target cursor.
- ☐ Position the cursor on the object to be moved.
- ☐ Press and hold the mouse button, the object is automatically selected and handles appear. Drag the object to the desired location and release the mouse.
- ☐ To duplicate an object while moving it, press the Option key while dragging. The duplicate cursor will appear, release the mouse then release the Option key.

Dynamic Snapping is automatically in effect when you position the cursor over one of the default snap locations, as defined by the locations of handles.

You may also force snapping using any of the predefined snap keys.

The Shift key may also be used to constrain the move by dragging operation to either the X or Y axis.

The Move Object tool may also be applied after you have selected objects in the drawing.

If the objects are selected first, position the Move Object cursor over one of the selected objects and drag the mouse; all selected objects will be moved.

Placing the Move Object cursor over the handle of a selected object will move the selected object from that automatic snap point: the object cannot be stretched with the Move Object tool.

To stretch an object, use the Selection Arrow tool.



Pan Tool

This is a fast and efficient way to move around your drawing.

To pan around your drawing:

- ☐ Select the Pan tool from the Tool palette.
- ☐ Move the cursor to the drawing area.
- ☐ Drag the hand to pan the drawing area.



NOTE: Dragging the cursor beyond the current viewable area will automatically pan the drawing window.

Text Tools



Text Tool

The Text tool allows you to add a line or paragraph of text to a drawing.

When a new PowerCADD document is opened, the text defaults (font, color, size, style, and so on) are those saved in the Untitled stationery drawing or in your custom stationery pad drawing.

Selecting the Text tool and choosing the appropriate text characteristics from the Text menu or the Text Attributes floating window may change these defaults.



NOTE: Picking the Text tool can automatically switch you to the Text Layer (or another layer name of your choosing) if the [Layer Tools](#) feature is checked on in the Layout Menu. If the Layer Tools feature is not checked on, text is drawn on the active layer.



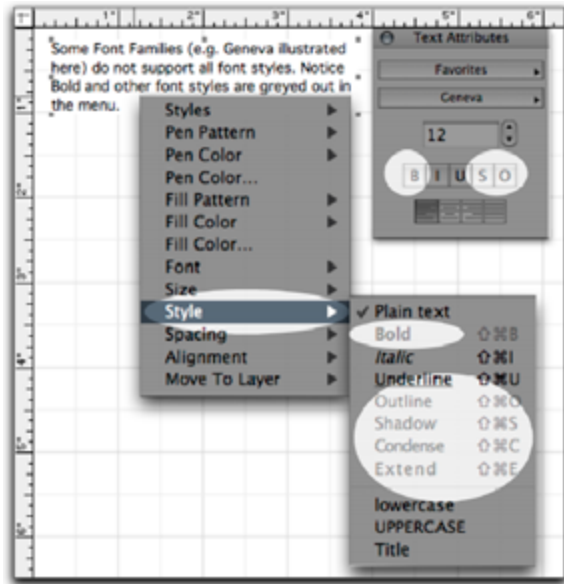
NOTE: After saving your PowerCADD 9 drawing, all text entries are automatically indexed by Spotlight in MacOS. Older PowerCADD drawings must be saved as PowerCADD 9 to index their text for Spotlight. For more information on using Spotlight to search for content in a document see Apple's help and support files included in MacOS.

PowerCADD takes full advantage of the technologies of the MacOS for smooth text rendering without being affected by font size or rotation angle.

Any font size, style or family (e.g. Helvetica, Times, etc) installed and available in your word processor or other applications can be used in your PowerCADD drawings. There are no special font requirements for PowerCADD.

Font sizes can range from as small as 1 point to 1000 point or larger.

If a particular font style is grayed out in the Text Menu, or if you are unable to assign your text a specific font style (e.g. bold or bold-italic) try changing to a different font family (e.g. change from Geneva to Arial). Alternatively you may wish to contact the font manufacturer to see if they offer the particular font style you are looking for.



To add text to a PowerCADD document:

- ☐ Select the Text tool from the Tool palette.
- ☐ Move the cursor to the drawing area where you wish to add text. When you move to the drawing area, the cursor turns into a Text Insertion cursor.
- ☐ Click in the area chosen for text (or click and drag), and the Text Insertion cursor becomes a blinking vertical bar.


Text Insertion Cursor



- ☐ Type the desired text on the keyboard.
- ☐ Correct immediate mistakes by using the Delete or Backspace keys.
- ☐ Use the Return key to add a new line to the text.

To add a paragraph to the drawing:

- ☐ Select the Text tool from the Tool Palette.
- ☐ Move the cursor onto the drawing area where you wish to add the paragraph.
- ☐ Drag to indicate the size of the paragraph. A temporary rectangle will be drawn indicating the left and right boundary of the paragraph.
- ☐ Type the desired text on the keyboard. Text will be placed inside the rectangle in paragraph format.
- ☐ Correct immediate mistakes by using the Delete or Backspace keys.

The Text tool also works like a word processor with words automatically wrapping onto the next line when the right boundary is ‘bumped’.

The rectangle shown below defines the temporary boundary of your paragraph. If your initial paragraph boundary is not large enough, the boundary will dynamically grow downwards while typing text.

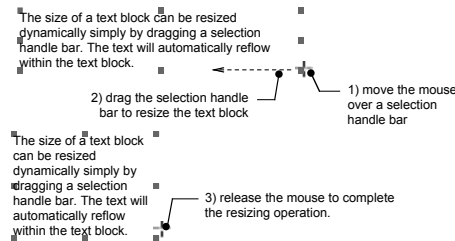
You have control over the formatting of text with in a single text block. Different font sizes, colors, styles, families, and attributes can be used within the same text block.

Resizing Text Blocks

Long single lines of text may also be reshaped into a paragraph block.

To dynamically resize and reflow a text block:

- ☐ Select a block of text in the drawing. The selection handles will appear.
- ☐ Select a selection handle and drag to resize the text block. The text is reflowed as you drag.
- ☐ Release the mouse when the text block is at the desired size.



The Edit Window may also be used to specify the exact width of a selected text block. When the text block changes sizes the text automatically reflows.

To use the Edit Window to resize and reflow a text block:

- ☐ Select a block of text in the drawing. The selection handles will appear.
- ☐ Press the Tab key to activate the Edit Window.
- ☐ Enter the desired value in the Width field of the Edit Window and press the Return or Enter key.
- ☐ Enter the desired value in the Height field of the Edit Window and press the Return on Enter key.

The text is automatically reflowed vertically to accommodate the specified width of the text block.

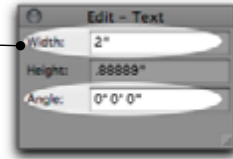
The Edit Window may be used to specify the exact width (or rotation angle) of a text block. The text is automatically reflowed vertically to accommodate the specified width

1) select the text block

2) enter the desired width in the Edit Window Width field and press the "Return" or "Enter" key

The Edit Window may be used to specify the exact width (or rotation angle) of a text block. The text is automatically reflowed vertically to accommodate the specified width

3) the text block is resized to the specified width and the text is automatically reflowed vertically



NOTE: The Edit Window can also be used to specify the rotation angle of a selected text block by entering a specific value in the Angle field of the Edit Window.

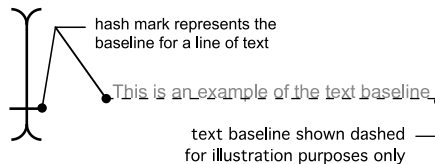
Adding Text to a Line of Existing Text

To add text to a line of existing text:

- ☐ Select the Text tool from the Tool palette.
- ☐ Use the cursor to select the text you wish to modify.
- ☐ Move the text cursor to the drawing area where you wish to modify text. Note when you move the Selection Arrow cursor to the drawing area, the arrow turns into a Text Insertion cursor.
- ☐ Move the Text Insertion cursor into position within the text block and click.
- ☐ You may now modify the text.



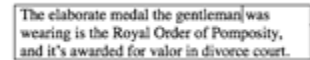
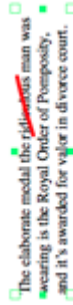
Double-click on a text block with the Selection Arrow tool selected to edit it.



Editing Rotated Text

Rotated text may be edited by clicking on the text with the Text Insertion cursor.

Rotated text, if selected with the Text Insertion cursor, will be rotated to zero for normal editing. When editing is complete and another selection is made, the text will return to its original rotation. A rotated text block can be resized, even when it is rotated.



Rotated Text Before Editing

Rotated Text During Editing

Deleting Characters from Existing Text

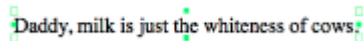
Position the Text Insertion cursor to the right of the text to be deleted. Backspace over the incorrect text.

Alternatively, you may select the text to be deleted as in standard word processors, and press the Backspace key to delete.

Changing the Appearance of Text in a Drawing

To change the characteristics of a block of text within a drawing document:

- ☐ Select the Selection Arrow tool from the Tool palette.
- ☐ Move the cursor to the drawing area and click in the block of text you wish to modify. The handles surrounding the block of text have been activated, as shown in the following example.



- ☐ Choose the appropriate text characteristics you wish to modify from the Text menu.

Characteristics include choice of font type, style, and size.

In addition, you may modify such text formatting features as justification and the case of letters.

After you have selected the text characteristics from the Text menu, the desired effects will be immediately noticeable on the screen.



See [Text](#) in the Menu chapter for details on working with fonts and text styles.



See [Text Attributes](#) in the Menu chapter for details on working with the Text Attributes floating window for setting font and text styles. Also see [Attributes](#) in the Menu chapter for details on working with the Attributes floating window for setting color, fill and other attributes of text blocks.



NOTE: To change the attributes of a particular letter, word or range of words in a text block use the text tool, drag to highlight the particular characters then change the desired attributes.

Moving Text in a Drawing

To move a block of text within a drawing document:

- ☐ Select the Selection Arrow tool from the Tool palette.
- ☐ Move the cursor to the drawing area and select the block of text you wish to move. Hold the mouse button down after you select the block of text.

The Selection cursor will turn into a Move cursor.

You may now reposition the block of text as you would any other drawing object.



NOTE: The Autoscroll feature is in effect while moving objects in the PowerCADD document window.

Deleting Text From a Drawing

To delete a block of text from a drawing document:

- ☐ Select the Selection Arrow tool from the Tool palette.
- ☐ Move the cursor to the drawing area and select the block of text you wish to delete.

The handles surrounding the block of text will appear.

- ☐ Press the Delete (or Backspace) key.
—or—
- ☐ Choose Clear or Cut from the Edit menu.
—or—
- ☐ Press Command-X.

Alternatively, when editing text you may drag over part of the text to highlight it. When text is highlighted, it may be cut or copied and pasted into the same text block or into a text block in any drawing (eliminating

unnecessary typing). The active text insertion point determines where text is pasted.

Still True?



NOTE: You can also drag and drop text from your PowerCADD drawing into your word processor or spreadsheet, **the Finder Desktop (creates a picture clipping file)** or another PowerCADD drawing.

Edit Text Window

Existing text may be edited in an enlarged Edit Text window.

Any attribute which can be assigned from the Text menu can be applied in the Edit Text window or from the [Text Attributes](#) floating window.

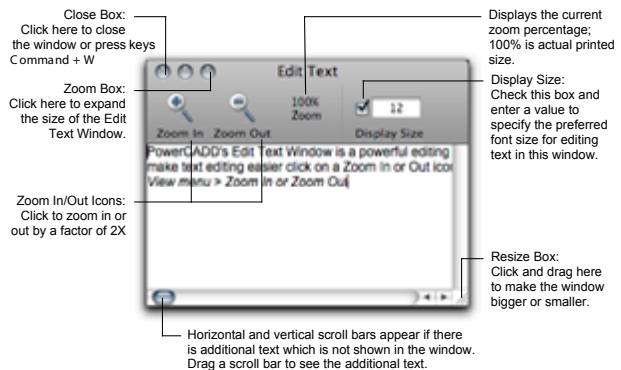
Pen color may be selected from the [Attr Menu](#) or from the [Attributes](#) Floating window.

To activate the Edit Text window:

- ☐ Select the text to be edited.
- ☐ Choose Edit from the Edit Menu.

The selected text opens in a new floating window on top of your drawing.

- ☐ Edit the text as needed.
- ☐ When finished, click on the Edit Text window's close box or press Command-W.



To make text editing easier, zooming in or out is available in the Edit Text window.

To zoom in or out in the Edit Text Window:

- ☐ Activate the Text Edit window as described earlier.
- ☐ Choose Zoom In from the View menu or use the key command you have assigned to that menu item.

- ☐ Text will display larger when zoomed in, and you may edit the text attributes using available menu items in the Text and Attr menus.
- ☐ When finished, click on the Edit Text window's close box.

To zoom out in the Edit Text Window, choose Zoom Out from the View menu or use the key command you have assigned to that menu item.



NOTE: Click on the Zoom In or Zoom Out Magnifying Glass icons in the Edit Text window to change the display size of text. This change affects the font size displayed in the window only; the actual font size attribute of the text is not affected. To make editing small font sizes easier, the Edit Text Window allows you specify a preferred font size. This change only affects the font size for editing in the Edit Text window; the actual font size attribute is not affected. Closing the Edit Text window will render the text to screen at the original font size.

To specify a display font size in the Edit Text Window:

- ☐ Activate the Text Edit window as described earlier.
- ☐ Check the Display Size box and enter a value in the Display Size field.

The next time you open the Edit Text window the zoom ratio is automatically set to simulate displaying text in the specified size.



Advanced Text Tool

The Advanced Text tool integrates the ability to create single lines of caption text and paragraph text blocks along with the ability to create and edit tables in your drawing.

Formatted text and tables may also be copied and pasted from other spreadsheet or word processing applications.

Advanced Text operates in two basic ways. A single click to create single lines of advanced caption style text or a click-and-drag to create advanced paragraph text.

The Advanced Text Editing window is used to enter or edit text or tables, and provides numerous advanced features including control over paragraphs and line spacing, kerning, and ligatures.



NOTE: Selecting the Advanced Text tool can automatically switch you to the Text Layer (or another layer name of your choosing) if the [Layer Tools](#) feature is checked on in the Layout Menu. If the Layer Tools feature is not checked on, Advanced Text is drawn on the active layer.



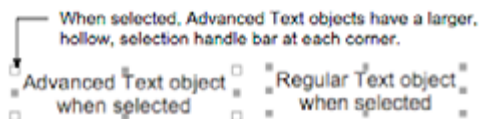
NOTE: After saving your PowerCADD 9 drawing, all advanced text entries are automatically indexed by Spotlight in the MacOS. Older PowerCADD drawings must be saved as PowerCADD 9 to index their text for Spotlight. For more information on using Spotlight to search for content in a document see Apple's help and support files included in the MacOS.



Also, see [Text](#) in the Menu chapter, and [Text Tool](#) in this chapter, for general information on moving and working with text. Also see [Attributes](#) in the Menu chapter for details on working with the Attributes floating window for setting color, fill and other attributes of text blocks. Also reference the Apple MacOS help files for information on working with the Font Panel and related system level text features.

Differences Between Advanced Text and Regular Text

Both advanced and traditional text objects are treated the same when using the Selection Filter to select text objects only. However, it is easy to tell the two apart when selected; the corner selection handles of Advanced Text objects are larger and hollow when compared to traditional text objects as pictured here:



Advanced Text takes advantage of numerous MacOS system level features that are not available with the traditional PowerCADD Text tool. To retain the advanced formatting features, when coordinating drawings that will be saved and opened in previous versions of PowerCADD, all Advanced Text objects including tables will be converted to PDF. The resulting PDF objects will print correctly in previous versions of PowerCADD but cannot be edited.

Creating Advanced Caption Text

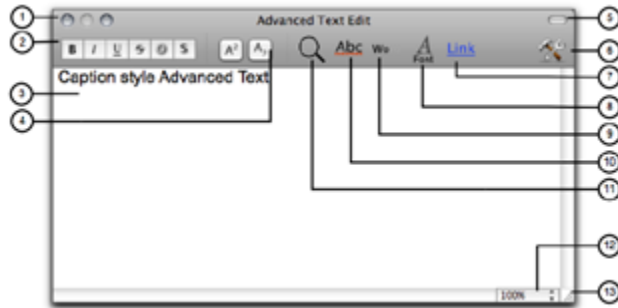
To create a single line of advanced caption style text:

- ☐ Select the Advanced Text tool from the Tool palette.
- ☐ Move the cursor into the drawing window and the cursor changes to a Text Insertion cursor.
- ☐ Click to define the left hand starting point of the text.
- ☐ The Advanced Caption Text Edit Window opens.
- ☐ Type the desired text using the keyboard.
- ☐ Close the Advanced Text Edit Window by clicking the close box, or pressing Command-W, or by clicking anywhere in the drawing window.

The text will be inserted into the drawing.

Advanced Caption Text Edit Window Features

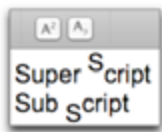
Access to various editing and formatting controls for Advanced Caption Text are accessed from the toolbar or contextual menus in the Advanced Caption Text edit window.



(1) **Window Buttons:** Click the left button to close the window (or press Command-W). Click the right most button to maximize the size of the window to fit your screen. When a window is maximized clicking the same button will minimize the window to its previous size. The center button is always inactive.

(2) **Font Styles:** When text is selected, click on an icon to apply the designated style. If a style button is dimmed the font family chosen (e.g. Geneva) may not support that particular style.

(3) **Window Body:** The primary area of the text edit window used to enter and edit text.



(4) **Super Script/Sub Script:** Clicking the Super Script icon will raise the selected characters above the baseline. Clicking the Sub Script icon will lower the selected characters below the baseline. The incremental amount of the super or sub script offset is proportional to the font size used. The super or sub script setting may be applied more than once by clicking multiple times on the appropriate icon. To remove super or sub script formatting, select the super or sub scripted characters and click on the opposing icon. For example, to restore super scripted characters select them and click on the sub script icon.

(5) **Toolbar Button:** Click this button to show or hide the Toolbar.

(6) **Customize:** Click to display a drop down panel containing items to customize the toolbar. You may add or remove items, change their location, or change the display size of icons in the toolbar. Control- or right-clicking on the toolbar will display a contextual menu to access the customize menu or change the display of the toolbar icons.

(7) **Link:** Select the desired characters and click to open the Link Destination Sheet. Type the desired internet address (e.g. <http://www.engsw.com>) and click OK. The selected characters will be formatted as blue and underlined in both the Advanced Text Edit Window and in the drawing. Links are not active in the main drawing window. To visit a link, open the Advanced Text Edit Window and click on the linked text.

(8) **Font:** Click to open the MacOS Font Panel. The Font Panel is a standard font selection window used across many MacOS applications to choose a particular font or to set the style, size and other formatting characteristics. For more information on working with the Font Panel search for 'Font Panel' in the help files that come with MacOS.

(9) **Word:** Click to see a list of suggestions to complete or change the current word.

To see and apply a suggested word:

- ☐ Position the text insertion cursor to the right of the word and click. The text insertion cursor will blink.
- ☐ Click *Word:* a list of suggested words will appear. If a list does not appear then there are no suggested words.
- ☐ Click on the desired suggested word and press the Return key to apply it. The up and down arrows on the keyboard will also navigate through the list of suggested words; press the Return key to apply the selected suggested word

(10) **Abc:** Click to open the MacOS Spelling and Grammar Checker Panel. The Spelling and Grammar checker is a standard window used across many MacOS applications. The Spelling and Grammar Checker Panel can also be used to select dictionaries or add words to your dictionary. If words in the Advanced Text Edit Window appear with a red underline then they have been flagged as being misspelled or perhaps they are spelled correctly but not in the dictionary. To enable or disable check as you type spell checking, Control- or right-click in a blank area of the Advanced Text Edit Window and choose the appropriate item from the contextual menu. For more information on working with the Spelling and Grammar checker search for 'Check Spelling' in the help files that come with MacOS.

(11) **Find:** Click to open the MacOS Find/Replace text dialog. The Find/Replace operation only looks inside of the currently active (open) Advanced Text Edit Window. Other Advanced Text blocks in the drawing are ignored even if they contain a word defined in the Find/Replace dialog.



TIP: To perform a global Find/Replace operation on all text in the drawing, choose Find... from the Text menu. The Find... menu item looks inside of both

regular and advanced text blocks, leader line objects and text inside of groups.

(12) **Zoom:** Click to set how large or small text is displayed when editing. A zoom percentage of 100% is actual printed size. The zoom percentage, whether larger or smaller, does not affect the true font size attribute (e.g. 12 points is still 12 points regardless of the zoom percentage assigned).

(13) **Resize box:** Click and drag to make the window larger or smaller.

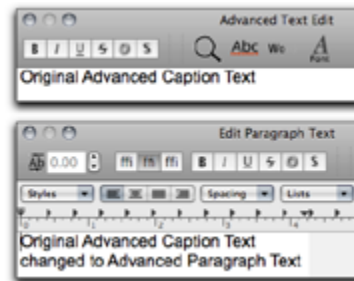
Converting to Paragraph Text

To change Advanced Caption Text to Advanced Paragraph Text:

- ☐ Select the Advanced Caption Text block in the drawing.
- ☐ Drag one of the selection handles to increase or decrease the size of the text block.

—or—

- ☐ Select the Advanced Caption Text block in the drawing.
- ☐ Press the Tab key to activate the Edit Window.
- ☐ Type a new value in the Width field and press the Return key.



HINT: To convert multiple Advanced Caption Text blocks to Advanced Paragraph Text blocks at the same time, Group them first (Arrange menu, Group). Then select the handle of the group and drag to increase the size of the group.

Creating Advanced Paragraph Text

Paragraph text has advantages over caption text in that you have access to additional options for character, line and paragraph formatting, the ability to create tables, or to copy and paste formatted table or text content from other applications.

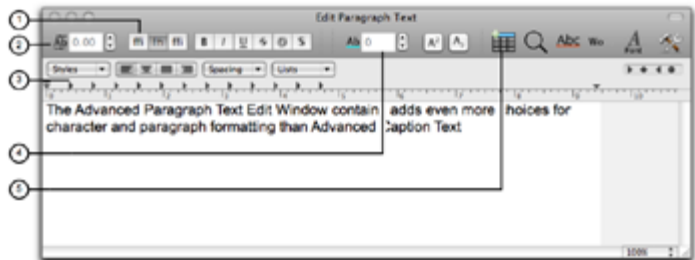
To create an Advanced Paragraph Text Block:

- ☐ Select the Advanced Text tool from the Tool Palette.
- ☐ Move the cursor into the drawing window; the cursor changes to a Text Insertion cursor.
- ☐ Click and drag to define a rectangle to set the size of the text block.
- ☐ The Advanced Paragraph Text Edit Window opens.
- ☐ Type the desired text using the keyboard.
- ☐ Close the Advanced Paragraph Text Edit Window by clicking the close box, or by pressing Command-W, or by clicking anywhere in the drawing window. The text will be inserted into the drawing.

Advanced Paragraph Text Edit Window Features

Various editing and formatting controls for Advanced Paragraph Text are accessed from the toolbar or contextual menus in the Advanced Paragraph Text Edit Window.

The Advanced Paragraph Text Edit Window contains all of the options available for Advanced Caption Text plus the following items: (see [Advanced Caption Text](#) discussed earlier in this section for additional descriptions).



(1) **Ligature Adjustments:** Allows you to combine two or more characters together to create a ligature. Æ ('A' and 'E') and æ ('a' and 'e') combined together are examples of ligatures. Ligatures are only supported for fonts with extended character sets or expert font sets so not all fonts installed in MacOS will support ligature adjustments.

To adjust ligatures:

- ☐ Select the characters to be adjusted in the Advanced Paragraph Text Edit Window.
- ☐ Click on the Ligature button to apply the desired ligature adjustment.



(2) Kerning Adjustments: Allows you to increase or decrease the spacing between characters. The value in the kerning field applies standard kerning to the selected characters using information provided by the font. Because kerning adjusts the natural space between characters, the overall effect of kerning may not be the same for all characters in a given font or font style. The effect of kerning will also vary between font families (e.g. Times versus Helvetica) and font types (e.g. Open Type versus True Type). The overall amount of kerning of selected characters can also be affected by the preceding or following characters (including spaces) in other parts of the line of text.

To adjust kerning:

- ☐ Select the characters to be adjusted in the Advanced Paragraph Text Edit Window.
- ☐ Enter a numeric value in the kerning field or click the up or down arrow icons. Positive values increase the spacing between characters. Negative values decrease the spacing between characters.
- ☐ To restore the default character spacing repeat the kerning adjustment but enter a value of zero in the kerning field



(3) Rulers: Formatting rulers provide access to control over tabs, left indents, alignment, font styles, paragraph spacing and lists. These items contain both standard and custom choices used across many MacOS applications.



Styles: The Style popup menu contains standard choices for font styles. If no text is selected, the chosen style is applied to all new text entered. If existing text is selected the new style is applied to the selection. To create your own favorite style, select the text with the desired format, color, font

type and choose Other... from the Styles popup. Favorite styles are shared among all MacOS applications which use the standard Styles popup menu.

Justification: Click on a justification icon to set the desired left, center, full, or right justification. If no text is selected, the new justification is applied to all new text entered. If existing text is selected the new justification settings are applied to the selection.

Spacing: The Spacing popup menu contains standard choices for adjusting line and paragraph spacing. If no text is selected, the spacing is applied to all new text entered. If existing text is selected the new spacing is applied to the selection. For additional control of both line and paragraph spacing choose Other... from the Spacing popup.

Lists: The Lists popup menu contains standard choices for adjusting line and paragraph spacing. If no text is selected, the spacing is applied to all new text entered. If existing text is selected the new spacing is applied to the selection. For additional control of the bullets or to add a prefix and suffix to a list item choose Other... from the Spacing popup.

Tabs: Existing tabs across the ruler may be moved by dragging them left or right. To remove an existing tab click and drag down and away from the ruler. To add a new tab (left, center, right or decimal tab) drag its corresponding icon from the choices in the top right onto the ruler. To adjust both the position of existing text and any associated tab, select the text first, then drag the tab in the ruler.

Indents: The horizontal bar at the left end of the ruler sets the indentation for the first line of a paragraph. If no text is selected the indentation will apply to new text entered. If existing text is selected the first line of the paragraph is indented and the related text is reflowed.



TIP: To change the right hand margin, exit the Advanced Paragraph Text Edit Window. Select the text block in the drawing and drag a selection handle wider or smaller. The text will automatically reflow.

(4) **Baseline Shift:** Enter a specific value or click the up or down icons to increase or decrease the selected characters up or down relative to the baseline of surrounding text. A positive value moves the selection above the baseline of the rest of the line; a negative value moves it below the baseline.

(5) **Tables:** Click this button to create a new table. Tables are discussed in detail later in this section.

Using Formatted Text from Other Applications

There may be times when it is quickest to reuse existing text that was typed and formatted in other applications.

To use text from another application which cannot be edited in PowerCADD:

- ☐ Use the print command to create a PDF file in the other application.
- ☐ Open a drawing and choose Place from the File menu.
- ☐ Place the PDF file into your drawing.
- ☐ The formatting of the original document is retained but cannot be edited in PowerCADD.

If the file contained more than one page, only the first page is placed. If you need all the pages of a multi-page document create a custom page size (see the help file included with MacOS) which is large enough to include your entire document on a single page. The PDF file will then place as a single item in PowerCADD.



TIP: To convert the PDF file to editable text select the PDF file containing the text, then choose PDF To Objects from the Tool menu. The PDF is converted to Advanced Text objects which may be edited. Note, paragraphs are converted to distinct lines when converted; to maintain paragraph formatting use the copy and paste method described below.

To use text from another application which can be edited in PowerCADD:

- ☐ Select all the text in the original file and choose Copy from the Edit menu.
- ☐ In the PowerCADD drawing, choose the Advanced Text Tool and click and drag to create a new Paragraph Text Block.
- ☐ In the Advanced Paragraph Text Edit Window choose Paste from the Edit menu.

The formatting of the original text, including font size, style, and tabs, is retained.

- ☐ Close the Advanced Text Edit window and the text is inserted into the drawing as an editable text block.

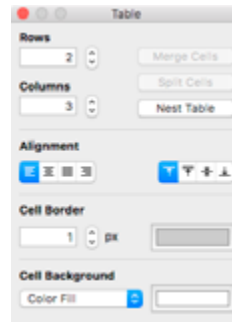


Creating Tables

The Advanced Paragraph Text tool is used to create new tables. The table may be inserted as part of an existing Advanced Paragraph Text Block, or be independent of other text blocks.

To create a new, independent, table:

- ☐ Select the Advanced Text tool from the Tool Palette.
- ☐ Click and drag to define a new Advanced Text block. **The Advanced Paragraph Text Edit Window opens.**
- ☐ Click on the Add Table icon in the Toolbar.
- ☐ A new empty table is created containing two rows and three columns. The Table Inspector window opens.



- ☐ Use the Table Inspector to set the properties of the table.
- ☐ Click in a cell to enter data. All font formatting features of the Advanced Paragraph Text Edit Window are available.
- ☐ Click the top left button in the Table Inspector window to close it.
- ☐ Close the Text Edit Window and the Table is inserted into the drawing.

To create a table that is part of an existing Advanced Paragraph Text Block:

- ☐ Select the Advanced Paragraph Text block in the drawing and press Command-E (or double-click to edit it).
- ☐ In the Advanced Text Window, move the cursor to where you want the table inserted and click: a flashing text insert icon will appear.
- ☐ Click on the Add Table tool icon in the Toolbar.
- ☐ A new empty table is created in the specified location.
- ☐ Make the desired entries into the table, close the Table Inspector and close the Advanced Text Edit Window. The table appears as part of your text block in the drawing.

Customizing Table Properties

Columns & Rows



Use the Table Inspector to adjust various properties of selected cells after a table is created.

Adjust the number of Columns or Rows: Type a value in the corresponding field or click the up or down arrows.

Merge Cells: When two or more cells are selected, click Merge Cells to combine them into a single cell. If both cells contained data, the data is retained and inserted onto two separate lines in the new cell.

Split Cells: When a merged cell is selected, click Split Cells to separate them. If the cell being split contained data, the new split cell will be empty.

Alignment

Text in a cell may be aligned using a combination of two methods.



Horizontal: Text in a cell may be aligned horizontally the same as traditional text outside of a table. Click in the cell, then click the left, center, full or right justification icons to apply the corresponding alignment.



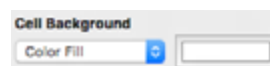
Vertical: Text within the cell may also be aligned vertically by clicking in the cell, then clicking on the corresponding top, baseline, center or bottom alignment icons.

Cell Borders and Backgrounds

The cell border and background for a single selected cell, or selected range of cells, may be adjusted for color and line weight.



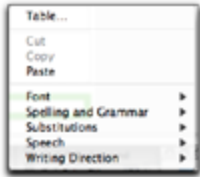
Cell Border: Enter a numeric value in the Cell Border field or click on the up or down arrow icons to change the size of the cell boarder in one pixel increments. Click on the color rectangle to select a cell border color from the MacOS Color Panel.



Cell Background: Choose the Color Fill popup menu to assign a color fill or no fill to a cell. A cell background of None will allow objects below the

table to be visible through the table. Choose Color Fill from the popup menu and click on the color rectangle to select a cell background color from the MacOS Color Panel. A background color of white will hide all objects below a table.

Editing an Existing Table



To edit an existing table that is either independent or part of another Advanced Paragraph Text block:

- ☐ Double-click on the Advanced Paragraph Text block containing the table.
- ☐ The Advanced Text Edit Window will open.
- ☐ Press the Control key and click on the existing table or click the existing table and click the New Table icon in the toolbar.
- ☐ Choose Table... from the contextual menu.
- ☐ The Table Inspector will open.
- ☐ Make the desired changes. Close the Table Inspector and close the Advanced Text Edit Window; the revised table is displayed in the drawing.

Using Formatted Text from Other Applications

There may be times when it is quickest to reuse existing tables created in another spreadsheet application such as Microsoft Excel or Apple Numbers.

To use a table from another application which cannot be edited in PowerCADD:

- ☐ Use the print command to create a PDF file in the other application.
- ☐ Open a drawing and choose Place from the File menu.
- ☐ Locate the PDF file and place that into your drawing.
- ☐ The formatting of the original document is retained but cannot be edited.

If the file contained more than one page, only the first page is placed. If you need all the pages of a multi-page document create a custom page size (see the help file included with MacOS) which is large enough to include your entire document on a single page. The PDF file will then place as a single item in PowerCADD.

To use a table from another application such as Excel or Numbers which can be edited in PowerCADD:

- ☐ Select the range of cells in the spreadsheet application and choose Copy from the Edit menu.
- ☐ In the PowerCADD drawing, choose the Advanced Text Tool and click and drag to create a new Paragraph Text Block.
- ☐ In the Advanced Paragraph Text Edit Window choose Paste from the Edit menu.
- ☐ A new table is created matching the table in the spreadsheet application. Control- or right-click on the table and choose Table... from the

Advanced Text Edit Window Contextual Menus

contextual menu to open the Table Inspector to make any changes.

- ☐ Close the Advanced Text Edit window and the table is inserted into the drawing as an editable text block.

Following is a list of locations in the Advanced Text Edit Window which will open a contextual menu specific to that location. In many cases the contextual menu provides access to additional features or editing options.

To apply a contextual menu item simply select it from the list of contextual menu items that appears.

- ☐ Control-click in a blank area in the body of the window.
- ☐ Control-click on a selected character(s).
- ☐ Control-click on a linked character(s).
- ☐ Control-click on the Toolbar.
- ☐ Control-click on an icon in the Toolbar.

Editing Advanced Text

To edit an existing Advanced Caption or Paragraph Text Block in the drawing simply select the text block and press Command-E or double-click on it with the Selection Arrow tool. In both cases the Advanced Text Edit Window will open.

Make the desired changes and close the window to have those changes appear in the drawing.

Changing Pen Color Attributes

You can change the pen color for all text in an Advanced Caption or Paragraph Text Block or just selected text within a text block.

To change the color of all text in any Advanced Text Block:

- ☐ Use the Selection Arrow tool and click once on the text block to select it.
- ☐ Choose the desired pen color from the Attributes Window, a contextual menu (Control- or right-click on the text block) or the Attribute menu.
- ☐ All text in the text block will change to reflect the selected color.

Hypertext link colors remain blue and underlined regardless of what pen color attribute is selected for the text block. If multiple Advanced Text Blocks are selected, the chosen color is applied to all selected items.

To change the color of selected text in an Advanced Text Block:

- ☐ Use the Selection Arrow tool and double-click on the text block to open the Advanced Text Edit Window (or press Command-E).
- ☐ Select the desired characters and choose a pen color from the Attribute menu.
- ☐ The selected text in the text block will change to reflect the chosen color.

Hypertext link colors remain blue and underlined regardless of what pen color attribute is selected for the text block.



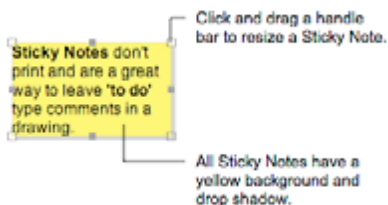
Sticky Note Tool

The Sticky Note tool allows you to insert non-printing type notes and comments in a drawing.

Inserting a Sticky Note can be an effective way to draw attention to comments regarding design or drawing changes that need to be made prior to issuing a drawing for client review.

To insert a new Sticky Note in a drawing:

- ☐ Select the Sticky Note tool from the Tool Palette.
- ☐ Move the cursor into the drawing window; the cursor changes to a Text Insertion cursor.
- ☐ Click and drag to define a text block for your Sticky Note.
- ☐ The Advanced Text Edit window opens with a yellow background.
- ☐ Type the desired text using the keyboard.
- ☐ Close the Advanced Text Edit window; the Sticky Note will be inserted into the drawing.



NOTE: After saving your PowerCADD 9 drawing, all sticky note entries are automatically indexed by Spotlight in MacOS. Older PowerCADD drawings must be saved as PowerCADD 9 to index their sticky notes for Spotlight. For more information on using Spotlight to search for content in a document see Apple's help and support files included in MacOS.



NOTE: The yellow background and drop shadow attributes of a Sticky Note cannot be modified.

To edit an existing Sticky Note in a drawing:

- ☐ Select the Sticky Note tool from the Tool Palette.
- ☐ Double-click on the existing Sticky Note in the drawing.
- ☐ Type the desired text using the keyboard.
- ☐ Close the Advanced Text Edit window;

The revised Sticky Note will be inserted into the drawing.



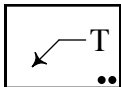
NOTE: Select the Selection Arrow tool and double-click on any Sticky Note to edit it.

To resize an existing Sticky Note in a drawing:

- ☐ Select the Selection Arrow tool from the Tool Palette.
- ☐ Click on the existing Sticky Note in the drawing to select it.
- ☐ Press and drag on an edit handle; the text will automatically reflow to match the new size.
- ☐ Release the mouse to complete the operation.



NOTE: Use the Edit Window to specify an exact width or rotation angle for a Sticky Note.

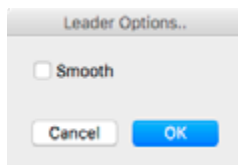


Leader Tool

The Leader tool allows you to create custom leaders with text. The tool has an option that allow you to always create a smooth looking leader line.

To set the behavior of the tool:

- ❑ Press the Option key and select the Leader tool from the Tool palette. The following dialog will appear:



- ❑ Select the Smooth checkbox and click OK.

To use the Leader tool:

- ❑ Select the Leader tool from the Tool palette.
- ❑ Place the cursor where you want the leader line's arrow to be located and press the mouse button.
- ❑ Drag to position the bend in the leader line and release.
- ❑ Move the cursor horizontally to position the leader's endpoint and click.



- ❑ Enter the text associated with the leader. The text cursor is automatically placed when the leader is finished so you can type information without selecting the Text tool. Text may be entered by standard word processing methods. A mouse click or the Enter key will complete the leader.
- ❑ Pressing the Esc key after you draw the first segment of the leader will abort the drawing operation and no object will be created. If you have created the leader line and see the text insertion cursor, the operation cannot be aborted. Pressing the Return key will automatically exit the drawing process, the leader line will be selected and pressing the Delete key will remove it from the drawing.



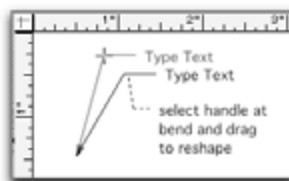
NOTE: After saving your PowerCADD 9 drawing, all leader line entries are automatically indexed by Spotlight in MacOS. Older PowerCADD drawings must be saved as PowerCADD 9 to index their leader lines for Spotlight. For more information on using Spotlight to search for content in a document see Apple's help and support files included in MacOS.



NOTE: Selecting the Leader tool can automatically switch you to the Text Layer (or another layer name of your choosing) if the [Layer Tools](#) feature is checked in the Layout menu. If the Layer Tools feature is not checked, leader lines and text are drawn on the active layer.

To edit a leader:

- ❑ To change the position of the leader text, select the text and move it as you would any object. To relocate the text horizontally or vertically, select it and press the Shift key. You may now move the text without disturbing the leader. The Shift key constrains the movement to vertical and horizontal.
- ❑ To extend the horizontal section of the leader or move it and the text to the other side, select the leader handle near the text and drag. The change can only be made horizontally. This process will not affect the angle of the leader.
- ❑ To change the angle or distance from the object, select the handle at the bend of the leader and drag. The angle and distance may be changed while the horizontal line and the text remain locked at horizontal.



- ❑ Use the Text tool to add text. Place the Text cursor where you wish to make the change, click and type or use the Edit command. Select the leader, choose [Edit](#) from the Edit menu, and type your changes.
- ❑ You may ungroup the leader line from the text. This operation will change the leader into a polygon. The leader can then be resized as you would resize a polygon, by selecting the handles and dragging. The text is now separate from the leader.

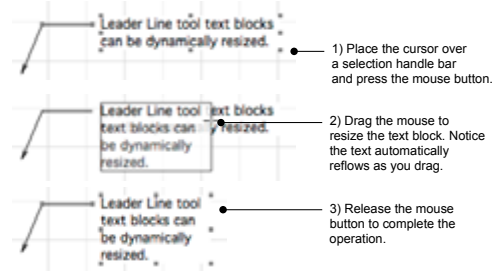
To draw a leader without text, press the Enter key when the Text cursor appears. The Leader tool will remain active so you may draw another leader.



NOTE: To create a smooth looking Leader, select the item(s) and pick Smooth from the [Tool menu, Polygon menu](#). You may also Unsmooth a previously smoothed Leader.

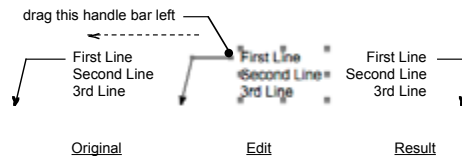
To change the size of a leader object text block:

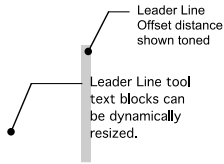
- ☐ Select the leader line. The selection handles will appear.
- ☐ Move the cursor so it is over a selection handle of the text block.
- ☐ Drag the handle to dynamically change the width of the text block.
- ☐ The height of the text block will change and the text will automatically reflow.
- ☐ Release the mouse to complete the operation.



The text alignment of Leader text will change when a Leader object is reshaped from left to right or from right to left; for example:

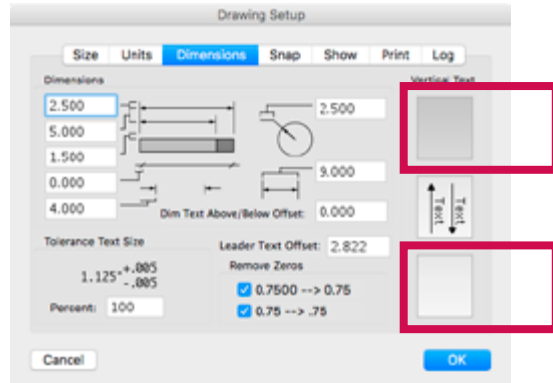
- ☐ Begin with a leader line with left justified text on the right side of the leader line as illustrated below.
- ☐ Drag the edit handle of the leader line left until the text block moves to the left of the leader line.
- ☐ The text justification will flip from left justified (original) to right justified (result) as shown below.





To specify the offset distance between the end of a leader line and the insertion point of the text block:

- ☐ Choose Drawing Setup from the Layout menu.
- ☐ Select the Dimensions tab, the following dialog will appear:



Leader Text Offset: 2.822

- ☐ Enter the desired value in the Leader Text Offset field. The value entered is at normal print size (100% printing or 1:1 scale) regardless of drawing scale.
- ☐ The value for Leader Text Offset are drawing specific and saved with the drawing.

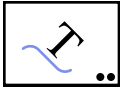
Editing Leader Text

To edit the content of Leader line text block:

- ☐ Select the Leader line containing the text block to be edited.
- ☐ Choose Edit from the Edit menu (or press Command-E).
- ☐ The Text Edit window will open. Make the desired changes and close the Text Edit window.
- ☐ Alternatively, you may also edit the text directly in the drawing window by choosing the Text tool and clicking in the text block.



NOTE: Select the Selection Arrow tool and double-click on any Leader Line text block to edit it.



Text on Path Tool

This tool maps a block of text to an object's path.

Text on Path has a variety of uses, including mapping text to rivers, streets or highways, labeling contour lines or applying text to circles and arcs to creating drawing seals. The possible uses are endless.

The text is placed at the beginning of the object and each character is rotated to match the slope of the object. This may be any object except symbols, and dimensions.

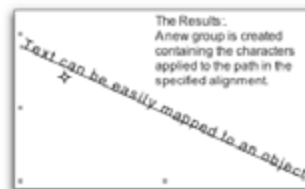
To use the Text on Path tool:

- ☐ Select the Text on Path tool from the Tool palette.
- ☐ Place the cursor on the text block and press.
- ☐ Drag the cursor to the object; the end you choose will be the end from which it starts placing text. A bridge line will connect the two objects to indicate the tool is active.
- ☐ Release the mouse button.

The result is a group containing individual characters on the path with the specified alignment.



NOTE: The Text on Path tool produces a unique text block for each character. As such MacOS Spotlight searches will only index each character, not the entire word they may represent.

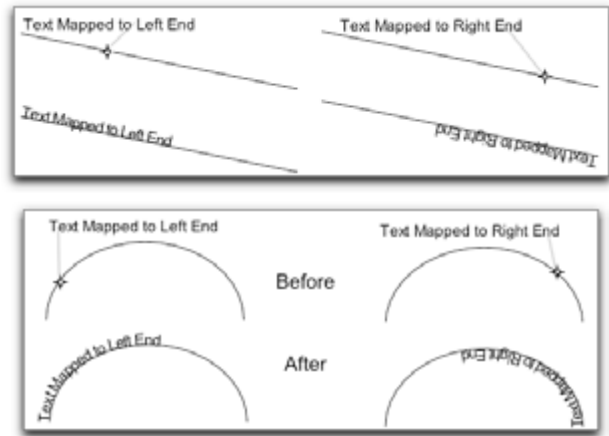


NOTE: The object being targeted to receive the text on path operation can be either a distinct object in the drawing or part of a Group of objects.



NOTE: The orientation of the text to the object depends on which end of the object you pick. For example, if you pick the left end of a line, the text will map along its top, reading left to right. If you pick the right end of a

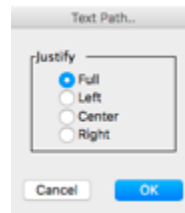
line, the text will map upside-down along its bottom, reading right to left, so justification will be reversed.



The text can be justified four ways on the object much like text is justified in a paragraph: left, right, center, and full.

Each justification setting works just like their namesakes in word processors.

To set the justification, activate the Text on Path dialog by pressing the Option key while selecting Text on Path. The following dialog appears:

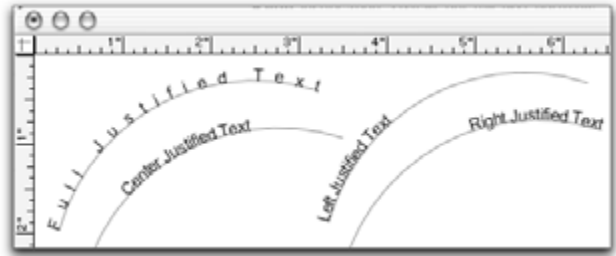


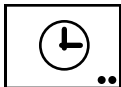
Full justification fits the text to the path, and will change the spacing between letters to accommodate the path's length.

Left justification spaces out the text normally, starting at the beginning of the path. If the text isn't long enough, blank space will be left at the end of the path. If the text is too long, a warning will appear, stating that it will truncate the extra text. Text from the end of the text block will be truncated.

Center justification spaces out the text normally, relative to the center of the path. If the text isn't long enough, blank space will be left at the beginning and end of the path. If the text is too long, a warning will appear, stating that it will truncate the extra text. Text from the beginning and end of the text block will be truncated.

Right justification spaces out the text normally, relative to the end of the path. If the text isn't long enough, blank space will be left at the beginning of the path. If the text is too long, a warning will appear, stating that it will truncate the extra text. Text from the beginning of the text block will be truncated.





Date Stamp Tool

The Date Stamp tool inserts the creation and modification times and dates into your documents as specified by the Date & Time settings in your MacOS.

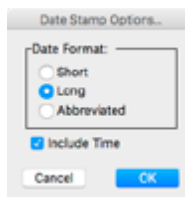
The Date Stamp object may be inserted anywhere in your drawing, and it is 'smart'. The object will automatically update each time you open or save a drawing.



NOTE: The Date Stamp tool is an External plug-in. In order for this tool to appear in the Tool menu it must be checked on when PowerCADD is launches. See [Loader Preferences](#) for more on turning External plug-ins On and Off.

To set the behavior of the tool:

- ❑ Press the Option key and select the Date Stamp tool from the Tool palette. The following dialog will appear:



- ❑ The examples below illustrate the optional settings available and how some of them will appear in your drawing.



To use the Date Stamp tool:

- ❑ Select the Date Stamp tool from the Tool palette.
- ❑ Place the cursor where you want the Date Stamp to be located and press the mouse button.
- ❑ Drag to position the object and release the mouse.

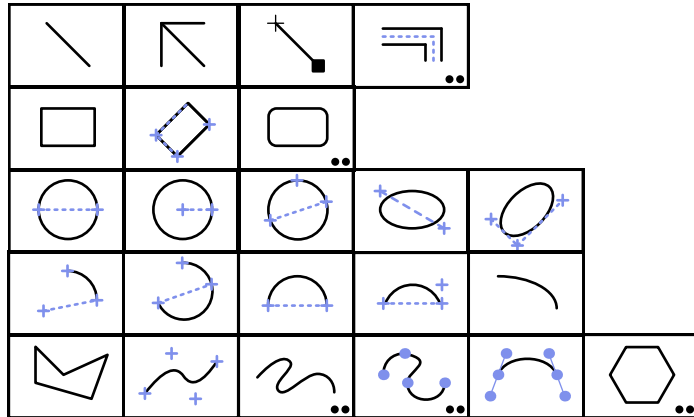
The date and/or time is inserted in the drawing.



CAUTION: You cannot change the font size or style of the smart Date Stamp objects. If you Ungroup a Date Stamp object it will be converted to a Placed object, and the object will no longer automatically update. To change the text attributes, set the attributes of the Date Stamp tool first, then insert a new Date Stamp object into the drawing.

Drawing Tools

Don't be deceived by the elegant and simple drawing tools section of the PowerCADD tool palette. These tools are powerful yet easy tools that enable you to create a complete range of basic to advanced geometry and designs quickly and easily. The following section describes each of the PowerCADD drawing tools.



Before describing each tool in detail, the following general information applies when working with all drawing tools.

- ☐ Dynamic Snapping may be used to modify the start or end points when drawing an object.
- ☐ Zoom on the fly is available. You can zoom in or out during the drawing operation.
- ☐ AutoScrolling is in effect. The screen will automatically scroll if you drag to draw an object which is larger than the currently displayed drawing area.
- ☐ Direct input via the Edit Window is available. Pressing the Tab key during a drawing operation will suspend the drawing operation and allow you to enter precise object dimensions in the Edit Window. Press the Return key when you are finished entering the information in the Edit Window and the object will be created.
- ☐ The the Shift key will toggle between constrained and unconstrained drawing or resizing operations. For example, to resize an existing circle, pressing the Shift key when dragging its handle will maintain the circle's aspect ratio. Pressing the Shift key when drawing an ellipse will constrain the shape into a circle.
- ☐ Any object may be edited using the mouse or the Edit Window after it is drawn.

Editing an Object Using the Mouse

To edit an object using the mouse after it was drawn:

- ☐ Select the object to be edited.
- ☐ Press the mouse button down on an object's edit handle.
- ☐ Drag the handle to reshape the object.
- ☐ Release the mouse button when the desired height and width have been achieved.
- ☐ By pressing the Shift key, the object can be reshaped while remaining in proportion to the original object.

Editing an Object Using the Edit Window

To edit an object using the Edit Window:

- ☐ Select the object to be edited.
- ☐ Activate the Edit Window by pressing the Tab key.
- ☐ Use the mouse or the Tab key to enter the desired edit field.
- ☐ Use standard text editing methods to change the edit field values.
- ☐ Use the Directional Arrow keys to jump to the start (left) or end (right) of content in an Edit Window field.
- ☐ Press the Return or Enter key to complete the change.

Draw by Center

While each tool typically draws beginning from an end point, you may also draw an object about its center. When using this method, begin at the center point.

To draw an object by its center:

- ☐ Select a drawing tool from the Tool palette.
- ☐ Move the cursor to the drawing area of the screen.
- ☐ Press the mouse and drag to start drawing the object.
- ☐ Tap the Space Bar.
- ☐ Drag away from the center point.

The object appears as you drag.

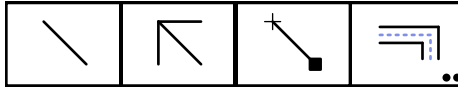
It can be dragged to any length and angle, with its center anchored at the point specified.

- ☐ Release the mouse to complete the object.



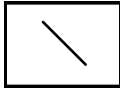
NOTE: Before releasing the mouse when drawing an object, tapping the Space Bar will toggle between drawing a line by its center or end point.

Line Drawing Tools



The third row of the Tool palette contains a suite of tools for creating line geometry.

Each tool may be used to create lines at any length or angle and with any type of attribute you choose (dashed lines, color, line weight, arrows, etc).

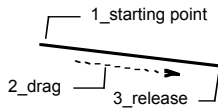


Line Tool

This tool is used to draw a line at any angle.

To use the Line tool:

- ☐ Select the Line tool from the Tool palette.
- ☐ Move the cursor to the drawing area of the screen.
- ☐ Press the mouse button where you want to start drawing (the starting point) and move the mouse in the desired direction.
- ☐ The line becomes longer the farther you move the cursor from the starting point.



Conversely, the line becomes shorter the closer the cursor is moved to the starting point.

As you move the cursor, notice the Dynamic Ruler Guides in the horizontal and vertical rulers and the values in the Edit Window which allow you to track your cursor movement.

- ☐ By moving the cursor about the starting point, you will be able to draw a line at any angle.
- ☐ Release the mouse button to complete the line.



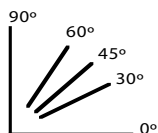
NOTE: By default the line is drawn at any arbitrary angle you choose. Pressing the Shift key when dragging will constrain the line to any of the preset snap angles. Snap angles may be set in the [Drawing Setup](#) dialog from the Layout Menu.



Multi-Angle Line Tool

The Multi-angle Line draws lines at the preset snap angles set in Drawing Setup of the Layout menu. The default angles are 0, 30, 45, 60, and 90 degrees and their complements.

By using this tool, the angle of lines are constrained or snapped as shown. You can also specify any angle in increments as small as one second of a degree. For information on changing the default or adding to the snap angles, see [Drawing Setup](#) in the Menu chapter.



To use the Multi-Angle Line tool:

- ☐ Select the Multi-Angle Line tool from the Tool palette.
- ☐ Move the cursor to the drawing area.
- ☐ Press the mouse button where you want to start drawing and move the mouse in the desired direction.
- ☐ The line becomes longer the farther you move the cursor from the starting point. Conversely, the line becomes shorter the closer the cursor is to the starting point.

If you try to draw a line with an angle of 35° or 85° from horizontal, PowerCADD will produce a line closest to the preset snap angle of 30° and 90°, respectively.

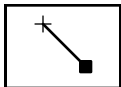
Lines with angles other than those listed above would also be similarly constrained.



NOTE: To draw lines at angles other than the preset snap angles press the Shift key to release the snap constraint.



NOTE: You can specify the preset snap angles. See [Drawing Setup, Snap, Constrain Snap Angles](#) for additional information.



Continuous Line Tool

The Continuous Line tool is used to draw lines between the last point clicked and the next mouse click.

Lines are drawn from the last mouse-up point to the current mouse-down location and they automatically connect at their common end points. Pressing and holding the mouse button will not only draw the line to the mouse-down location, but will allow the end of the line to be stretched or moved to an appropriate location.

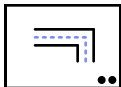
The next click will draw a new line connecting it to the last line or last mouse click.

To use the Continuous Line tool:

- ☐ Using the Selection Arrow tool, click the mouse in the drawing window to set the start point for the Continuous Line tool.
- ☐ Select the Continuous Line tool from the Tool palette and move the cursor to the drawing area.
- ☐ Click and drag the mouse button; a new line will start from the previous mouse click and end when you release the mouse.
- ☐ The line becomes longer the farther you move the cursor from the starting point. Conversely, the line becomes shorter the closer the cursor is to the starting point.



CAUTION: After selecting the Selection Arrow, if you click in the drawing and the next operation is the Continuous Line tool, a line will be drawn starting from where you clicked in the drawing.



Parallel Line Tool

The Parallel Line tool may be used to draw parallel lines or a parallel polygon.

Although the Parallel Line tool was designed for drawing walls in house plans, it can be used to draw many other shapes, which require parallel lines or require the finished shape be a closed polygon.

The closed polygon appears to have a hole in its center.

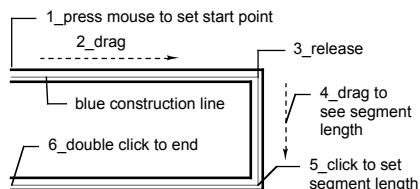
The polygon can be filled, reshaped or changed just as any other polygon.



NOTE: Selecting the Parallel Line tool can automatically switch you to the Wall Layer (or another layer name of your choosing) if the [Layer Tools](#) feature is checked on in the Layout Menu. If the Layer Tools feature is not checked on, objects are drawn on the active layer.

To use the Parallel Line tool:

- ☐ Select the Parallel Line tool from the Tool palette.
- ☐ Move the cursor to the drawing area.
- ☐ Press the mouse button where you want to start drawing and move the mouse in the desired direction. You will notice three lines being drawn. The outermost two solid lines will remain when the drawing is done. The blue centerline is a visual aid to assist you in constructing your parallel lines. The centerline will disappear when you finish constructing the object.
- ☐ The lines become longer the farther you move the cursor from the starting point. Conversely, the lines become shorter the closer the cursor is moved to the starting point.
- ☐ Click where you want to place each of the intersection points or corner.
- ☐ Change the direction of the mouse. Notice the angle is constrained to either 90° or 45°. If you press the Shift key while moving the mouse, you can drag the line to any desired angle.
- ☐ Click at the next desired point and continue drawing.
- ☐ Double-click to stop the drawing process.



When using the Parallel Line tool, you will see two solid lines that represent the offsets and a ghost line in the center, which represents the centerline.

Tap the Space Bar to shift the offset to the left, right or center positions while drawing. This shift is not permanent and will revert back to the original position after the current object is created.

To end a drawing operation with the Parallel Line tool, double-click the mouse where you want to end.

Draw by center is not available for the Parallel Line tool.

Pressing the Space Bar will cycle between left, center and right offsets.



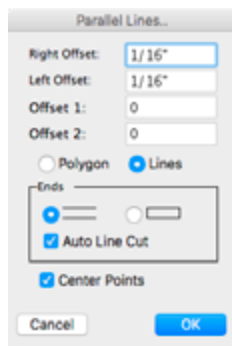
NOTE: You can also specify a precise length and angle for each segment as you draw using direct input in the Edit Window.

To specify a length and angle as you draw:

- ☐ Start to draw using the Parallel Line tool.
- ☐ Press the Tab key and release the mouse.
- ☐ Enter the specific length or angle in the appropriate field of the Edit Window and press the Enter or Return key.
- ☐ If you entered a specific angle, drag the mouse in the drawing to define the length and click the mouse button. If you entered a specific line length, drag your mouse to define the angle and click the mouse button.

To change the right and left offset (the space between the parallel lines):

- ☐ Press the Option key and select the Parallel Line tool from the Tool palette. The following dialog will appear:

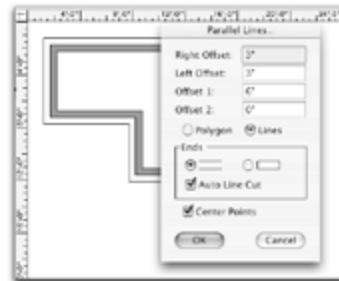


- ☐ The Parallel line tool requires at least one value be entered in the Right or Left Offset fields. The value is to scale based on values you entered for Drawing Setup in the Layout Menu. Enter your offset values then press the Return key or click OK to continue. To set an offset, enter half of the total offset required. For example, if the wall is to be 6" wide, the right offset should be 3" and the left offset should be 3".
- ☐ To draw parallel lines, rather than a polygon, select Lines rather than Polygon.

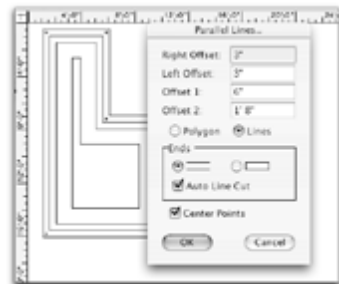
If you normally dimension by the center of the wall, click Center Points and a non-printing point will be placed when drawing parallel lines.



There is an option for additional offsets to the Parallel Line tool. If you would like additional offsets, your object must be a closed shape and the Lines option must be set. The following example shows only one offset.



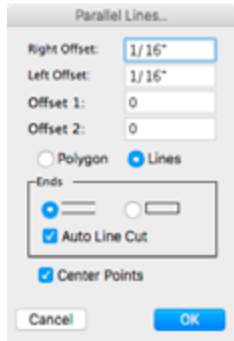
The following example shows both offsets being used.



Both offsets are measured as the distance from the outside line. If you want a 3-inch offset from the outside line for Offset 1 and Offset 2, you would use 3 inches for Offset 1 and 6 inches for Offset 2.

To set an offset, enter half of the total offset required. For example, if the wall is to be 6" wide, the right offset should be 3" and the left offset should be 3". If you set the right offset to 6" and the left offset to 0", you could create the walls of a building utilizing the outer dimensions by drawing counterclockwise.

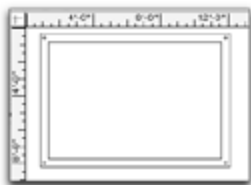
Additional optional settings for the Parallel Line tool are:



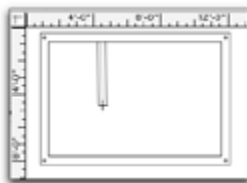
Polygon creates the parallel lines as a polygon that can be moved and manipulated.

Lines create the parallel lines as a set of lines placed in the drawing. All line segments remain as individual objects unless they are grouped or connected using Attach from the Tool menu.

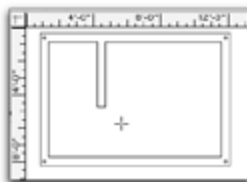
Auto Line Cut allows you to place parallel lines and cut other lines at their intersections. For example, you have constructed the outer walls of a floor plan using parallel lines. You now need to create interior walls that connect to the outer walls with cutting and intersection cleanup. Place the cursor on the point of the exterior wall where you wish to begin the interior wall. Press and drag the parallel line to construct the wall and click twice to end the object. The interior wall has joined itself to the exterior wall without intersecting lines in the way.



Construct a solid wall floor plan.

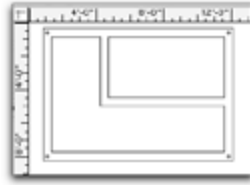


To place a wall, construct a parallel line from a point on the exterior wall.



The added interior wall will be placed with the intersecting lines removed.

To create a wall between two walls cutting both intersections, place the cursor on the exterior wall and drag a parallel line to the interior wall and release.



The opening will be inserted with the intersecting lines cleaned up and the short piece removed.

Poché

If a fill pattern or color is assigned to the Parallel Line tool, polygons matching the pattern or color will be assigned to the cavity between the two original wall lines. This method of poché does not require the parallel lines to be a polygon.



To assign attributes or to poché with the Parallel Line tool:

- ☐ Select the Parallel Line tool from the Tool palette.
- ☐ Make sure no object in the drawing is selected.
- ☐ If you would like a pattern in the wall cavity, choose Fill from the Attribute menu and determine the pattern of choice from the pop-out menu.

If you would prefer a grayscale or color instead of a pattern,

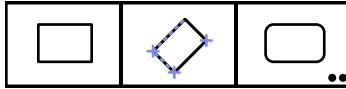
- ☐ Choose Fill from the Attribute menu and choose black from the pop-out menu.
- ☐ Choose Fill Color from the Attribute menu and determine the color of choice from the pop-out menu.

Once you have set the attribute for the Parallel Line tool, it will remain the default until changed.

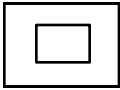


NOTE: The Paint Bucket tool may also be used to create a polygon that may be filled and used to poché the walls. The polygon created using the Paint Bucket tool may be on its own layer or on the same layer as the walls.

Rectangle Tools



PowerCADD provides three rectangle tools: Constrained Rectangle, Rotated Rectangle and Rounded Rectangle. All tools create rectangles, which can be subsequently edited with the mouse or the Edit Window.

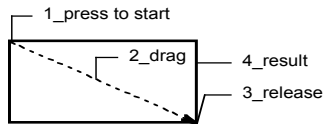


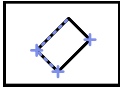
Constrained Rectangle Tool

This tool is used to draw rectangles constrained to the horizontal and vertical axes.

To create a Constrained Rectangle:

- ☐ Select the Constrained Rectangle tool from the Tool palette.
- ☐ Move the cursor onto the drawing where you wish to begin the rectangle.
- ☐ Press the mouse button to establish the starting point.
- ☐ Continue to hold the mouse button and drag away from the starting point.
- ☐ Release the mouse button to complete the rectangle when it is the desired size and shape.



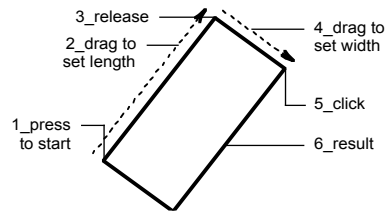


Rotated Rectangle Tool

This tool is used to draw rotated rectangles at any angle without constraint to the horizontal or vertical axes.

To create a Rotated Rectangle:

- ☐ Select the Rotated Rectangle tool from the Tool palette.
- ☐ Move the cursor onto the drawing area where you wish to begin the rectangle.
- ☐ Press to establish the starting point.
- ☐ Continue to hold the mouse button as you drag away from the starting point.
- ☐ Release the mouse button when the desired axis angle and direction has been defined.
- ☐ Move the mouse in any direction to define the desired characteristics of the rectangle.
- ☐ Click to establish the rectangle.



Pressing the Shift key while creating the rotated rectangle will allow you to constrain the axis of the rectangle to the preset snap angles.

Once the axis is established, it can be edited by using the Rotate command or the Edit Window. The Edit Window can be used to establish the axis angle during creation of the rectangle, or it can be used to edit an existing rectangle.

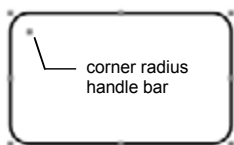


Round Rectangle Tool

This tool is used to draw squares and rectangles with rounded corners.

The use of the Round Rectangle tool is similar to the Rectangle tool:

- ☐ Select the Round Rectangle tool from the Tool palette.
- ☐ Move the cursor to the drawing area where you want to place your rectangle.
- ☐ Press the mouse button to set the starting point.
- ☐ Move the cursor diagonally away from the starting point. As you move the cursor away from the starting point the object becomes larger. If you move the cursor back toward the starting point, the rectangle becomes smaller.
- ☐ Release the mouse button when the rectangle or square is the correct size.

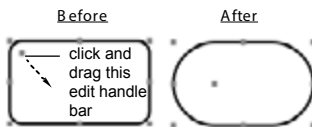


Notice the extra edit handle that appears in the upper left corner when the round rectangle is selected.

This corner radius handle is used to change the radius of the corners. Selecting and moving the corner radius handle can easily alter the shape of the curved corners.

To change the radius of the rectangular object's corners:

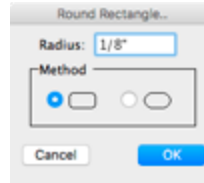
- ☐ Select the Selection Arrow tool.
- ☐ Move the Selection Arrow to the drawing area and select the rounded rectangle (or square) you wish to modify. Its handles should now be visible.
- ☐ Select the corner radius handle and hold the mouse button down.
- ☐ While holding the mouse button down, move the corner radius handle. As you move the handle, note the change in the corner radius.
- ☐ Release the mouse button when you see the desired radius.



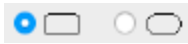
NOTE: The corner radius handle can be dragged to a zero radius producing a “round” rectangle that has square corners.

The default radius of the corner curvatures may be set as follows:

- ❑ Press the Option key and select the Round Rectangle tool from the Tool palette. The following dialog will appear:



The dialog allows you to specify the type of round rectangle that will be created.



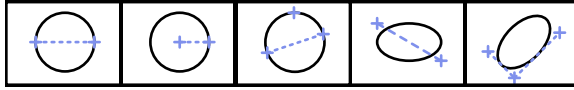
It offers two forms of initial creation: fillet rectangle or round end rectangle. Both options create the object with a corner edit handle to change the corner radius after creation.

The dialog box also allows you to specify the radius of the filleted corners.

To specify a radius, simply select the current radius value and replace it with the new value. This default setting will remain in effect until you change it.

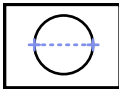
- ❑ Select the object type desired. After you have made your selection, press the Return key or click OK.

Circle and Ellipse Tools



PowerCADD provides a suite of three circle and two ellipse tools for creating a wide range of simple and complex designs.

All tools create circles or ellipses, which can be subsequently edited with the mouse or the [Edit Window](#) as described at the start of the Drawing Tools section.

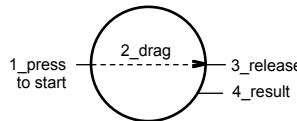


Diameter Circle Tool

This tool allows you to construct circles based upon a given diameter.

To create a circle based on its diameter:

- ☐ To see the dimensions of the circle while drawing, open the Edit Window by choosing Edit from the Window menu.
- ☐ Select the Diameter Circle tool from the Tool palette.
- ☐ Place the cursor where you wish to start the circle, press the mouse button. The point you specify will represent a point on the circumference of the circle.
- ☐ Drag the mouse away from the starting point and watch the Edit Window until you reach the value desired. Release the mouse button. As you drag away from the starting point, a temporary blue construction line will appear to aid you in constructing the circle.



Once established, the circle can be changed using standard PowerCADD methods.

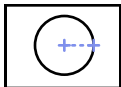
The Diameter Circle tool is useful when you need to create a circle, which passes through two points or is a specific diameter.

To create a circle which passes through two points:

- ☐ To observe the dimension of the circle while drawing, open the Edit Window.
- ☐ Select the Diameter Circle tool from the Tool palette.
- ☐ Place the cursor on the first of the two points and press the mouse button.
- ☐ While watching the Edit Window, drag the mouse until the desired diameter or the second point has been reached, release the mouse button.

- While establishing the diameter, notice as you drag the cursor away from the starting point you may also rotate the circle about the starting point. While dragging, move the cursor about the starting point to view the rotation. This will allow you to place the circle so its perimeter passes through two points and still retains its diameter.

Once established, the circle can be changed using standard PowerCADD methods.

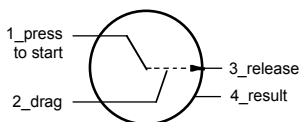


Radius Circle Tool

This tool allows you to create circles based upon a given radius.

To create a circle based on a given radius

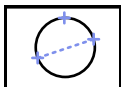
- ❑ To see the dimensions of the circle while drawing, open the Edit Window by choosing Edit from the Window menu.
- ❑ Select the Radius Circle tool from the Tool palette.
- ❑ Place the cursor where you wish to locate the center of the circle and press the mouse button.
- ❑ Drag away from the starting point and watch the Edit Window until you reached the desired radius.
- ❑ Release the mouse button. As you drag the circle out, a temporary blue construction line will appear from the center of the circle to the circumference to aid you in the construction of the circle.



Once established, the circle can be edited using standard PowerCADD methods.

This tool is also helpful when a circle is needed with its center at a specific point and its circumference touching another point.

Start the circle at the specified point, drag the cursor to the second point, and release the mouse button.

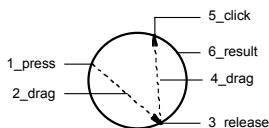


Three-Point Circle Tool

This tool allows you to create a circle that must pass through three points.

To create a circle based on three points:

- ☐ Select the Three-Point Circle tool from the Tool palette.
- ☐ Place the cursor on the first of the three points and press the mouse button.
- ☐ Drag to the second point and release. An expandable circle will appear which passes through the two specified points and the cursor.
- ☐ Click on the third point.

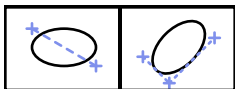


Once established the circle can be changed using standard PowerCADD methods.



Points placed so they form a straight line, or close to a straight line, will create enormous circles that will extend a great distance off the drawing page.

Ellipse Tools



PowerCADD provides two ellipse tools: a Constrained Ellipse tool and a Rotated Ellipse tool. Both tools are used to create ellipses, which can be subsequently edited with the mouse or the Edit Window.

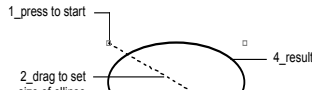


Constrained Ellipse Tool

This tool is used to draw ellipses constrained to the horizontal and vertical axes.

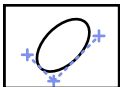
To create a Constrained Ellipse:

- ☐ Select the Constrained Ellipse tool from the Tool palette.
- ☐ Move the cursor onto the drawing where you wish to begin the ellipse.
- ☐ Press the mouse button to establish the starting point.
- ☐ Continue to hold the mouse button as you drag away from the starting point.
- ☐ When the ellipse is the desired shape, release the mouse button.



Pressing the Shift key after you begin drawing will constrain the horizontal and vertical proportions of the ellipse.

For example the ellipse tool can be used to create a circle, where the horizontal and vertical proportions of the ellipse are equal.

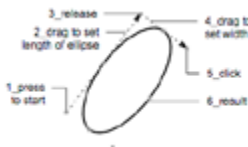


Rotated Ellipse Tool

This tool is used to draw ellipses at any angle without constraint to the horizontal or vertical axes.

To create a rotated ellipse:

- ☐ Select the Rotated Ellipse tool from the Tool palette.
- ☐ Move the cursor onto the drawing where you wish to begin the ellipse.
- ☐ Press to establish the starting point.
- ☐ Continue to hold the mouse button as you drag away from the starting point.
- ☐ Release the mouse button when the desired axis angle and direction has been defined.
- ☐ Move the mouse in any direction to define the desired characteristics of the ellipse.
- ☐ Click to complete the ellipse.

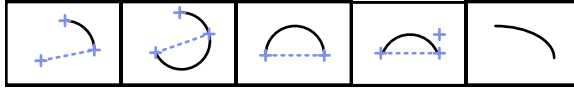


Pressing the Shift key while creating the rotated ellipse will allow you to constrain the axis of the ellipse to the preset snap angles.

Once the axis is established, it can be edited by using the Rotate command or the Edit Window.

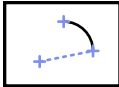
The Edit Window can be used to establish the axis angle during creation of the ellipse, or it can be used to edit an existing ellipse as described earlier in this section.

Arc Tools



PowerCADD provides a suite of four circular arc tools and one elliptical arc tool for creating a wide range of simple and complex designs.

All tools create arcs, which can be subsequently edited with the mouse or the Edit window as described at the start of the Drawing Tools section.



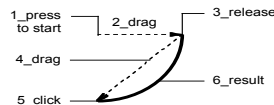
Radius Arc Tool

This tool allows you to draw an arc based on a given radius.

A radial arc is a portion of a circle which can represent any part of a circle from 0° 0' 1" to 360° 0' 0".

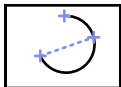
To create a radial arc:

- ☐ To see the dimensions of the arc while drawing, open the Edit Window by choosing Edit from the Window menu.
- ☐ Select the Radius Arc tool from the Tool palette.
- ☐ Place the cursor in a location representing the center of the radial arc. Press the mouse button. Since a radial arc is a portion of a circle, it is created in much the same way as a circle. The center of a radial arc would be the center of a circle if the radial arc were extended 360°.
- ☐ Drag away from the starting point and watch the Edit Window until you reach the desired radius and release the mouse button.
- ☐ As you drag the cursor, a temporary radial line will appear to aid in the construction of the arc.
- ☐ Move the cursor in the direction you wish for the arc to extend. When the arc has reached the desired angular degree, click to establish the arc.



The radial arc can be edited and filled like any object in PowerCADD.

Press the Shift key to constrain the radius arc to the preset snap angles in the Drawing Setup dialog.



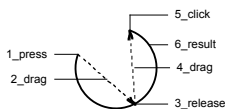
Three-Point Arc Tool

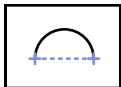
This tool allows you to create an arc that passes through three points.

To create a circular arc based on three points:

- ☐ Select the Three Point Arc tool from the Tool palette.
- ☐ Place the cursor on the first of the three points and press the mouse button.
- ☐ Drag the cursor to the second point and release. An expandable arc will appear that passes through the two specified points and the cursor.
- ☐ Click on the third point.

The arc is established and can be changed using standard PowerCADD editing methods.





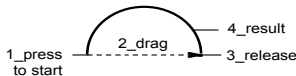
Arc by Diameter Tool

This tool allows you to draw a circular arc by specifying a diameter.

Arc by Diameter is a two-point arc tool which produces a 180° circular arc.

To draw an arc:

- ❑ Place the cursor at the desired location for the first point of the diameter line.
- ❑ Press and drag. A ghost line will appear indicating the length and angle of the diameter line. Release to complete the arc.



When creating an arc, you can change its horizontal or vertical orientation by tapping the Space Bar while dragging the diameter ghost line.

To constrain the diameter line to the preset snap angles, press the Shift key while drawing the diameter.



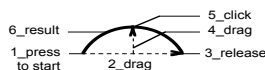
Arc by Chord Tool

This tool allows you to draw a circular arc by first specifying a chord line and then a height.

Arc by Chord is a three-point arc tool which produces circular arcs given only a chord length and height.

To draw an arc:

- ☐ Select the Arc by Chord tool.
- ☐ Place the cursor at the desired location for the first point of the chord line.
- ☐ Press and drag. A ghost line will appear indicating the length and angle of the chord line. Release to end the chord line.
- ☐ Move the mouse to specify the height of the arc and click to complete the arc.



To constrain the chord line to the preset snap angles, press the Shift key while drawing the chord.



NOTE: Arcs that have little curvature may place their center point off the drawing page.



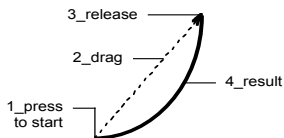
Elliptical Arc Tool

This tool allows you to create an elliptical arc of any size.

Elliptical arcs are quarter sections of ellipses or circles.

To draw an elliptical arc:

- ❑ Select the Elliptical Arc tool from the Tool palette.
- ❑ Place the cursor where you wish to begin the arc and press the mouse button.
- ❑ Drag away from the starting point until you have created an arc of appropriate size and release the mouse button.



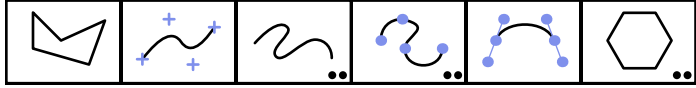
The arc has been established and can be edited using standard PowerCADD methods.

The elliptical arc can be filled and edited just like any other object.

When creating an elliptical arc, you can change its horizontal or vertical orientation by tapping the Space Bar while dragging the arc.

Pressing the Shift key while creating an elliptical arc will constrain the arc to the preset snap angles. The constraint will apply to a line that passes through the start and finish points of the bounding rectangle.

Polygon and Bézier Tools



Before describing each Polygon and Bézier tool in detail, the following general information applies when working with all these tools.

- ☐ Dynamic Snapping may be used to modify the start or end points of any polygon or Bézier both during object creation and when editing.
- ☐ AutoScrolling is in effect. The screen will automatically scroll if you drag to draw an object which is larger than the currently displayed drawing area.
- ☐ Direct input via the Edit Window is only accepted for the Polygon and Regular Polygon tools.
- ☐ Polygons and Béziers may be edited after they're created using the mouse by selecting the object and picking Reshape from the Edit Menu.



See sections in the manual regarding the [Tool Menu](#), [Polygon Menu](#) and [Edit Menu, Reshape](#) for additional features and information on working with Polygons and Béziers.



Polygon Tool

This tool allows you to create open or closed polygon shapes with two or more sides.

Polygons are objects with at least two sides and are normally drawn closed like hexagons and triangles.

To create an open or closed polygon:

- ☐ Select the Polygon tool from the Tool palette.
- ☐ Place the cursor where you wish to start the polygon and press the mouse button.
- ☐ Move the cursor to the end of the first side. You will see the first side of the polygon being formed as you move away from the starting point. Release the mouse button to establish the first side of the polygon.
- ☐ Move the cursor to the end of the second side of the polygon. Again you will see the line segment being formed as you move the mouse. Click the mouse button to establish the second side of the polygon.
- ☐ Continue this pattern to add as many sides as you wish.
- ☐ To create a closed polygon continue the process of adding sides until you have returned to the starting point. Upon reaching the starting point, place the cursor directly over the first point created and click. The polygon will become an editable object and the Polygon Drawing tool will remain selected.



- ☐ To create an open polygon, add as many sides as necessary and click twice on the end point of the last side entered. The line segments will become an editable object and the Polygon Drawing tool will remain selected.



Closed and open polygons can be reshaped and filled with patterns.

Press the Shift key to constrain the side being drawn to the preset snap angles specified in Drawing Setup.

Press the Option key to prevent closure and to allow placing duplicate control points on top of each other.

This is helpful if two points in the polygon are very close together.

If you get near an existing point, a snap function may attach the points causing the polygon to close and become an object. To prevent this, press the Option key and the polygon will not close.

Press the Enter key to end the current polygon and leave the Polygon tool selected.

Reshaping a Polygon

To reshape a polygon using the mouse:

- ☐ Select the polygon to be edited.
- ☐ Choose Reshape from the Edit menu.
- ☐ Each vertex of the polygon will display an edit handle.
- ☐ Use the mouse to drag each handle to the desired location.
- ☐ Press the Option key and select an edit handle to delete it. The cursor will change to include a (–) sign when the mouse is over the edit handle.
- ☐ Press the Option key and click anywhere on the polygon to add a new point. The cursor will include a (+) sign when the mouse is on the polygon.
- ☐ Press the Enter key or click in the drawing window to complete the edit operation.



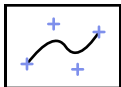
NOTE: You can also use the Edit Window to create open or closed polygon shapes quickly accurately.

The ability to directly enter data in the Edit Window while creating an object can have significant advantages. A single polygon can be filled to create color renderings, or used to perform area and perimeter calculation for land use or space planning analysis among many other benefits. For example, you might have received a photocopy reduction of a surveyor's land parcel drawing, and you'd like to quickly draw it based on the layout information provided.

To draw a polygon using direct data entry in the Edit Window:

- ☐ Select the Polygon Drawing tool from the Tool palette.
- ☐ Place the cursor where you wish to start the polygon and press the mouse button.
- ☐ Start to drag the mouse to draw the first segment of the polygon.
- ☐ Without releasing the mouse button, press the Tab key then release the mouse.
- ☐ The Edit Window will be activated. Press the Tab key to navigate in the Edit Window to the Length Field.
- ☐ Type a value in the Length Field and press the Return key.
- ☐ Drag the mouse in the drawing window to define the angle. Press the Shift key to constrain the movement to the preset snap angles.

- ☐ Click the mouse to complete the first segment.
- ☐ Drag the mouse to begin the next segment.
- ☐ Press the Tab key then release the mouse.
- ☐ Press the Tab key again to navigate in the Edit Window to the Angle Field.
- ☐ Type a value in the Angle Field and press the Return key.
- ☐ Drag the mouse in the drawing window to define the length.
- ☐ Click the mouse and second segment is completed.
- ☐ Repeat these steps for each segment and press the Esc key when all segments have been drawn.



Smooth Polygon Tool

Smooth Polygons, also known as B-Spline curves, are a type of mathematical curve similar to a French curve used in manual drafting.

A Smooth Polygon, or B-Spline, generates continuous curves based on the straight polygon segments you draw.

To create an open or closed Smooth Polygon:

- ☐ Select the Smooth Polygon tool from the Tool palette.
- ☐ Place the cursor where you wish to start the polygon and click.
- ☐ Move the cursor to the end of the first side. You will see the first side of the smooth polygon being formed as you move away from the starting point. Release the mouse button. The first side of the smooth polygon has been established.
- ☐ Move the cursor to the end of the second side of the smooth polygon and click. Again you will see the line segment being formed as you move the mouse. The second side of the smooth polygon has been established.
- ☐ Continue this pattern to add as many sides as you wish.
- ☐ To create a closed polygon continue the process of adding sides until you have returned to the starting point. Upon reaching the starting point, place the cursor directly over the first point created and click. The smooth polygon will become an editable object and the Smooth Polygon tool will remain selected.



NOTE: A smooth polygon may also be created by drawing a regular polygon and choosing Smooth from the Polygon submenu in the Tool menu.

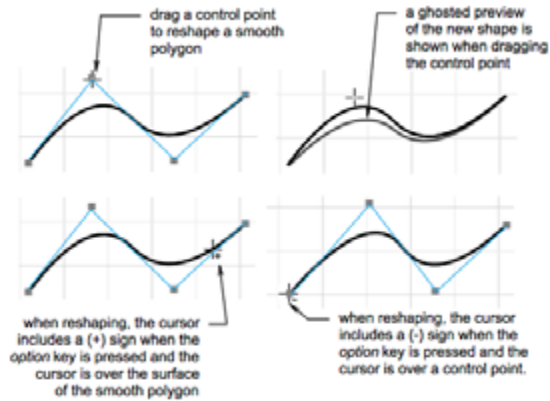


NOTE: To convert a smooth polygon into a regular, straight-sided polygon, choose UnSmooth from the Polygon submenu in the Tool menu.

Reshaping a Smooth Polygon

To reshape a smooth polygon using the mouse:

- ☐ Select the smooth polygon to be edited.
- ☐ Choose Reshape from the Edit menu.
- ☐ Each control point for the smooth polygon will display a handle.
- ☐ Drag each handle to the desired location.
- ☐ Option-click on an edit handle to delete it. The cursor will change to include a (-) sign when the mouse is over the edit handle.
- ☐ Option-click anywhere on the smooth polygon to add a new control point. The cursor will include a (+) sign when the mouse is on the smooth polygon.
- ☐ Press the Esc or Enter key to complete the edit operation.





Freehand Tool

The Freehand tool allows you to create unstructured freehand lines and shapes.

To use the Freehand tool:

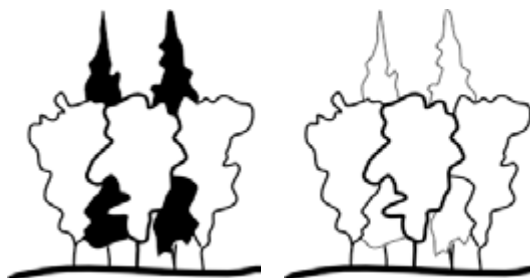
- ☐ Select the Freehand tool from the Tool palette.
- ☐ Place the cursor where you wish to begin the freehand line and press the mouse button.
- ☐ Move the cursor in any direction and the unstructured line will appear. This tool is much like drawing with a pencil or pen. You are allowed to draw a freehand line in any direction without constraint.
- ☐ Release the mouse button when the desired object has been created to end the freehand line.

The Freehand Drawing tool will remain selected to allow additional drawing without reselecting the tool.

The Freehand tool uses intelligent thinning of points or vertices during the drawing process to produce polygons matching the mouse movement. The resulting objects can be filled, moved, smoothed, or reshaped as needed to add or delete unneeded points.

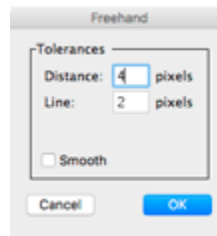


NOTE: The Freehand tool (or any PowerCADD tool) can also be used with a graphics tablet and stylus instead of a mouse. The Freehand and Freehand Bézier tools in particular benefit from the more intuitive feel of sketching with a stylus verses dragging with a mouse.





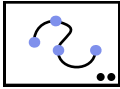
- ❑ Press the option key and select the Freehand tool. The following dialog will appear:



Distance controls the number of screen pixels the mouse must move before a point is added to the polygon. Typically, you will use a value of 4 to 8 pixels. Smaller numbers produce more finely detailed polygons; larger numbers produce coarser polygons.

Line controls a filter that's used after the polygon is drawn to throw away points that are roughly in-line. If you enter a value of 2 for Line, then for any three points, A, B and C, the point B is thrown away if it falls within 2 screen pixels of the line AC. It is best to leave this value at 1 or 2, for most purposes.

Smooth converts the polygon to a B-spline. This is the same as choosing the Smooth command.



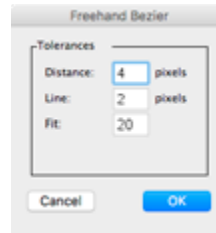
Freehand Bézier Tool

The Freehand Bézier tool creates a Bézier curve from the freehand shape that is drawn.

Like the other Freehand tool, the Freehand Bézier points are intelligently thinned during the drawing process. The resulting Bézier curve may be reshaped by using the Bézier editing methods discussed below.

To set the behavior of the tool:

- ☐ Press the Option key and select the Freehand Bézier tool. The following dialog will appear:



Distance controls the number of screen pixels the mouse must move before a point is added to the polygon. Typically, you will use a value of 4 to 8 pixels. Smaller numbers produce more finely detailed polygons; larger numbers produce coarser polygons.

Line controls a filter that's used after the polygon is drawn to throw away points that are roughly in-line. If you enter a value of 2 for Line, then for any three points, A, B and C, the point B is thrown away if it falls within 2 screen pixels of the line AC. It is best to leave this value at 1 or 2, for most purposes.

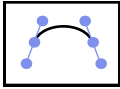
Fit controls the behavior of the Bézier curve fitting that's done after you draw. Smaller numbers result in more Bézier segments; larger numbers result in fewer Bézier segments. The range of allowable numbers is 1 to 400, however it is best to use values around 20. Extremely high or low numbers sometimes produce strange results.

Reshaping a Smooth Polygon

To reshape a polygon using the mouse:

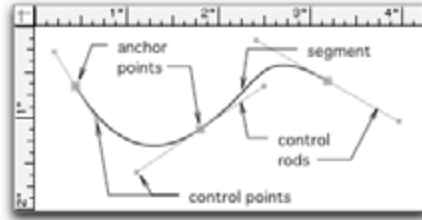
- ☐ Select the Freehand Bézier to be edited.
- ☐ Choose Reshape from the Edit Menu.
- ☐ Adjust the control rods to reflect the new shape.
- ☐ Press the Esc key to complete the edit operation.

You can also add, subtract or modify the control rods of a Freehand Bézier using a combination of key commands the mouse. See the [Bézier Tool](#) later in this section for additional information.



Bézier Tool

Bézier curves are smooth mathematical curves that pass through specified adjustable points. Béziers are the electronic equivalent of the French curve.



Bézier curve composed of two Bézier segments

A Bézier segment consists of three basic elements: Anchor points, control points, and controls rods.

Anchor points represent the end points of a segment.

A **control point** is a point associated with an anchor point that affects the direction and shape of the curve when dragged. Increasing the distance between a control point and its anchor point increases the pull on the curve; conversely, decreasing the distance decreases the pull on the curve. If the distance is zero, a sharp corner is produced in the curve.

A **control rod** represents the distance between a control point and its anchor point. It is important to note the curve through an anchor point is always tangent to the control rod connected to the anchor point.

When the Bézier Curve tool is in use, you will see a pen-like cursor. There are seven variations of this cursor depending on what type of operation you are performing.


Drawing Cursor

The Drawing cursor is the first cursor that will appear when you begin constructing a Bézier curve. The plus (+) indicates the creation or addition of anchor points.


Reshape Cursor

The Reshape cursor will appear when the Reshape command is applied to a selected Bézier curve.


Delete Cursor

The Delete cursor will appear when the Option key is pressed during drawing or reshaping. The minus (–) indicate the removal of anchor points and control points. Pressing the Option key and clicking on the center of an anchor point deletes the anchor point. Pressing the Option key and clicking on the end of a control rod point deletes the control rod segment.



Add Cursor

The Add cursor will appear when Option key is pressed during drawing and reshaping and clicking anywhere on a curve. The plus (++) indicates the addition of anchor points.



Close Curve Cursor

The Close curve cursor will appear when Option-Control keys are pressed during drawing and clicking on the first point in the Bézier. The Bézier will automatically close.



Move Segment Cursor

The Move Segment cursor will appear when the Command key is pressed during drawing or reshaping and clicking anywhere on the surface of the curve. Drag the cursor to move the segment in or out to change the shape of the curve.



Open Curve Cursor

The Knife cursor will appear when Option-Control keys are pressed and you select an existing anchor point of a closed Bézier. The Bézier curve is broken at the anchor point and may be dragged apart.



Break/Extend Cursor

The Break/Extend cursor will appear when the Command key is pressed during drawing or reshaping and you select a control point or anchor point. The break and extend operations allow you to lengthen, shorten, or break the control rods to create discontinuous curves or right angle bends. You can also expand a previously collapsed or broken control rod by pressing the Command key and clicking on the node with the collapsed controlled rod.

Drawing Béziers

To draw a Bézier:

- ☐ Select the Bézier tool from the Tool Palette.
- ☐ Move the cursor to the location you wish to start the curve. Press and drag. An anchor point and its control points will appear. When the control rod is the desired length and angle, release the mouse button.
- ☐ Move the cursor into position for the end point of the first Bézier segment and press. Drag to locate the control points and rods as described above. To create a straight segment, click to place an anchor but do not drag out the control points.
- ☐ If another anchor point is needed, follow the directions for locating another anchor point. You may add as many Bézier segments as you wish using this procedure. If you want to create a closed path, drag one end anchor point of a segment and place it on top of the curve's other end anchor point.
- ☐ To end the Bézier drawing process, press the Esc or Enter keys or double-click.

You may undo the last curve segment while drawing or reshaping by pressing Command-Z.

Multiple curve segments can be removed one at a time by pressing the Delete key for each segment.

To undo the current drawing or reshaping session, press the Enter key, then press Command-Z.

To edit an existing Bézier curve, select a curve and choose Reshape from the Edit menu. Anchor points, control points and rods will reappear for you to make adjustments.

Adjusting Béziers

Bézier curves have nearly-limitless flexibility. Once a curve is created, it can be adjusted to produce almost any shape including long flowing paths, intricate scroll-like patterns, and straight lines. All of these shapes are made by variations of basic adjustments. Adjustments to curve segments can be made any time during the Bézier drawing process by moving anchor points, control points, or by dragging the curve itself.

To move an anchor point while drawing or when using the Reshape command:

- ☐ Place the cursor over the desired anchor point, press and drag to a new position. The curve will change as the anchor point is being moved, but the control points and rods will remain at their set length and angle.
- ☐ When the anchor point is in the desired location, release the mouse.

To move a control point while drawing or when using the Reshape command:

- ☐ Place the cursor over the desired control point. Press and drag to a new position. You may change the length of each rod independently. Lengthening rods pulls on the curve in the direction of the rod length change; shortening the rod relaxes the pull of the curve. Rotating the rod will change its angle. If the anchor point associated with the control point is connecting two Bézier segments, both segments will change.
- ☐ When the control point is in the desired location, release the mouse.

To drag a curve segment while drawing or when using the Reshape command:

- ☐ Place the cursor over the desired curve segment. Press and drag to the desired location, then release.



NOTE: While drawing and reshaping Béziers, snapping is automatically turned off. This enables you to make very minor changes to curves, without interference. To activate a snap, press the corresponding snap key on the keyboard.

Advanced Bézier Curves

The Bézier Curve tool has many advanced capabilities which are keystroke and mouse combinations; that is, you must press a key while clicking or dragging with the mouse. The following information illustrates these combinations according to the end result on the curve.

Adding and Deleting Anchor Points

Anchor points can be added to the end of existing curves, which adds additional Bézier segments. You may also add anchors onto a curve itself, between its existing endpoints.

To add an anchor point to the end of an open Bézier curve:

- ☐ Select a curve and choose Reshape from the Edit menu.
- ☐ The Bézier will automatically be extended from its end point.
- ☐ Move the cursor to the desired location, click to add an additional control point and drag to shape the curve.

To add an anchor point to the start of an open Bézier curve:

- ☐ Select the Bézier tool.
- ☐ Click the end point of the Bézier.
- ☐ The Bézier will automatically be extended from the point you clicked.

To add an anchor point which creates a straight segment or sharp corner:

- ☐ Select a curve and choose Reshape from the Edit menu.
- ☐ Move the cursor to the desired location and click. A new anchor point is added with no control rods.

To add an anchor point onto an existing curve segment:

- ☐ Select a curve and choose Reshape from the Edit menu.
- ☐ Move the cursor on top of the segment, press the Option key and click.

To delete an anchor point from a curve:

- ☐ Select a curve and choose Reshape from the Edit menu.
- ☐ Move the cursor over the anchor point to be deleted, press the Option key and click.

Adding and Deleting Control Points

You may add or delete control points and rods to existing anchor points while drawing or reshaping. Once added, they can be dragged and shaped as needed.

To add a control point:

- ☐ Move the cursor over the desired anchor point, press the Command key and drag.

To delete a control point:

- ☐ Move the cursor over the control point to be deleted, press the Option key and click.

Creating Discontinuous Curves

A discontinuous curve is a curve whose path makes an abrupt change in direction—a sharp corner. The sharp corner is created by breaking a control rod into two independently-editable control rods. When the control rod is broken, the two control rods appear hinged at the anchor point. You may create discontinuous curves while drawing or reshaping.

To create a discontinuous curve:

- ☐ Move the cursor over a control point. While pressing the Command key, press and drag. The control rod will break at the anchor point, allowing you to rotate the rod.

To change a discontinuous curve back to a continuous curve:

- ☐ Move the cursor over the control point of a discontinuous curve. Press the Command key and click the control point.

Pulling on the Curve

You may reshape a curve segment by pulling directly on the curve while drawing or reshaping. This gives you lots of flexibility by allowing you to stretch and even shear curves

You may:

- ☐ Press and drag the curve to adjust the length of only the inside control rods, that is, the rods closest to the segment that is moved. The curve will remain continuous.
- ☐ Press the Command key while pressing and dragging the curve. This will break and adjust the control rods associated with the selected curve and produce a discontinuous curve.
- ☐ Press the Control key while pressing and dragging the curve. This will adjust the selected curve segment and its adjacent curves, and will produce a discontinuous curve.

Closing a Curve

You may close a curve while drawing or editing:

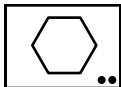
- ☐ Press the Option-Control keys and drag over the first point of the curve.
- ☐ The cursor will change to a close cursor.
- ☐ Click to close the curve.

Breaking a Curve

You may break a closed curve while drawing:

- ☐ Press the Option-Control keys and select an anchor point of a closed curve.
- ☐ The curve will be broken at that anchor point.
- ☐ Drag the anchor points apart to open the curve.

To break a closed or open curve after it is drawn, use the Break tool.



Regular Polygon Tool

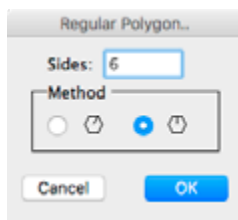
The Regular Polygon tool is used to draw polygons with three or more sides of equal length and angles.

With the Regular Polygon tool, polygons with equal sides and equal angles can be constructed with great ease. Regular polygons can be filled with patterns and reshaped.

The Regular Polygon tool allows you to specify the number of sides the polygon will have and how the polygon will be created. There are two methods of creating polygons. You may create the polygon by drawing from the shape's center to a vertex (outside radius) or you may draw from the shape's center to the midpoint of one surface (inside radius). Once a method is set, it remains in effect until you change it.

To set the behavior of the tool:

- ❑ Press the Option key and select the Regular Polygon tool. The following dialog will appear:

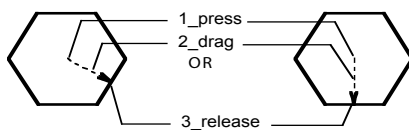


- ❑ Type the number of sides desired (3 or more) and set the method of construction. Press the Return key or click OK.

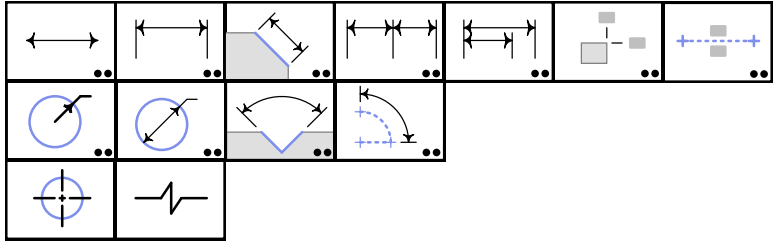
To draw a regular polygon:

- ❑ Move the cursor to the drawing area where you wish to construct the polygon.
- ❑ Press the mouse button to set the starting point, then drag the cursor from the starting point in any direction. You will notice the polygon's shape is instantly modified to show you what it will look like. The line representing the polygon's radius line is a visual aid only and will disappear when the polygon is complete.
- ❑ Press the Shift key while drawing to constrain the construction line to the preset snap angles.
- ❑ When you see the desired polygon, release the mouse button.

The regular polygon will appear.



Dimensioning Tools



The dimensioning tools of the PowerCADD Tool palette consist of a suite of versatile, easy-to-use dimensioning tools. These tools enable you to dimension lines, angles, circles, and add centerlines and long breaks. The following section describes each of the PowerCADD drawing tools in detail.



NOTE: Selecting a Dimension tool can automatically switch you to the Dimension Layer (or another layer name of your choosing) if the [Layer Tools](#) feature is checked in the Layout menu. If the Layer Tools feature is not checked, dimensions are drawn on the active layer.

Before describing each Dimensioning tool in detail, it is important to be aware of the following fundamental concepts, which apply to all dimension tools:

- ☐ Dynamic Snapping may be used to modify the start or end points of any dimension both during object creation, when moving objects or positioning dimension lines.
- ☐ AutoScrolling is in effect. The screen will automatically scroll if you drag to draw an object which is larger than the currently displayed drawing area.
- ☐ By default, the units of measure applied by a dimension tool are as specified in the Drawing Setup dialog. You can override these preferences using optional settings described later in this section. See [Modifying Dimensions](#) later in this section.
- ☐ A drawing may be dimensioned in single or mixed units. For Example, you can draw in inches but dimension in millimeters or both inches and millimeters.
- ☐ The [Dimensions Floating Window](#) may be used to change attributes of any dimension in your drawing. See the Window Menu section of this manual for additional information.
- ☐ Changing the size of a dimension line will automatically update the associated dimension text.
- ☐ Setting the text attributes for one dimension tool sets all dimension tools to match.
- ☐ The size and font of the dimension text may be modified with commands from the [Text menu](#), [Text Attributes Window](#) or from a

contextual menu.

- ❑ The dimension text can be moved on the drawing without disturbing the dimension line.
- ❑ To change the dimension default settings, press the Option key and select any dimension tool. For more details about changing the Dimension tool defaults, or setting accuracy and tolerance, see [Setting Dimension Defaults](#), later in this chapter.
- ❑ A dimension line and its accompanying measurement or leader lines may be modified after it is drawn. For more information about modifying dimensions after they have been drawn, see [Edit](#) in the Menus chapter and “Modify Dimension” later in this chapter.



NOTE: Dimension text supports font styles such as **bold**, *italic*, and underline. In the MacOS, not all font families include a **bold**, *italic* or other special font styles.

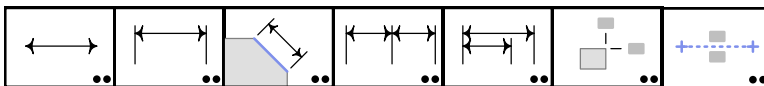


NOTE: Use the Move Points tool to change the length of your dimensions and objects at the same time.

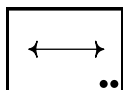


See sections in the manual regarding the [Attribute \(Attr\) Menu, Arrows](#) and [Layout Menu, Drawing Setup, Dimensions Tab](#) for additional information on setting dimension tool preferences for your drawing.

Linear Dimensioning Tools



Each of PowerCADD's linear dimensioning tools is optimized for various standard drawing and dimensioning techniques traditionally used in manual drafting. These tools allow you to efficiently and accurately create linear dimensions describing object geometry or the spaces and relationships between objects to effectively communicate your designs.



Dimension Line Tool

The Dimension Line tool is used to draw lines at any angle and automatically includes the length dimension of the constructed lines.

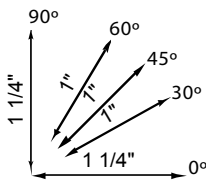
Nearly all technical drawings require objects be dimensioned in a unit of measurement such as feet, inches, meters or centimeters. In addition, the dimension lines usually have arrows on each end of the line delimiting the distance being measured. A dimension line indicates the direction and length of a dimension. Drawing with the Dimension Line tool measures the distance you have drawn and puts the dimension in the center of the dimension line. This tool is normally set to draw lines ending with dimension indicators. The dimension line end indicators can be arrows, circles, or slashes.

Drawing the dimension lines manually is a time-consuming and tedious task. With the Dimension Line tools, this process is greatly simplified.

Like the Line tool, the Dimension Line tool is useful to draw dimension lines at any angle. In addition, the Dimension Line tool creates a line, which includes its length in the currently selected unit of measurement

To use the Dimension Line tool:

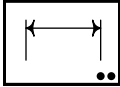
- ☐ Select the Dimension Line tool from the Tool palette.
- ☐ Move the cursor to the drawing area of the screen.
- ☐ Press the mouse where you want to start drawing (the starting point) and move the mouse in the desired direction.
- ☐ By rotating the line about the starting point, you will be able to draw a line having various preset angles from the horizontal. Normally the angles will be constrained to 0, 30, 45, 60, and 90 degrees (like the Multi-angle Line tool). By pressing the Shift key, after drawing has started, you may draw lines at any angle you desire.
- ☐ Release the mouse button upon completing the line. The line and its corresponding dimensions will be drawn.



Notice the lines shown in the illustration include arrows at each end of the line segments and the length of each line in inches.



NOTE: Changing the length of the dimension line causes the dimension text to reflect the new dimension.



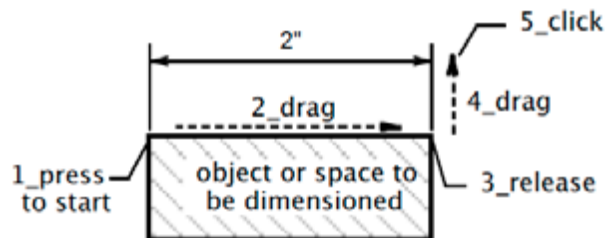
Witness Line Dimension Tool

The Witness Line Dimension tool is used to draw dimension lines with one or two extension lines.

An extension line “extends” from a point on the drawing to which a dimension line refers (see the illustration below). The Witness Line Dimension tool is constrained to the preset snap angles specified in the Drawing Setup: Snap tab under the Layout menu. Pressing the Shift key will override the preset snap angles, allowing you to draw a Witness Line Dimension at any angle.

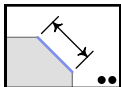
To use the Witness Line Dimension Line tool:

- ☐ Select the Witness Line Dimension tool from the Tool palette.
- ☐ Move the cursor to the point on the object you wish to dimension.
- ☐ Press the mouse at the point where you wish to establish the dimension starting point. Move the mouse to the end of the dimension. The line you see being drawn is a dimension line. When you have created the desired line, release the mouse button.
- ☐ Move the cursor away from the line. As you begin to move away, you will see perpendicular lines appear from the sides of the dimension line. These lines are extension lines. By moving the cursor you can set the angle of the dimension extension lines to any of the snap angles. By pressing the Shift key you can constrain the dimension parallel to the first line.
- ☐ Click at the approximate location and angle of the dimension text. You will now see the entire dimension line with corresponding extension lines.



The Witness Line Dimension tool can also be used to dimension the space between objects and they may be at any angle. Some examples of various dimensioning conditions include:



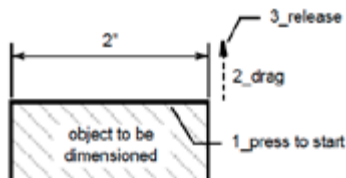


Face Dimension Tool

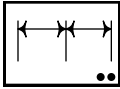
The Face Dimension tool is used to dimension a flat surface at any angle by selecting the surface and dragging the dimension. The Face Dimension tool can be used repeatedly without reselecting the tool.

To use the Face Dimension tool:

- ☐ Select the Face Dimension tool from the Tool palette.
- ☐ Place the cursor on the surface to be dimensioned, press the mouse button.
- ☐ Drag in the direction you wish to locate the dimension. As you drag, a dimension line and two leaders will appear. This parallel line can be offset to different angles depending on the dimension you need to specify. Dimension lines will snap to the angles specified in Drawing Setup.
- ☐ When the cursor is in the appropriate location, release the mouse button and the dimension will be established.



Pressing the Shift key when drawing the dimension will release the dimension from the preset snap angles and snap it parallel to the surface being dimensioned.

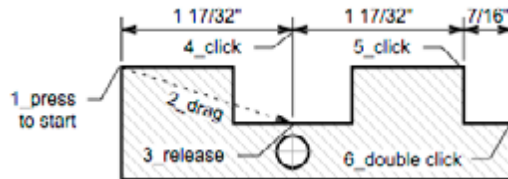


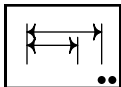
Chain Dimension Tool

The Chain Dimension tool is used to dimension a drawing with a series of dimensions that fall in a straight line.

To use the Chain Dimension tool:

- ☐ Select the Chain Dimension tool from the Tool palette.
- ☐ Place the cursor where you want to start the first leader line, press the mouse button.
- ☐ Drag in the direction to be dimensioned. This line should span the same distance as the intended dimension. Release the mouse button.
- ☐ Move away from the line you just created. An expanding dimension line will appear. The dimension can be set to any angle and at any distance away from the object surface.
- ☐ Place the cursor where you want to place the dimension line and click. The first dimension is now established. All subsequent dimensions will be located in a straight line with the first dimension.
- ☐ To establish the next dimension; click at the next dimension point. You may place as many string dimensions as you wish without re-establishing the initial dimension.
- ☐ To stop a dimension string, click twice at the last dimension point or press the Esc or Enter key. This will stop the dimension string and leave the tool selected. You may then begin another dimension string.
- ☐ To deselect the Chain Dimension tool, click in a blank space in the drawing or select another tool.



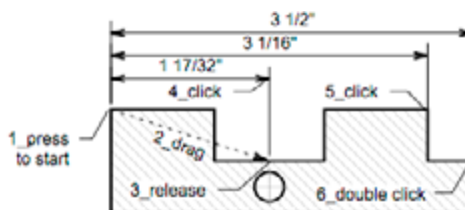


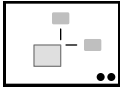
Stack Dimension Tool

The Stack Dimension tool is used to dimension a drawing with a stack of evenly spaced parallel dimensions originating from a common dimension line.

To use the Stack Dimension tool:

- ❑ Select the Stack Dimension tool from the Tool palette.
- ❑ Place the cursor where you want to start the first dimension and press the mouse button.
- ❑ Drag in the direction of the dimension point. This line should span the same distance as the dimension line. Release the mouse button.
- ❑ Move away from the line you just created. An expanding dimension line will appear. The line, parallel to the line you created, represents the dimension line. The perpendicular lines represent the extension lines. These lines can be set to any angle and at any distance away from the object surface.
- ❑ Move to where you want to place the dimension line and click. The first dimension is now established. All subsequent dimensions will be located so a parallel stack will be formed by the stack dimension lines.
- ❑ To locate the next dimension; select the next dimension point. You may place as many stack dimensions as you wish without reselecting the tool.
- ❑ To stop a stack dimension, click twice at the end of the last dimension or press the Esc or Enter key. This will stop the stack dimension and leave the tool selected. You are then free to begin another stack dimension.
- ❑ If you want to deselect the tool, you may click in a blank space in the drawing or select another tool.



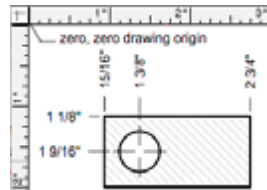


Base Line Dimension Tool

This tool allows you to place a dimension on a point that tells you how far that point is away from the Drawing Origin. The dimensions are given in absolute values, representing how far the point is away from the current Drawing Origin.

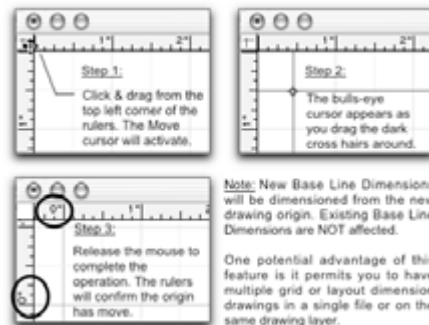
To use the Base Line Dimension tool:

- ☐ Select the Base Line Dimension tool from the Tool palette.
- ☐ Place the cursor at the point you wish to dimension, press the mouse button.
- ☐ Drag the cursor vertically for an X-axis dimension and horizontally for a Y-axis dimension. This tool will dimension one axis at a time. If you want both X and Y coordinates dimensioned, place a dimension line both vertically and horizontally.
- ☐ At a suitable distance away from the dimension point release the mouse button. The point-to-point drag will represent the leader line. The dimension text will be placed at the end of the line.



To resize the leader, or if you change the scale, select the handle of the leader line by the text to resize. If the drawing scale has been changed, this stretching operation will recalculate the dimension to match the units setting.

Dimensions are values as measured from the 0,0 drawing origin. To change the drawing origin, 0,0 point:

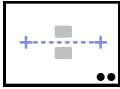


You may use the snap options with this tool. For example, to dimension the center point of a rectangle, hold the C key while drawing the dimension line to snap it to the center of the rectangle.

To move the dimension text, simply select it and move it just as you would any object.



NOTE: The Drawing Origin can be repositioned and even snapped to an existing object in the drawing. Changing the Drawing Origin will not modify existing Baseline Dimensions in the drawing; all new Baseline Dimensions will reflect the changed drawing origin.



Survey Dimension Tool

This tool allows you to add bearing and distance dimensions to your drawings.

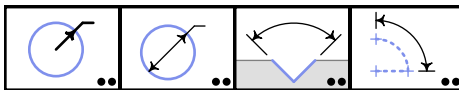
The tool can be used to dimension polygons, lines, and other objects. The dimension text is inserted and rotated to match the angle and bearing of the object and the dimension text is inserted above the line

To use the Survey Dimension tool:

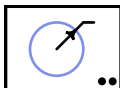
- ☐ Select the Survey Dimension tool from the Tool palette.
- ☐ Place the cursor at the point to start the dimension and press the mouse button.
- ☐ Drag in the direction of the bearing. A ghost line connects the starting point with the cursor.
- ☐ Release at the end of the dimension. The distance and bearing appear.
- ☐ To change the method in which bearings and distances are displayed (such as azimuth or bearing), change the unit and angle defaults using Drawing Setup: Units in the Layout menu.

The information inserted for the dimension length and bearing are normal PowerCADD text objects. As such, the values for length and bearing do not automatically recalculate after an object is reshaped, resized or rescaled.

Radial & Angular Dimensioning Tools



Each of PowerCADD's radial and angular dimensioning tools is optimized for various standard drawing and dimensioning techniques traditionally used in manual drafting. These tools allow you to efficiently and accurately create radial and angular dimensions describing object geometry or the spaces and relationships between objects to effectively communicate your designs.

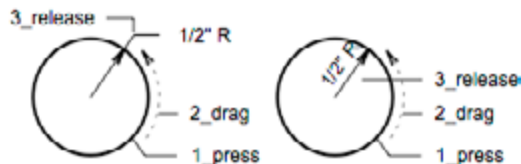


Radial Dimension Tool

The Radial Dimension tool allows you to dimension the radius of a circle or arc. The tool may be used repeatedly without having to reselect it.

To use the Radial Dimension tool:

- ☐ Select the Radial Dimension tool from the Tool palette.
- ☐ Place the cursor on the circle or arc you wish to dimension, press the mouse button.
- ☐ The radial dimension will appear. The placement of the dimension text is specified by the location of the cursor when the mouse button is released. If the cursor is inside the circle, the text will be placed inside the circle; if the cursor is outside the circle, the text will be placed outside.
- ☐ When the cursor is in an appropriate location, release the mouse button, and the dimension is established.

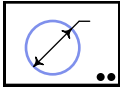


To edit a radial dimension:

- ☐ To change the location of the dimension text, select it and drag the text to a new location. If you change the location of the radial line, the text will snap back to its original position.
- ☐ To reset the angle or distance from the object, select the handle located at the bend of the radial dimension leader and drag. The angle and distance may be changed while the horizontal line and the text remain locked at horizontal.

To change the dimension default settings, press the Option key and select the dimension tool. You may set the defaults of the circular dimension tools independently.

To move the dimension text, simply select it and move it just as you would any object.

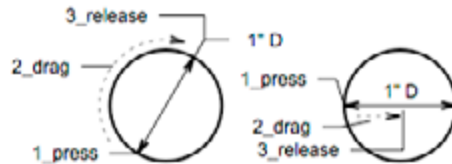


Diametrical Dimension Tool

This tool is used to dimension the diameters of circles and arcs.

To use the Diametrical Dimension tool:

- ❑ Select the Diametrical Dimension tool from the Tool palette.
- ❑ Place the cursor on a circle or arc, press the mouse button.
- ❑ A diameter dimension line will appear. The placement of the dimension text is specified by the location of the cursor when the mouse button is released. If the cursor is inside the circle, the text will be placed inside the circle; if the cursor is outside the circle, the text will be placed outside.
- ❑ A diameter dimension line will appear. The placement of the dimension text is specified by the location of the cursor when the mouse button is released. If the cursor is inside the circle, the text will be placed inside the circle; if the cursor is outside the circle, the text will be placed outside.
- ❑ When the mouse button is released, the dimension line is placed and the dimension text will appear.

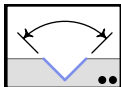


To change the dimension default settings, press the Option key and select the dimension tool. You may set the defaults of the circular dimension tools independently.

To edit the Diametrical Dimension:

- ❑ To reset the angle of the dimension line or the distance from the object, select the handle located at the bend of the leader and drag. The angle and distance may be changed while the horizontal line and text remain locked at horizontal.
- ❑ When resizing or relocating the dimension lines which are inside the circle or arc, the selection handle will be located on one of the arrowheads. If the dimension text is in the way, move the text momentarily.

To move the dimension text, simply select it and move it just as you would any object.

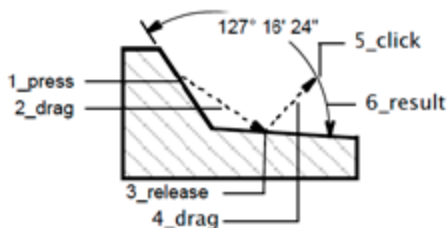


Two-Line Angular Dimension Tool

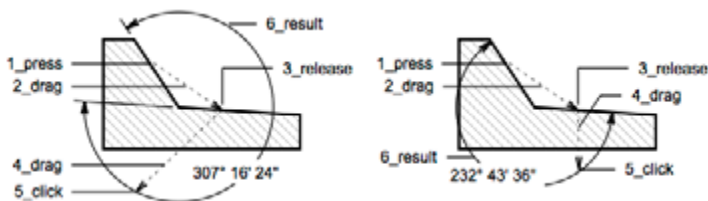
This tool is used to create a dimension arc between two lines which intersect or will intersect if extended. PowerCADD will insert leader lines which extend from the original lines.

To use the Two-Line Angular Dimension tool:

- ❑ Select the Two-Line Angular Dimension tool from the Tool palette.
- ❑ Place the cursor on the first of two lines that require a dimension, press the mouse button.
- ❑ Drag to the second line. A bridge line will appear indicating the tool is active. Release the mouse button.
- ❑ The dimension will appear either inside or outside the angle depending on the cursor location. Place the cursor where you want the dimension to be located and click.



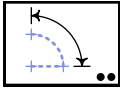
If you drag the cursor outside of the angle, the dimension will flip to the outside or complement angle.



The dimension text can be moved around the drawing without disturbing the dimension line. Simply select it and move it as you would any object. The text remains associated with the dimension.

The Two-Line Angular Dimension tool can be used repeatedly without reselecting the tool or objects. If you deselect the tool by accident, the Return or Enter key will reactivate the last tool selected.

The angular dimension can be edited. If you select the dimension and select one of the handles, the dimension can be resized by dragging. Dragging the arrow tip handles repositions the dimension and text without changing the dimension angle. Dragging the leader handle will change the dimension angle.

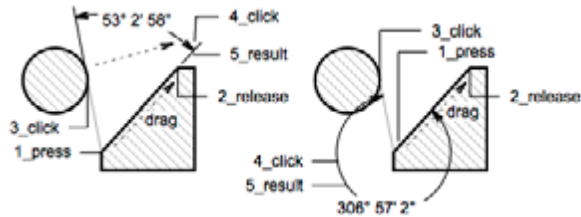


Four-Point Angular Dimension Tool

This tool is used to create an angular dimension anywhere in the drawing.

To use the Four-Point Angular Dimension tool:

- ☐ Select the Four-Point Angular Dimension tool from the Tool palette.
- ☐ Place the cursor on the apex (1) of the angle to be dimensioned, press the mouse button. You do not have to define a specific point to be the apex. You may place it anywhere you wish.
- ☐ Drag from the apex to define the first side of the angle (2) and release the mouse button.
- ☐ As you move from the end of the first line you will see a ghost line that represents the second line of the angle. Drag the end of this line to a point that will define the angle (3). Click to establish the angle (4).
- ☐ The dimension has been defined and the final click will define the location of the dimension arc (4). Drag the cursor inside the defined lines to get the included angle, drag the cursor outside of the defined lines to get the complement of that angle. Place the cursor where you want the dimension to be located and click.



- ☐ To move the dimension arc, select the arrow tip handle and drag it into location. The dimension may be changed by dragging the leader tip handle. When the dimension is stretched, it will be recalculated. To move dimension text select it and move it into location.

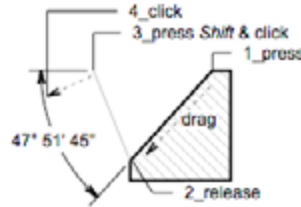
The Four-Point Angular Dimension tool also allows you to dimension angles from a constrained snap angle. For example if you had a triangle which you wanted to know its angles with respect to a preset snap angle, you could use the Four-Point Angular Dimension tool to insert the dimension with the correct extension lines.

To dimension from a constrained angle:

- ☐ Select the Four-Point Angular Dimension tool from the Tool palette.
- ☐ Place the cursor on the apex of the angle to be dimensioned, press the mouse button (1).
- ☐ Drag from the apex to define the first side of the angle and release the mouse button. The line you just specified should be a side of an object (2).
- ☐ Press the Shift key to constrain the ghost line to the preset snap angles specified in Preferences. As you move the cursor to the third point, the

ghost line will snap to the preset angles. In this case the dimension is snapped to horizontal (3).

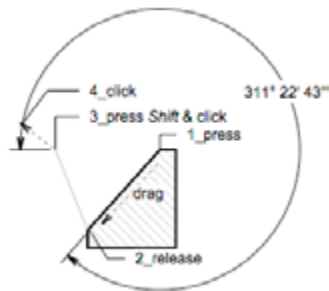
- ❑ While pressing the Shift key, select a point that falls on the second side of the angle. The dimension arc will appear upon placement of the third point.
- ❑ Place the cursor where you want the dimension arc to be located and click (4).
- ❑ Upon placement of the fourth and final point, the dimension text will appear and an extension line from the apex of the angle to the dimension arc will appear along the constrained line (4).



Constraining a line tells PowerCADD you wish to place an extension line from the apex to the dimension arc. Constraining the line also specifies which side of the angle is to receive the extension.

The Four-Point Angular Dimension tool can be used repeatedly without having to select objects. If you drag the cursor outside of the angle, the dimension will flip to the outside or complement angle.

If the movement from point 3 to 4 is away from the origin, the constrained line will extend back to the apex of the angle.



The dimension text can be moved around the drawing without disturbing the dimension line. Simply select it and move it as you would any object.

The Four-Point Angular Dimension can be edited. If you select the dimension and select one of the handles, the dimension can be resized by dragging.

Setting Dimension Defaults

Option-clicking on Dimension tools presents a dialog box for setting the defaults of all the dimension tools.

You may specify where the text will appear, where the arrows will appear, unit measurement system, angular measurement system, unit measurement accuracy, angular measurement accuracy, additional text present in the dimension, tolerance parameters, and if the text will rotate with the dimension line.



See [Drawing Setup, Dimensions](#) for additional information on setting dimension preferences for your drawing.

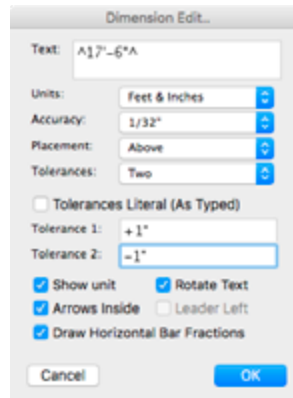
All Linear, Radial, and Angular Dimension tools operate with the same dialog settings. If the dialog defaults are changed for one linear dimension tool, all linear dimension tools take on the setting.

To change the default settings for all linear dimension tools:

- ❑ Double-click any Linear Dimension tool in the Tool palette.

Double-clicking ensures no other objects are selected.

- ❑ Press the Option key and click on any linear dimension tool in the Tool palette. The following dialog will appear.



The Text box, at the top of the dialog, allows you to add text prefix or suffix to a dimension such as radial and diametrical symbols.

As a default, `^DIM^` is present in the box. This is actually a symbol telling PowerCADD the actual measurement numbers are to be placed in the dimension. If this is removed, no measurement will be displayed. If you inadvertently remove the symbol, simply type *Shift-6*, *DIM*, *Shift-6*. To add text to the dimension, enter the text using standard PowerCADD methods either before or after the `^DIM^` information in the Text Box.

Units: Feet & Inches

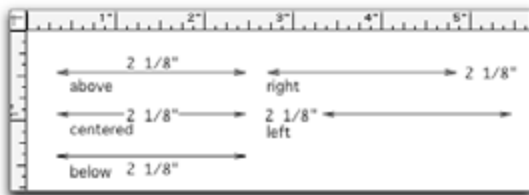
Units allows you to set the measurement system you wish to use, such as metric or imperial. The dimension's units can be set independently of the unit setting in Drawing Setup.

Accuracy: 1/32"

Accuracy allows you to set the unit accuracy of the dimension, for example, 1/2" or 1/256". The dimensions unit accuracy can be set independently of the unit accuracy setting in Drawing Setup.

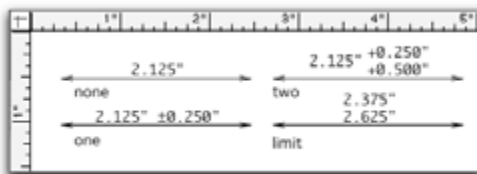
Placement: Above

Placement allows you to specify where the dimension text will appear in relation to the dimension line. You can position the text below the line, above the line, on the line, and so on.



Tolerances: Two

Tolerances allows you to specify if the dimension will have a tolerance, the type of tolerance and the number of tolerance values. You have the option of using one or two tolerances (which display as a dimension and an upper and lower tolerance), a Limit (which displays as two dimension stacked on top of each other denoting an upper and lower limit), or no tolerance at all. Tolerances of one and two allow you to enter the desired values.



☐ Tolerances Literal (As Typed)

Tolerances Literal allows you to specify the tolerances entered to display exactly as typed, or if PowerCADD will remove all insignificant figures.

For example, if you entered .25000, you could select the check box and that value would be entered. If the check box is deselected, the displayed value will be .250. Think of it as specifying "Tolerances as noted" except you are entering the values in the Dimension Edit dialog instead of manually for each dimension in the drawing.

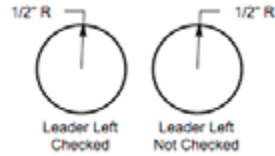
☒ Rotate Text

Rotate Text allows you to specify if the dimension text will rotate with the dimension line, or if the dimension text will remain horizontal as the dimension line is set to a different angle.



☒ Leader Left

Leader Left allows you to specify if the leader line and dimension text appears to the left or right of the dimension arrow when using Radial and Diametrical dimension tools. This box is dimmed for all Linear and Angular dimension tools.

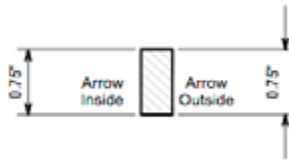


☒ Show unit

Show Unit will add the measurement unit to the end of the dimension text, such as *km* or *mm*.

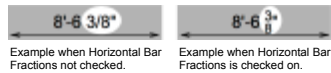
☒ Arrows Inside

Arrows Inside allows you to specify if the dimension arrows will appear inside the dimension area or outside the dimension area.



☒ Draw Horizontal Bar Fractions

Draw Horizontal Bar Fractions allows you to specify how fractional dimensions are displayed.



☐ Make the desired changes and click OK.

All Linear Dimensions will use the specified settings.

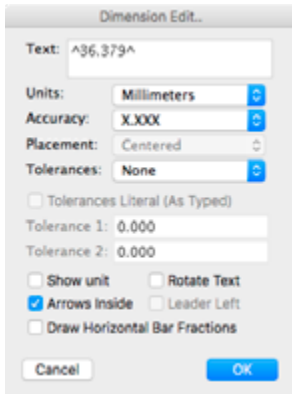
Circular Dimension tools can be set independent of each other. If one is changed, the other is not affected.

The Angular Dimension tools operate with the same dialog settings. Changing one Angular Dimension tool changes both.

Setting Dimension Defaults

In addition to editing the typical object attributes (e.g. line weight, arrows, color, etc.) you can also modify various properties after a dimension has been placed in the drawing.

Changing the drawing scale or layer scale does not change the dimension until the dimension line is stretched or scaled by using Scale from the Edit menu or by opening and closing a dimensions Edit dialog box. This feature allows you have objects drawn at different scales appear on the same layer or in the same drawing just as you could do in manual drafting. When you shorten or lengthen a dimension line, PowerCADD recalculates the dimension according to the active layer's scale and current units. If you stretch or modify dimension lines on a detail make sure the desired scale has been selected first.



Dimension Edit Dialog

You can use the Dimension Edit dialog box to modify any dimension in the drawing. If you select multiple dimensions, the changes made in the Dimension Edit dialog are applied to all selected dimensions.

To edit a dimension (or all selected dimensions) using the Dimension Edit dialog:

- ☐ Select the dimension objects to edit.
- ☐ Pick Edit from the Edit Menu and the Dimension Edit dialog will appear.
- ☐ Adjust the desired settings and click OK.

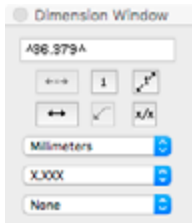
See the following diagram and information discussed in “Setting Dimension Defaults” for a complete description of each setting.

The selected dimensions will be automatically updated to reflect the new changes.

If the drawing scale or layer scale were changed before editing the dimensions, the dimensions will be automatically recalculated.



NOTE: You can dimension in units which are different than those you drew in. This can be handy for preparing a dual dimensioned drawing in both imperial and metric units. After you have dimensioned your drawing once, select all the dimensions, paste them onto a new layer and use the Dimension Edit window to change the units (for example from inches to millimeters) and dimension text placement (for example from above to below the dimension line).



Dimension Window

You can use the [Dimension Window](#) to modify any dimension in the drawing. If you select multiple dimensions, the changes made in the Dimension Window are applied to all selected dimensions.

The Dimension Window is a unique PowerCADD External and must be loaded in order to take advantage of this feature. Reference the [Loading External](#) section of this manual for additional information on how to load the Dimension Window External.

Modify Dimension

To edit a dimension (or all selected dimensions) using the Dimension Window:

- ☐ Select the dimension objects you'd like to edit.
- ☐ Select the Dimension Window in the Anchor Window.
- ☐ Adjust the desired settings.

The selected dimensions will be automatically updated to reflect the new changes.

If the drawing scale or layer scale were changed before editing the dimensions, the dimensions will be automatically recalculated.

The Dimension Window provides access to the same optional settings as the Dimension Edit dialog.

See [Setting Dimension Defaults](#) previously in this section for a complete description of each setting.

Using the Mouse

Dimension text can be repositioned without disturbing the dimension line or its relationship to the text.

To do this, place the cursor near the top edge of the text, press and drag the text to a new location.

To constrain the dimension text parallel to the dimension line, press the Shift key while dragging the text.

The size and font of dimension text may be modified using the commands under the Text menu.

To prevent the accidental selection of the dimension line itself, especially on a dimension whose text is *on* the dimension line rather than above or below, press the Command key while selecting dimension text for moving. Pressing the Command key allows you to temporarily deactivate snapping.

Lengthening an existing dimension line automatically causes the dimension text to change reflecting the new line length. However, moving a line drawn with the Dimension Line tool does not change the dimension text or the length of the line.

Pressing the Shift key while moving the dimension line set will constrain movement to the horizontal and vertical axis only.

Pressing the Shift key will constrain the dimension parallel to the line that you drew to specify the dimension points.

Pressing the Shift key while moving the dimension set will constrain movement to the horizontal and vertical axes only.

Pressing the Shift key while moving dimension text will constrain the text movement along the length of the dimension line.

The Shift key constraint is very useful when you must dimension off of a construction line that is not exactly vertical or horizontal.

Dimensions previously created can be edited.

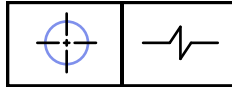
If you select the dimension and select one of the handles, the dimension can be moved within the leaders by dragging.

To resize the dimension, select the handle located on the end of the leader line and drag to extend or shorten the dimension.

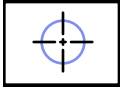
When the dimension is released, the measurement will be recalculated and displayed on the line.

Dimensions may also be edited using the Edit Window.

Supplementary Dimensioning Tools



PowerCADD includes supplementary dimensioning tools which quickly create industry standard drawing symbols for center and break lines.



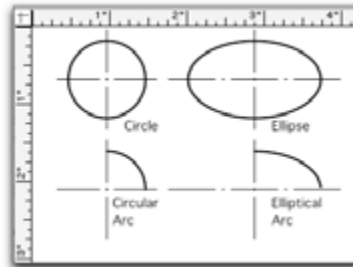
Centerline Tool

This tool allows you to insert a centerline symbol in any circle, ellipse, arc, rectangle, or rounded rectangle. These objects may be in a group or distinct objects in a drawing.

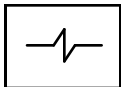
To use the Centerline tool:

- ☐ Select the Centerline tool from the Tool palette.
- ☐ Place the cursor on the object you wish to identify with a centerline symbol.
- ☐ Click and the centerline will be placed in the drawing.

The Centerline tool will remain selected until you click in a blank space or select another tool. If you need several centerlines of many different sizes, you select each object which requires a centerline. The Centerline tool takes care of creating the correct size centerline symbol for each object.



NOTE: To place a centerline on the corner of a rounded rectangle, use the Centerline tool found in the External Mechanical Tools palette.

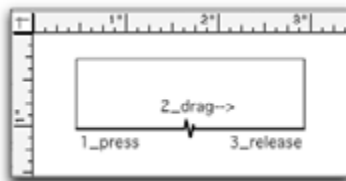


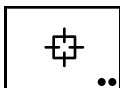
Long Break Tool

This tool draws a long break using any attributes you specify.

To use the Long Break tool:

- ☐ Select the Long Break tool from the Tool palette.
- ☐ Place the cursor on one end of the object you would like to illustrate as being broken.
- ☐ Drag to the other end of the break and release. Use the Shift key to constrain drawing the long break line to the preset snap angles.



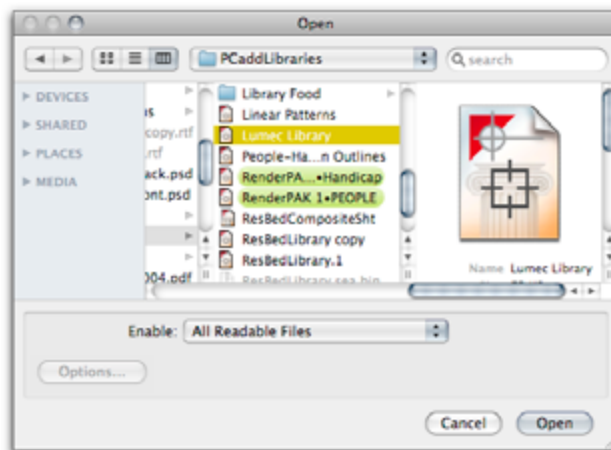


Symbol Tool

This tool allows you to place library symbols in any part of the drawing.

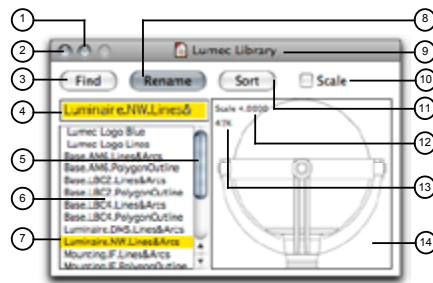
To use the Symbol tool:

- ☐ Choose Open from the File menu and a dialog will appear.



- ☐ From the Enable pop-up menu, choose the type of PowerCADD Symbol Library you would like to open. If you are not sure what version of PowerCADD was used to create the symbol library choose All Readable Files.
- ☐ The symbol library selected in the Open dialog opens in its own window.
- ☐ In the symbol library window, click on the name of the symbol you want to insert into the drawing.
- ☐ Return to the drawing by either using the Window command under the Window menu, the Open Recent list or by placing the cursor in the drawing window and clicking.
- ☐ With the library open and the correct symbol selected in the library window, select the Symbol tool.
- ☐ Place the cursor in the location you want to place the symbol and click, or click and drag.
- ☐ To add additional symbols, simply place the cursor and click. The current symbol will remain in effect until you bring the library to the front and select another symbol.

Symbol Library Window The following diagram describes the basic elements of a Symbol Library Window.



(1) **Minimize Box:** Click here to collapse the Symbol Library to the Dock.

(2) **Close Box:** Click here to close a Symbol Library or press Command-W.

(3) **Find Button:** Used to search for a specific symbol name in a library. To find a specific symbol type its name or portion of its name and click *Find*.

(4) **Name Field:** Data entry field to find, name, or rename symbols.

(5) **Scroll Bars:** Scroll up or down the Symbol Name List to see additional symbols in a library. If no scroll bars are displayed all symbols in the library are being displayed in the Symbol Name List.

(6) **Symbol Name List:** Displays the names of all symbols in a library.

(7) **Symbol Name:** Click on a symbol name to see a thumbnail of it in the Preview Pane.

(8) **Rename Button:** Used to Rename an existing symbol in a library you assign a name to a new symbol. To name a new symbol type the name in the Name Field immediately after pasting it into the library and click *Rename*. To rename an old symbol select the symbol from the Symbol Name List, type a new name in the Name Field Type and click *Rename*.

(9) **Title Bar:** Displays the name of the Symbol Library. Press the Command key and click on the name of the symbol library to display its file path.

(10) **Scale:** Check this box to have symbols automatically rescaled to match the scale of the drawing. When unchecked the symbols are inserted at their original scale (they are not scaled to match the drawing scale).

(11) **Sort Button:** Press this button to sort the Symbol Name List in A.S.C.I.I (similar to alphabetical) sort order.

(12) Drawing Scale: Displays the drawing scale that the original symbol was created in.

(13) Preview Scale: Displays the relative size of the thumbnail shown in the Preview Pane. 100% is a full size preview.

(14) Preview Pane: Displays a thumbnail of the selected Symbol Name.

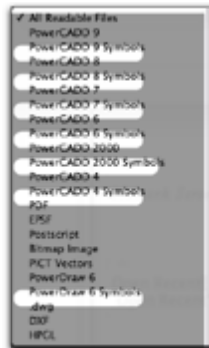


For more information on working with Symbols, see [Symbol Palette](#) in the Menu chapter.



NOTE: You can specify which version of PowerCADD or PowerDraw Symbol Libraries you would like to open.

Selecting a specific type of file will automatically filter the list of items displayed to match the specified type of file. If a particular library you want to open is dimmed in the list, choose All Readable Files.



For example, to see only Symbol Libraries created using PowerCADD 2000, choose PowerCADD 2000 Symbols from the list. Only symbol libraries saved in PowerCADD 2000 will be displayed; all other files will be dimmed.



For more information on symbol libraries, please refer to [New Library](#) in the Menus chapter.

Symbol Palette



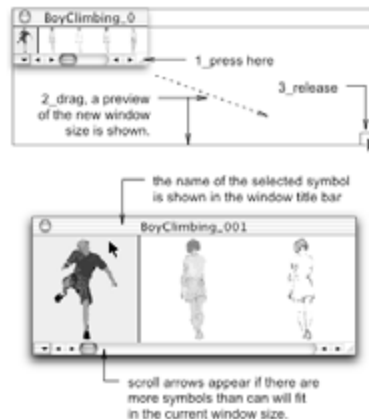
The Symbol palette floats on top of your drawing window and allows you to choose the symbols to place without leaving your drawing window.

NOTE: The Symbol Palette is an External plug-in. In order for this window to appear in the Window menu it must be checked on when PowerCADD is launched. See 'Preferences', 'Loader' for more on turning External plug-ins On and Off.



The Symbol palette is an alternate to the Symbol tool for placing symbols in your drawing. Symbols are no different if placed using the Symbol library dialog in conjunction with the Symbol tool than if placed by using the floating Symbol palette. Before a symbol library has been opened, your floating Symbol palette appears with no symbols.

When you have a symbol library open or you create a symbol library, the Symbol palette displays your symbols as a row of thumbnails.



Drag the window size box to adjust the palette size.

When you choose a symbol in the Symbol palette, the Symbol tool is automatically selected. You may choose a different symbol by clicking on the picture of the symbol. You are ready to place your symbol by simply clicking in your drawing at the appropriate location.

You may also drag and drop a symbol into your drawing by clicking on your symbol and dragging it into the desired location. The symbol will be placed about its center or its placement point.

To deselect the Symbol tool, double-click in the same location or select another tool from the Tool palette.

When you place a symbol, its center point appears at the cursor location and is used as the placement point. For a point other than the center to be used for placement, a reference point must be the last object added to the symbol. The reference point becomes the placement point.

The Door Insertion tool, which can also use symbols, ignores placement points. It always places the bottom of the door on the wall line.

Groups and Symbols

A symbol can be inserted in a drawing two ways:

- ☐ As a group
- ☐ As a symbol

A group is a grouped object. Although PowerCADD treats a group as one object, each part of the group retains its own identity. Each occurrence of the group requires a complete description of each part, and each description requires space in the file.

A symbol is merely a reference to a description of the symbol, therefore requiring less file space. The description occurs only once, but references to the description can occur many times.

Advantage of groups:

Groups redraw faster than symbols and their object attributes may be changed (e.g. line weights and colors).

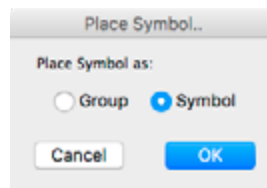
Advantages of symbols:

- ☐ Symbols use less file space than groups.
- ☐ **Information (such as part number or size) attached to a symbol appears on reports generated by Symbol Report. The Set Symbol Tags menu item is used to assign the information.**
- ☐ Object attributes, such as line weight, color, etc. cannot be edited.

Obsolete?

To set the default for the Symbol tool:

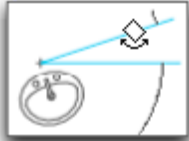
- ☐ Press the Option key and select the Symbol tool from the Tool palette. The following dialog will appear.



- ☐ Select Group or Symbol and click OK.

Rotating Symbols

A key command must be assigned to Rotate in the Arrange menu (R is the default key assigned, but it can be any key or key combination you choose to assign).



To rotate a symbol during placement:

- ☐ Select a symbol from an open Symbol Library and click in the active drawing window.
- ☐ Select the Symbol tool from the tool palette and move the mouse into the drawing window.
- ☐ Press the mouse button. A ghosted symbol appears.
- ☐ Enter the rotation mode by tapping the R key (or the key you have assigned). A ghost symbol and the Rotation cursor appear.
- ☐ Drag to rotate the symbol. The symbol rotates about its placement point and is constrained to the preset snap angles. The Shift key releases the constraint.
- ☐ When the correct rotation is achieved, press the F key to flip the symbol's orientation or release the mouse button to insert the symbol.
- ☐ To exit the rotation mode, press and release the R key a second time.

The symbol can then be dragged to another location or flipped.

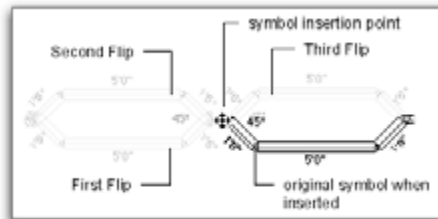
Flipping Symbols

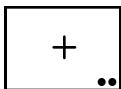
A key command must be assigned to Flip Horizontal in the Arrange menu (F is the default key assigned, but it can be any key or key combination you choose to assign).

To flip a symbol during placement:

- ☐ Select the Symbol Tool and press the mouse button. A ghosted preview appears of the symbol being placed.
- ☐ Press the F key (or the key you have assigned to Flip Horizontal). A ghosted preview appears of the symbol in its new flipped position.
- ☐ Keep pressing the F key to toggle through the symbol's four horizontal and vertical orientations.
- ☐ When the symbol is in the desired orientation, press the R key to rotate the symbol to the desired angle or release the mouse to insert the symbol at its default rotation angle.

graphic below is terrible





Point Tool

This tool allows you to establish reference points which can be snapped to during the drawing process or to specify the insertion point for symbols.

Sometimes it is difficult to position an object precisely on a given point in the drawing area. By establishing a reference point, and selecting the object snap option in the Drawing Setup, Snap tab under the Layout menu, a drawn object can be placed with greater precision.

To place a reference point in the drawing:

- ☐ Select the Point tool from the Tool palette.
- ☐ Place the cursor in the drawing where you wish to locate a reference point and click. You will see a “+” symbol appear which represents the insertion point.

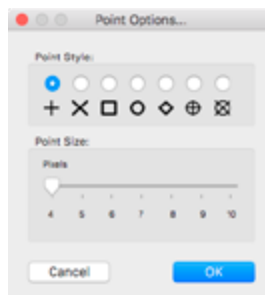
You may place as many points as you need by placing the cursor and clicking the mouse.

To deselect the Point tool, select another tool or place a reference point directly on top of the last placed point.

You can select the size and type of graphic to use as a reference point.

To set the default for the Point tool:

- ☐ Press the Option key and select the Point tool from the Tool palette. The Point Options dialog will appear:



- ☐ Select a radio button to choose the desired Point Style.
- ☐ Drag the Point Size slider to set the desired size, in pixels, of the Reference Point.
- ☐ Click OK.
- ☐ Click the mouse in the drawing window to insert the reference point.

These settings will remain in effect until you change them.

The X, Y location of reference points, like any other object, can be edited

with the Edit Window. Reference points also act as normal snap points when Object Snap is selected.

Reference points are generally used as temporary markers or magnets to locate other items. Points represent an excellent method of marking locations that will be snapped to.

By default, Reference Points do not print nor are they included when using Export from the File menu to create raster images for use on the internet or other image editing applications.

Reference points can be left in the drawing indefinitely.



NOTE: Reference Points can be printed by enabling the Print Points option, in the PowerCADD print settings. See 'Print' in the menu chapter for more information on optional PowerCADD print settings.

To remove the reference points quickly and easily:

- ☐ Set the Selection Filter to points only.
- ☐ You may now select and delete reference points from an area without deleting other objects.

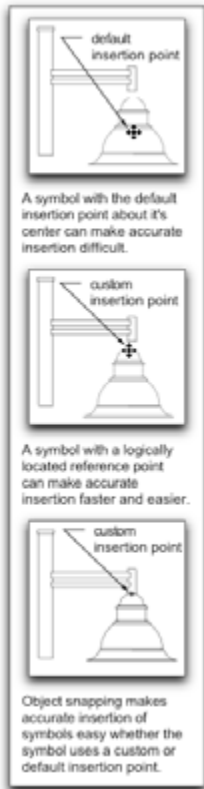
If you want to remove all reference points from the drawing, choose Select All from the Edit menu and then Select Clear.

All reference points will now be removed from the drawing, and you have not lost valuable time looking for points.

Be sure to reset the Selection Filter window to select all objects, otherwise you will only be allowed to select reference points and some snap commands will not work correctly.

To reset the Selection Filter window, Option-click on the Points box or select All. All items should now be activated.

Changing a Symbol's Insertion Point



When a Reference Point is the last object added before pasting a symbol into a library, the reference point will become the insertion point for that symbol.

By default, the center of a symbol is its insertion point. However, in many drawing conditions, it may be beneficial to insert a symbol about a point other than its center.

For example, when inserting a toilet or vanity fixture into a plan, it may be more practical to have the insertion point be the minimum offset from the corner of a wall or counter top.

When creating symbols for use in elevation and section drawings such as a street lamp, the logical insertion point is the bottom of the light fixture where it meets the ground.

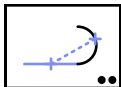
To use a Reference Point to change a symbol's insertion point:

- ☐ Draw the symbol using any type of PowerCADD object, collection of objects or drawing techniques.
- ☐ Select the Reference Point tool from the Tool palette and click to insert the reference point in the desired location.

This location will be the insertion point for the symbol and does not have to touch the object. It is critical the Reference Point be the top most object (i.e. the last object you add to the symbol).

- ☐ Select all the objects intended to be the symbol including the reference point and choose Copy from the Edit menu.
- ☐ Open a new or existing Symbols Library window and choose Paste from the Edit menu.
- ☐ Rename your symbol and save the library.

When you use the symbol tool, the Reference Point will be the insertion point for the symbol. The insertion point is also the origin for all Flip or Rotate operations applied to the symbol as it is being inserted into the drawing. Reference flipping and rotating symbols earlier in this section for additional information.



Attach Tool

This tool allows you to attach separate objects together into a new, single polygon, Bézier, or group of objects containing a new polygon.

While similar to the [Attach menu item](#), the Attach tool allows you to either automatically select all adjoining items or to pick the specific items to be attached. If the endpoints of the objects to be attached are close but do not meet exactly, short line segments are automatically added to connect them.

To use the Attach tool:

- ☐ Select the Attach tool from the tool palette.
- ☐ Click on the first object.

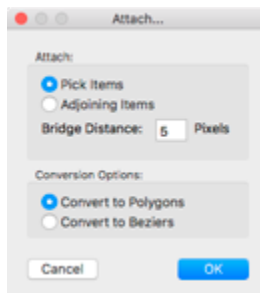
All adjoining objects are automatically selected and a new object is created.

Whether the Attach tool automatically selects adjoining objects, or requires you to click and drag to select objects to be attached, is determined by an optional tool setting described later in this section.

The Attach tool has optional settings which affect how it interacts with the objects in the drawing and the type of geometry created. Settings made in the optional settings dialog remain in effect until you change them.

To change the settings for the Attach tool:

- ☐ Press the Option key and select the Attach tool from the Tool palette. The following dialog will appear:



Pick Items: Requires you to perform click and drag to select the objects to be attached. Click on the first object, drag, and release on the second object: the two objects are attached. The tool remains active and you may continue to attach additional objects to the result.

Adjoining Items: Allows the tool to work with a single click. Click on the first object and all adjoining objects, or objects whose endpoints are within the specified Bridge Distance, are attached.

Bridge Distance: Allows you to specify the maximum distance between object end points in pixels. Objects which are farther apart than the specified distance are not attached.

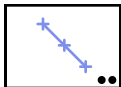
Convert to Polygons: All objects to be attached are converted to a polygon; the original geometry is changed. For example, if the objects to be attached included a line and an arc, the result is a polygon.

Convert to Béziers: All objects to be attached are converted to a Bézier curve; the original geometry is changed. For example, if the objects to be attached included a line and an arc, the original arc is replaced with a Bézier curve.

- ☐ Make the desired changes and click OK.
- ☐ Apply the Attach tool as described earlier.



CAUTION: When Conversion Options are set to Convert to Polygons or Convert to Béziers, higher level geometry such as arcs and elliptical arcs are replaced. To retain the original higher level geometry, use the Attach menu item and set its options for No Conversion.



Objects On Line Tool

This tool allows you to place Reference Points, or objects from the Clipboard or a Symbol Library along a linear path.

To use the Objects On Line tool:

- ☐ Select an object in the drawing and choose Copy from the Edit menu.

See Optional settings below on how to insert a Reference Point or Symbol.

- ☐ Select the Objects On Line tool from the Tool palette.
- ☐ Click and drag in the drawing to specify a line.
- ☐ A ghosted outline appears of the objects being inserted.
- ☐ Release the mouse: objects are inserted at their center points along the specified linear path.
- ☐ The inserted items are automatically grouped and selected.

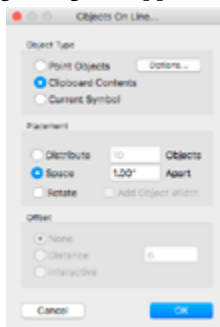


NOTE: Press the Shift key to constrain the linear path to preset snap angles. Use object snapping keys to snap the linear path to existing geometry in the drawing. Press the Tab key as you drag to specify the linear path; enter the desired length and angle in the Edit Window and press the Return or Enter key.

The Objects On Line tool has optional settings which affect how it interacts with the objects in the drawing and the type of objects inserted. Settings made in the optional settings dialog remain in effect until you change them.

To change the settings for the Objects On Line tool:

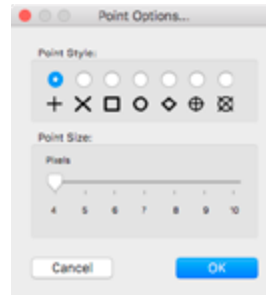
- ☐ Press the Option key and select the Objects On Line tool from the Tool palette. The following dialog will appear:



Point Objects: Inserts Reference Points into the drawing.

Point Options: Click to open the Point Options dialog and choose the desired point style and size. Settings in this dialog are saved with the

Objects On Line tool: they are separate to the Objects On Path and Reference Point tool optional settings.



Clipboard Contents: Inserts the contents of the Clipboard into the drawing. In the main drawing window, select the object and choose Copy from the Edit menu; the object is added to the Clipboard.

Current Symbol: Inserts the current symbol displayed in the active Symbol Library window.

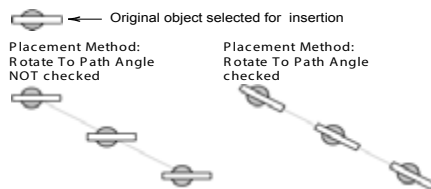
Distribute: Evenly inserts the number of objects specified in the Objects field. The spacing between objects is determined by the length of the path drawn with the Objects on Line tool.

Objects Field: Allows you to specify the quantity of objects inserted when using the Distribute placement method.

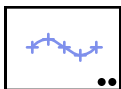
Space: Inserts objects based on the distance specified in the Apart field. The total number of objects is determined by the length of the path drawn with the Objects on Line tool.

Apart Field: Allows you to specify the spacing between objects inserted when using the Space placement method.

Rotate To Path Angle: Inserts objects so they rotate perpendicular to the path drawn with the Objects on Line tool. This option is only available when Object Type is set to Clipboard Contents or Current Symbol.



- ☐ Make the desired changes and click OK.
- ☐ Apply the Objects On Line tool as described earlier.



Objects On Path Tool

This tool allows you to place Reference Points, or objects from the Clipboard or a Symbol Library using an existing object in the drawing as the specified path.

To use the Objects On Path tool:

- ☐ Select an object in the drawing and choose Copy from the Edit menu.

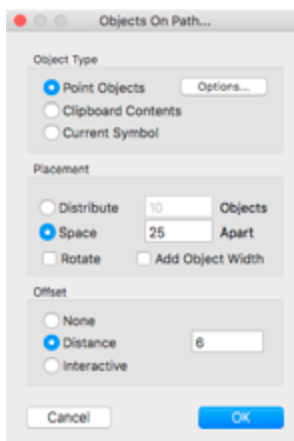
See Optional settings below on how to insert a Reference Point or Symbol.

- ☐ Select the Objects On Path tool from the tool palette.
- ☐ Click on the object in the drawing to choose the path the objects should be placed on. The object may be a line, polygon, arc or Bézier curve.
- ☐ Objects are inserted at their center points along the chosen path.
- ☐ The inserted items are automatically grouped and selected.

The Objects On Path tool has optional settings which affect how it interacts with the objects in the drawing and the type of objects inserted. Settings made in the optional settings dialog remain in effect until you change them.

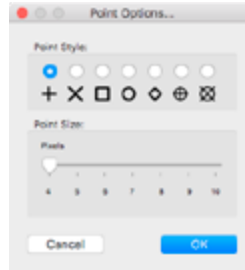
To change the settings for the Objects On Path tool:

- ☐ Press the Option key and select the Objects On Path tool from the Tool palette. The following dialog will appear:



Point Objects: Inserts Reference Points into the drawing.

Point Options: Click to open the Point Options dialog and choose the desired point style and size. Settings in this dialog are saved with the Objects On Path tool: they are separate from the Objects On Line and Reference Point tool optional settings.



Clipboard Contents: Inserts the contents of the Clipboard into the drawing. In the main drawing window, select the object and choose Copy from the Edit menu; the object is added to the Clipboard.

Current Symbol: Inserts the current symbol displayed in the active Symbol Library window.

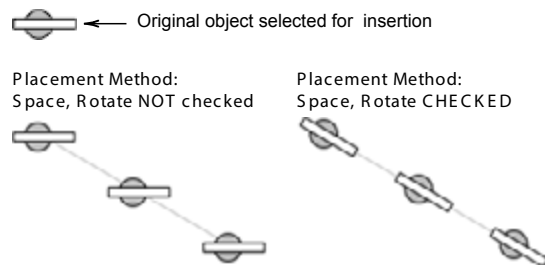
Distribute: Evenly inserts the number of objects specified in the Objects field. The spacing between objects is determined by the length of the path chosen with the Objects on Path tool.

Objects Field: Allows you to specify the quantity of objects inserted when using the Distribute placement method.

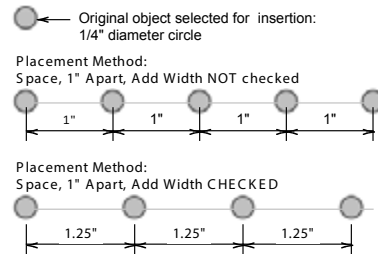
Space: Inserts objects based on the distance specified in the Apart field. The total number of objects is determined by the length of the path chosen with the Objects on Path tool.

Apart Field: Allows you to specify the spacing between objects inserted when using the Space placement method.

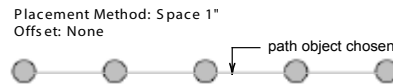
Rotate: Inserts objects so they rotate perpendicular to the path chosen with the Objects on Path tool.



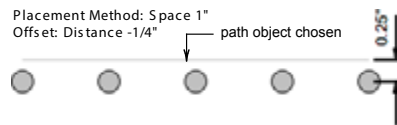
Add Object Width: Works with both Distribute and Space placement methods to increase the spacing between the objects inserted by the width of the original object selected for insertion. See below:



None: Uses no (zero) offset spacing; the objects are inserted directly along the path chosen with the Objects on Path tool.



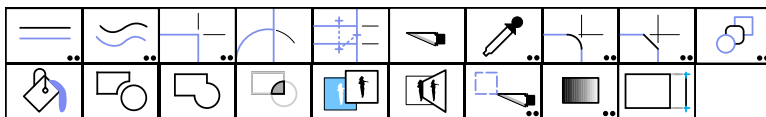
Distance: Allows you to specify a numerical offset for the inserted objects based on the drawing units and scale. Using the target cursor, press on the object defining the path; drag in the direction (up, down, left, right) the objects should be offset and release the mouse.



Interactive: Allows you to dynamically specify the offset for objects using a press and drag operation. Using the target cursor, press on the object defining the path and drag to specify the offset; release the mouse to complete the operation.

- ☐ Make the desired changes and click OK.
- ☐ Apply the Objects On Path tool as described earlier.

Modifier Tools



PowerCADD's Modifier Tools consist of a suite of versatile, easy-to-use tools generally used to enhance existing objects in the drawing. The tools enable you to create new derivative geometry from existing objects, apply special effects or perform Boolean operations to clip and combine objects. Each tool is optimized to fit a particular part of your work flow. The following section describes each of the PowerCADD Modifier Tools in detail.

Before describing each Modifier tool in detail, it is important to be aware of the following fundamental concepts which apply to all Modifier tools:

- ❑ In all cases, these tools interact with objects already in the drawing. How the tools interact and the results will vary with each tool. In some cases the existing object geometry is deleted and replaced with a new object. In other cases the existing geometry is modified or a new object is created. The specifics of each tool are described in detail later in this section.
- ❑ Dynamic Snapping may be used to modify the start or end points of many tool operations. For example, when using the Break tool, snap commands may be used to specify the precise location of the operation.
- ❑ Tools requiring dimensional data such as an offset or radius support direct entry of those values using the Edit Window.
- ❑ When working with Edit All Layers turned On, new objects are always created on the active drawing layer regardless of which layer the original objects were located.



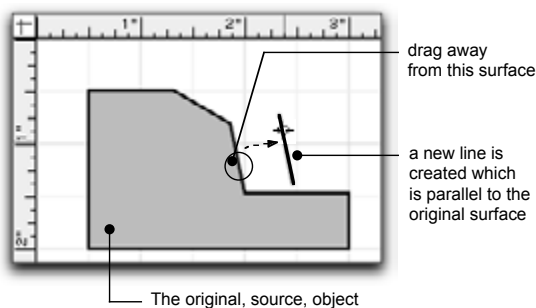
Offset Line Tool

This tool makes a new line which is offset from an existing object in the drawing.

The Offset Line tool has optional settings and attribute settings which, once set, remain in effect until you change them. The tool may be used repeatedly, remaining active until you select another tool or click in an empty area of the drawing.

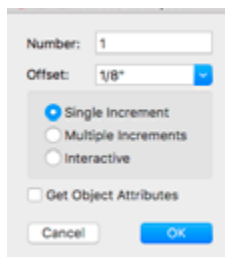
To use the Offset Line tool:

- ☐ Select the Offset Line tool from the Tool palette.
- ☐ Place the cursor on the object to be offset, press the mouse button.
- ☐ Move the cursor in the direction you wish the offset to occur and release the mouse. As soon as a direction is specified, the offset is performed.



To change the default settings for the Offset Line tool:

- ☐ Press the Option key and select the Offset Line tool from the Tool palette. The following dialog will appear:



Number: Determines how many new lines will be created with a single use of the tool. If you know the number needed, and the offset is the same, they can be created in one step.

Offset: Determines the distance, based on the units scale of the drawing (or Layer Scale Attributes) the new line will be shifted from the original

object and is performed in the direction of mouse movement. If Single Increment is chosen the offset distance will increase in multiples of the specified offset value. Up to nine offset values are saved. If more than nine offsets are entered, the oldest (value entered first) is deleted; the most recently added value appears at the top of the list. To choose an offset click on the popup menu icon and select it from the list.

Single Increment: The offset will be limited to a single offset distance as specified in the offset value field.

Multiple Increments: The offset distance will increase in multiples of the specified offset value the farther you drag the mouse. For example, if the offset is set to 1/8 inch, the offset distance will increase in increments of 1/8 inch as you drag the mouse, creating a new object offset at 1/4, 3/8, 1/2 inch or more. The final offset distance depends on how far you drag the mouse from the original object. The current offset value is displayed in the Edit Window as you drag to offset.

Interactive: Determine the offset by dragging in the drawing window. If this is not selected, you type the desired offset in the Offset field described below. When performing an interactive offset you may use snap commands to snap the offset to another object in the drawing. You may also press the Tab key during the offset operation and enter a specific distance in the Edit Window.

Get Object Attributes means the attributes of the new, offset, object will match the attributes of the original, source, object. All attributes, as defined in the Attributes floating window, are matched including opacity and shadow settings.

Assigning Attributes

To assign default attributes to the Offset Line tool:

- ☐ Double-click the Offset Line tool in the Tool palette.
- ☐ Assign the desired attributes using the Attr menu or the Attributes floating window.

When the Get Object Attributes box is not checked, attributes assigned to the Offset Line tool are applied to the new offset line. When the Get Objects Attributes box is checked, attributes assigned to the tool are ignored and the attributes of the source object in the drawing are applied to the new offset line.

Using the Edit Window

You can specify the offset value on the fly during the offset operation by tabbing into the Edit Window. Specifying the offset value in the Edit Window affects only the current offset operation. The settings specified in the Offset Line dialog are not affected.

To specify an offset using the Edit Window:

- ☐ Select the Offset Line tool from the Tool palette.
- ☐ Press and drag on the object to be offset. Do not release the mouse.
- ☐ Press the Tab key. If the Edit Window isn't already open, it will open automatically.
- ☐ Type the offset value and press the Return key.

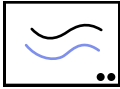
Edit All Layers

When working with Edit All Layers turned On, new objects are created on the same layer as the original, regardless of the active drawing layer when the Offset Line tool was applied.

Groups and Symbols

You can create an offset line from objects inside of a group. This includes symbols which are inserted into a drawing as a Group. You cannot create an offset line from an object inserted from a Symbol Library as a Symbol.

Generally you can perform a offset line operation from any opened or closed object with a straight surface. Objects from which a parallel line cannot be offset from are Béziers, smooth polygons, text, symbols, bitmaps, placed objects, and points.



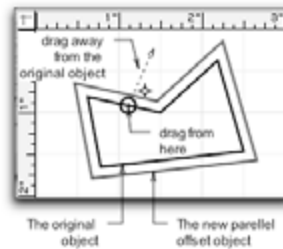
Parallel Offset Tool

This tool allows you to make a duplicate of an object offset from the original.

The Parallel Offset tool has optional settings and attribute settings which, once set, remain in effect until you change them. The tool may be used repeatedly, remaining active until you select another tool or click in an empty area of the drawing.

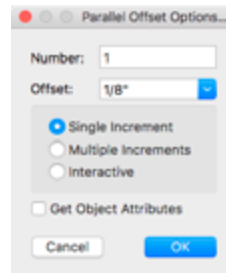
To use the Parallel Offset tool:

- ☐ Select the Parallel Offset tool from the Tool palette.
- ☐ Place the cursor on the object to be offset, press the mouse button.
- ☐ Move the cursor in the direction you wish the offset to occur and release the mouse. As soon as a direction is specified, the offset is performed.



To change the default settings for the Parallel Offset tool:

- ☐ Press the Option key and select the Parallel Offset tool from the Tool palette. The following dialog will appear:



Number: Determines how many new lines will be created with a single use of the tool. If you know the number needed, and the offset is the same, they can be created in one step.

Offset: Determines the distance, based on the units scale of the drawing (or Layer Scale Attributes) the new line will be shifted from the original object and is performed in the direction of mouse movement. If Single Increment is chosen the offset distance will increase in multiples of the specified offset value. Up to nine offset values are saved. If more than nine offsets are entered, the oldest (value entered first) is deleted; the most recently added value appears at the top of the list. To choose an

offset click on the popup menu icon and select it from the list.

Single Increment: The offset will be limited to a single offset distance as specified in the offset value field.

Multiple Increments: The offset distance will increase in multiples of the specified offset value the farther you drag the mouse. For example, if the offset is set to 1/8 inch, the offset distance will increase in increments of 1/8 inch as you drag the mouse, creating a new object offset at 1/4, 3/8, 1/2 inch or more. The final offset distance depends on how far you drag the mouse from the original object. The current offset value is displayed in the Edit Window as you drag to offset.

Interactive: Determine the offset by dragging in the drawing window. If this is not selected, you type the desired offset in the Offset field described below. When performing an interactive offset you may use snap commands to snap the offset to another object in the drawing. You may also press the Tab key during the offset operation and enter a specific distance in the Edit Window.

Get Object Attributes means the attributes of the new, offset, object will match the attributes of the original, source, object. All attributes, as defined in the Attributes floating window, are matched including opacity and shadow settings.

Assigning Attributes

To assign default attributes to the Parallel Offset tool:

- ☐ Double-click the Parallel Offset tool in the tool palette.
- ☐ Assign the desired attributes using the Attr menu or the Attributes floating window.

When the Get Object Attributes box is not checked, attributes assigned to the Parallel Offset tool are applied to the new offset line.

When the Get Objects Attributes box is checked, attributes assigned to the tool are ignored and the attributes of the source object in the drawing are applied to the new offset line.

Using the Edit Window You can specify the offset value on the fly during the offset operation by tabbing into the Edit Window. Specifying the offset value in the Edit Window affects only the current offset operation. The settings specified in the Parallel Offset dialog are not affected.

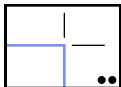
To specify an offset using the Edit Window:

- ☐ Select the Parallel Offset tool from the tool palette.
- ☐ Press and drag on the object to be offset. Do not release the mouse.
- ☐ Press the Tab key. If the Edit Window isn't already open, it will open automatically.
- ☐ Type the offset value and press the Return key..

Edit All Layers When working with Edit All Layers turned on, new objects are created on the same layer as the original, regardless of which layer was the active drawing layer when the Parallel Offset tool was applied.

Groups and Symbols You can perform a parallel offset of objects inside of a group. This includes symbols that are inserted into a drawing as a Group. You cannot perform a parallel offset of objects inserted from a Symbol Library as a Symbol.

Generally you can perform a parallel offset on Bézier curves, circles, polygons, lines, rectangles and any other open or closed geometric shape. Objects which cannot be offset are text, the text block component of a leader line, (the leader line itself may be offset) symbols, bitmaps, placed objects, groups, points, dimension lines and hatches.



Line Trim/Extend Tool

This tool cuts or extends lines at their intersection so they meet at a converging point in space.

The Line Trim/Extend tool is an intelligent modifying tool; it will make a decision based on its settings, the objects selected and their position relative to each other. The tool will remain active until you click in a blank area of the drawing or select another tool. Default settings chosen in the dialog are remembered and remain in effect until you change them.



NOTE: A line outside a group may be trimmed or extended to an object inside a group. To trim or extend lines inside of a group, use the Edit Group command.



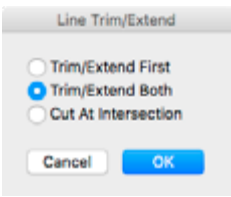
NOTE: The Line Trim/Extend tool will only trim or extend single lines; to trim or extend multiple lines at the same time use the [Multi-Trim/Extend tool](#) described later in this chapter.



NOTE: Lines may be trimmed or extended to Guidelines, open or closed polygons, rectangles, and rotated rectangles which are distinct objects in the drawing or part of a group of objects.

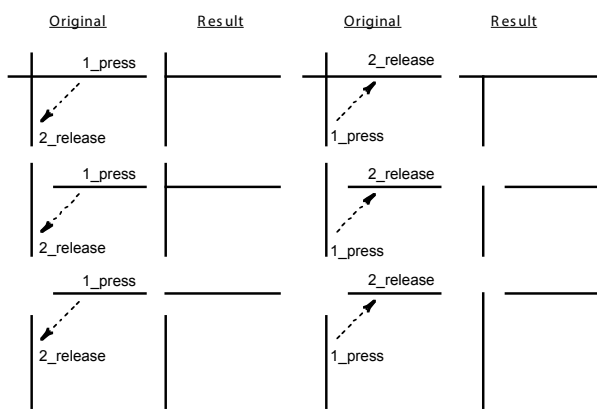
To set the behavior of the tool:

- ❑ Press the Option key and select the Line Trim/Extend tool. The following dialog will appear.



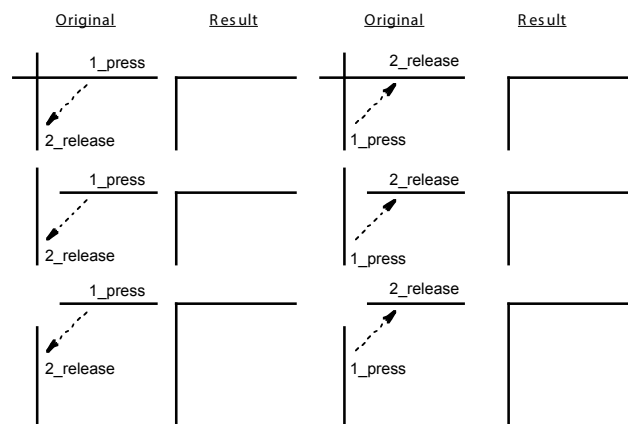
Trim/Extend First

The first line chosen will be either trimmed or extended to the intersection of the second line chosen. If the lines overlap, the first line is trimmed to where it meets the second line and the trimmed piece is deleted. If the lines do not overlap, the first line is extended to where it meets the second line.



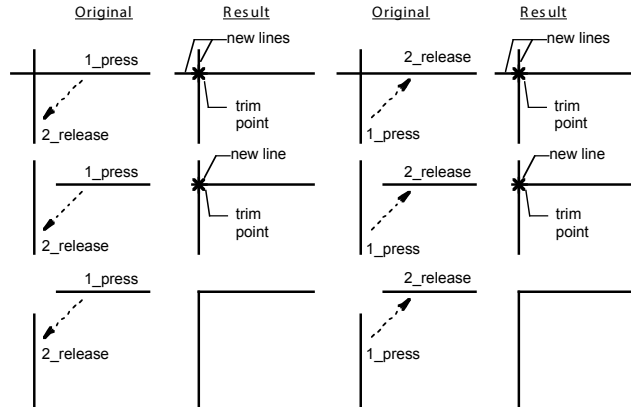
Trim/Extend Both

Both lines chosen will be either trimmed or extended to meet at their intersection. If the lines overlap, they are trimmed where they meet and the trimmed pieces are deleted. If the lines do not overlap, they are extended to where they meet.



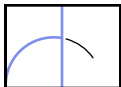
Cut At Intersection

The first line chosen will be either trimmed or extended to the intersection of the second line chosen. If the lines overlap, both lines are trimmed and the overlapping portions become new lines. If the lines do not overlap, the first line chosen is extended or trimmed to meet the second line. If a trim operation occurred the line segment is not deleted; a new line is created at the intersection.



To Trim or Extend lines:

- ☐ Select the Line Trim/Extend tool from the Tool palette.
- ☐ Place the cursor on the first of the two lines to be trimmed or extended and press.
- ☐ Drag the cursor to the second line. A blue construction line will connect the two lines to indicate the tool is active. Release the mouse button to trim.



Arc Trim/Extend Tool

This tool cuts or extends arcs to their intersections so they meet at a converging point in space.

The tool will remain active until you click in a blank area of the drawing or select another tool.

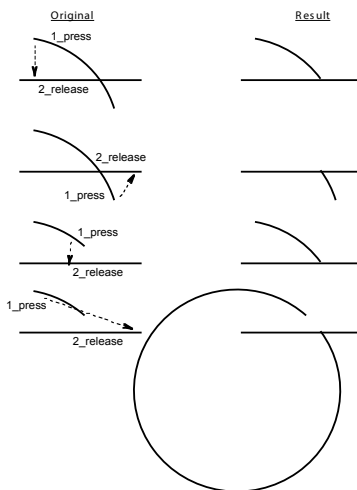
The first object picked will be the object modified. It will be trimmed or extended to meet the second object picked, at the intersection of the two. The first object picked can only be an arc.

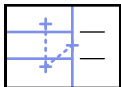
The second object can be Guidelines, open or closed polygons, rectangles, rotated rectangles, and lines which are distinct objects in the drawing or part of a group of objects.

To Trim or Extend Arcs:

- ☐ Select the Arc Trim/Extend tool from the Tool palette.
- ☐ Place the cursor on the arc you wish to extend or trim and press the mouse button. The end picked first is the end which will be retained.
- ☐ Drag the cursor to the second object. A blue construction line will connect the two objects to indicate the tool is active.
- ☐ Release the mouse button.

The arc will extend or trim to the intersection of the second.





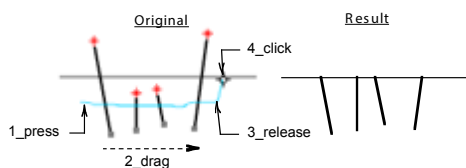
Multi-Trim/Extend Tool

This tool may be used to trim or extend multiple lines at the same time to a single target object such as a line, rectangle, rotated rectangle, or polygon segment.

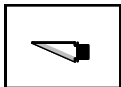
To trim and extend multiple lines at the same time:

- ☐ Select the Multi-Trim/Extend tool from the Tool palette.
- ☐ Place the cursor near the objects to be trimmed and extended.
- ☐ Drag the swipe cursor across the objects to be trimmed and extended then release the mouse.
- ☐ Drag the target cursor to the line you want the lines trimmed and extended to and release the mouse button.

The Trim/Extend tool will trim and extend the selected lines to their intersection point on the line.



NOTE: Using the Multi-Trim/Extend tool lines may be trimmed or extended to Guidelines, open or closed polygons, rectangles, and rotated rectangles which are distinct objects in the drawing or part of a group of objects.

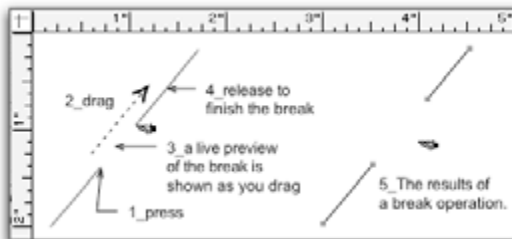


Break Tool

This tool allows you to create an opening in an object at any point along the perimeter of the object. You may also slice through objects.

To use the Break tool:

- ☐ Select the Break tool from the Tool palette.
- ☐ Place the cursor on the object to be broken and press. This will specify the first point of the break.
- ☐ Drag the cursor in the direction you wish the break to occur.
- ☐ Release the mouse and the break will be inserted.



The Break tool will remain active until you select another tool or click in the drawing. Break can be used repeatedly without having to preselect objects or tools; once a break is inserted, you are free to start a new break.

If the line you wish to break extends out of the current view, simply begin the break as described and drag the cursor off the screen to scroll the page. The break will continue until you stop the scroll and release.



NOTE: The Break tool always breaks the top-most object. If you are working in a crowded drawing or working with Edit All Layers On, it can be difficult to determine which object is on top.

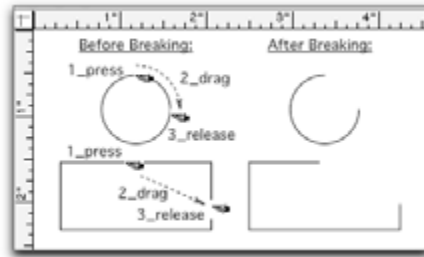
To select the object to be broken in a crowded drawing:

- ☐ Choose the Break tool from the tool palette.
- ☐ Press the Option key, the target cursor will appear, and click anywhere on the object to be broken.
- ☐ Release the Option key and place the cursor on the object to be broken and press the mouse button. This will specify the first point of the break.
- ☐ Drag the cursor in the direction you wish the break to occur.
- ☐ Release the mouse and the break will be inserted.

This method ensures you are breaking the desired object and does not modify the objects layer position or object order.

Breaking Curves and Polygons

The Break tool will cut along the circumference of circles and the around the corners of rectangles and polygons. Simply place the cursor on the point where the break will begin, then press and drag the cursor to the desired end point.



If you begin a break on an object and press the Option key, the break will toggle from the area you specified, to the remainder of the object leaving the section you specified. For example, if you choose to break out one small section of a circle and press the Option key, the active break line would toggle to the remaining larger section of the circle.

Snipping

A cut or *snip* may be made by simply clicking on an object in the desired location.

Slicing

The Break tool may be used to slice through an object or several objects on the same layer; the results are very much like those when using the Cut with Line command.

To slice an object or objects:

- ☐ Select the Break tool. Place the cursor on or above the object or objects to be cut.
- ☐ Press and drag. A blue construction line will appear indicating the path of the cut.
- ☐ Drag to the desired location and release.

All objects which were crossed by the cutting line will be cut and will remain selected.

If Edit All Layers is On, all visible objects crossing the cutting line will be cut regardless of which layer they are on. To cut only objects on the active layer, turn Edit All Layers Off.

You may also use the Break tool to slice bitmap images. If the raster image was inserted using the Place command, it must be converted to a bitmap first using the Object to Bitmap command.

Editing a Break

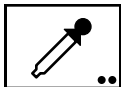
To create a break of specific length in a selected line, you may use the Edit Window.

To edit a break in a line:

- ☐ Make sure the Edit Window is active.
- ☐ Select the Break tool from the Tool palette.
- ☐ Place the cursor on the line to be broken and press. This will specify the first point of the break.
- ☐ Drag the cursor along the line in the direction you wish the break to occur. Press the Tab key, then release the mouse button.
- ☐ Tab to the Length field in the Edit Window.
- ☐ Type the desired length of the line break, then click anywhere in the drawing or press the Enter key. The section you specified will be highlighting on the line. Click to complete the break.



NOTE: Break will not work on grouped objects.



Eyedropper Tool

This tool allows you to copy the attributes of an object and apply those attributes to other objects.

To adopt an object's attributes:

- ☐ Select the Eyedropper tool from the Tool palette.
- ☐ When you press the Option key the cursor will display as an empty eyedropper ready to pick up or copy the attributes of the object you select:
- ☐ Click on the object with the Eyedropper tool.

The attributes will be picked up or copied.

The Eyedropper tool is now ready to apply those attributes to other objects. The tool will retain those attributes even if you select another tool and come back to it at a later time during your drawing session.

Defaults can also be set for the Eyedropper tool as with the other PowerCADD tools. Press the Control key and select the tool and choose the attribute settings you wish to use from the Contextual menu.

To apply an object's attributes:

- ☐ Select the Eyedropper tool from the Tool palette. The cursor will display as a full eyedropper:
- ☐ Click on the object with the Eyedropper.

The attributes will be applied from the Eyedropper tool to the object.



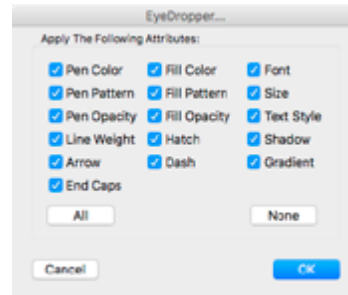
NOTE: Attributes are not remembered from session to session and will use the default settings until changed.

You may modify the attributes of individual objects within a group. Select the group, choose Edit Group from the Edit menu, then use the Eyedropper tool to change the attributes of one or more objects as desired. Choose Exit Groups from the Edit menu when finished.

The Eyedropper tool can also be used with text styles, hatch patterns, transparency, shadows, and gradient fills created by the Gradient tool.

To specify which object attributes the Eyedropper tool should pick up and apply:

- ❑ Press the Option key and select on the Eyedropper tool in the Tool palette. The following dialog will appear:



- ❑ Check the boxes corresponding to the object attributes you like to pick up and apply.
- ❑ Click OK and apply the Eyedropper tool.

The settings in this dialog remain in effect until you change them.

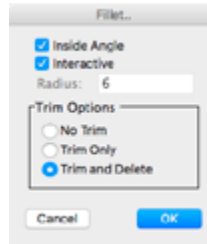
Fillet Tool

This tool draws an arc tangent to two lines, circles, or circular arcs in any combination.

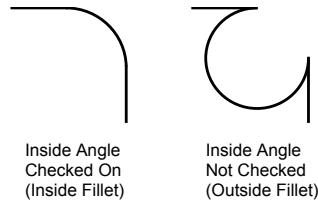
A fillet is a rounded-corner, which may be applied to connect intersecting or converging arc-arc, arc-line, line-line, line-circle, or arc-circle objects.

Objects that do not intersect will be extended.

To set the behavior of the tool, press the Option key and select the Fillet tool from the Tool palette. The following dialog will appear:



- ☐ Inside Angle dictates the direction of the fillet. You may create inside or outside fillets.

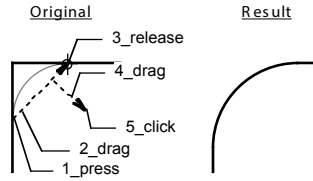


- ☐ Interactive means the fillet is not a predetermined size; you will determine the size of the radius by dragging. If Interactive is not selected, you type in the radius of the fillet.

The Trim Options determine the outcome of the fillet area:

- ☐ No Trim inserts a radius between the two objects and leaves the original lines intact.
- ☐ Trim Only inserts a radius between the two lines and cuts the original lines at the tangent intersection of the lines and the fillet. This allows you to delete all or part of the line segments between the fillet and the original line intersection.
- ☐ Trim and Delete inserts a radius between the two lines and cuts the original lines at the tangent intersection of the lines and the fillet. In the same operation, the trimmed line segments are deleted. The sharp line intersection is converted to a smooth round corner. Trim works with circles, but no pieces are deleted. The arc or circle is cut at the tangent point.

After setting the Fillet defaults, they remain in effect until you change them again. The Fillet tool will remain selected until you select another tool or deselect by clicking on a blank area of the drawing.



To use the Fillet tool:

- ☐ Select the Fillet tool from the Tool palette.
- ☐ Place the cursor on the first of the two lines to be connected by a fillet, press the mouse button.
- ☐ Drag to the second line. A blue construction line will connect the two lines to indicate the tool is active. Release the mouse button. A small fillet will appear in the corner of the intersection.
- ☐ After releasing the mouse button, move the mouse to drag the fillet to the appropriate size. Once an appropriate size has been reached, click to establish the fillet. If Interactive is not selected in the dialog box, the fillet will snap to the preset radius.

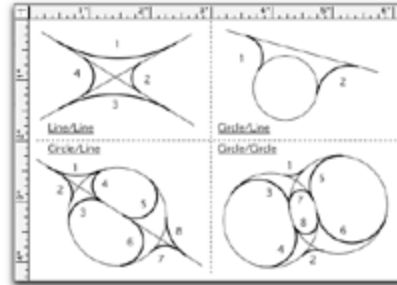
Fillet results in three separate objects: the two sides and the rounded corner. Each object remains selected for further modification.



NOTE: When filleting objects which are part of a Group, No Trim settings are automatically applied regardless of the Fillet tools default settings. Trim Only and Trim and Delete are not available. The arc resulting from the fillet operation is not added to the Group.

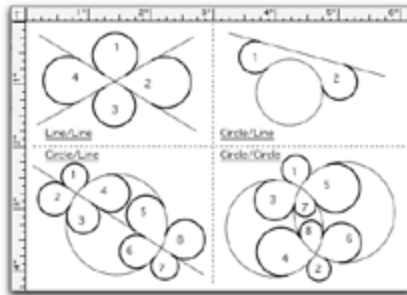
Fillet can be used repeatedly without preselecting objects. You establish a bridge line between two lines and go on to the next filleting operation.

The following is an example of the possible combinations of tangents for inside angles. Inside angles imply arcs with arc angles of less than 180 degrees.



The arcs or lines may be cut at the tangent point by setting trim on. Trim cuts, but does not throw away. Depending upon the desired end result, trimmed or cut arcs can later be reshaped, or alternately the unwanted arc can be deleted.

The following is an example of the possible combinations of tangents for outside angles. Arcs with outside angles have sweep angles greater than 180 degrees.



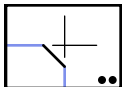
Editing a Fillet

To create a fillet of specific radius or diameter you may use the Edit Window as you are creating the fillet.

To edit a fillet on the fly:

- ☐ Make sure the Edit Window is active.
- ☐ Press the Option key and select the Fillet tool from the Tool palette.
- ☐ Click the Interactive radio button and click OK.
- ☐ Place the cursor on the first of the two lines, arcs or circles to be connected by a fillet, press the mouse button.
- ☐ Drag to the second line, arc or circle. A blue construction line will connect the two objects indicating the fillet operation is active.
- ☐ As you drag to define the fillet length, the Edit Window's radius and diameter fields display the current values.
- ☐ Tab to the radius or diameter field in the Edit Window.
- ☐ Type the desired radius of the fillet, then click anywhere in the drawing or press the Enter key.

A fillet of the specified radius or diameter will be inserted in the drawing.



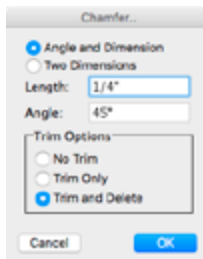
Chamfer Tool

This tool produces a beveled corner connecting two lines.

A chamfer can only be applied to two lines which intersect, or which will intersect if extended. Non-intersecting lines will automatically be extended if needed.

A Chamfer results in three separate objects: the two sides and the chamfer line. Each object remains selected for further modification.

To set the behavior of the tool, press the Option key and select the Chamfer tool from the Tool palette. The following dialog will appear:



Angle and Dimension allows you to create a chamfer that begins at a preselected distance from the intersection point on the last line created and extends out at a preselected angle to intersect with the second line.

Two Dimensions allows you to create a chamfer which begins at a preselected distance from the intersection point on the two lines. The dimensions you enter will specify how far from the intersection point the chamfer will begin on each line. Length One specifies how far from the intersection on the most recent line. Length Two specifies the distance from the intersection point on the second line. A chamfer line is then created between the two points.

Trim Options determine the outcome of the Chamfer tool.

No Trim places a chamfer between two lines and leaves the original lines intact.

Trim Only places a chamfer between two lines and cuts the original lines at the intersections of the original lines and the chamfer line. The trimmed line segments will not be deleted.

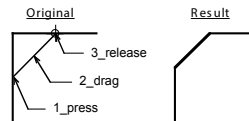
Trim and Delete places a chamfer between the two lines and removes the line segments that continue from the chamfer to the original intersection point. The sharp line intersection is converted into a beveled corner.

After setting the Chamfer defaults, they remain in effect until you change them again. Chamfer will also remain active until you select another tool or deselect by clicking on a blank area.

To use the Chamfer tool:

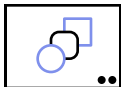
- ☐ Select the Chamfer tool from the Tool palette.
- ☐ Place the cursor on the first of two lines to be connected by a chamfer, press the mouse button.
- ☐ Drag the cursor to the second line. A blue construction line will connect the two lines to indicate the tool is active. Release the mouse button.

When the mouse button is released, the chamfer will be placed in the drawing.



When you select two lines by using the bridge line, the angle of the chamfer will measure the specified degrees from the most current line. The length specifies where the angle will begin with respect to the line intersection.

PowerCADD will find the most recent line of the two, measure the specified distance from the intersection and start the chamfer line at that point. The chamfer line is then locked to the specified angle and drawn until it reaches the second line.



Blend Objects Tool

This tool blends shapes and colors from one object to another in a specified number of steps. In addition to blending color attributes, the Blend Objects tool will also blend dash patterns, line weights, pen patterns and fill patterns.

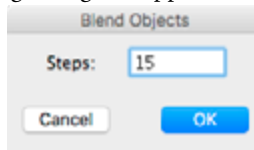
Blend objects may be used to add contour lines to a map or blend colors for a graduated fill for a variety of rendering effects.



NOTE: The Blend Objects tool is an External plug-in. In order for this tool to appear in the Tool palette it must be checked on when PowerCADD is launched. See [Loader Preferences](#) for more on turning External plug-ins On and Off.

To change the number of steps when blending objects:

- ☐ Press the Option key and select the Blend Objects tool from the Tool palette. The following dialog will appear:



- ☐ Enter the number of steps desired and click OK.

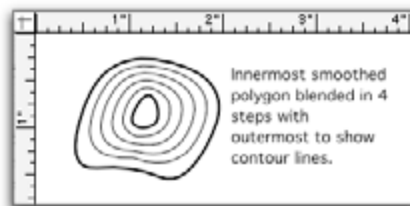
This will remain the default number of steps until you change it.

Blending Shapes

To blend shapes:

- ☐ Draw the starting object and the ending object to be blended.
- ☐ Select the Blend Objects tool.
- ☐ Select one of the two objects to be blended and drag a bridge line to the second object.

Blend Objects will place the steps between the first object drawn and the last object drawn.



Blending Colors

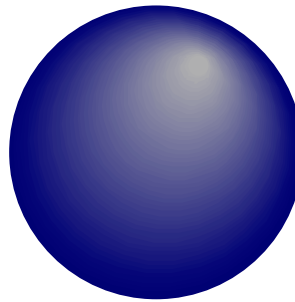
Blending colors is done by filling the starting object with one color and the ending object with another color. New objects are created which are a blend of the first object and last object's color. This is similar to how the Mix button works in the Color Edit dialog. The colors applied to blended objects are independent of the current drawing color palette.

To get a more realistic range of colors, specify more steps in the Blend Objects dialog or adjust the color value of the first or last objects in the blend and reapply the tool. The more shades between the starting color and ending color, the better final result will be.



NOTE: The Gradient Fill Tool may also be used to create complex gradient color blends.

Press Command-Period (.) to cancel a blend operation.



NOTE: If the first object selected to be blended includes a shadow, that shadow is applied to all blended objects. Opacity settings for object fill and pen attributes are also blended.

Blending Fill & Pen Patterns

Blending fill patterns is done by filling the starting object with one pattern and the ending object with another pattern. New objects are created which acquire their pattern based on the values in the fill and pen pattern palettes.



Paint Bucket Tool

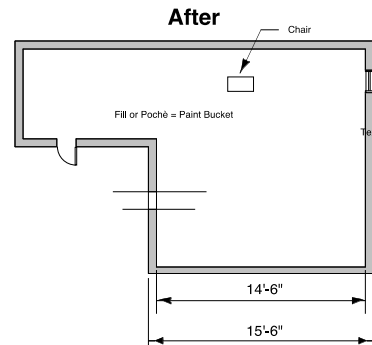
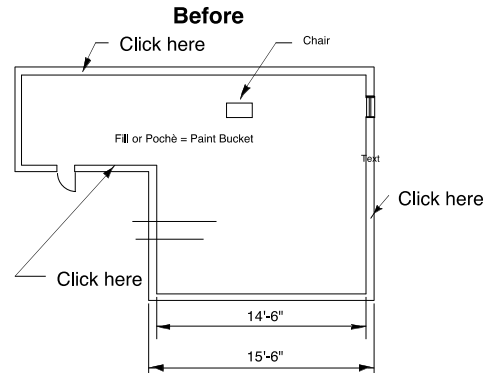
This tool fills any enclosed area of a drawing.

It is a quick way to trace around the bounds of an enclosed space and add a new filled polygon to the drawing. In many ways, it works like the Paint Bucket tool in most painting programs. The enclosed space may be created by individual lines, polygons, arcs, or Béziers. Dimensions, leaders lines, text and points are ignored.

If it is used on an area that is not enclosed, it will *leak*, filling all connected objects, including the background. Possible uses of the Paint Bucket tool include creating a poché filled wall effect, filling areas to perform quantity take-off's for area and perimeter calculations, or to create color renderings.

To use the Paint Bucket tool:

- ☐ Select the Paint Bucket tool from the Tool palette.
- ☐ Click in an enclosed area in the drawing.
- ☐ If you click in an incorrect area, Command-period will abort the fill operation.

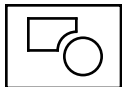


A polygon will be created that traces the edge of the enclosure and will be placed on the active layer.

You can also manually thin a polygon generated with the Paint Bucket tool. To manually thin a polygon (reduce the total number of points used to describe its shape), select the polygon then choose Thin Polygon from the [Polygon menu under the Tool menu](#). In some instances better thinning results can be achieved by applying the Distribute Points command before choosing Thin Polygon. See the [Tool Menu, Polygon Menu](#) for additional information.



NOTE: The Paint Bucket tool operates on the screen image, so the setting of Edit All Layers has no effect on the operation. With Edit All Layers On, the Paint Bucket makes a new polygon on the Active layer.



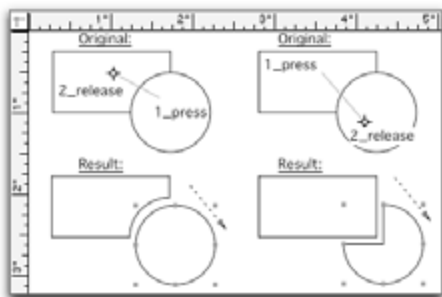
Clip Tool

This tool removes the area from one object it shares with an overlapping object. If the objects do not overlap, a warning beep will be issued.

Clipping, in this context, is the cutting or removing of one surface by another. The cutting action is similar to a cookie-cutter effect. The shape of the overlap dictates the shape of the cut on the lower surface. The Clip tool remains active until you select another tool or deselect by clicking on a blank space.

To use the Clip tool:

- ☐ Select the Clip tool from the Tool palette.
- ☐ Place the cursor on the first of the two objects that will perform the clipping action, press the mouse button.
- ☐ Drag the cursor to the second object. A blue construction line will connect the two objects to indicate the tool is active.
- ☐ When the mouse button is released, the clip will take place.



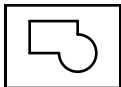
If two objects are arranged in such a fashion that the upper object fits entirely inside the lower object, clipping will not occur, and the tool will fall out.



CAUTION: The original object geometry (e.g. a circle) is always replaced with a new polygon reflecting the clipped shape. If it is important to keep the original object geometry, always work on a copy of the object before performing a clipping operation.



NOTE: If Edit All Layers is On and the objects are on two different layers, the new clipped object is created on the Active layer.



Combine Tool

This tool converts two overlapping objects into a single polygon. The resulting shape is the outline of the two parts.

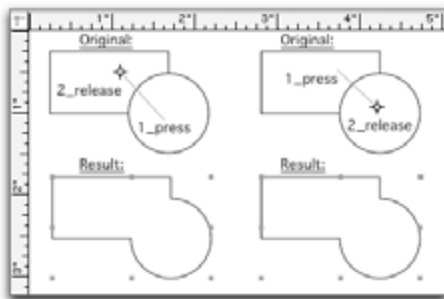
Combine is useful to create shapes which are difficult to draw directly. The Combine tool takes two items and turns them into one object with the geometry of the combined outline. The result is a polygon which can be reshaped or modified.

Objects to be combined must be contiguous or overlapping for Combine to take effect. If the objects do not overlap or if one object is grouped, the tool will fall out.

To use the Combine tool:

- ☐ Select the Combine tool from the Tool palette.
- ☐ Place the cursor on the first of two objects to be combined, press the mouse button.
- ☐ Drag the cursor to the second object. A blue construction line will connect the two objects to indicate the tool is active. Release the mouse button.

When the mouse button is released, the selected objects are combined into a single polygon.



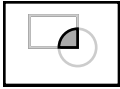
NOTE: When combining objects with different attributes (e.g. a blue rectangle and a red circle), the attributes of the first object clicked on are applied to the new combined object.



CAUTION: The original object geometry (e.g. a circle and rectangle) is always replaced with a new polygon reflecting the combined shape. If it is important to keep the original object geometry, always work on a copies of the objects before performing a combine operation.



NOTE: If Edit All Layers is On and the objects are on two different layers, the new object is created on the active layer.



Union Tool

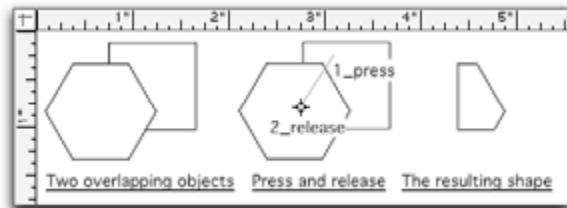
This tool creates an object from the overlapping part of two overlapping objects.

As opposed to the Clip tool, the Union tool keeps the overlapping area and removes all material except the overlapping area.

The objects to be combined must be contiguous or overlapping for Union to take effect. If the objects do not overlap or if one object is grouped, the tool will fall out.

To use the Union tool:

- ☐ Select the Union tool from the Tool palette.
- ☐ Place the cursor on the first of the two overlapping objects and press the mouse button.
- ☐ Drag the cursor to the second object. A bridge line will connect the two objects to indicate the tool is active.
- ☐ Release the mouse button. The resulting object is the overlapping portion of the two initial objects.



NOTE: When objects have different attributes (e.g. a blue rectangle and a red circle), the attributes of the first object clicked on are applied to the new combined object.



CAUTION: The original object geometry (e.g. a circle and rectangle) are always replaced with a new polygon reflecting the combined union shape. If it is important to keep the original object geometry, always work on a copies of the objects before performing a union operation.



NOTE: If Edit All Layers is On and the objects are on two different layers, the Combined object is created on the Active layer.



Bitmap Effect Tool

This tool allows you enhance bitmap graphics in your drawings with a variety of QuickTime special effects.

The Bitmap Effect tool can only be applied to bitmap images. To apply bitmap effects to normal PowerCADD object geometry, use the Object to Bitmap menu item in the Tool menu to convert any PowerCADD object or group of objects into a bitmap before applying the Bitmap Effect tool. The Bitmap Effect tool requires two bitmap images and creates a combined image based on the settings applied.

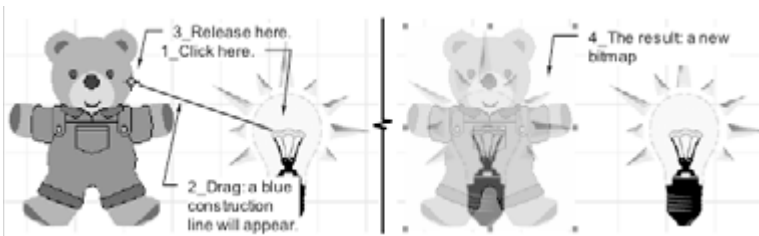


NOTE: The Bitmap Effect tool is part of the Bitmap Utils External plug-in. In order for this tool to appear in the Tool palette Bitmap Utils must be checked on when PowerCADD is launches. See [Loader Preferences](#) for more on turning External plug-ins On and Off.

To use the Bitmap Effect tool:

- ☐ Select the Bitmap Effect tool from the Tool palette. The cursor will change to a target cursor.
- ☐ Place the cursor on the first bitmap image and press the mouse button (the images do not need to overlap).
- ☐ Drag the cursor to the second bitmap object. A blue construction line will connect the two bitmaps to indicate the tool is active.
- ☐ Release the Mouse button.
- ☐ The QuickTime effects dialog box will open. Select the desired effect and click OK. A preview window allows you to sample various effects before clicking OK.

The second bitmap is replaced with a new bitmap.



NOTE: The Bitmap Effect tool requires a full installation of Apple's QuickTime software on your computer however Apple has been making many changes and has introduced a number of new technologies to replace these capabilities. Apple's Core Image technology is now widely used in the latest MacOS as well as on the iPhone and iPad. The Core Image Filter tool in PixelTools (in WildTools) provides modern replacements for these bitmap effects.

The Bitmap Effect tool supports full color, gray scale or black and white bitmap images as source objects. Possible uses of Bitmap Effect include simulating window or water reflections in renderings, perform image transitions and compositions or adjusting the opacity of an object.



NOTE: If the two bitmaps are different sizes, the first bitmap selected will be scaled to match the size of the second bitmap selected.



Bitmap Transformation Tool

This tool maps any bitmap to any four-sided closed polygon. The polygon may be in any projection plane including isometric or perspective.

The Bitmap Transformation tool can only be applied to any full color, gray scale or black & white bitmap images. To apply Bitmap Transformation tool to normal PowerCADD object geometry, use the [Object to Bitmap](#) in the Tool menu to convert any PowerCADD object or group of objects into a bitmap first.

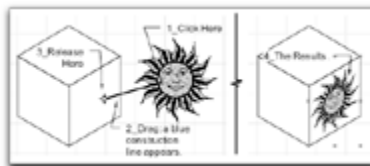


NOTE: The Bitmap Transformation Tool is part of the Bitmap Utils External plug-in. In order for this tool to appear in the Tool palette Bitmap Utils must be checked on when PowerCADD is launches. See [Loader Preferences](#) for more on turning External plug-ins On and Off.

To use the Bitmap Transformation tool:

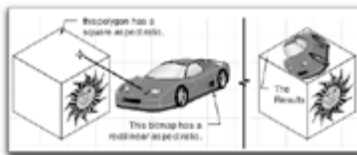
- ☐ Select the Bitmap Transformation tool from the Tool palette. The cursor will change to a target cursor.
- ☐ Place the cursor on the bitmap image and drag to a four-sided polygon. A blue construction line will connect the two objects to indicate the tool is active.
- ☐ Release the Mouse button.

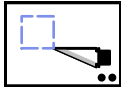
The bitmap is mapped to match the proportions of the polygon.



The aspect ratio of the bitmap is modified to match the aspect ratio of the polygon. If the aspect ratio's of the bitmap and polygon are significant, the bitmap may appear to be distorted as illustrated here. This is normal behavior given the different aspect ratios of the objects.

In addition to creating interesting visual effects, the Bitmap Perspective tool can also be a significant time saving device. For example, digital or scanned photographs may be mapped onto a polygon drawn in perspective as a backdrop for design and presentation drawings. This technique saves drawing time and provides a much more dynamic presentation to the client.





Fence Cut Tool

This tool allows you to delete or copy information in a specific area of the drawing. The Fence Cut operation can delete the data inside or outside the fenced area or copy the fenced area to the clipboard.

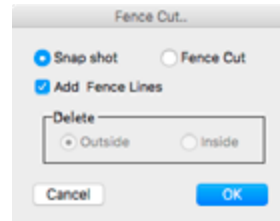
Performing a Fence Cut can be a fast and efficient means to use existing drawing content to create new drawing when only a portion of the original drawing is required.



NOTE: The Fence Cut Tool is an External plug-in. In order for this tool to appear in the Tool palette it must be checked on when PowerCADD is launches. See [Loader Preferences](#).

To set the behavior of the tool:

- ❑ Press the Option key and select the Fence Cut tool from the Tool palette. The following dialog will appear:



Snap Shot: Copies the portion of the drawing enclosed by the fence into the Clipboard, the original information in the drawing is not modified. Layer information is not retained; objects on different layers will appear on the active layer when pasted. Once in the Clipboard, the information may be pasted into a new drawing or into another location in the same drawing. If Add Fence Lines was checked on, a new rectangle is added matching the fenced area.

Fence Cut: A Fence Cut is used in conjunction with Delete Outside or Delete Inside to remove information from the drawing. Layer information is not retained; objects on different layers will appear on the active layer when pasted. If Add Fence Lines was checked, a new rectangle is added in the drawing matching the fenced area.

Snap Shot and Fence Cut settings retain scale and layer information. Use Paste Special, At Scale, Through Layers, to automatically rescale the Fence Cut or Fence Snap Shot data when pasting into a new or existing drawing. Layer Scale settings take precedent over Drawing Setup scale when performing the Fence Cut or Snap Shot operation.

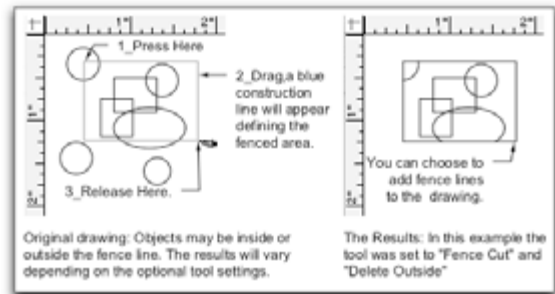
Add Fence Lines: Used in conjunction with Snap Shot or Fence Cut, when checked, a rectangle is added defining the fenced area. This can be particularly helpful when using Snap Shot if the operation needs to be repeated again after a drawing revision is made.

Delete: Used in conjunction with Fence Cut, there are two options for deleting objects from the drawing. Delete Outside deletes everything outside the fenced area on the visible layers. Delete Inside deletes everything inside the fenced area on visible layers.

To use the Fence Cut tool:

- ☐ Select the Fence Cut tool from the Tool palette. The cursor will change to a knife cursor.
- ☐ Hold the mouse button down and drag to draw a rectangle (fence) defining the area you want the Fence Cut tool to be applied.
- ☐ Release the mouse.

The Fence Cut operation is completed using your specified settings.

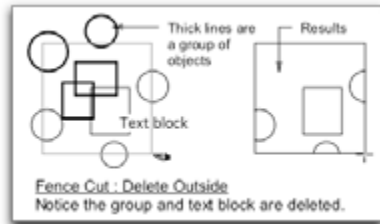
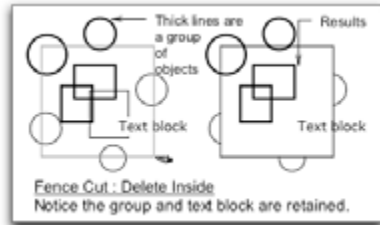
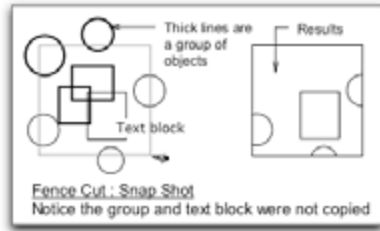


Edit All Layers

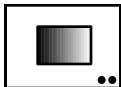
The Fence Cut tool is smart and works in conjunction with Edit All Layers being On or Off. If Edit All Layers is On, all visible information is affected by the Fence Cut tool as per the specified settings. If Edit All Layers is Off only the active layer is affected by the Fence Cut tool. This provides a great deal of drawing management flexibility.

Symbols, Groups and Text Objects

There are unique conditions which apply when performing a Fence Cut or Snap Shot on groups, symbols or text with the Fence Cut tool as illustrated below.



NOTE: A Save or Save As is recommended on a drawing prior to performing a Fence Cut Delete Inside or Delete Outside operation. While Undo may be applied after a Delete Inside or Outside operation, the results will vary depending on how many objects were affected by the Fence Cut operation and how many Undo Levels you have specified for the Undo menu item.



Gradient Tool

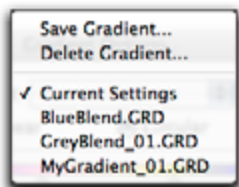
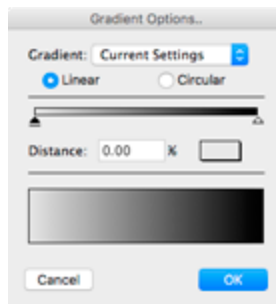
This tool allows you to enhance your drawing with shaded linear or circular gradient fills. Gradients may be applied to any PowerCADD object including Béziers, circles, arc, ellipses and rectangles.

NOTE: The Gradient Tool is an External plug-in. In order for this tool to appear in the Tool palette it must be checked on when PowerCADD is launches. See [Loader Preferences](#) for more on turning External plug-ins On and Off.

The Gradient tool has optional user control settings to specify the colors and type of gradient being applied. Any changes made in the Gradient Options dialog are remembered until you change them.

To set the behavior of the tool:

- ❑ Press the Option key and select the Gradient tool from the Tool palette. The following dialog will appear.



Current Settings: The current settings pop-up menu allows you to save and recall gradient templates. Using gradient templates is an efficient way to organize all the components of a particular gradient profile for reuse in any drawing. To use a previously saved gradient, simply choose it from the list and the changes will be reflected in the Gradient Dialog. To save a new gradient template, simply choose Save Gradient from the list, name the item and click OK. To delete a template, Choose Delete Gradient from the list, choose the template name and click Delete.

Circular: Click the Circular radio button to apply a circular gradient to an object. The center point of the gradient may be modified in the Preview Pane or in the drawing after the gradient has been applied to an object.

Linear: Click the Linear radio button to apply a linear gradient to an object. The angle of the gradient may be modified in the Preview Pane or in the drawing after the gradient has been applied to an object.

Color Point: Color Points are the building blocks of a gradient profile.

Active Color Points are indicated by a solid black triangle, inactive Color Points are white triangles. To activate a Color Point, click on it, then drag left or right to change its position and modify the gradients profile. To remove a Color Point, press on the color point, drag downward and release the mouse or press the Delete key. To add a Color Point, click anywhere on the Color Bar. To preview the effects of adding or removing a Color Point press on a color point then drag it on and off the Color Bar, the Preview Pane will update to reflect the changes.

Color Bar: The Color Bar is used to add, remove or modify the position of color points used to create a gradient profile. You may have as many color points as you like on the Color Bar. As new Color Points are added, the Color Bar and Preview Pane reflect those changes.

Distance: The Distance Field reflects the horizontal position of a selected Color Point on the Color Bar, allowing you to accurately specify a Color Point position in the gradient profile. When specifying a Color Point's position, 0% is the absolute left of the Color Bar, 100% is the absolute right. To specify a specific position for a Color Point, select the Color Point then type the specific value in the Distance Field.

Color Square: The color of the currently selected Color Point is shown in the Color Square. To change the color of the selected Color Point, click on the Color Square. The MacOS color panel will be displayed allowing you to specify a color from any standard Mac OS color palette or to select a color from any item visible on screen.

Preview Pane: Any change made to a gradient profile is immediately update in the Preview Pane. To set the center point for a circular gradient, click anywhere in the Preview Pane. To set the angle for a linear gradient, click and drag the mouse in the Preview Pane.

Removing Gradients

To remove a gradient applied to any object, select the object and choose a pattern from the Fill Pattern menu in the Attr menu. To change a gradient from Linear to Circular or Circular to Linear, assign the object a new fill pattern from the Attr menu first, then apply the new gradient fill.

Editing Gradients



To edit the gradient profile currently applied to an object:

- ☐ Select the Gradient tool from the Tool palette. The cursor will change to a target cursor with two dots.
- ☐ Press the Option key and click the mouse on the gradient filled object.
- ☐ The Gradient Options dialog will open.
- ☐ Adjust the gradient profile as desired and click OK.
- ☐ The object will reflect the new gradient fill settings.

To edit the angle of a linear gradient currently applied to an object:

- ☐ Select the Gradient tool from the tool palette.
- ☐ Click and drag on the linear filled gradient object.
- ☐ The angle you specify during the drag operation will specify the new angle. Press the Shift key to have the gradient conform to the preset snap angles.
- ☐ Release the Mouse button.

To edit the center point of a circular gradient currently applied to an object:

- ☐ Select the Gradient tool from the Tool palette.
- ☐ Click inside the circular filled gradient object.
- ☐ The location you click on will be the new center of the circular gradient.

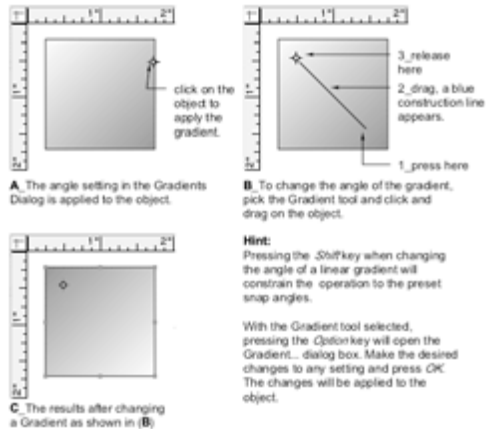


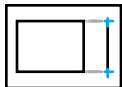
NOTE: Gradient templates are saved in the User folder: Library: Preferences: PowerCADD: Gradients Folder. You may share gradient templates with others simply by copying the appropriate file in the Gradients Folder.

To use the Gradient tool:

- ☐ Select the Gradient tool from the Tool palette. The cursor will change to a target cursor.
- ☐ Click the mouse on the object to receive the gradient fill.

The object is filled with the specified gradient.





Move Points Tool

This tool allows you move all selected vertices in a selected area to a new location.

One of the primary advantages of the Move Points tool is its ability to save you time by moving selected points of existing geometry as opposed to deleting the objects and redrawing. For example, using the Move Points tool it is possible adjust the length of all the selected walls in a drawing, including their associated dimensions, and move them together to increase or decrease their length or change their angle.

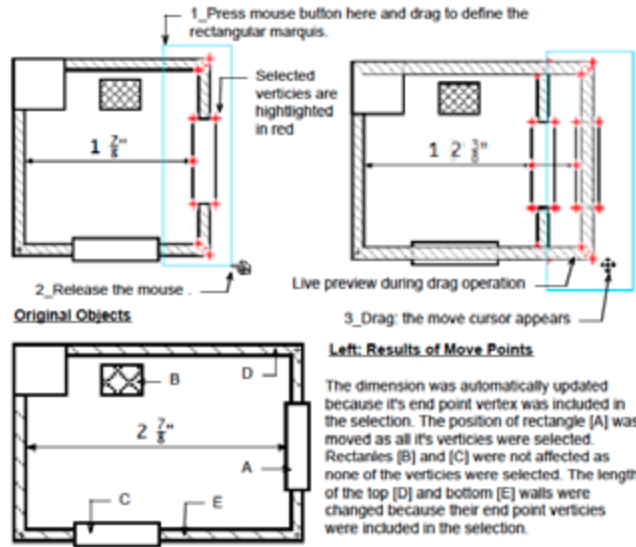
Points may be moved any arbitrary length and angles, or specific lengths and angles using the mouse or a combination of the mouse and a dialog box. Snap commands may also be used to define the start or end point of a Move Points operation.



NOTE: The Move Points Tool is an External plug-in. In order for this tool to appear in the Tool palette it must be checked on when PowerCADD is launches. See [Loader Preferences](#) for more on turning External plug-ins On and Off.

To use the Move Points tool:

- ❑ Select the Move Points tool from the Tool palette. The cursor will change to a finger cursor.
- ❑ Click and drag to draw a rectangular marquee which encloses the vertices to be moved and release the mouse.
- ❑ Selected vertices are highlighted in red and the cursor changes to a move cursor.
- ❑ Drag the mouse to move the selected vertices. Press the Shift key to constrain the move to the X or Y axis.
- ❑ Release the mouse to complete the move points operation.

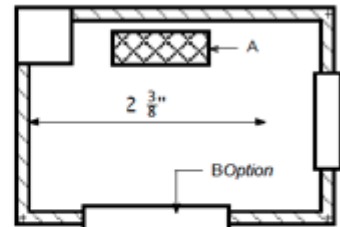
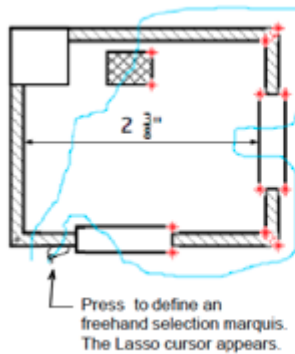


Move Points Freehand Selection

In a drawing containing many objects it may not be practical to select the vertices to be moved using a rectangular selection.

To make a freehand selection using the Move Points tool:

- ☐ Select the Move Points tool from the Tool palette. The cursor will change to a finger cursor. Press the Option key, the cursor will change to a lasso.
- ☐ Click and drag to draw a freehand marquee which encloses the vertices to be moved and release the mouse.
- ☐ Selected vertices are highlighted in red and the cursor changes to a move cursor.
- ☐ Drag the mouse to move the selected vertices. Press the Shift key to constrain the move to the X or Y axis.
- ☐ Release the mouse to complete the move points operation.



The Results:

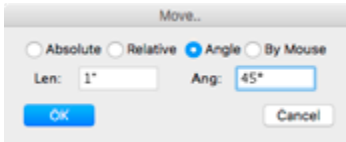
The dimension was NOT updated because it's end point was not included in the freehand selection. Rectangles [A] and [B] are longer because their end points were selected.

Move Points a Specified Distance

There are instances where you may know the precise length or angle the selected points should be moved. For example, a building wall may need to be moved precisely 6" to the right or perhaps an entire parking lot may need to be moved 11.25 meters up or down.

To move selected points a specified distance and direction using the Move Points tool:

- ☐ Select the Move Point tool from the Tool palette. The cursor will change to a finger cursor. Press the Option key and the cursor will change to a lasso.
- ☐ Click and drag to draw a marquee which encloses the vertices to be moved and release the mouse.
- ☐ Selected vertices are highlighted in red and the cursor changes to a move cursor.
- ☐ Begin dragging the mouse to move the points, then press the Tab key.
- ☐ The following dialog box will appear



The Move dialog provides four move methods:

Absolute: allows you to specify the absolute X,Y coordinate location.

Relative: moves objects the specified amount relative to their current X,Y location.

Angle: moves objects the specified length and angle.

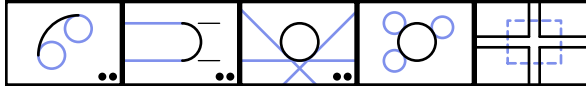
By Mouse: moves objects dynamically using the mouse and dragging on screen.

- ☐ Enter the specific values and click OK.



NOTE: To move the vertices of objects which are different layers using the Move Points tool, open the Layers Window and select the Edit All Layers checkbox.

Arc Tools and Wall Heal/Join



NOTE: All tools in this row of the tool palette are External plug-ins. In order for Arc Tools tool to appear in the Tool palette the Arc Functions External must be checked on when PowerCADD is launches. The Wall Heal/Join tool requires the Wall Tools External to be loaded. See [Loader Preferences](#) for more on turning External plug-ins On and Off.

Arc Tools provide specialized tools that extend the capability of PowerCADD for mechanical engineers, industrial designers or anyone who works with circles, circular arcs or arc tangent geometry. They appear as the last row of tools in the main Tool palette.

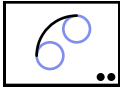
For the purpose of describing the arc tools, circles and circular arcs will subsequently be referred to as *arcs*. Elliptical arcs are not included in the reference to arcs.

Arcs can be drawn tangent to:

- ☐ Two arcs (Tangent Arc tool)
- ☐ Two parallel lines (Slot tool)

Circles can be drawn tangent to:

- ☐ Three arcs (Three-Circle Circle tool)
- ☐ Three lines (Three-Line Circle tool)

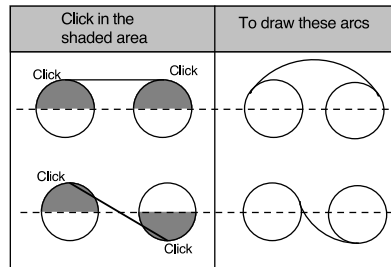


Tangent Arc Tool

This tool draws an arc tangent to two arcs.

A simple technique is used to indicate where the tangent arc should intersect the arcs. Imagine a line drawn through the centers of the arcs. The line divides each arc into two sections. To draw a tangent arc intersecting an arc in a particular section, simply click that section.

The following examples show how this works:

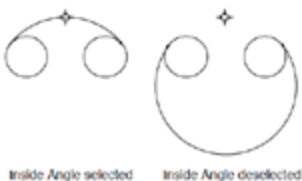
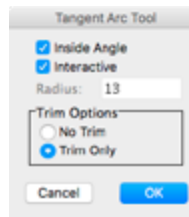


With filled arcs, click anywhere in the section. With unfilled arcs, select the perimeter of the arc.

The radius of the arc can be specified with the keyboard or interactively by dragging the mouse. There is also an option of arcs greater or less than 180° .

To set the behavior of the tool:

- ☐ Press the Option key and select the Tangent Arc tool from the Tool palette. The following dialog will appear:

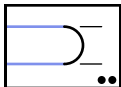


- ☐ Select Inside Angle to place arcs of less than 180° and deselect for arcs of more than 180° .
- ☐ If you know what the radius should be, deselect Interactive and enter the radius. If Interactive is selected, the radius of the ghost arc is changed by moving the mouse.
- ☐ When you enter a radius and only one position for an arc of that radius is possible, the tangent arc is placed immediately after you select the arcs. If more than one position is possible, the ghost arc snaps to each position as you move the mouse.

- ☐ There are two options for trimming: Trim Only or No Trim. Trim Only cuts both original arcs at the intersection points. If No Trim is selected, the arcs will be left intact.

To use the Tangent Arc tool:

- ☐ Select the Tangent Arc tool from the Tool palette.
- ☐ Select the first arc by clicking one of its two sections. A bridge line connects the arc with the cursor indicating the tool is active.
- ☐ Select the second arc by clicking one of its sections. When the arc has been selected, a ghost tangent arc appears.
- ☐ Move the cursor to adjust the arc's radius and position.
- ☐ Click to place the arc.

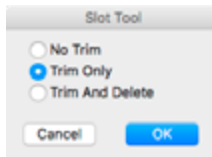


Slot Tool

This tool draws a circular arc tangent to two parallel lines, creating a slot.

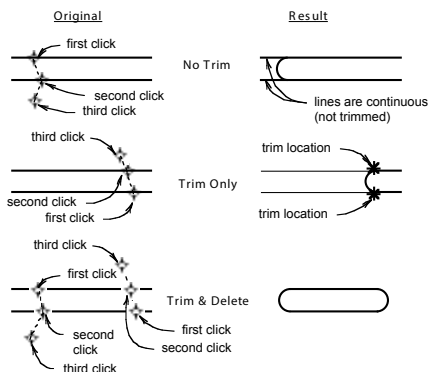
To set the behavior of the tool:

- ❑ Press the Option key and select the Slot tool from the Tool palette. The following dialog will appear:



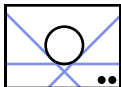
- ❑ There are three options for trimming: No Trim, Trim Only, and Trim And Delete.
- ❑ If No Trim is selected, the lines will remain intact.
- ❑ Trim Only cuts the lines at their arc intersections or extends the lines to the arc intersections.
- ❑ Trim And Delete cuts the lines and deletes the line segments forming a slot.

The following illustration shows an example of Trim And Delete.



To use the Slot tool:

- ❑ Move the cursor over the first of two lines defining the parallel lines which will create the slot. The target cursor will appear.
- ❑ Click on the first line and drag to the second line. A blue bridge line appears connecting the two points.
- ❑ Click on the second line; a ghosted circular arc appears.
- ❑ Drag the mouse left and right to position the arc horizontally. Drag the mouse up and down to set the direction of the arc.
- ❑ When the arc is in the desired location, click; a circular arc is inserted which is tangent to the two parallel lines.
- ❑ Repeat the process for the other end of the slot.

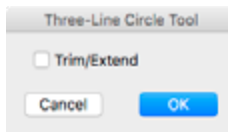


Three-Line Circle Tool

This tool draws a circle tangent to three lines.

To set the behavior of the tool:

- ☐ Press the Option key and select the Three-Line Circle tool from the Tool palette. The following dialog will appear:

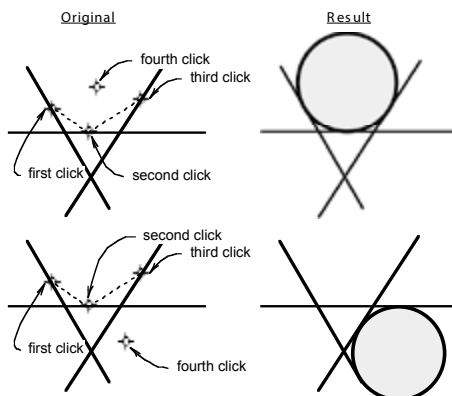


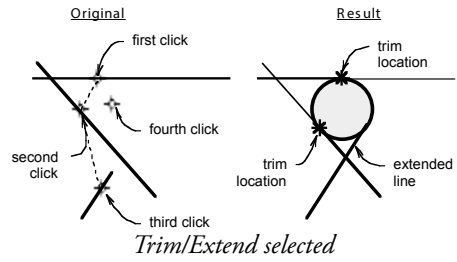
- ☐ Select the Trim/Extend check box to trim or extend the lines at their intersections with the circle.
- ☐ Click OK.

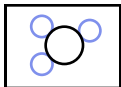
To use the Three-Line Circle tool:

- ☐ Select the Three-Line Circle tool from the Tool palette.
- ☐ Move the cursor over the first of three lines required to define the tangent surfaces for the circle. The target cursor will appear.
- ☐ Click on the first line and drag to the second line. A blue bridge line appears connecting the two points.
- ☐ Click on the second line and drag to the third line. A blue bridge line appears connecting all three points.
- ☐ Click on the third line; a ghosted circle appears.
- ☐ Move the mouse until the ghosted circle is in the desired quadrant and click.
- ☐ A circle is inserted which is tangent to the three selected lines.

A circle is inserted which is tangent to the three selected lines. Lines which were too short are extended. Lines which were long enough are trimmed where the line and circle intersect.





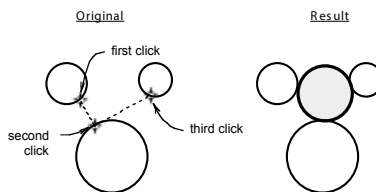


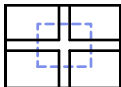
Three-Circle Circle Tool

This tool draws a circle tangent to three circles.

To use the Three-Circle Circle tool:

- ❑ Select the Three-Circle Circle tool from the Tool palette.
- ❑ Move the cursor over the first of three circles required to define the tangent surface for the circle. The target cursor will appear.
- ❑ Click on the first circle and drag to the second circle. A blue bridge line appears connecting the two points.
- ❑ Click on the second circle and drag to the third circle. A blue bridge line appears connecting all three points.
- ❑ Click on the third circle.
- ❑ A circle is inserted which is tangent to the three selected circles.





Heal/Join Tool

This tool cleans up and edits a variety of lines and intersections.

It provides a simple and elegant way to edit, manipulate, and clean up parallel lines.

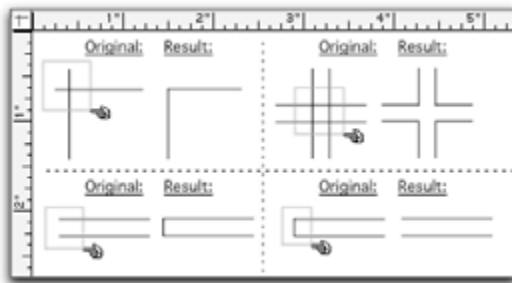
The Heal/Join tool can be used in a variety of situations and has the intelligence to determine the action needed. Diagonal lines, as well as horizontal and vertical lines, may be healed.

There is no need to pre-select an option, but there are five basic cleanup categories:

- ☐ Extend or trim T, X, and L intersections.
- ☐ Extend or trim lines.
- ☐ Join lines or heal lines.
- ☐ Remove or add caps.
- ☐ Clean up T, X, and L intersections.

To use the Heal/Join tool:

- ☐ Select the Heal/Join tool from the Tool palette.
- ☐ Draw a selection marquee around the area to be manipulated.
- ☐ Continue selecting areas or click to quit.



NOTE: Heal/Join does not work on ambiguous selections.



NOTE: The Wall tool external must be loaded for the Heal/Join tool to appear in the main Tool palette.

The following chart illustrates some of the ways in which the tool can be used. There are three columns of ‘before’ and ‘after’ examples. Carefully examine the selected area in each Action box and compare it with the adjacent Result box.

Wall Trim/Heal Tool - Chart of Selections

Action	Result	Action	Result	Action	Result
T Intersections		Extend/Trim Lines		L Intersections	
				No Results	
		Caps On/Off			
Heal/Join				X Intersections	

External Tool Palettes

This section describes the external tool Palettes. Before proceeding, you should become familiar with the basics of PowerCADD. See [PowerCADD Tutorial](#).

External tool palettes are separate floating palettes which contain one or more related tools.

The external tool palettes are completely separate from the Tool palette and appear floating on the PowerCADD screen. Each palette is a collection of tools designed for special needs or tasks. Because these External are separate from PowerCADD, they can be loaded or unloaded by choosing [Loader Preferences](#) from PowerCADD Preferences.

External Tool Palettes include:

- ☐ Mechanical Tools (requires the Mechanical External to be loaded).
- ☐ Wall Tools (requires the Wall Tools External to be loaded).
- ☐ Notation Tools (requires the Notations External to be loaded).

Before describing each External Tool Palette in detail, it is important to be aware of the following fundamental concepts, which apply to all External Tool Palettes:

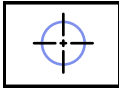
- ☐ For an External Tool Palette to be available its corresponding External Plug-in must be loaded.
- ☐ The orientation of an external tool palette may be adjusted from vertical to horizontal by Control- or right-clicking on the External tool palette, and choosing Change Palette Orientation from the contextual menu.
- ☐ The settings for each palette may be customized by pressing Control- or right-clicking on the External tool palette, and choosing Palette Settings from the contextual menu. [For a complete description](#) of all Palette Settings reference the detailed diagram at the start of the Tools chapter.
- ☐ Use the Commands tab in the Preferences dialog to assign custom key commands to any External tool.
- ☐ The size of all tool palettes may be adjusted using the [View tab in the PowerCADD Preferences dialog](#). [For a complete description](#) reference the detailed steps discussed at the start of the Tools chapter.

Mechanical Tool Palette

The Mechanical Tool palette provides specialized tools for mechanical drafting and design, extending the capability of PowerCADD for mechanical engineers.

The Mechanical Tool palette consists of seven tools:

- ☐ Centerline
- ☐ S-Break
- ☐ Drilled Hole
- ☐ Tapped Hole
- ☐ Top View Tapped Hole
- ☐ Hex Bolt
- ☐ Hex Nut



Centerline Tool

This tool places a centerline on objects. Centerlines can be placed on circles, ellipses, arcs, rectangles, rotated objects, etc. Centerlines can also be placed on objects such as hex nuts, objects within groups, and the corners of rounded rectangles.

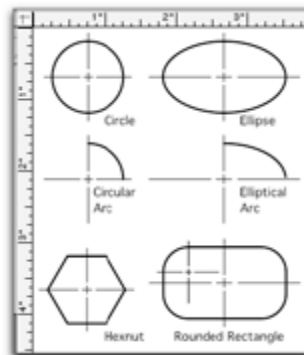
A centerline can be modified by ungrouping it. If there is not enough room in the drawing for a full centerline, part of the centerline can be deleted.

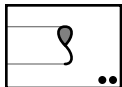
To use the Centerline tool:

- ☐ Select the Centerline tool from the Mechanical Tool palette.
- ☐ Place the cursor on the object on which the centerline is to be placed. Unfilled objects may be selected on their perimeter.
- ☐ Click, and the centerline appears.

To place a centerline on the corner of a rounded rectangle, place the cursor on the rectangle and press. Drag the centerline to the corner and release.

The Centerline tool remains selected until you click in a blank space or select another tool. The tool creates the correct size centerline for each object selected.





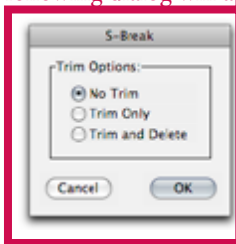
S-Break Tool

This tool draws an S-break.

The S-Break tool can be used to add an S-break to a shaft, pipe, or other cylindrical piece.

To set the behavior of the tool:

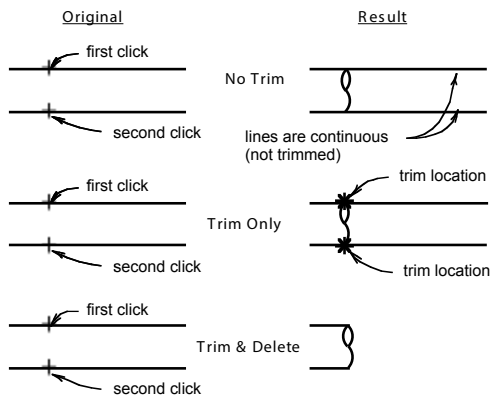
- ❑ Press the Option key and select the S-Break tool from the Mechanical Tool palette. *The following dialog will appear: (it does not)*



- ❑ There are three options for trimming: No Trim, Trim Only, and Trim and Delete.
- ❑ If No Trim is selected, the lines of the shaft are not changed.
- ❑ Trim Only cuts the lines at the break.
- ❑ Trim and Delete cuts the lines and deletes the line segments beyond the break. When set to Trim Only or Trim and Delete, the attributes assigned to the S-Break tool are applied to the remaining lines of the shaft.

To use the S-Break tool:

- ❑ Select the S-Break tool from the Mechanical Tool palette.
- ❑ Place the cursor on the outside of the shaft.
- ❑ Drag to the other side and release.

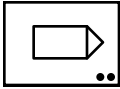


NOTE: Using Perpendicular (P) snapping to start the drawing operation and Surface (S) snapping to end the drawing operation is an efficient way to draw S-Breaks accurately between any two parallel lines regardless of their angles.



S-breaks can be placed on either end of a shaft. To place a break on the right end of a horizontal shaft, begin the break at the top of the shaft. To place a break on the left end of a horizontal shaft, begin the break at the bottom of the shaft.

The S-Break is constrained to the preset snap angles. Press the Shift key to release the constraint.



Drilled Hole Tool

This tool draws multiple objects depicting a drilled hole. The resulting drilled hole object is a group

Drilled holes can be drawn by

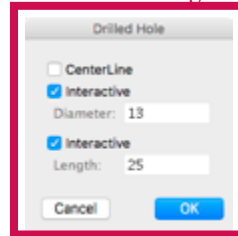
- ☐ Dragging with the mouse.
- ☐ Entering the length or diameter, or both.

Both blind and through holes can be drawn. The Drilled Hole determines from your actions when to draw a through hole. In addition, centerlines can be drawn along with the hole.

When a dashing default for the tool has been set, hidden views are drawn.

To set the behavior of the tool:

- ☐ Press the Option key and select the Drilled Hole tool from the Mechanical Tool palette. The following dialog will appear: (Not!)

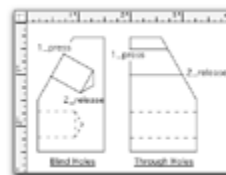


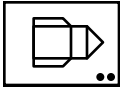
- ☐ Select Centerline to place a centerline along with the hole.
- ☐ Interactive means the drilled hole is not a predetermined size; you will determine the size of the diameter and length by dragging. If the Interactive boxes are not checked, you enter the diameter and length.

To use the Drilled Hole tool:

- ☐ Select the Drilled Hole tool from the Mechanical Tool palette.
- ☐ Place the cursor at the center of the hole.
- ☐ Drag to the hole depth and width.
- ☐ Release when the appropriate size is reached.

To draw a through hole, drag until the cursor is directly over a surface and release. PowerCADD automatically recognizes the difference between a blind hole and a through hole. To snap the hole perpendicular to a surface, press and hold the P key (Perpendicular Snapping) when starting the hole.





Tapped Hole Tool

This tool draws multiple objects depicting a tapped hole. The resulting tapped hole object is a group.

Tapped holes can be drawn by:

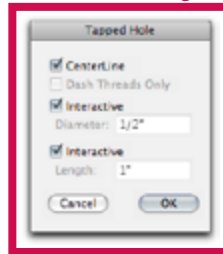
- ☐ Dragging with the mouse.
- ☐ Entering the length or diameter, or both.

The length is the depth of the thread. The diameter is the thread diameter. Both blind and through holes can be drawn. The Tapped Hole determines from your actions when to draw a through hole. In addition, centerlines can be drawn along with the hole.

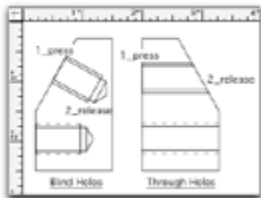
When a dashing default for the tool has been set, hidden or section views are drawn. For section views, the tool dashes the thread form only.

To set the behavior of the tool:

- ☐ Press the Option key and select the Tapped Hole tool from the Mechanical Tool palette. **The following dialog will appear: (Not!)**



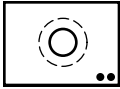
- ☐ Select Centerline to place a centerline along with the hole.
- ☐ Select Dash Threads Only to dash only the thread form. This option is only available when the tool has been set for dashed lines.
- ☐ Interactive means that the tapped hole is not a predetermined size; you will determine the size of the diameter and length by dragging.
- ☐ If Interactive is not checked, you enter the diameter and length.



To use the Tapped Hole tool:

- ☐ Select the Tapped Hole tool from the Mechanical Tool palette.
- ☐ Place the cursor at the center of the hole.
- ☐ Drag to the thread depth.
- ☐ Release when the appropriate size is reached.

To draw a through hole, drag until the cursor is directly over a surface and release. To draw a hole from an angled surface, use Perpendicular snap when starting the hole.



Tapped Hole Top Tool

This tool draws a top view of a tapped hole.

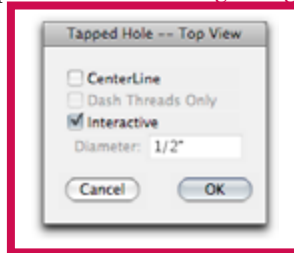
Top view holes can be drawn by

- ☐ Dragging with the mouse.
- ☐ Entering the diameter.

The diameter is the thread diameter, or the bolt size.

To set the behavior of the tool:

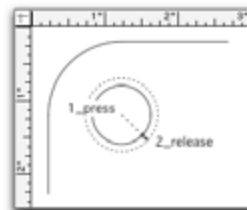
- ☐ Press the Option key and select the Tapped Hole Top tool from the Mechanical Tool palette. **The following dialog will appear: (Not!)**

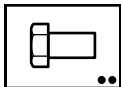


- ☐ Select Centerline to place a centerline along with the hole.
- ☐ Select Dash Threads Only to dash the thread form only. This option is only available when the tool has been set for dashed lines.
- ☐ Interactive means that the tapped hole is not a predetermined size; you will determine the size of the diameter by dragging.
- ☐ If Interactive is not checked, enter the diameter in the dialog.

To use the Top View Tapped Hole tool:

- ☐ Select the Top View Tapped Hole tool from the Mechanical Tool palette.
- ☐ Place the cursor at the center of the hole.
- ☐ Drag to the outside diameter.
- ☐ Release when the appropriate size is reached.





Hex Bolt Tool

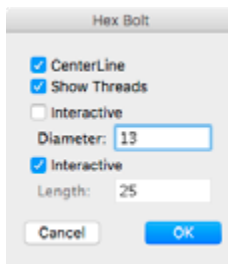
This tool draws a hex bolt.

Hex bolts can be drawn by

- ☐ Dragging with the mouse.
- ☐ Entering the length or diameter, or both.

To set the behavior of the tool:

- ☐ Press the Option key and select the Tapped Hole Top tool from the Mechanical Tool palette. The following dialog will appear:



	Show Threads selected	Show Threads deselected
Dashing		
No dashing		

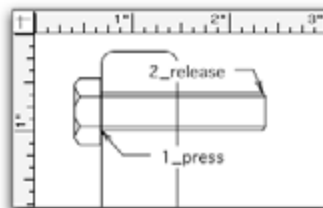
- ☐ Select Centerline to place a centerline along with the bolt.
- ☐ Select Show Threads to draw bolts with ASA schematic threads. If this is not checked, the thread form is shown. If this is not checked and a dashing default has been set for the tool, the thread form only is dashed.
- ☐ Interactive mean that the hex bolt is not a predetermined size; you will determine the size of the diameter and length by dragging.
- ☐ If Interactive is not checked, you enter the diameter and length.

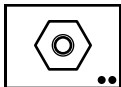
The bottom of the head appears at the point where you press the mouse button.

To use the Hex Bolt tool:

- ☐ Select the Hex Bolt tool from the Mechanical Tool palette.
- ☐ Place the cursor at the bottom-center of the head.
- ☐ Drag to the end of the bolt.
- ☐ Release when the appropriate size is reached.

To draw a bolt from an angled surface, use Perpendicular snap when starting the bolt.





Hex Nut Tool

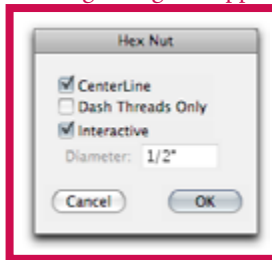
This tool draws tool the top view of a hex nut.

Hex nuts can be drawn by

- ☐ Dragging with the mouse.
- ☐ Entering the diameter of the threads.

To set the behavior of the tool:

- ☐ Press the Option key and select the Hex Note tool from the Mechanical Tool palette. **The following dialog will appear: (Not!)**

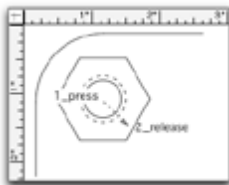


- ☐ Select Centerline to place a centerline along with the nut.
- ☐ Select Dash Threads Only to dash only the thread form. This option is only available when the tool has been set for dashed lines.
- ☐ Interactive means that the hex nut is not a predetermined size; you will determine the size of the diameter by dragging.
- ☐ If Interactive is not checked, you enter the diameter.

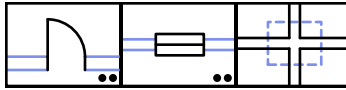
The diameter is the outside thread diameter. Centerlines can be drawn along with the nut.

To use the Hex Nut tool:

- ☐ Select the Hex Nut tool from the Mechanical Tool palette.
- ☐ Place the cursor at the center of the nut.
- ☐ Drag to the outside diameter.
- ☐ Release when the appropriate size is reached.



Wall Tool Palette



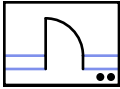
The Wall Tool palette provides specialized tools for walls in architectural drafting and design.

With the Wall Tool Palette you can

- ☐ Place doors (Door Insertion tool).
- ☐ Place windows (Window Insertion tool).
- ☐ Extend, trim, and heal wall lines (Wall Heal/Join tool).

The Door Insertion and Window Insertion tools draw doors and windows to your specifications. The walls are cut and healed automatically. These tools can also use symbols from symbol libraries.

Wall Heal/Join cleans up T, X, and L intersections. It performs other tasks also, such as restoring lines that have been broken and capping wall ends. If you delete a door or window, the Wall Heal/Join tool may be used to repair the open wall lines.



Door Insertion Tool

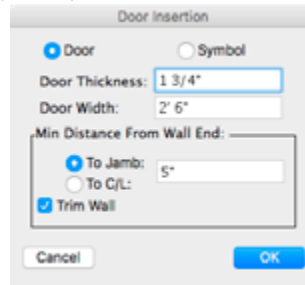
This tool inserts doors into walls (two parallel lines), cuts, and heals the walls. It makes it easy to place doors into a drawing.

Doors can be inserted by

- ☐ Entering their specifications.
- ☐ Using doors from symbol libraries

To set the behavior of the tool:

Press the Option key and select the Door Insertion tool from the Wall Tool palette. The following dialog will appear:



- ☐ You can specify the door thickness and width. The jamb and centerline options control the minimum distance from the door to the end of the wall. The tool automatically prevents you from moving the door closer to the wall end than the distance set.
- ☐ When you select Trim Wall, the wall is automatically cut and healed as the door is placed. This is extremely useful when exchanging files with less capable CAD systems that cannot hide lines. Trim Wall can be used with both the parametric and symbol method of inserting doors. These settings will remain in effect until you change them.

Door Insertion measures the distance from the door to the end of the wall, allowing the door to be positioned in exactly the right place. It can insert doors in diagonal, horizontal, and vertical walls. The walls are automatically cut and healed, making door insertion virtually effortless.

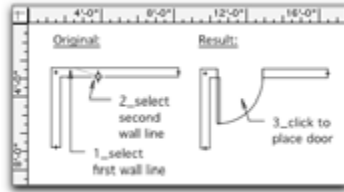


NOTE: Selecting the Door Insertion tool can automatically switch you to the Door Layer (or another layer name of your choosing) if the [Layer Tools](#) feature is checked in the Layout Menu. If the Layer Tools feature is not checked, doors are inserted on the active layer.

To use the Door Insertion tool:

- ☐ Select the Door Insertion tool from the Wall Tool palette.
- ☐ Select one of the wall lines. A bridge line will connect the wall line and the cursor, showing the tool is active.

- ☐ Select the second wall line. A ghost door will appear.
- ☐ Move the cursor to move the door to location. To reverse the direction of swing, bump the door against one of the wall ends. To change the side of the wall on which the door opens, move the door to the other side of the wall.
- ☐ Click to insert the door.



To start over if an incorrect line is selected, press the Enter key. The tool will remain active.

Using Symbols with Door Insertion



The Door Insertion tool can also be used to insert doors (or other symbols) from a symbol library between two parallel lines.

Correct orientation of the door in the symbol library is mandatory to achieve predictable results. The door must open up, and to the left, as shown in the illustration.

The symbol is automatically rotated when used with vertical and diagonal walls.

Symbols can be inserted as instances or groups. Door Insertion uses the settings of the Symbol tool to determine whether to place a symbol as an instance or a group.

When placing the door as a symbol, placement points are ignored. The bottom of the symbol is always placed on the door line.

Some symbol libraries were designed with an opaque rectangle. When these are used with the regular Symbol tool, this rectangle covers-up the wall lines. Since these symbols normally have placement points for correct placement, they will be unsatisfactory in their placement location.

Both the parametric method and the regular Symbol tool placement method (which covers up the wall) are perfectly acceptable. They are just different techniques.

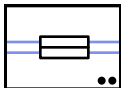
To use the Door Insertion tool with symbols:

- ☐ Press the Option key and select the Door Insertion tool from the Wall Tool palette.

- ☐ Select the symbol button in the dialog.
- ☐ Set the options desired and click OK.
- ☐ Open a library containing door symbols.
- ☐ Select the door symbol to use.
- ☐ Click in the drawing window to make it active.
- ☐ Select the two wall lines.
- ☐ Position the door symbol and click to place.



NOTE: Doors can only be inserted into two parallel lines, not polygons.



Window Insertion Tool

This tool inserts windows into walls (two parallel lines), cuts, and heals the walls. The tool makes it easy to place windows into a drawing.

Windows can be inserted by

- ☐ Entering their dimensions.
- ☐ Using windows from symbol libraries.

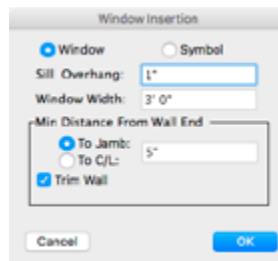
The Window Insertion tool can measure the distance from the window to the end of the wall, allowing the window to be positioned in exactly the right place. It can insert windows in diagonal, horizontal, and vertical walls. The walls are automatically cut and healed, making window insertion virtually effortless.



NOTE: Selecting the Window Insertion tool can automatically switch you to the Window Layer (or another layer name of your choosing) if the [Layer Tools](#) feature is checked in the Layout Menu. If the Layer Tools feature is not checked, windows are inserted on the active layer.

To set the behavior of the tool:

Press the Option key and select the Window Insertion tool from the Wall Tool palette. The following dialog will appear:

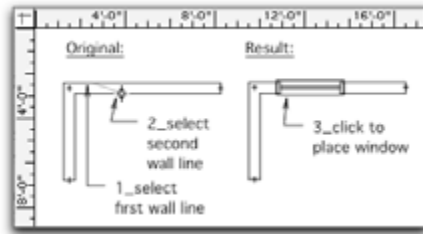


- ☐ You can specify sill overhang and window width. The jamb and centerline options allow the minimum distance from the end of the wall to the window to be specified. The tool automatically prevents you from moving the window closer to the wall end than the distance set.
- ☐ When you select Trim Wall, the wall is automatically cut and healed as the window is placed.

To use the Window Insertion tool:

- ☐ Select the Window Insertion tool from the Wall Tool palette.
- ☐ Select a wall line. A bridge line will appear, connecting the wall line with the cursor.
- ☐ Select the second wall line, and a ghost window will appear.
- ☐ Move the cursor to move the window to the desired location.

- ❑ Click and the window will be inserted.



To start over if an incorrect line is selected, press the Enter key. The tool will remain active.

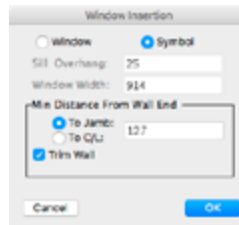
Using Symbols with Window Insertion

The Window Insertion tool can also be used to insert windows (or other symbols) from a symbol library between two parallel lines.

The symbol must be designed for a horizontal wall. The symbol is automatically rotated when used with vertical and diagonal walls.

To use the Window Insertion tool with symbols:

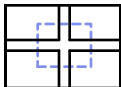
- ❑ Press the Option key and select the Window Insertion tool from the Wall Tool palette.
- ❑ Select the symbol button in the dialog.
- ❑ Set the appropriate options and click OK.



- ❑ Open a library with window symbols.
- ❑ Select a symbol designed for a horizontal wall.
- ❑ Click in the drawing window to make it active.
- ❑ Select the two wall lines. A ghost of the window symbol will appear.
- ❑ Position the window symbol and click to place.



NOTE: Windows can only be inserted into two parallel lines, not polygons.

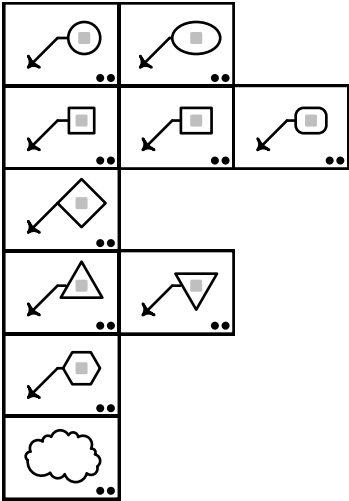


Wall Heal-Join Tool

This tool cleans up and edits a variety of wall lines and intersections.

The tool provides a simple and elegant way to edit, manipulate, and clean up walls and lines. Although the tool is named Wall Heal/Join, it is not just for architects. Any professional who needs to trim, extend, or join lines will quickly find this tool useful.

The Wall Heal/Join tool also appears in Row 14 of the Tool palette. To learn more about the Wall Heal/Join tool please see the descriptions provided for tools in Row 14 of the Tool palette.



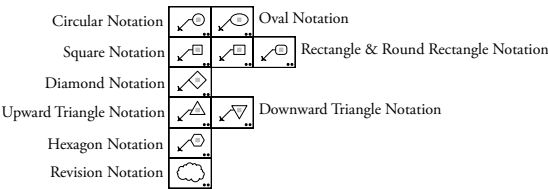
Optional Settings

Notation Tool Palette

With the Notation Tool palette you can place notation symbols in a drawing with customized text. Notations can be used for a variety of specialty labeling tasks ranging from revision clouds and detail keys to door, window, or furniture labels for schedules.

Since all the Notation Tools share the same basic method of operation and optional settings, we will use the Circular Notation tool for illustration and demonstration purposes below. Notation Tools which operation differently will be clearly noted. Each Notation Tool, as diagramed below, inserts a bubble note in the drawing with a shape corresponding to the tools name and icon. For example, the Circular Notation tool inserts circular shaped bubble notes, and Diamond Notation tool inserts diamond shaped bubble notes.

Each notation tool has optional settings which allow you to specify the type of leader line, the size of the bubble note and whether dividers for multiple text fields should be used.

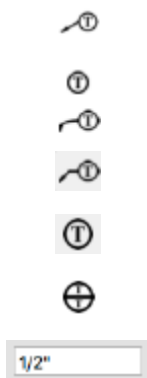


To specify the optional settings for any Notation Tool:

- Press the Option key and select a notation tool. A dialog box will appear:



The name of the dialog box will match the name of the tool. In this example pressing the Option key while selecting the Circular Notation tool the dialog changes to Circular Options.



Straight Line Leader: Click this icon to insert bubble notes with a straight leader line.

No Leader: Click this icon to insert bubble notes with no leader line.

Smooth Polygon Leader: Click this icon to insert bubble notes with a smooth polygon leader line.

Straight Polygon Leader: Click this icon to insert bubble notes with a straight polygon leader line.

No Divider: Click this icon to insert bubble notes with only one text block.

Divider Line: Click this icon to insert bubble notes with two text blocks. Each text block is separated by a horizontal line.

Width Field: Enter the default width for bubble notes here in metric or imperial units. The sizes are for 1:1 or full print size, drawing or layer scale is ignored when specifying this value. The width is fixed and does not change if the font size or amount of text entered is larger than the width specified.

OK button: Click to confirm your changes and close the dialog box. The settings are remembered and remain in effect until you change them. Existing bubble notes in a drawing are not affected. The settings for each Notation Tool are distinct; for example Circular Notations may have different settings than Diamond Notations.

Cancel Button: Click to close the dialog box and abort the setting changes. Any changes made in the dialog box are ignored.

- ☐ Make the desired changes and click OK.

The selected Notation Tool will now draw bubble notes with the specified settings. These settings are in effect until you change them.

Setting Notation Tool Default Attributes

Setting the default line weight, arrow style, dash, color, fill and text style for a notation tool is just like setting default attributes for any other PowerCADD tool. You may use the [Attr menu](#), [Attributes window](#), [Text Attributes Window](#), [Text menu](#) or Contextual Menus (or any combination thereof) to set default attributes. Refer to the start of the Tools chapter, [Change a Tool's Default Attributes](#) for additional information.



NOTE: The default text justification for Notation tools is centered. However you may adjust the justification using the [Text Attributes Window](#), or the [Text menu](#)

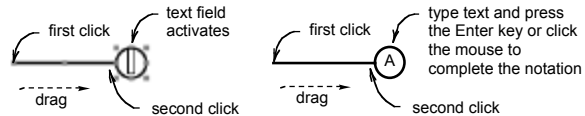
Using the Notation Tools

All Notation Tools follow the same basic method of operation. We'll use the Circular Notation tool to illustrate the various methods for inserting a notation bubble and text in a drawing. However, the same methods apply for all other Notation Tools.

Drawing Straight Line Leader Notations

To use a Notation Tool to draw a Straight Line Leader:

- ❑ Select a Notation tool from the Notation Tool palette.
- ❑ Press the Option key and select the tool to open the optional settings dialog and select the Straight Line Leader icon. Click OK.
- ❑ Place the cursor in the drawing window where you want the leader to start and click.
- ❑ Drag to set the length and angle of the straight line leader. Click the mouse button.
- ❑ The text field will automatically activate. Type the desired text and press the Enter key.
- ❑ The notation is inserted in the drawing.

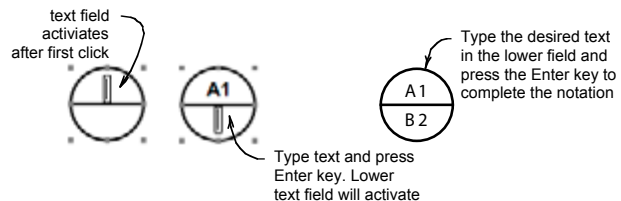


NOTE: The width of the notation bubble is fixed. Text that is too large to fit will extend beyond the width of the notation. Pressing the Return key when entering text will insert a new line which could extend past the width of the notation bubble.

Drawing No Leader Notations

To use a Notation Tool to draw No Leader with two divided text fields:

- ❑ Select a Notation tool from the Notation Tool palette.
- ❑ Press the Option key and select the tool to open the optional settings dialog and select the No Leader icon. Click OK.
- ❑ Place the cursor in the drawing window where you want the notation to appear and click the mouse button.
- ❑ The upper text field will activate. Type the desired text and press the Enter key.
- ❑ The lower text field will activate. Type the desired text and press the Enter key.
- ❑ The notation is inserted in the drawing.

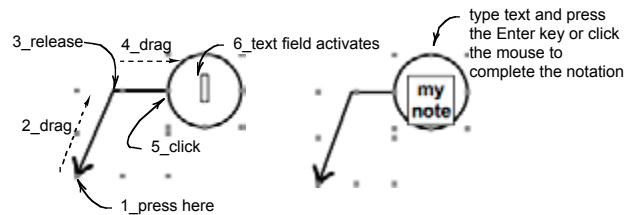


Drawing Straight Polygon Leader Notations

To use a Notation Tool to draw a Straight Polygon Leader with one text field:

- ☐ Select a Notation tool from the Notation Tool palette.
- ☐ Press the Option key and select the tool to open the optional settings dialog and select the Straight Polygon Leader icon, select the No Divider icon and click OK.
- ☐ Place the cursor in the drawing window where you want the leader to start. Press the mouse button.
- ☐ Drag to position the bend in the leader line and release.
- ☐ Move the cursor horizontally to position the leaders endpoint and click.
- ☐ The text field will automatically activate. Type the text associated with the notation. To insert more than one line of text in a note press the Return key.
- ☐ Press the Enter key or click to complete the notation.

NOTE: The same technique applies to create a notation using a Smooth Polygon Leader. Simply select the Smooth Polygon Leader in the optional settings dialog before starting your notation.

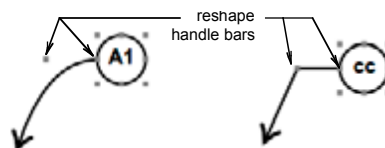


NOTE: Upward and Downward Triangle Notation tools cannot have dividers. The Divider option is dimmed in their optional setting dialog boxes.

Reshaping Existing Notations Leader Lines

To reshape an existing notation leader line:

- ☐ Select the notation with the leader line to be reshaped.
- ☐ Selection handles appear.
- ☐ Move the mouse over a selection handle and drag to reshape the leader line.
- ☐ Release to complete the reshape operation.



Editing Existing Notation Text

There are two methods for editing the text associated with a notation already in a drawing. Directly in the drawing window using the Text tool or using the Text Edit window.

To edit existing notation text using the Text tool:

- ☐ Select the Text tool from the main PowerCADD Tool palette.
- ☐ Click the notation text to be edited. A text insertion point will appear.
- ☐ Make the desired changes and click outside the notation.

To edit existing notation text using the Text Edit Window:

- ☐ Select the notation containing the text to be edited.
- ☐ Choose Edit from the Edit menu. The Text Edit window will open.
- ☐ Make the desired changes and close the Text Edit window.
- ☐ The notation text is updated in the drawing.
- ☐ Click the notation text to be edited. A text insertion point will appear.
- ☐ Make the desired changes and click outside the notation.



Also, see [Text Tool](#), and [Edit Text Window](#), discussed earlier in this chapter for more information on working with Text in PowerCADD.

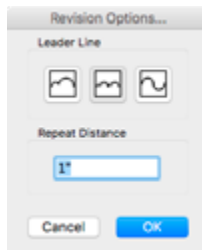


Revision Tool

This tool creates a Bézier curve in an open or closed shape with a circular, scalloped, edge. Traditionally these types of shapes were used to identify areas of drawings which had been revised. However, the shapes can also be used to illustrate areas of plant material such as trees, shrubs, or other illustrative effects.

To set the behavior of the tool:

- ☐ Press the Option key and select the Revision tool from the Notation Tool palette. The following dialog will appear:



Fixed Repeat: With this setting, the scalloped edges are drawn at fixed repeat distance which cannot be changed for this drawing method. The farther you drag between segments the larger the curves will be.

Freehand: Select this icon to draw revision clouds using an unconstrained freehand stroke. As you drag the revision cloud is drawn. A custom repeat distance may be specified.

Variable Repeat: With this setting the scalloped edges are drawn at a user specified repeat distance. The farther you drag the more scallops repeat along the segment at the interval specified in the Repeat Distance field.

Repeat Distance: Enter the default distance the mouse must move before the pattern repeats. The distance may be specified in metric or imperial units. The values are for 1:1 or full print size, drawing or layer scale is ignored when specifying this value. Shorter distances create smaller scalloped edge.

- ☐ Make the desired changes and click OK.

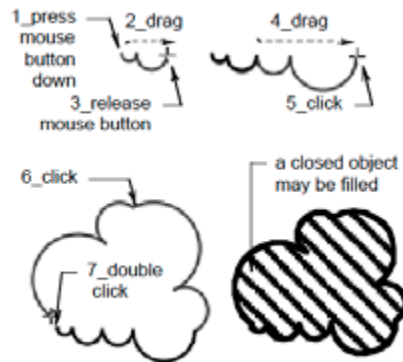
The Revision Tool will now draw with the specified settings. These settings are in effect until you change the tool settings.

Fixed Repeat



To use the Revision Tool with a Fixed Repeat setting:

- ☐ Press the Option key and select the Revision tool from the Notation Tool palette.
- ☐ Select the Fixed Repeat icon and click OK.
- ☐ Place the cursor in the drawing window where you want the revision bubble and click.
- ☐ Drag to begin defining the first segment. A ghosted curve will appear.
- ☐ Release the mouse button to complete defining the first segment.
- ☐ Drag to position the next segment. The farther you drag the mouse the larger the curves will be.
- ☐ Click to define the new segment.
- ☐ Repeat the process until you have defined the desired number of segments.
- ☐ Double-click to complete the drawing operation.

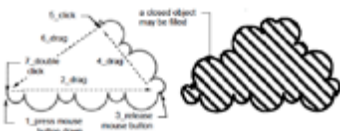


Variable Repeat



To use the Revision Tool with a Variable Repeat setting:

- ☐ Press the Option key and select the Revision tool from the Notation Tool palette.
- ☐ Select the Variable Repeat icon, enter a distance in the Repeat Distance field (example: 1 inch) and click OK.
- ☐ Place the cursor in the drawing window where you want the revision bubble and click.
- ☐ Drag to begin defining the first segment. A ghosted curve will appear. Dragging left or right sets the length of the segment. Dragging up or down sets the angle.
- ☐ Release the mouse button to complete defining the first segment. The farther you drag the more scallops repeat along the segment at the interval specified in the Repeat Distance field.
- ☐ Drag to position the next segment.
- ☐ Click to define the new segment.
- ☐ Repeat the process until you have defined the desired number of segments.
- ☐ Double-click to complete the drawing operation.





NOTE: Entering a smaller Repeat Distance value will create a scalloped edge with smaller curves.

A Repeat Distance of 1"



A Repeat Distance of 0.25"

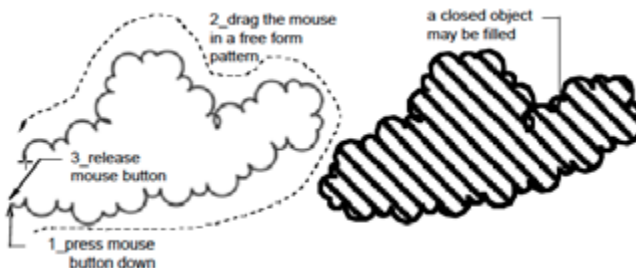


Freehand



To use the Revision Tool with a Freehand setting:

- ☐ Press the Option key and select the Revision tool from the Notation Tool palette.
- ☐ Select the Freehand icon, enter a distance in the Repeat Distance field (example: 0.5 inch) and click OK.
- ☐ Place the cursor in the drawing window where you want the revision bubble to begin and click.
- ☐ Drag to draw the revision cloud. The edges are drawn as you drag the mouse.
- ☐ Release the mouse button to complete the drawing operation.



NOTE: Entering a smaller Repeat Distance value will create a scalloped edge with smaller curves.

A Repeat Distance of 1"



A Repeat Distance of 0.25"



Menu

Menus

This chapter will describe the menu commands and functions in PowerCADD including External menu items.

Generally, all menu items require one or more objects to be selected before applying the menu item. Some menu items are Externals and the particular external must be loaded in order for the items to appear in the appropriate menu.

Menu Bar

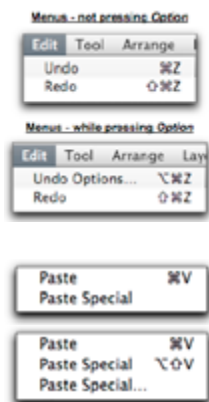
The Menu bar located at the top of the screen contains various pull-down menus of PowerCADD options and commands which you can choose with your mouse.

PowerCADD File Edit Tool Arrange Layout View Attr Text References Window Help

NOTE: The three dots (...) following some of the menu commands indicate special commands. When you choose these commands, a typical Macintosh dialog box will appear, automatically.

A typical example is the Print and Page Setup menu items as shown on the left. When chosen by the Selection cursor, a dialog will appear. This and other dialogs will contain various parameters which must be set before PowerCADD will perform a task.

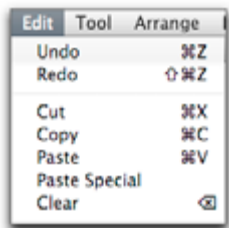
Option Key Commands



Many PowerCADD commands have optional features which are accessed by pressing the Option key while selecting a command. Any command followed by three periods (...) will present you with a dialog box, if selected while pressing the Option key. If you select the command without pressing the Option key, PowerCADD will use the last settings assigned in the dialog.

The use of the Option key with the triple-period commands gives you access to additional advanced features. An example of how a menu display changes can be seen in the Edit menu with Undo and Undo Options... as shown on the left.

If you assign a custom key command which includes the Option key, an additional item appears in the menu. In the example shown here, a custom key command of Shift-Option-V was assigned to the Paste Special menu item. Notice the additional entry in the Edit menu.



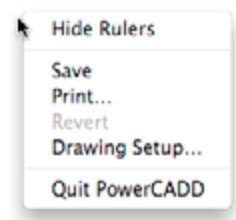
The Edit menu is an excellent illustration of keyboard shortcuts. The commands shown at the left are common to most Macintosh software. Although the commands may be changed, you should attempt to keep the command keystroke sequences as standard as possible with other Macintosh programs. Command keystroke sequences may be added to blank menu items or you may change existing commands as explained later in this chapter.

Contextual Menus

Many PowerCADD menu items may be accessed by pressing the Control key and clicking (or right-clicking) in various areas of the drawing window or on an object. They provide access to a range of features, some of which appear in other PowerCADD menus or floating windows, and others which appear exclusively in a contextual menu.

Contextual menus are intelligent in that they recognize the type of object you are Control- or right-clicking and the menu items presented are adjusted accordingly. For example, the contextual menu for lines has appropriate items for adjusting line weight, color and other typical object attributes. The contextual menu that appears for Dimensions includes menu items for adjusting text size and font type in addition to typical object attribute items. In this section we'll describe the most common contextual menu items which appear when you Control- or right-click on an object or in the drawing window.

Drawing Window Contextual Menu



To access the Drawing Window contextual menu:

- ☐ Control- or right-click in a blank drawing area.
- ☐ Choose an item from the contextual menu.

Hide Rulers: Turns off the drawing rulers in the active drawing. If the drawing rulers are currently hidden (off) this menu item will change to read **Show Rulers**.

Save: Stores the current PowerCADD file to a specified disk under the most recently designated file name. If the drawing has not been previously saved you will be prompted for a file name and location.

Print: Sends the active drawing to output devices such as ink jet or laser printers.

Revert: Replaces the active drawing with the version last saved under the same file name. If the drawing has never been saved this item is dimmed.

Drawing Setup: Opens the Drawing Setup dialog for the active drawing where you may set specific drawing parameters.

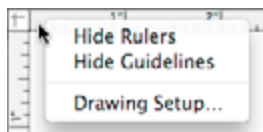
Menus

Quit PowerCADD: Closes all drawings and exits PowerCADD.



For more information on working with a drawing, see [The Layout Menu](#) later in this chapter. For more information see [Print](#), [Save](#), [Revert](#) and [Quit](#), in The PowerCADD Menu.

Drawing Rulers Contextual Menu



To access the Drawing Rulers contextual menu:

- ☐ Control-click on the horizontal or vertical drawing ruler. If drawing rulers are turned off, this contextual menu item is not available.
- ☐ Choose an item from the contextual menu.

Hide Rulers: Turns off the drawing rulers in the active drawing.

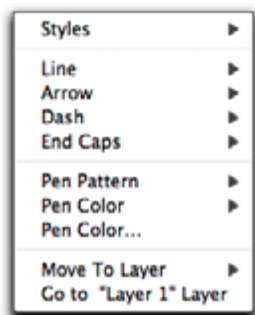
Show/Hide Guidelines: If guidelines are hidden, choose Show to display them. If guidelines are showing, choose Hide to turn them off.

Drawing Setup: Opens the Drawing Setup dialog for the active drawing where you may set specific drawing parameters.



See “The Layout Menu” later in this chapter for more information on working with [Drawing Setup](#) and [Guidelines](#).

Typical Object Attribute Contextual Menu



To access the attribute menu for Lines and other objects:

- ☐ Control- or right-click on a Line in the drawing (contextual menu pictured left).
- ☐ Choose an item from the contextual menu.
- ☐ If a line is not selected the change is applied to the item you Control-clicked on. If multiple objects are selected the change is applied to all selected items.

All objects have access to typical attributes which may be set via a contextual menu:

Styles: Allows you to select from a list of user defined Attribute Styles. If no custom styles are available, choose New to define a new attribute style (see “Making New Attribute Styles” later in this chapter).

Line: Allows you to change the line weights of objects.

Arrow: Allows you to change the arrow attributes of objects.

Dash: Allows you to change the dash patterns of objects.

End Caps: Allows you to change the type of end cap used at the ends of lines or other open ended geometry.

Pen Pattern: Allows you to assign a pattern to lines and borders that outline objects.

Pen Color: Allows you select a color or gray scale value for lines and borders that outline objects from the PowerCADD color palette.

Pen Color... (three dots): Provides direct access to the MacOS Color Panel to select a color or gray scale value for lines and borders that outline objects.

Move To Layer: Allows you to select which visible drawing layer an object should be moved to. The layer the object is currently on, and hidden layers, are dimmed (unavailable) in the list. The list of layers presented is from the active drawing. Moving an object removes it from its current layer onto the selected layer in the same X,Y location.

Go to “Layer 1” Layer: Allows you to change the active drawing layer to match an existing objects layer. The layer name of the object will be displayed instead of “Layer 1” if it is not on the current active drawing layer.



NOTE: Go to “Layer 1” Layer is available from a contextual if an object is selected.



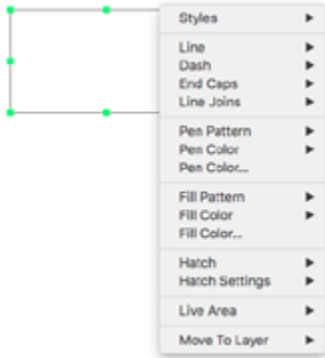
For more information on working with object attributes, see [The Attr Menu](#) later in this chapter.

Additional Object Attributes Contextual Menu

To access the Attributes menu for closed or complex geometry objects such as rectangles, Bezier curves, polygons, circles and arcs:

- ❑ Control- or right-click on an object in the drawing.
- ❑ Choose an item from the contextual menu.
- ❑ If an object is not selected, the change is applied to the item you Control- or right-clicked on. If multiple objects are selected, the change is applied to all selected items.

In addition to typical object attributes, some objects also display additional attribute choices:



Fill Pattern: Allows you to assign a pattern used to fill an area defined by the borders of a closed or open object.

Fill Color: Allows you select a color or gray scale value for fill patterns from the PowerCADD color palette.

Fill Color... (three dots): Allows you select a color or gray scale value for fill patterns from the MacOS Color Panel.

Hatch: Allows you to assign a vector based pattern to fill an area defined by the borders of a closed or open object.

Hatch Settings: Allows you to control various hatch attributes of associative hatch patterns, including hatch origin, pen color, line weight and scale.

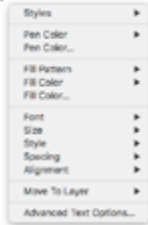
Live Area: Allows you to insert a smart text object in the drawing displaying the area and/or perimeter of an object.



See [Live Area](#) and [pen](#), [fill](#) and [hatch](#) attributes in the Attr Menu later in this chapter for more information on working with these attributes.

Text Attributes Contextual Menu

Feel? Well, let's just say flying an A-2 is like making love to a fat lady who's had too much to drink: there's a lot to work with, it's unresponsive, you're never quite sure when you're there, and it's big-time ugly.



To access the contextual menu for text objects:

- ☐ Control- or right-click on a text object in the drawing.
- ☐ Choose an item from the contextual menu.
- ☐ If no text object is selected the change is applied to the item you Control-clicked on. If multiple text objects are selected the change is applied to all selected items.

In addition to the typical object attributes such as pen color and fill pattern (see [Typical](#) & [Additional](#) Object Attribute Contextual Menus), special contextual menu items related to Text are also displayed:

Font: Allows you to select the font family that you want used for your text. You may select from any active font installed in MacOS.

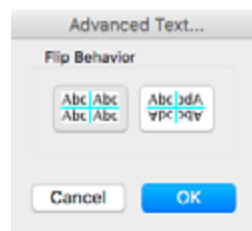
Size: Allows you to select the size, in points, of the text.

Style: Allows you to select the style of a font to be used. For example **bold** or *italic*. You may also select the case of text, for example lower, UPPER or Title Case.

Spacing: Allows you to control the vertical distance between lines of text from a standard list of line spacing, incremental leading, or custom leading amounts.

Alignment: Allows you choose from Left, Center, Right or Full justified text alignments.

Advanced Text Options....: Allows you to control how text is displayed when it is mirrored, or flipped vertically or horizontally. Choosing this contextual menu item opens the following dialog.



Click on the Flip Behavior icon to select the desired option.



For more information on working with text, see the [Text Menu](#) later in this chapter.



Blossom is a special contextual menu which brings up the standard PowerCADD tool palette anywhere on screen when you assign a custom key command to Blossom.

You can also create your own custom tool palette which appears instead of the default PowerCADD tool palette

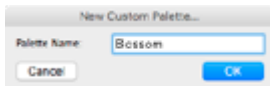
To use a custom Blossom tool palette:

- ☐ Choose Preferences from the PowerCADD menu and the Preferences dialog will open.
- ☐ Select the Commands Tab.
- ☐ From Single Commands choose Misc., XTNL Misc., Blossom.
- ☐ Press the desired key command sequence then click OK to close the Preferences dialog.
- ☐ In the drawing window, press your custom key command sequence; the Blossom tool palette will appear displaying the standard PowerCADD Tool Palette. If you created a custom Blossom tool palette (described later in this section), your custom Blossom tool palette is displayed.
- ☐ Tools selected from the Blossom tool palette function just like tools in the regular PowerCADD Tool palette.



TIP: Some multi-button mice allow you to assign a key command sequence to a specific button. If your hardware has that capability try assigning your custom key command one of the special mouse buttons for even easier access to Blossom.

For more information on working with custom key commands, see [Commands Preferences](#) later in this chapter.



To create a custom Blossom tool palette:

- ☐ Choose New Custom Palette from the Layout menu.
- ☐ In the New Custom Palette dialog, name the palette *Blossom*. The name is case sensitive; notice there is an upper case 'B' and the rest of the text is lower case.
- ☐ A new, empty, custom tool palette will appear on screen.
- ☐ Press the Command key while dragging a tool from any tool palette onto the new Blossom tool palette.

The next time you press your custom command sequence in the drawing window, your custom Blossom tool palette will appear instead of the standard PowerCADD tool palette.

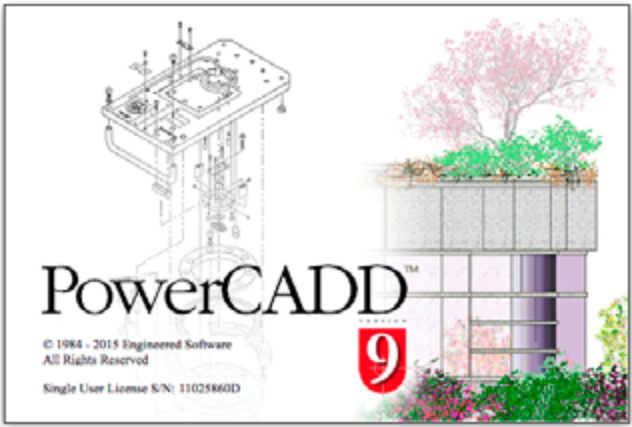


TIP: A custom Blossom tool palette can be a very effective way to manage tools you use on a regular basis. For example, you could create a Blossom tool palette which contains any standard PowerCADD tools and any

External tools (for example WildTools) you use regularly. This can save a significant amount of time when compared to moving in and out of multiple tool palettes during a typical drawing session.

PowerCADD Menu

The PowerCADD menu tracks important information about the application, computer memory usage, drawing statistics and allows you to set application preferences.



About PowerCADD...	
PowerCADD Profile...	
Preferences...	⌘,
Services	▶
Hide PowerCADD	⌘H
Hide Others	⌘H
Show All	
Quit PowerCADD	⌘Q

About PowerCADD contains your software serial number. Choose About PowerCADD from the PowerCADD menu, the serial number is displayed in the bottom left of the splash screen as pictured above.

PowerCADD Profile provides a Software Profile (information about the application) and a Drawing Profile (information about your drawing).

To use PowerCADD Profile:

- ❑ Choose PowerCADD Profile from the PowerCADD menu and the following dialog will appear:



Software Profile: Contains your PowerCADD serial number, the application version number and your operating system version number. This information can be helpful when requesting technical support.

Drawing Profile: Contains a general profile about the current active drawing including the drawing name, number of objects on the current active layer, total number of layers in the drawing and total number of objects in the drawing. This information is provided to assist you in managing your drawings.

Preferences

Preferences allow you to customize PowerCADD's general behavior to suit your drawing and working style. Preference settings are saved with the application and apply to all drawings. To set particular drawing settings, see Drawing Setup described later in this chapter.

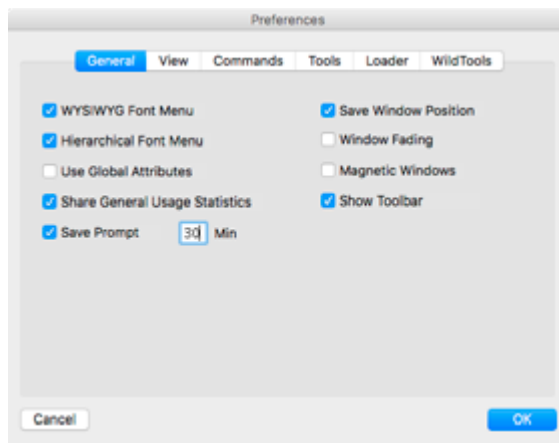
Within the Preferences tabbed dialog, you are able to set and control PowerCADD's preferences, from how you view your document, changing your Command keys to deciding which External will be loaded the next time you open PowerCADD.

To use Preferences:

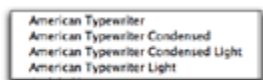
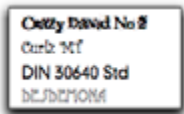
- ☐ Choose Preferences from the PowerCADD menu and the Preferences dialog will open.
- ☐ Each tab contains groups of options you can set. Select a tab to see the different options.
- ☐ Check a box or specify a particular value for the desired settings and click OK.
- ☐ The preference settings are set and will affect all drawings.

General Preferences

The **General** preferences tab allows you to specify the behavior of general PowerCADD features.



WYSIWYG Font Menu: Allows you to specify whether the font menu displays items in their true font style or in a generic font. Changes made to this preference do not take effect until the next time you open PowerCADD.



Global Attributes

WYSIWYG Font Menu not checked:

- ☐ The font menu displays all fonts in the same, default font. Generally this method of font display is faster and may help increase performance on slower computers or when many fonts are loaded.

WYSIWYG Font Menu checked:

- ☐ The font menu displays all fonts in their particular style. This method can make it easier to determine which font you might want to use.

Hierarchical Font Menu: Allows you specify whether the font menu displays any styles available for font in an associated submenu. Changes made to this preference do not take effect until the next time you open PowerCADD.

Hierarchical Font Menu not checked:

- ☐ The font menu displays the name of the primary font family.

Hierarchical Font Menu checked:

- ☐ The font menu displays all font styles available for specific font family in a submenu.

Global Attributes allows you to share a common set of attribute menu items across all drawings. When checked, the menu display for seven attributes are changed: Pen (line weights), Color (fill and pen), Hatch patterns, Fill Patterns, Arrows, Dashes, and Attribute Styles. Menu choices displayed when using the Attribute menu, Defaults Window, Contextual Menu, or Attributes Window are affected for all new or existing drawings. When unchecked the attributes saved with the document are displayed in the menus.

Before exploring the specifics of using Global Attributes, it is important to understand some fundamental rules regarding object attributes and Global Attributes in general:

- ☐ Global Attributes are stored in the Defaults Table file found in the PowerCADD Preferences folder in your [User Library Folder](#). Once this file is seeded with global attributes, it may be shared with others.
- ☐ When Global Attributes is on (checked), attributes displayed in menus are from the Global Attributes Default Table.
- ☐ When Global Attributes is off (not checked), attributes saved with the specific document are displayed in the menus.
- ☐ When you assign an attribute to an object in PowerCADD, that attribute is saved with the object. Turning Global Attributes on or off will never change the attributes of objects already in a drawing.
- ☐ When Global Attributes is on, attributes you add to a particular drawing are only appended to the Global Attributes Default Table.

These attributes are then available in all other drawings. If you later turn Global Attributes Off, attributes you added will not appear in the documents Attribute menu.

- ❑ When Global Attributes is Off, attributes you add to a particular drawing are available only in that drawing. They are not appended to the Global Attributes Default Table.
- ❑ Turning Global Attributes Off returns a drawing's attribute menus to their original state before Global Attributes was turned On. That is, the drawing behaves as if Global Attributes was never used.

Turning Global Attributes On

To use Global Attributes in all drawings:

- ❑ Open the drawing containing the attributes for Pen (line weights), Color (fill and pen), Hatch patterns, Fill Patterns, Arrows, Dashes, and Attribute Styles. This is the drawing used to seed the Global Attributes Default Table.
- ❑ Choose Preferences from the PowerCADD menu and select the General tab.
- ❑ Check the Use Global Attributes box and click OK.
- ❑ A dialog appears asking you to confirm, click OK.
- ❑ The Global Attributes Default Table has been seeded.
- ❑ The items displayed in Attribute menu, Attributes window, Default window and Contextual menus in all old and new drawings will display the same Global Attributes.

Updating Global Attributes

To add a new Global Attribute which will be available in all old and new drawing:

- ❑ Open any existing drawing or create a new drawing.
- ❑ Add the desired attribute.
- ❑ The new attribute is appended to the Attribute menu and is now available in all drawings.

The same process holds true for deleting or modifying a Global Attribute. Making the change in any open drawing will update the Attribute menu display in all drawings.



NOTE: Even though updates to Global Attributes are displayed immediately in all drawings, changes are not saved to the Global Attributes, Default Table file until you Quit PowerCADD.

User Library Folder

In MacOS Sierra and later, the user library files are hidden among many others, however you can show and hide the hidden files with the new key command of Command-Shift-(period) and avoid having to use Terminal to show hidden files.

Sharing Global Attributes

Sharing a Global Attributes Default Table file can be an effective method to share standard attributes in a multiple-user environment. In this scenario we'll presume Global Attributes was already on for both users and user one has Global Attributes they want to share with user two.

To share Global Attributes:

- ☐ User one: Quit PowerCADD. Any recent changes to the Global Attributes Default Table file are saved to disk.
- ☐ Open the PowerCADD preferences folder (in your User: Library folder) and copy the Default Table file to your desktop.
- ☐ User two: Quit PowerCADD and copy the Default Table file into the PowerCADD Preferences folder on your computer.
- ☐ Both copies of PowerCADD will now display the same Global Attributes the next time a drawing is opened.

Turning Global Attributes Off

Turning Global Attributes Off will only modify the items displayed in attributes menus. Attributes added to a menu while Global Attributes was on will not appear in the menus. Object attributes already in the drawing are not changed. The attribute menus in each drawing will now display their original choices.

Global Attributes vs Stationery Drawings

Both Global Attributes and Stationery Pad Drawings are powerful methods to manage your workflow and drawing environment. Given the broad range of types of drawing and design disciplines PowerCADD is used in, we don't offer a cut-and-dry recommendation as to which method might be best for your particular situation.

Some items to consider when deciding whether to use Global Attributes or Stationery Pad Drawings include:

- ☐ Do you have multiple custom Stationery Pad Drawings, each with different attributes (perhaps different line weights, color palettes or fill pattern palettes)? Global Attributes may not be the best solution since all drawings share common attributes. Using a Stationery Pad may be a better solution.
- ☐ Do you use a single Stationery Pad for everything? Global Attributes may be worth considering.
- ☐ Are you in a multi-user environment? While sharing Global Attributes can be an effective way to keep everyone on the same page, it doesn't prohibit each user from adding their own items to their local Global Attributes Default Table file. Those changes will not be available unless the Default Table file is copied to all computers. Stationery Pads can be stored on a central server and can contain custom attribute files which may be worth considering.
- ☐ Do you share files with other PowerCADD users? If everyone doesn't have the same Global Attributes Default file, the menu items available will not be the same across all computers. Working with

Global Attributes off always saves any attribute menu changes with the drawing so menu selections are the same on all computers.

Those are just some of the broader stroke items to consider. Take the time to explore the workflow and production management methods available to both Stationery Pads and Global Attributes. Each method will have pros and cons which vary with your particular workflow.

Share Usage Statistics: Enables automatic logging of which tools, menu commands, and general types of objects you typically use. This information is automatically shared with Engineered Software to help fine-tune PowerCADD performance and features.

Save Prompt: Allows you to specify how many minutes will pass before PowerCADD prompts you to save the drawing. Note, drawings are not automatically saved, you will be presented with a dialog asking you to proceed or cancel the Save operation. The Save prompt dialog appends as a sheet to the documents title bar. If multiple drawings are open, each drawing may have its own Save Prompt Sheet appended to its document window.

Save Window Positions: When checked, PowerCADD will remember the position and size of your drawing window when it is saved and closed. When unchecked all drawing windows open with the window in the top left corner of the screen (the default location) and fill the entire screen. When Save Window Positions is checked, PowerCADD also remembers up to ten MacOS User Names when saving the window size and position.

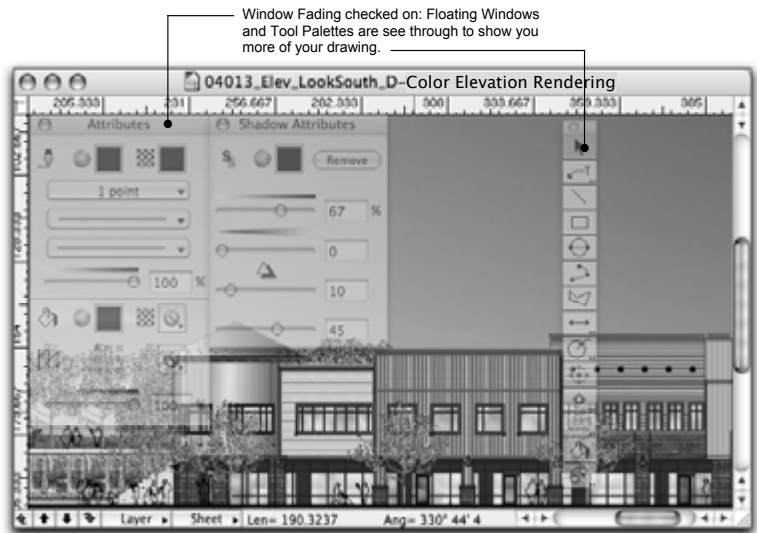
During the course of drawing production, it is common for a drawing to be opened by different users. It is also common for each of those users to have different computers which may have different monitor sizes. This typically results in each user having a different preference for both the size and location for the drawing window.

By remembering who (the MacOS User ID) last saved the drawing, PowerCADD can open the drawing window in your preferred size and location. The next time someone else opens the same drawing on their computer, it will be in their preferred location and size. This process allows for a drawing to make multiple round trips while still remembering each person's preferred window size and location.

NOTE: A drawing window can never be saved or opened in a position that will leave it entirely off screen. If a screen is too small to display the last saved location, the drawing will be positioned in the top left corner and sized to fit the maximum screen area. Any subsequent customization as to size and location is remembered when the file is saved.



Window Fading controls whether floating windows and tool palettes can automatically fade. When checked, the Auto-Fade settings associated with each floating window or tool palette are enabled. When unchecked the auto-fading is disabled for all floating windows and tool palettes. Window fading can be an excellent work space saver, allowing you to see more of your drawing. Each window can have unique window fading properties.



Magnetic Windows controls whether floating windows and tool palettes can interact with each other or the edges of the computer screen. When checked, the Magnetic Edges settings associated with each floating window or tool palette are enabled and the windows will stick together when touching. Moving one window will move all windows that are stuck together. When unchecked the Magnetic Edges settings are disabled for all floating windows or tool palettes. To release windows which are stuck together press the Command key and move the window. Each window can have the magnetic property turned on or off.



For more information on Auto-Fade and Magnetic Edge settings for tool palettes, see [The Tool Palette](#) in the Tools chapter. For more information on Auto-Fade and Magnetic Edge settings for floating windows, see [Floating Windows](#) in the PowerCADD Tutorial.

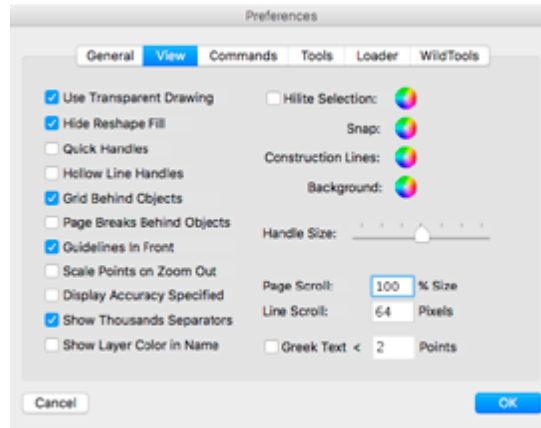
Show Toolbar controls whether the toolbar on the top of the drawing window is visible when opening drawings. When Show Toolbar is checked all drawings display the toolbar by default. When Show Toolbar is unchecked the toolbar is hidden by default. You can change the visible or hidden status of any drawing window toolbar by clicking the oval button in the top right corner of a drawing window.



For more information on working the drawing window Toolbar, see [The Document Window](#) in the PowerCADD Tutorial.

View Preferences

The View preferences tab allows you to specify how objects in the drawing are viewed on screen.

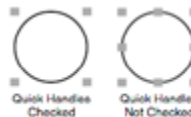


TIP: The Preference setting for High Quality Drags found in PowerCADD 8 has been discontinued. By default, dragging to move multiple objects will always display all selected objects. The Control key may be used to temporarily toggle High Quality Drag Off. For example, pressing the Control key during a move by dragging operation will temporarily show only the object you first selected. (check ???)

Use Transparent Drawing when checked, objects with a solid fill or line pattern are transparent when dragging them. When unchecked, solid patterns render as opaque when dragging them.

Hide Reshape Fill allows you to hide the fills for Béziers, polygons, arcs and circles while reshaping an object.

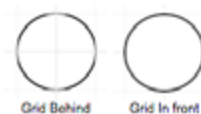
Quick Handles shows four object handle for selected objects instead of eight.



Hollow Line Handles allows you to display a handle at the end point of objects drawn with line tools. The hollow line handle indicates the end point of the line. The start point of a line is fixed when entering new line lengths using the Edit Window; the hollow line end point indicator will be moved.

Grid Behind Objects allows you to display the drawing grid lines either in front or behind objects. When checked on, objects will be rendered on top of (in front) of the drawing grid lines. When unchecked, the drawing grid

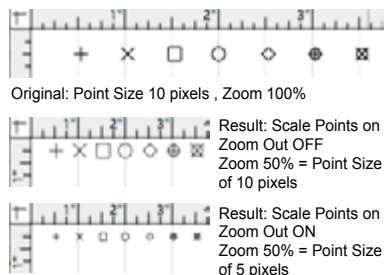
line renders on top of all objects, including objects with solid fill patterns.



Page Breaks Behind Objects allows you to display page breaks either in front or behind objects. When checked On, objects will be rendered on top of (in front) of page breaks.

Guidelines In Front allows you to display Guidelines either in front or behind objects. When unchecked, objects will be rendered on top of (in front) of Guidelines. When checked, Guidelines are rendered on top of all objects. In previous versions of PowerCADD this was accessed by an Option-click on a drawing ruler.

Scale Points on Zoom Out: When checked (On), Points in the drawing inserted using the Reference Point tool will be rescaled smaller as you zoom past normal zoom. When not checked (Off), Points in the drawing are not rescaled, they retain their original size in pixels as specified in the Point tool settings dialog.

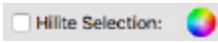


Display Accuracy Specified makes the Edit Window and other dialogs reflect its numerical information rounded to the accuracy set in Drawing Setup, Units. Placed dimensions are also rounded. For example, if the drawing setup accuracy is $\frac{1}{4}$ ", a object with a length of $1 \frac{1}{8}$ " in the drawing would be rounded off when displayed in the Edit Window to 1" if Display Accuracy Specified is checked. An alert dialog will appear requiring you to confirm this action.

Show Thousands Separators includes a comma when displaying values in the Edit Window. For example, with Show Thousands Separators checked one thousand is displayed as 1,000. With Show Thousands Separators not checked one thousand is displayed as 1000.

Show Layer Color In Name affects the display of layer names in the floating Layers Window for those layers with a Layer Color Attribute

assigned. When unchecked, only the layer status indicator is displayed in the specified layer color. When checked, the entire layer name and the layer status indicator is displayed in the specified layer color.



Hilite Color



Hilite Selection enables you to change the color of the object and its handles while it is selected. Click on the color wheel to pick a Hilite color from the MacOS color picker. If Hilite Selection is checked the Hilite color affects both the handle color and hilite color of a selected object. If Hilite Selection is not checked, the Hilite color affects the color of any object's selection handles only.

TIP: Even though you may not choose Hilite Selection, the Hilite color is used throughout the program in many places. WildTools uses the Hilite color when tools pick objects in the drawing. To see this in action, try using a bright green as the Hilite color and notice how the color is used as you use the program.

Snap Color allows you to specify any color for the snap indicators used on screen and in the Dynamic Snap Indicator floating window. To change the snap color click the Snap Color button and choose a color from the MacOS Color Panel.

Background Color allows you to specify any color for the drawing background. The background color doesn't print. To change the color click the Background Color button and choose a color from the MacOS Color Panel.

Construction Lines allows you to specify a color for the construction lines which appear when creating and editing objects. This color is also applied to Guidelines. To change the construction line color, click the Construction Line Color button and choose a color from the MacOS Color Panel.

Handle Size allows you to adjust the size of the selection handles displayed when an object is selected. Drag the slider bar to the right to increase the handle bar size.

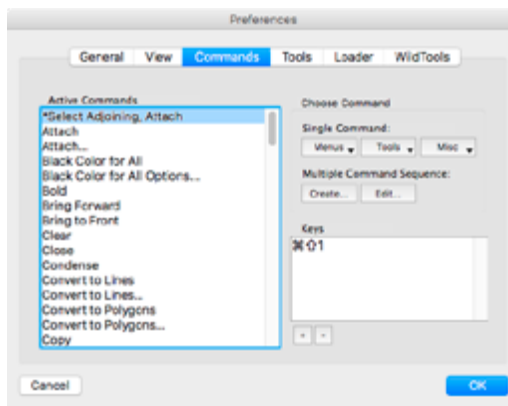
Page Scroll allows you to specify, as a percentage how far the page will scroll each time the area between the scroll arrows is clicked. For example, if the page scroll is set to 75%, only 75% of the drawing visible on the screen will scroll in one step. This does not affect the amount of scrolling per scroll bar arrow (see Line Scroll below).

Line Scroll allows you to specify how far, in pixels, the drawing will move each time you click on a scroll arrow.

Greek Text allows you to specify the text point size threshold where text is redrawn to screen as a gray box instead of normal text. Greeking occurs when text will render equal to or smaller than the specified point size on screen, it is not linked to the font size you specify when entering text. For example, if greeking is set to 6 points and text entered as 12 points, the text will render as a gray box when the drawing is viewed at 50% zoom. Greeking does not affected printing, all text will print at the specified font size based on the page setup settings. Greeking text can help increase redraw performance, particular in drawings containing large amount of small text.

Commands Preferences

The Commands preferences tab allows you to assign a key sequence to any tool or menu item in PowerCADD.



Keys displays the actual key sequence assigned to the function selected in the Active Commands list. You may have more than one key sequence assigned to any Active Command.

Active Commands displays all functions that presently have an assigned key sequence. Multiple Command Sequences always appear at the top of the list.

The Choose Command section contains choices for creating single or multiple command sequences which can be assigned to key command sequence you specify.

Single Commands

Each PowerCADD Tool, Menu item, or special Miscellaneous items can have a custom key sequence assigned.

Tools is a pop-up menu which allows you to specify the tool to which you wish to assign a key sequence.

Menus is a pop-up menu which allows you to specify the menu item to which you wish to assign a key sequence.

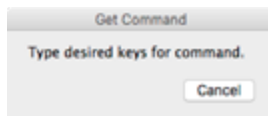
Miscellaneous is a pop-up menu which allows you to specify the miscellaneous functions to which you wish to assign a key sequence.

Keys displays the actual key sequence assigned to the function selected in the Defined Functions window. You may have more than one key sequence assigned to any Defined Function.

Adding a Single Command

To create a single command key sequence:

- ☐ Click the Menus, Tools or Miscellaneous pop-up to choose an item.
- ☐ The Get Command dialog appears; press the desired keys you want assigned to the selected item. The keys must be pressed one at a time and held until all keys to be assigned are down.



- ☐ The key assignment is displayed in the Keys list.
- ☐ Click OK. The key assignment is complete and is remembered unless you change it.

You may add key stroke assignments to as many items as you wish before you leave the dialog box. To use the assigned key command return to a drawing window and press the custom assigned key sequence.

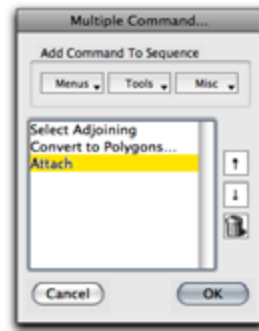
Multiple Command Sequence

Multiple PowerCADD menu items can be sequenced together and assigned to a single custom key sequence. A multiple command sequence can save significant time by taking a series of repetitive tasks in a workflow and linking them into one step.

Create Multiple Command Sequence

To create a multiple command key sequence:

- ❑ Click Create, the Multiple Command dialog options appears.



- ❑ Click the Menus pop-up to choose a menu item to be included in the multiple command sequence.
- ❑ Continue to choose Menu, Tools or Misc. items from their respective pop-up menus. Each new item is added sequentially to the list.
- ❑ Click OK; the Get Command dialog appears.
- ❑ Press the desired keys you want assigned to the command sequence. The keys must be pressed one at a time and held until all keys to be assigned are down.
- ❑ The key assignment is displayed in the Keys list.
- ❑ Click OK. The key assignment is complete and is remembered unless you change it.

You may create additional multiple command key sequences and key stroke assignments to as many items as you wish before you leave the dialog box. To use the multiple command sequence, return to a drawing window and press the custom assigned key sequence.



TIP: We recommend tools included in multiple command key sequences be the last item in the sequence.

Change Order or Delete an Item in a Multiple Command Sequence

To change the order or delete an item in a Multiple Command Sequence:

- ❑ Select the item name in the Multiple Command Dialog.
- ❑ Click the Up Arrow icon to move the item up the list. Click the Down Arrow icon to move the item down the list
- ❑ Click the Trash Can icon to delete an item from the sequence.

Editing a Multiple Command Sequence

To edit an existing multiple command key sequence:

- ☐ Select the Multiple Command Sequence from the Active Commands list. Multiple commands are always displayed at the top of the list.
- ☐ Click the Edit button; the Multiple Command dialog appears.
- ☐ Click the the Up Arrow icon to move the item up the list. Press the Down Arrow icon to move the item down the list.
- ☐ Click the Trash Can icon to delete an item from the sequence.
- ☐ Click OK to save the changes and return to the Commands dialog.
- ☐ Click OK to return to the drawing window.

Deleting Command Keys

To delete a single or multiple command key sequence:

- ☐ Select the item from the Active Commands List.
- ☐ Select the key commands in the Keys list.
- ☐ Click the Delete (-) icon.
- ☐ Click OK. The key assignment is deleted from the list.

If multiple key commands have been assigned to an Active Command, all the key sequences must be deleted to remove the item from the Active Command list. To see the change, you must open and close the Preferences dialog.

Adding Command Keys

To add an additional command key sequence to an existing item in the Active Commands list:

- ☐ Select the item from the Active Commands List.
- ☐ Click the Add (+) icon; the Get Command dialog appears.
- ☐ Press the desired keys you want assigned to the command sequence. The keys must be pressed one at a time and held until all keys to be assigned are down.
- ☐ Click OK to return to the drawing window.



NOTE: To set an F-key like F1 through F15 which appear across the top of most keyboards, press the F-key itself as opposed to the letter *F* and then the number.



CAUTION: When choosing key commands for PowerCADD you should also consider which custom key commands are assigned as global keyboard shortcuts in MacOS. For example F1 is typically assigned to Exposé.

If you find the commands you have set are not as you had expected or if you need to return to the original command key settings, open the PowerCADD Preferences Folder located inside the [User Library Folder](#) in MacOS. Locate and throw away the PowerCADD Commands file. The next time PowerCADD is opened, a new Commands file will be created.

If you wish to view or print a list of the current command key defaults, open

the PowerCADD Key Defaults file located in the PowerCADD Preferences folder. This document can be opened by word processing packages. The document will give you an alphabetized list of the functions and their associated key sequences. PowerCADD will update this file each time a change is made, therefore the listing is always current.

Seeing and Printing the List of Command Keys

The PowerCADD Key Defaults file contains all the PowerCADD standard key commands and your custom key command are saved in a single file which may be opened and printed for reference. The PowerCADD Key Defaults file is updated each time it is changed so the listing is always current.

To see or print the PowerCADD Key Defaults file:

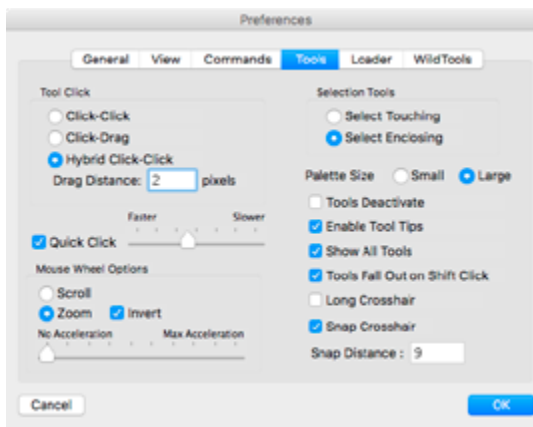
- ☐ Navigate to the following folder: User/Library/Preferences/PowerCADD ([which may be hidden](#))
- ☐ Select the file named PowerCADD Key Defaults and drag it onto the application icon of any word processing application. For example: drag the icon onto the Text Edit application icon in the Applications folder or in the Dock.
- ☐ The PowerCADD Key Defaults file will open.

Backing Up Custom Command Keys

It is likely that over time you will create various customized command key sequences to suit your particular workflow. To make backup of the custom command keys simply make a copy of the PowerCADD Key Defaults file to another location on your hard drive or to other backup media. If you are using the MacOS Time Machine, the Key Defaults file will be included with that backup by default. Restoring a lost or damaged Key Defaults file is as simple as quitting PowerCADD, locating the backup copy of the Key Defaults file and copying it back to the User/Library/Preferences/PowerCADD folder [which may be hidden](#).

Tools Preferences

The Tools preferences tab allows you to set the general behavior of the drawing tools.



Tool Click

Tool Click allows you to specify what type of mouse action you like when using the drawing tools. Choose the tool click style that works best for you.

Click-Click: Requires two mouse clicks to draw an object. You do not have to keep the mouse button pressed when drawing an object.

To draw using the click-click method:

- ☐ Pick a drawing tool from the tool palette, for example the Line tool.
- ☐ Click where you want to start the line. The line follows you as you drag.
- ☐ Drag the mouse to specify the length of the line.
- ☐ Click where you want the line to end.

Click-Drag: Drawing with a click and drag operation requires a single mouse click and is much like drawing with a pen on a piece of paper.

To draw using the click-drag method:

- ☐ Pick a drawing tool from the tool palette, for example the Line tool.
- ☐ Click where you want to start the line.
- ☐ Drag the mouse to specify the length of the line.
- ☐ Release the mouse where you want the line to end.

Hybrid Click-Click is the default setting allows you to draw with either a click-click or click-drag method with any tool.

Drag Distance allows you specify a distance in pixels you have to drag when using hybrid-click before the tool provides visual feedback. For example, a drag distance setting of 72 would require dragging with the line tool approximately one inch before you started to see a preview of the line.

A value of two pixels (the default) means dragging only a short distance will begin to show the object's preview.

Quick Click allows you to specify the speed of a mouse click PowerCADD will use when working with tools. Dragging the slider bar to Faster tells PowerCADD to consider a fast mouse click as a valid operation. Dragging the slider bar to Slower tells PowerCADD to only consider slower mouse clicks as valid operations. For example, with the slider set to Slower and the Text tool chosen, a fast mouse click inside of an existing text block will have no effect. To edit the text block, either set the Quick Click slider to faster or click the mouse slower (a longer pause) when clicking inside the text block.

Mouse Wheel Options

Mouse Wheel Options allows you to customize the behavior of a mouse scroll wheel or the touch pad. If you are using a touch pad on a laptop please reference the MacOS help files for information on how to set up System Preferences, Trackpad Gestures.

Scroll: When enabled, a mouse scroll wheel or touch pad will scroll the drawing or overview windows.

Zoom: When enabled, a mouse scroll wheel or touch pad will zoom in or out of the drawing or overview windows. The zoom in or out behavior can be changed by choosing Invert or adjusting the Acceleration slider.

Invert: When unchecked, rolling a scroll wheel forward will zoom in, rolling a scroll wheel backward will zoom out. When checked the behavior is reversed: rolling a scroll wheel forward will zoom out and rolling backward will zoom in. The settings behave the same if you are using a touch pad on a laptop.

Acceleration: The acceleration slider allows you to adjust how fast the zoom in and zoom out occurs when mouse scroll wheel zooming is enabled. The degree of control this slider provides will vary with the type of mouse and any MacOS system level drivers installed.

Selection Tools

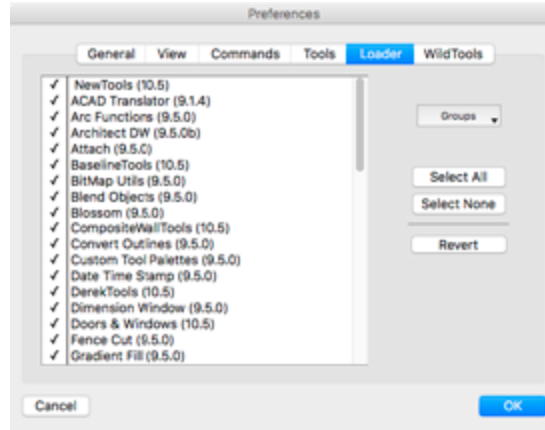
You can choose to have the Selection Arrow tool behave two fundamentally different ways.

- ☐ **Select Touching** will pick all objects which touch a selection marquee.
- ☐ **Select Enclosing** will pick only those objects which are entirely inside the selection marquee. You can also temporarily switch on the fly between the two selection modes by pressing the Command key when dragging with the Selection Arrow tool. See the Tools section of this manual for more information.

Palette Size	Palette Size allows you specify the size of PowerCADD tool palettes. Select Small to display all tool palettes at approximately 30 pixels wide. Select Large to display all tool palettes at approximately 45 pixels wide.
Tools Deactivate	<p>Tools Deactivate allows you to specify what should happen after you've used a drawing tool.</p> <p>Tool Deactivate not checked:</p> <ul style="list-style-type: none"> <input type="checkbox"/> After you draw an object, the tool remains active allowing you to draw another object with the same tool. The tool remains active until you select a different tool or Shift-click on the screen (See Tools Fall Out on Shift Click below for more information). <p>Tools Deactivate checked:</p> <ul style="list-style-type: none"> <input type="checkbox"/> After you draw an object, the tool automatically deactivates and the Arrow Selection tool is activated.
Enable Tool Tips	Enable Tool Tips displays the name of a tool when you hold the mouse over its icon in the tool palette.
Show All Tools	Show All Tools displays all the tools in the palette when you select any tool. When unchecked only a tool's row is displayed when you select a tool.
Tools Fall Out on Shift Click	Tools Fall Out on Shift Click will allow you to press Shift key and click the mouse to cause a tool to fall out and the Arrow Selection tool to activate.
Long Crosshair	Long Crosshair allows you to turn the long crosshair On or Off. The crosshair is a gray line running the entire length and width of the drawing window and can be useful when you must align an object with another that is not adjacent.
Snap Crosshair	Snap Crosshair works in conjunction with the Long Crosshair in the Tools tab. With snap crosshair checked On, the crosshair will move in response to snap commands.
Snap Distance	Snap Distance is the distance in pixels in which an automatic snap will take place when your mouse is near a basic snap point.

Loader Preferences

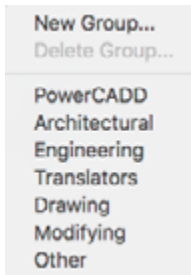
The Loader preferences tab allows you to specify which PowerCADD Externals should load.



A checkmark precedes the names and version numbers of all currently loaded (active) Externals. All of the Externals can be activated or deactivated by clicking Select All or Select None respectively. To return to the status they had before the dialog appeared, click Revert.



NOTE: Choose Quit from the File menu and launch PowerCADD again for the changes made in Loader to take effect.



To turn an external on or off, select the checkmark next to the external's name. After you've specified which externals you want on and off, you can save those settings as Group.

To make a custom external Group:

- ☐ Click on the Groups pop-up menu.
- ☐ Choose New Group.
- ☐ Name the group and click OK

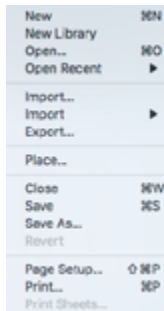
To load a saved External Group simply choose it from the Groups pop-up list. You will need to quit then relaunch PowerCADD to load that group of externals. The default groups (PowerCADD, Architectural, Engineering, Translators, Drawing, Modifying, Other) cannot be edited or deleted from the list.

To Delete a custom external Group choose the Delete Group menu item, select the group name and press Delete.

If you have installed third party PowerCADD Externals, choose PowerCADD to see only the standard PowerCADD suite of tools and menu options.

Services	Services is an Apple menu item. All items listed in the Services menu and submenus pertain to Apple MacOS level features and are unavailable when in PowerCADD.
Hide PowerCADD	Hide PowerCADD will hide all PowerCADD drawing windows, tool palettes and floating windows. The application and drawings are still running but are hidden from view. To bring the PowerCADD windows back, click on the PowerCADD icon in the Dock or choose Show All from any other application menu.
Hide Others	Hide Others is an Apple menu item. Choosing Hide Others will hide windows and palettes associated with all other applications currently running in MacOS except PowerCADD.
Show All	Show All is an Apple menu item. Choosing Show All will show all windows and palettes associated with all applications currently running in MacOS. If another application was hidden it will be come visible.
Quit PowerCADD	<p>Quit PowerCADD leaves the current drawing and closes PowerCADD.</p> <p>To quit PowerCADD and the current drawing:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Choose Quit PowerCADD from the PowerCADD menu. <input type="checkbox"/> If you have not saved the current drawing, you will be asked to do so before proceeding to Quit. <p>If you have multiple drawings open which have changed, a warning dialog will open listing the number of open documents and request the next course of action.</p>
<i>Discard Changes</i>	Will close all open drawings without saving any changes. All changes made to a drawing since it was last saves are lost.
<i>Cancel</i>	Will abort the Quit operation and return you to the drawing session.
<i>Review Changes</i>	Will allow you to review each drawing and confirm whether changes should be saved or ignored.

File Menu



The File menu consists of PowerCADD commands for working with a drawing file as a whole as compared to other menus which allow you to work with specific objects in a drawing.

New

New creates an original, unused PowerCADD document from either a customized Default Stationery pad or a default untitled document.

To create a new document:

- ☐ From the File menu choose New.
- ☐ A default untitled document is opened to being drawing in.

Using Stationery

Using Stationery allows you to choose a custom start-up document when choosing New from the File menu. Think of a stationery drawing as a template to quickly get you started. The stationery drawing may contain common drawing elements such as sheet size, a drawing title block and other common attributes such as line weights, colors or drawing scales.

Opening Stationery

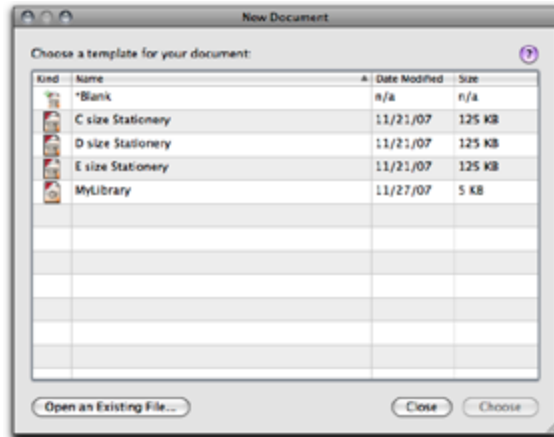
To open a custom stationery pad:

- ☐ Add a PowerCADD drawing to the Stationery folder as described later in this section.
- ☐ Choose New from the File menu or press Command-N and the New Document dialog will open.
- ☐ Select the desired stationery file from the Name list and press Choose or double-click on the file name in the list.
- ☐ A copy of the selected stationery file opens to begin drawing in.



NOTE: Pressing the Option key and choosing New from the File menu will temporarily override stationery choices in the New Document dialog and open a default untitled document (*Blank as shown in the New Document window).

Open New Document Dialog



Click the help button to read additional information about this dialog.

The body of the window displays a list of available files in the Stationery folder specifying the **Kind** of document, the document **Name**, the **Date Modified**, and the documents **Size**. To resize columns, click and drag the line dividing the column heading names. To change the position of the columns press Option and drag the column name. Clicking a column name will sort the list based on that column; click on the same column name to switch the sort between ascending and descending order.



NOTE: If there is only one item in the Stationery folder, that file will open, and if there are no items in the Stationery folder choosing New from the File menu will open the default blank, 8.5x11 inch untitled document.

Open an Existing File...: Click to display a standard MacOS Open dialog and open an existing file.

Close: Click this button to close the dialog and not open a stationery drawing.

Choose: This button will be activated when a stationery drawing is selected in the list. Click Choose or double-click a file name to open it.

Adding Stationery

Custom stationery files are saved in the PowerCADD Application folder, Stationery folder.

To create a custom stationery pad and have it appear in the New Document dialog:

- ☐ Start with any PowerCADD drawing that contains the standard sheet size, layers, line weights, colors and objects you would like to use at the start of every drawing.
- ☐ Save and close the drawing.
- ☐ In the Finder, move the PowerCADD drawing file you want added to the New Document list into the PowerCADD Application folder, Stationery folder.
- ☐ The next time you choose New from the File menu the file will appear in the New Document dialog.

To remove a stationery drawing from the New Document simply remove the document from the PowerCADD application folder, Stationery folder. The Blank file is a generic Untitled document and cannot be removed from the list. The Stationery folder may contain PowerCADD documents, PowerCADD Symbol Libraries, or any other type of file PowerCADD can open. Changes made to the Stationery folder are immediate, you do not need to quit and relaunch PowerCADD to see the change.

Working with Stationery in a Networked Environment

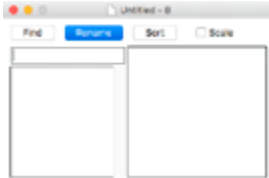
If you are in a network environment and want to keep your stationery drawings on a network server or networked hard drive, use an alias (pointer) to the file. For example, to use a file on your network server, connect to the share point and make an alias of the file into your local PowerCADD folder, Stationery Folder.



CAUTION: Use *extreme* caution when working with stationery which is stored on a server. An existing stationery open file can be overwritten, that is, a stationery file can replace an existing stationery file of the same name. To be safe, *always* make a backup of your stationery in locations other than the server.

New Library

Symbol Library

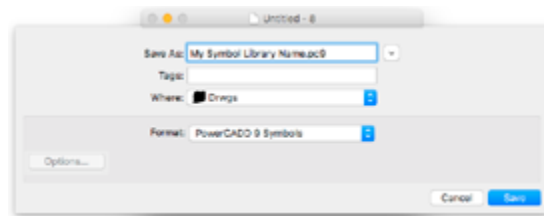


New Library creates an original unused library suitable for storing symbols.

A Symbol Library will allow you to create frequently used objects and save them in a library format for later use. The symbols you paste in a library are available anytime you are using PowerCADD. Once the symbol is created and pasted in a library you will never have to draw the object again; all you need to do is open the library, choose a symbol and insert it into your drawing.

To create a Symbol Library:

- ☐ Create an object on the screen using standard PowerCADD methods.
- ☐ Select the items you wish to save as a symbol.
- ☐ Choose Copy from the Edit menu.
- ☐ Choose New Library from the File menu and a new Symbol Library will open.
- ☐ Choose Paste from the Edit menu and the items you selected in the drawing area will be grouped and displayed in the right-hand field of the window.
- ☐ A name has been given to the symbol which is highlighted in the left-hand side of the window. To rename the symbol, type in a new name while both fields are still highlighted and click the Rename button. The name you entered will be assigned to that symbol.
- ☐ While the Symbol Library window is active, choose Save from the File menu. Choose PowerCADD 9 Symbols from the Format pop-out as the following dialog shows:
- ☐ Enter the name of the new library and click Save. The library has now been saved as a file that may be recalled later.



- ☐ Click in any portion of the drawing window that is visible to return to the drawing area.
- ☐ Create another symbol and select all items that pertain to the symbol.
- ☐ Choose Copy from the Edit menu.
- ☐ To bring the open library window to the front, choose the name you assigned to the new library from the Window's menu. The library window will be brought to the front, and you may paste in the second symbol as described above.

You may place as many symbols in the Symbol Library as you wish. The larger the library becomes, the more memory it takes to open it.

Deleting Symbols

To Delete a symbol from a Symbol Library:

- ☐ With the Symbol Library window active, select the symbol you wish to delete.
- ☐ Choose Cut or Clear from the Edit menu. A warning message will appear.
- ☐ Click OK and the symbol will be deleted.



NOTE: If the symbol to be deleted is part of another symbol, creating a nested symbol, it cannot be deleted. The original (nested) symbol must be deleted first.



NOTE: If the objects you copy reside on two or more different layers, which have different layer scales, the objects will be pasted into the library according to the current Drawing Scale. A message will display letting you know the objects are at a different scale. If all objects selected have the same scale, the objects will be pasted into the library at their scale. If the active layer has a Layer Scale set and all objects are on that layer, the symbol will be inserted into the library at the layer's scale.



For more information on Layer Scale, see [Layers](#).

Symbol Palette

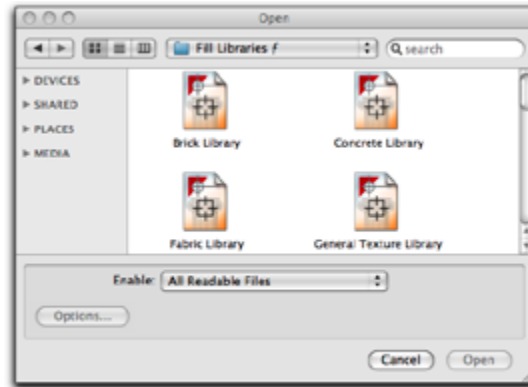


An alternative method of placing symbols is with the Symbol palette. Since it is an External, it must be loaded in order for it to appear. The Symbol palette displays your library as a row of thumbnail views of your symbols as illustrated.

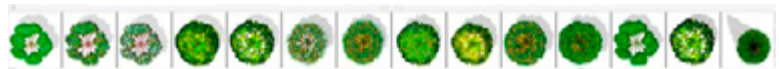
For more information on Symbol Palette, see [Symbol Tool](#) in the Tool chapter.

To recall a symbol from a Symbols Library:

- ☐ Choose Open from the File menu and a dialog will appear:



- ☐ Locate and select the desired library. From the Enable pop-up menu, choose the type of PowerCADD Symbol Library you would like to open. If you are not sure what version of PowerCADD was used to create the symbol library, choose All Readable Files. Click Open (or double-click on the library name.)
- ☐ The Symbols Library window will appear with the first symbol selected.
- ☐ Select the correct symbol by using the scroll bar and clicking on the name or use the Find option.
- ☐ Once the correct symbol is visible in the right-hand box click on an exposed section of the drawing window to return to the drawing area.
- ☐ Select the Symbol tool from the Tool Palette.
- ☐ Place the cursor in the general location you wish to place the symbol and click. The symbol will be placed in the drawing according to its reference point.



- ☐ You may continue to place additional symbols in the drawing by simply clicking in the correct location. Every symbol placed will be the same until you bring the Symbol Library window to the front and change the symbol that is showing in the right-hand field.

When several items are selected in a drawing and pasted in the library as a symbol, the items are automatically grouped. The symbol may be ungrouped after being placed in the drawing to allow for modification.

Symbols are inserted in the drawing by the symbol's center. A reference point may be used as the symbol insertion point. For example, each time you create a symbol, place a Reference Point at a strategic location in the symbol. If the Reference Point is the last item created in the symbol, the point in the symbol will correspond the insertion point. The Reference Point will show on the screen, but it will not plot or print.

The symbol is inserted with a click or a press and drag action. A symbol will show a ghost rectangle while being placed. Holding the Option key while placing allows you to see all of the objects in the symbol.



Open...

For more information on working with symbols, points and flipping or rotating symbols see Tool chapter, [Point Tool](#) and [Symbol Tool](#).

Open presents you with a dialog allowing you to retrieve documents in other formats, depending upon which Externals are in the Plugins folder.

To open previously created and saved PowerCADD documents:

- ☐ From the File menu choose Open.
- ☐ A dialog will appear.



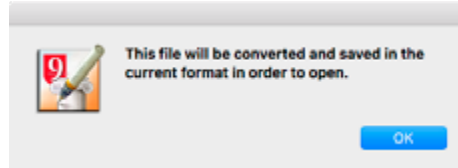
- ☐ Double-click on a file to open it or click once on the file and click Open.

Legacy PowerCADD Drawings and Symbol Libraries

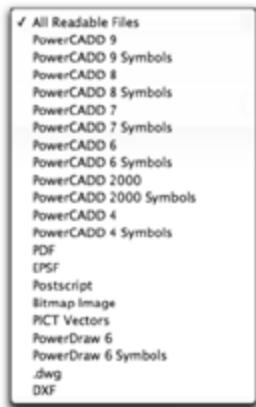
Legacy PowerCADD drawings and symbol libraries are automatically upgraded to the most current PowerCADD file format when opened.

To open legacy PowerCADD documents or symbol libraries:

- ❑ Open the file as described above in this section. The following dialog will appear:



- ❑ Click Cancel to abort the operation. The legacy file is not opened.
- ❑ Click OK and the file is opened and a copy of the legacy file is automatically saved in the same location as the original legacy drawing or symbol library. There is no need to choose save. You can just close all windows.



PowerCADD 9 files and PowerCADD 9 Symbols are the normal method of storing drawings for subsequent use within PowerCADD. Library files are Symbol Libraries which can be opened in the drawing to add previously created symbols. You may also open documents and libraries created from older versions of PowerCADD or PowerDraw by selecting the corresponding file type from the list.

PDF documents (*.pdf) are files created in Adobe Acrobat PDF format. By default, the content of a PDF file cannot be edited. To convert a PDF for editing, choose PDF To Objects from the Tool menu.

EPSF files (*.eps) are Encapsulated Postscript Files created by many desktop publishing and illustration applications. By default, the content of an EPSF document cannot be edited. To convert an EPSF file for editing, choose PDF To Objects from the Tool Menu.

Postscript (*.ps) is a page description language commonly used for printing high quality text and graphics. Postscript files which are saved (as opposed to sending to a physical printer) can be opened but by default cannot be edited. To convert a Postscript file after opening it, choose PDF To Objects from the Tool menu.

The Alpha Channel (transparency) of bitmap images imported is retained. Imported PDF (*.pdf), EPSF (*.eps), illustrator (*.ai) or Postscript (*.ps) files may be converted to objects for subsequent editing.

Bitmap Image files (*.bmp, *.pict, *.jpg, *.tif, *.psd, etc.) are raster (pixel based) image files created by various image editing applications.

PICT Vectors (*.pict or *.pct) files are generally restricted to screen resolution and represent a somewhat universal method for transferring data between most drawing programs and other applications.

If you have QuickTime installed and have the Graphics Export External loaded, you can also open the following raster file formats: SGI, PhotoShop, BMP, JPEG, PICT Bitmap, PNG, MacPaint, TIFF, TGA, JP2 and QuickTime Image format.

DXF, dwg and the other translating options are Externals included with PowerCADD designed to open those specific file types. These Externals must be checked in Preferences, Loader for them to be available for use.

Network Servers

If your files are saved and opened from a network server, open and saving performance will vary with your network configuration. See your local network administrator for particular procedures regarding working with files across the network or stored on your network server.

Open Recent

Open Recent lists the last files which have been opened and saved by PowerCADD.

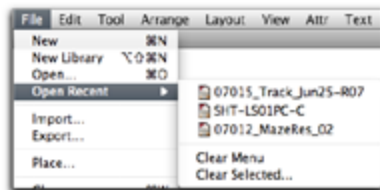
To open a recently saved file which has been closed:

- ☐ Choose Open Recent from the File menu.
- ☐ Select the file name from the list.

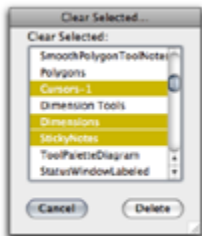
Clearing All Recent Items

To clear the list of recently opened items:

- ☐ Choose Open Recent from the File Menu.
- ☐ Choose Clear Menu from the Open Recent submenu.



Clearing Selected Recent Items

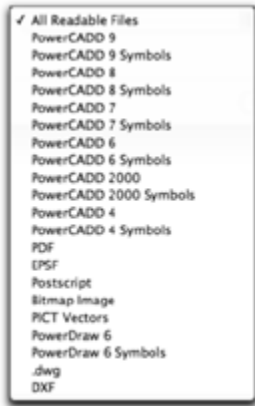


To clear selected items from the list of recently opened items:

- ☐ Choose Open Recent from the File Menu.
- ☐ Choose Clear Selected... from the Open Recent submenu. The Clear Selected dialog will appear.
- ☐ Select the file name(s) to be deleted and click Delete. To delete consecutive files, Shift-click on the file names. To delete non-consecutive files, Command-click on the file names.
- ☐ Click OK to close the dialog box.
- ☐ The selected file names are removed from the list of recent files.

Deleting items from the list cannot be undone. To add an item back into the list of recent files open and save the file.

Import



Import allows you to import a variety of file formats into an existing drawing.

File formats available for import include all the file types listed in the Open command with the exception of Symbol Libraries.

To import a file into a drawing:

- ☐ Open an existing drawing or make a new PowerCADD drawing.
- ☐ Choose Import from the File Menu.
- ☐ The Import dialog box will open. Navigate to the file to be imported.
- ☐ Double-click on the file to be imported or click on the file name and click Open.
- ☐ The selected file is imported into the drawing.

The type of editing you can perform on the imported file will depend on the type of file. For example, an imported PowerCADD or .dwg file will have objects which may be edited.

An imported PDF or QuickTime Raster formats such as JPEG may only be cropped or rotated. Bitmap effects may be applied to imported raster images by first using the Objects to Bitmap command in the Tool menu.

When Importing .dwg, .dxf or PowerCADD files which contained information on multiple layers, all data on visible and invisible (hidden) layers is imported into the current active drawing layer.



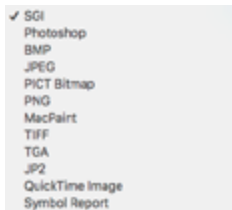
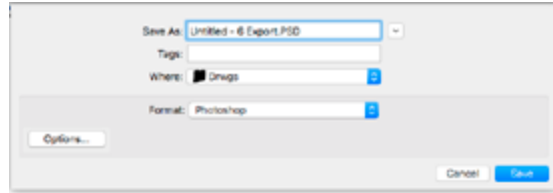
The Alpha Channel (transparency) of bitmap images imported is retained. Imported PDF (*.pdf), EPSF (*.eps), illustrator (*.ai) or Postscript (*.ps) files may be converted to objects for subsequent editing.

Export

Export allows you to save a drawing, or portions of the drawing in a variety of QuickTime raster file formats or generate reports for data in a drawing.

To use Export:

- ❑ Choose Export from the File Menu and the following dialog will appear:

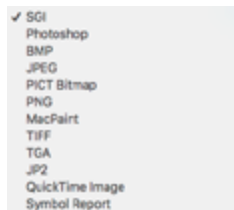


- ❑ In the Save As field type a name for your file. As a default, the word 'Export' is automatically appended to the name.
- ❑ Click on the Format pop-up menu to choose a file format. The correct suffix for the file format is automatically appended to the file name.
- ❑ From the Where pop-up specify the location you'd like to save your file.
- ❑ If you'd like to set optional settings click Options, otherwise click Save.

If the Options button is dimmed then no options are available for the selected export format.

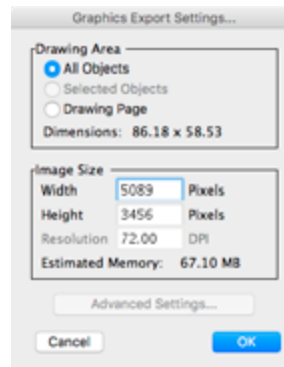
- ❑ The export operation is completed, and you are returned to the active drawing window.

Exporting Raster File Formats



The Graphic Export External must be loaded in order to export a drawing or portions of a drawing, in a raster file format. Some raster file formats are linked to Apple's QuickTime software on your computer. If some of the raster file formats shown here do not appear in your list then visit Apple Computers website to learn more about installing any required updates to QuickTime.

The raster file formats have several optional settings providing additional control over the file size, resolution, and quality of the exported file. Optional settings are remembered until you change them. To access optional settings, click Options in the Export dialog box and the following dialog will appear:



Drawing Area settings allow you to specify how much of a drawing is exported.

All Objects will export every visible object including objects off the drawing page except point objects.

Selected Objects will export all visible objects which fall within the bounding box of the selected object or objects. For example, with one rectangle selected, all visible objects which are inside the selected rectangle are exported. This method is an excellent way to export only a portion of a drawing for use on a web page, emailing or use in a report.

Drawing Page will export all visible objects which fall on the white drawing area.

Dimensions are listed detailing the size of the drawing area being exported. Dimension units are as specified in Drawing Setup, Units. For example, if the drawing is setup for millimeters the dimensions are listed in centimeters. If the drawing setup is for feet and inches the dimensions are listed in decimal inches. Reviewing the dimensions is an excellent method to cross check just how large the area is you are exporting.

Image Size settings allow you to specify how many pixels are exported which affects the images physical dimensions and file size.

Width is expressed in pixels matching the width of the exported area. Changing the width value will automatically update the Height and Resolution fields to maintain the correct aspect ratio. Not all export formats support changing this value.

Height is expressed in pixels matching the height of the exported area. Changing the height value will automatically update the Width and Resolution fields to maintain the correct aspect ratio. Not all export formats support changing this value.

Resolution is expressed in Dots Per Inch. Changing the Resolution value will automatically update the Height and Width fields to maintain the correct aspect ratio. A value of 72 DPI will export the image at 1:1 (e.g. an 8.5 x 11 page will be 8.5 x 11 inches). A value of 144 DPI will export the image twice as large and there will be twice as many pixels. A value of 36 DPI will export half the size with half as many pixels. Not all export formats support changing this value.

Estimated Memory will display the estimated memory required to export the file based on the Resolution, Height and Width Settings. If you have insufficient memory, the entire file may not be written or the file may not open correctly. It is always a good idea to preview the Estimated Memory to gain a sense of how large the file will be when exported and saved to disk. This can be particularly important if you are exporting graphics for web or email or other situations where large file sizes may not be practical.

The **Advanced Settings** button provides access to additional settings which are unique to each export format. Options include changing the bit depth or amount and type of compression applied to the exported image.

Exporting Drawing Formats



PowerCADD provides various methods to add text based descriptions to individual objects, grouped objects, or symbols in a drawing using Symbol Tags. Think of these descriptions as special data tags you might assign to objects to generate cost estimate, bills of materials, quantity take-offs or perhaps even use for facility management purposes. Once the data tags have been assigned you can use Export to generate a plain text file report which can be opened in other applications to complete your value added analysis.

It may be helpful to think of PowerCADD as a graphical front end to a database. It is often faster, easier and in some cases more accurate to make changes graphically, then export a report and import that data into your spread sheet or other third party application. If you use standard templates on various projects, it is even possible to have graphs or other detailed calculations automatically update when you import an updated report.

All this is obsolete?

Exporting Symbol Reports

Symbol Report creates a text file listing all symbols or groups in a drawing. The report includes a count of the objects, their name, and up to five additional text fields. The Set Symbol Tags External must be loaded for this export option to be available.

The report lists items:

- ☐ Inserted as symbols from a Symbol Library
- ☐ Inserted as groups from a Symbol Library
- ☐ Normal groups in a drawing that have been assigned names or text attributes using the Set Symbol Tags command
- ☐ Items inserted from a Symbol Library as a group or symbol that had previously been assigned additional data tags using the Set Symbol Tags command

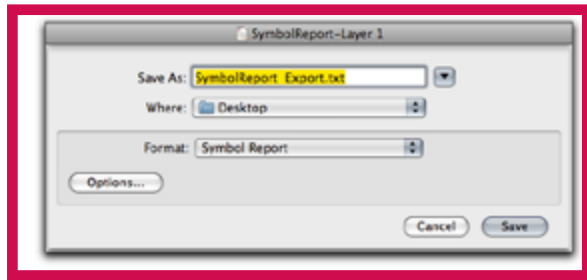
The report heading consists of the title of the drawing and whether the reported items are from all layers or only visible layers. Only items on visible layers are reported.

The remainder of the report is divided into two sections: Symbols and Groups. The group section lists items inserted from a Symbol Library as groups and groups that have been assigned names or additional Symbol Tags in the drawing. For each item, the name and number of occurrences is given. Any information attached to instance symbols is also reported.

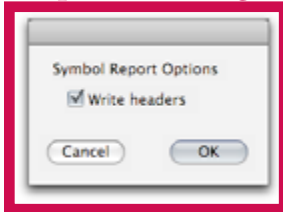
To export a Symbol Report:

- ☐ Choose Export from the File menu. The following dialog will appear:

Menus



Symbol Reports Optional Settings



- ☐ Choose Symbol Report from the Format pop-up menu.
- ☐ Enter a name for the report and click Save.

To access the optional settings:

- ☐ Choose Export from the file menu and click Options.
- ☐ The Symbol Report Options dialog will appear.
- ☐ Checking Write Headers will write all data fields assigned with Set Symbol Tags and a description at the start of the report.
- ☐ Click OK to close the dialog box.

These settings remain in effect until you change them. Examine the examples below of reports with Write Headers checked and unchecked, then opened in Excel.

*Write Headers
Not Checked*

	A	B	C
1	Chaenomeles japonica	24	
2	Lonicera involucrata	29	
3	Pinus mugo 'pumilio'	27	
4	Bench	13	
5	Trash Can	3	
6	Tree Grate	6	

*Write Headers
Checked*

Write Headers Checked.txt						
	A	B	C	D	E	F
1	SYMBOL REPORT					
2	Drawing: Your Drawing Name Here					
3	* This report contains all items.					
4						
5	Symbol Name	Count				
6	-----					
7	Chaenomeles japonica	24	Red Flowering Quince	5 gal.	container	
8	Lonicera involucrata	29	Twinsy Honeyuckle	5 gal.	container	
9	Pinus mugo 'pumilio'	27	Shaw's Mugo Pine	2 gal.	container	
10	Pinus mugo 'pumilio'	2	Pumilio Ash	2" cal.	sho er w.b.	single stem - 7' crotch
11	Acer platanoides 'Emerald Queen'	4	Emerald Queen Norway Maple	2" cal.	sho er w.b.	single stem - 7' crotch
12	Cercis canadensis	8	Redstem Blackbud	2 1/2" cal.	sho	single stem
13	Pinus nigra	5	Austrian Pine	4" min. ht to 6' ht.	sho er w.b.	full & bushy
14						
15	Group Name	Count				
16	-----					
17	Bench	13				
18	Trash Can	3				
19	Tree Grate	6				

Exporting a Symbol Report for Only Selected Objects

- ☐ To export a symbol report for only selected objects in a drawing:
- ☐ Use any combination of selection techniques to select the objects to be reported.
- ☐ Choose Export from the File menu. Choose Symbol Report from the Format pop-up menu and click Save.
- ☐ A confirmation dialog appears as pictured here.



- ❑ Click OK to export a report for only the selected objects. Click Save All to export a report for all visible layers in the drawing; the selected objects will be included in the report. Click Cancel to abort the export symbol report operation.

If no objects are selected all visible layers are automatically included in the exported report and the confirmation dialog will not appear.

Place...

Place allows the placement of PICT, Encapsulated PostScript (EPSF), paint (bit image) files, Adobe PDF files and QuickTime supported file formats into PowerCADD drawings. A placed graphic can be changed in a limited number of ways.

Place provides an efficient way to put various graphic files, including company logos, site plans, details, maps or digital photos into a drawing.

A placed graphic may be reduced to fit a standard sheet size and transferred into a word processing document while maintaining its high degree of accuracy. Placed graphics are also useful for view-only projects, for which modifications are not desirable.

Placed graphics cannot be edited. However, they can be scaled (resized), moved, printed, cropped, and rotated. Scaling and moving are done with standard PowerCADD techniques. To scale a graphic proportionally, press the Shift key while resizing or use the Scale command from the Edit menu.

While the Placed graphic cannot be edited, you may apply Bitmap effects by first using Convert to Bitmap from the Tool menu. If the Placed object has been cropped, converting it to a bitmap makes the crop operation permanent. New cropping and uncropping operations may be performed on the new bitmap.

Place has several major advantages including:

- ☐ Place saves memory.
- ☐ Place protects graphics from editing.

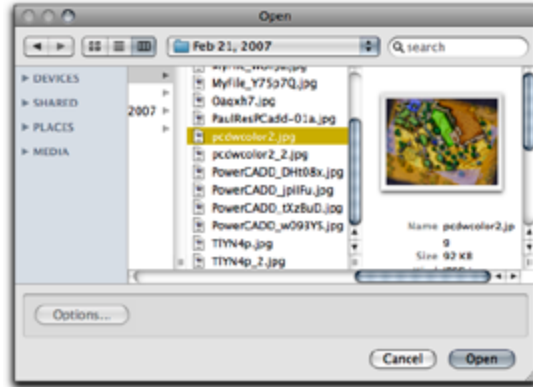
A placed PICT file uses about eight times less memory than an imported PICT file. For instance, a one-megabyte file may occupy only 150K when placed. Therefore, with Place, scanned images that could otherwise not be opened due to memory restrictions are accessible.

When a placed graphic is scaled, its text—not just the text location—is scaled also. With this feature, an E-size drawing can be saved as **PICT (??? obsolete???)**, placed, and scaled to fit an 8-1/2 x 11" sheet of paper. Details can also be isolated, placed, and printed on a single sheet.

Since internal changes to a placed graphic are not possible, most accidental and unauthorized alterations are prevented. Thus, Place can be used to create view-only documents. You can also combine a series of pictures into a single document in order to view a project at various stages.

To use Place:

- ❑ Choose Place from the File menu. A dialog appears.



- ❑ Choose a PICT, Paint, EPSF, PDF or any QuickTime format file and click Open. A cursor representing the upper-left corner of the graphic will appear.
- ❑ Move the cursor to the desired location and click. The graphic appears.

EPSF files are often saved with a preview image, which is used to display the file on the screen. For EPSF files that do not contain a preview image, a gray box represents the graphic on PowerCADD's screen. To view the graphic, print the drawing.

Cropping

Unlike scaling, cropping does not resize a graphic. Instead, cropping trims the graphic. When you crop a graphic, the entire graphic is retained. The cropped portion can be restored at any time.



To crop a graphic:

- ☐ Select the graphic.
- ☐ Press the Command key, place the cursor on an edit handle, and the Crop cursor appears.
- ☐ Drag to start. A ghost rectangle appears and a preview of the cropping operation is shown.
- ☐ Drag the corner of the rectangle to crop the picture. When you release the mouse button, the graphic is cropped along the edges of the rectangle.

To restore a cropped portion (uncrop), hold the Command key and drag a corner of the rectangle until the rectangle is large enough to include the cropped portion.



TIP: Cropping is constrained to maintain the original aspect ratio of the image. To unconstrain the operation, press Shift-Command while dragging a corner handlebar of the selected object. To crop an object about a mid-point handle bar, press Shift-Command.



TIP: The Alpha Channel (transparency) of bitmap images is retained when using Place to insert bitmap images. Placing PDF (*.pdf), EPSF (*.eps), illustrator (*.ai) or Postscript (*.ps) files may be converted to objects for subsequent editing using PDF to Objects from the Tool menu.



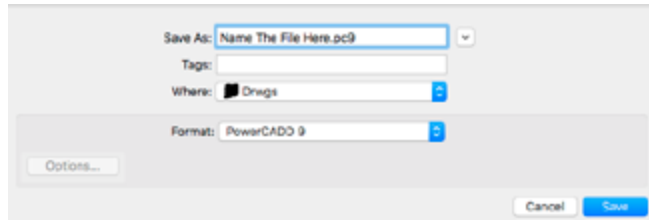
TIP: You can also crop bitmap objects using the same technique as described for Placed objects.

Close

Close shuts the current window.

To close a PowerCADD document:

- ☐ Choose Close from the File menu, or press Command-W, or click the drawing close button in the top left corner.
- ☐ If you have not saved the current file immediately before closing, a dialog will appear asking if you would like to save before closing. The file is saved to the same location from which it was opened.
- ☐ If the file had been previously saved, pressing the Return key or clicking Save will save the file and close it. If this was a new document, pressing the Return key or clicking Save will open a dialog prompting you to name the file and specify a location prior to saving.



Formats for saving depend upon which External's are checked in Preferences, Loader. The standard formats include PowerCADD 8 and PowerCADD 9.

Save

Save stores the current PowerCADD file to a specified disk under the most recently designated file name.

After a document is saved, the current drawing is left open for further use.

To Save a PowerCADD file to the file folder from which the document was opened:

- ☐ Choose Save from the File menu.



NOTE: PowerCADD will prompt you periodically to remind you to save the current drawing. Save is not automatic, and you will have to respond Yes or No by selecting the appropriate buttons.



TIP: PowerCADD 9 files are saved with a Preview which may be viewed in the MacOS Finder using Quick Look. The preview presented in Quick Look is the last saved state of your PowerCADD 9 drawing. Legacy PowerCADD drawings will need to be opened and saved as PowerCADD 9 to take advantage of Quick Look. For more information on using Quick Look see Apple's help and support files included in MacOS.

Save As...

Save As allows you to store the current PowerCADD document under a new file name, or in a different file format.



To save a current drawing under a different file name.

- ☐ Choose Save As from the File menu.
- ☐ Choose the file format in which you wish to save.
- ☐ Name your document and click Save.

You may choose any file format available from the list. Choose PowerCADD 9 to create a file which may be read only by PowerCADD 9. Choose PowerCADD 8 to create a file which may be read only by PowerCADD version 7 and version 8. Versions of PowerCADD older than version 7 cannot open PowerCADD v7 or v8 files.



For more information on .dwg and .dxf files, see [DXF & DWG Translators](#).

Revert

Revert removes the current drawing and replaces it with the drawing last saved under the same file name.

To Revert to the last saved drawing under the current file name:

- ☐ Choose Revert from the File menu.

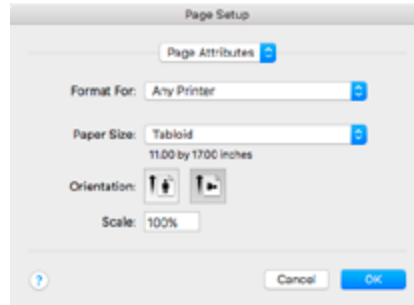
When Revert is used, all undos are cleared.

Page Setup...

Page Setup sets the printing parameters.

To use the Page Setup options:

- ❑ From the File menu choose Page Setup. The following dialog will appear:



Settings allows you to pick from Page Attributes (shown above), custom sheet sizes or displays a summary of your page settings.

Format for allows you to specify the type of printer you want to use. Each printer has unique features including color or different sheet sizes or ink adjustment settings.

Paper Size lets you pick from various sheets sizes based on the type printer picked in the Format for pop-up.

Orientation allows you to set the page to be portrait (vertical) or landscape (horizontal).

Scale allows you to specify a percentage to enlarge or reduce the printed output. A value of 100% is considered 'normal' or 1:1 printing and typically used to print your drawings at the scale specified in Drawing Setup, Units.

The first time you choose Page Setup and click OK, the Drawing Setup Size dialog box will appear allowing you to set various options. PowerCADD supports page sizes up to 200 x 200 inches and provides a preview of your drawing in the Drawing Setup Size dialog.



For more information on Drawing Setup, [Drawing Size](#) and [Print options](#) see [Layout Menu](#). For more information on choosing page sizes and working with margin or tiled sheets of paper, see [How to Get Started](#) in the PowerCADD Tutorial.



Print...

Print sends drawing file information to output devices like ink jet or laser printers or may be used with features native in MacOS to send a fax or save an Adobe Acrobat PDF formatted file.

You may print the active drawing at normal size or print to fit your printer's current page size.

To use the Print command:

- ❑ Once you have established the page setup parameters, choose Print from the File menu. Choose PowerCADD Settings from the pop-up menu and the PowerCADD options will appear:



Printer lets you select the printer to use. The printer selected here should match the printer you selected in Page Setup.

Presets let you save and select from preset printing setups which can include a variety of variables including color ink settings and sheet sizes. PowerCADD specific print settings are not saved with presets.

PowerCADD Optional Settings

The Optional Settings pop-up allows you to specify various optional settings which vary with the printer you have selected. With the latest MacOS, the PowerCADD Optional Settings are displayed automatically when choosing Print.

Save Settings for Standard. When checked, choices set for Print Area, Print To, Fit to Page, Print Time Stamp, print Points, and Gray Pen Pattern for Dimmed or Transparent Layers will be remembered and applied each time you choose Print.

Print Area

Normal will print normally; all visible objects in the drawing are printed. This is the default setting for each printing session.

Selection will print selected items. This setting is off by default but is remembered between printing sessions. Details of Print Selection are described later in this section.

Window will print the area of a drawing displayed in the active drawing window. To print a drawing window, adjust the desired view by zooming, scrolling, and resizing the drawing window or use a preset view. When your window is set, choose Print and click Window. This setting may also be used in conjunction with Fit to Page and Print To setting options. This setting is off by default and is not remembered between printing sessions.

Print Partial Pages when selected will print all pages of your document even if there is only a sliver on a page.



NOTE: When printing a window to fit, the current orientation of the drawing and the shape of the window you specify will affect how much of the printed page is used. For example, if your drawing is landscape and the windowed area you set is a little taller than it is wide, the printed result will be smaller than if the current orientation was portrait. To optimize the printed page, temporarily set the orientation (Page Setup in the File menu) of the drawing according to the shape of the window you have set.



NOTE: The Fit and Window checkboxes will return to their inactive settings and Normal will be selected by default when the print session is complete.

Print To

The Print To settings affect the position of items on the printed page. The selected Print To setting is remembered between print sessions and affects all documents. Center of Printable Area and Center of Paper are particularly useful when using Print Selection, Print Window or Fit to Page optional settings

Default will print objects based on the default margins and positioning as defined by the selected print driver. When selected in conjunction with Fit to Page, Print Window or Print Select, the top left corner of the drawing window is the anchor point for the printed items. This setting is on by default and is remembered between printing sessions.

Center of Printable Area prints objects in the center of the printable area as defined by the page margins. For example, if an 8.5 x 11 inch page has margins of 1 inch left, right and bottom but 3 inches top, the center of the printable area is not equal to the center of the paper because of the larger top margin. Objects will be printed relative to the center of the printable area and may appear off center relative to the physical paper size. This setting is off by default and is remembered between printing sessions.

Center of Paper prints objects in the center of the physical paper size as defined in page setup; page margins are ignored. For example, if an 8.5 x 11 inch page has margins which are 1 inch left, right and bottom and 3 inches top, the objects will be printed in the center of the 8.5 x 11 paper size and may appear off center relative to the printable area due to the unequal margin dimensions. This setting is off by default and is remembered between printing sessions.

If your page margins are equal dimensions or the differences between them are very small you may not perceive any difference between selecting Center of Printable Area or Center of Paper.



TIP: To see the different effects of various combinations of Print Area, Print To, and other optional settings use the MacOS Preview button found in the Print dialog. The previewed drawings may be saved as PDF files for detailed review and kept on file for later reference.

Additional Options

Fit to Page will force the contents of the active drawing to print on the available paper size. For example, you may print an E-sized drawing onto a letter size (8.5" x 11") page. This setting is Off by default and is not remembered between printing sessions.

Time Stamp prints the file name, date, and time at the top center of the page. If the document is tiled on multiple pages, a time stamp will be printed on each page. This setting is Off by default and is remembered between printing sessions.

Print Points prints the reference points in the drawing — this is Off by default and is not remembered between printing sessions.

Gray Pattern for Dimmed or Transparent Layers is an application-wide setting to help address Postscript and PDF workflows and flattening of transparent objects. Due to limitations in how transparent content is rasterized at the operating system level during the Postscript and PDF printing process, the output may appear to become fuzzy or less sharp.

When enabled, the WYSIWYG relationship between screen display and printed output is affected for objects with a Dimmed layer status or objects residing on layers with a Layer Opacity setting other than 100%. Filled patterns are not printed, and text, bitmaps and placed objects are printed as if their opacity were 100% (solid); objects on these layers are printed with a gray pen pattern instead. Settings applied for Color printing options in Drawing Setup, Print tab still apply.

When off or disabled, the transparency setting set when a layer status is Dimmed or Layer Opacity settings are changed will be applied to all objects when rendered to screen and when printed. In the case of non-postscript printers, leaving this box unchecked (Off) is recommended to maintain the WYSIWYG relationship between the screen display and printed output.

Button Row

The buttons along the bottom of the Print dialog will vary depending on which version of the MacOS you are running.



Help button opens Apple's Help View application with additional information on using the Print dialog box.

PDF taps into the MacOS feature set for saving or faxing PDF files or saving PDF as Postscript. The list of options available will vary with your version of MacOS and any third party applications you may have installed.

To learn more about how to take advantage of the various MacOS system level features for creating and working with PDF files, search the MacOS help files in the Finder for 'PDF Services'.

Preview opens your file in Apple's Preview application (or the application you have specified to display PDF files) for viewing prior to printing or saving as PDF.

Supplies launches your web browser and takes you to an online store.

Cancel aborts the printing operation and returns you to the PowerCADD drawing.

Print proceeds with sending the data to the specified printer based on your chosen printing options.



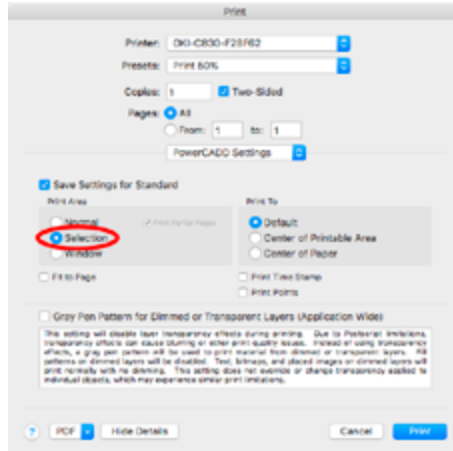
For more information on additional drawing specific print options, see information in this chapter on [Drawing Setup, Print tab](#) and [Drawing Size](#).

Print Selection

Print Selection prints all selected objects based on settings in the Print dialog box. The extents of the selected objects define a clipping area of information that will be printed. All visible objects inside the clipping area are printed.

To use Print Selection:

- ❑ Select the objects in the drawing you want to print and choose Print from the File menu and the following dialog will appear:



- ❑ Note the status of the Print Area, Selection radio button is remembered between printing sessions.
- ❑ Choose any additional options and click Print.

Print Sheets

Print Sheets prints....

Edit Menu

Undo	⌘Z
Redo	⇧⌘Z
Cut	⌘X
Copy	⌘C
Paste	⌘V
Paste Special	
Clear	⌘
Duplicate	⌘D
Select All	⌘A
Select Special...	
Select Adjoining	
Select With Object	
Select Inverse	
Deselect All	⇧⌘A
Edit Group	
Exit Groups	
Edit...	⌘E
Reshape	⌘R
Scale...	
Mirror	
Move	
Shear...	
Start Dictation...	fn fn

The Edit menu contains items that allow you to change the physical appearance or location of an object. You may also duplicate objects or create arrays of many forms.

Some menu items such as Undo, Paste Special, Duplicate, Mirror and Move have special optional settings. Press the Option key when selecting the Edit menu will show three dots beside menu items with optional settings. Pressing the Option key when you choose those menu items will present a dialog box.

Paste Special, Select Special, and Select Adjoining are Externals and must be loaded in order to appear in the Edit menu.

Undo

Undo restores the drawing to its status before the last change to the document.

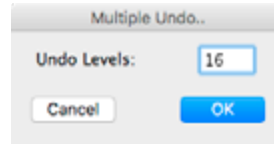
There are multiple levels of undo, which can help you correct mistakes you make while drawing. You can even undo an operation after you have chosen the Save command (but not if you have reverted or closed and reopened the file).

To undo the last action:

- ☐ Choose Undo from the Edit menu.
- ☐ You should immediately see the last command issued be reversed or undone.

To set the number of undo/redo levels:

- ☐ Press the Option key while choosing the Edit menu and Undo changes to Undo Options. Choose Undo Options and the Multiple Undo dialog will appear.



- ☐ Enter the levels of Undo and click OK.

When you change the levels of Undo, any previous changes will be cleared from the Undo buffer, so you will be able to undo only the actions you perform after you had changed the Undo levels.



CAUTION: If Revert is used to restore your drawing to the one last saved, all undos will be cleared.



The default number of undos is set to sixteen. A larger quantity of undos requires more memory. If you find performance feels sluggish or if you are running into low memory conditions, reduce the number of undo levels.

Redo

Redo reverts the last undo made to the document.

Redo, like Undo, has multiple levels which can help you when you mistakenly used Undo several steps back and have now changed your mind. The number of levels set for Undo is also applicable to Redo.

Cut

Cut removes and places the currently selected object on the Clipboard. This has the effect of removing the object from the current layer of the drawing.

To cut an object from the current layer of a drawing:

- ☐ Select the desired object from the drawing.
- ☐ Choose Cut from the Edit menu.

The selected objects will be removed from the drawing.

The method for moving an object between layers is by cutting (or copying) and pasting. Cutting an object, changing layers, and pasting causes the object to be pasted in exactly the same coordinates from which it was cut/copied. The object can then be moved by dragging or other means.



NOTE: To recover from an accidental Cut, choose Undo or Paste from the Edit menu before going any further. Undo or Paste will re-insert the objects which were mistakenly removed from the drawing.

Copy

Copy makes an exact copy of a selected object and places the copy on the Clipboard. The original object is not modified.

To copy a selected object from the current layer of a drawing:

- ☐ Select the desired object from the drawing.
- ☐ Choose Copy from the Edit menu.

An exact copy of selected objects will be made and placed on the Clipboard for your further use.

Copy has the effect of leaving the selected object in the drawing area and placing a duplicate copy of the object on the Clipboard for further use, whereas Cut actually removes the object from the drawing area and places a copy on the Clipboard.



NOTE: Objects Cut or Copied and Pasted between PowerCADD drawings will retain Shadow and Opacity attributes. Shadow and Opacity attributes may be lost when pasting data into other applications.

Paste

Paste places the contents of the Clipboard onto the drawing area.

To paste the contents of the Clipboard onto the drawing area:

- ☐ First, make sure the objects you wish to place in the drawing are on the Clipboard by selecting the objects and performing a Cut or Copy operation.
- ☐ Choose Paste from the Edit menu.

An exact copy of the contents of the Clipboard will be placed on the drawing.

Objects are pasted in the exact same coordinate location from the drawing which they were cut/copied. As they remain selected, you may immediately reposition them.

An object can be moved between layers or between drawings by cutting, copying, and pasting.



NOTE: Paste ignores layer information and objects are always pasted onto the current active layer. To have Paste operations remember layer information use Paste Special.

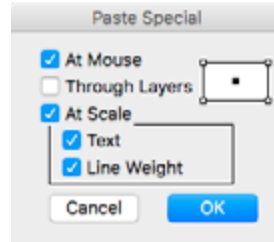
Paste Special...



The Paste Special optional dialog box allows you to paste an object from the Clipboard to a point specified by the mouse, paste at another scale, and paste multiple objects through layers.

Choosing Paste Special allows you to position objects and control certain object attributes when they are pasted into the drawing.

Choosing Paste Special while pressing the Option key allows you specify the specific behavior of the pasting operation. Changes made in the Paste Special dialog remain in effect until you change them.



At Mouse pastes the selected objects relative to the mouse, instead of pasting them at their original x,y location. The rectangle in the upper right of the dialog box is used to set the position of the objects relative to the mouse; the black box indicates the active position. To set the position, click on a white box and it will become black confirming the new active position.

Through Layers maintains layer information when pasting the selected objects. For example, if you copy a group of objects, in which each object is on a separate layer, the objects will be placed on the separate layers. If the layers associated with the object do not exist in the drawing, Paste Special will create the layers to accommodate the pasted object.

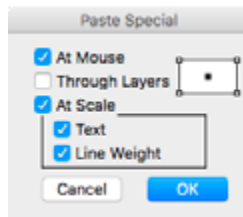
At Scale maintains the dimensions of the selected objects when pasting onto another layer or drawing with a different scale. It will enlarge or reduce their physical size according to the new scale. For example, you can copy an 8' x 8' square from a drawing at 1/4" = 1'0" scale and paste it into a drawing at 1/8" = 1'0". The square will be 8' x 8' at 1/8" = 1'0" scale. This option is extremely useful if you need to copy a portion of your drawing and paste it into another drawing to make an enlarged detail.

At Scale Text will rescale text along with the drawing objects. For example, you can copy text with a font size of 12 points from a drawing at 1/4" = 1'-0" and paste it into a drawing at 1/8"=1'-0". The font size will be 6 points after pasting (one half the size in this example). If Scale Text is not checked the font size is not changed.

Line Weights will rescale the line weight attribute along with the drawing objects. For example, you can copy an object with a line weight of 4 points from a drawing at $1/4" = 1'-0"$ and paste it into a drawing at $1/8" = 1'-0"$. The object will be rescaled and its line weight will also be changed to 2 points (one half the size in this example). If Line Weights is not checked the line weight attribute is not changed.

To use the Paste Special options:

- ☐ Copy the desired object using the Copy command. The Cut command may also be used.
- ☐ Press the Option key and choose Paste Special from the Edit menu. A dialog appears



- ☐ Click one of the placement point locations. The black box indicates the selected point.
- ☐ Click At Scale and Through Layers as desired.
- ☐ Click OK. The object will appear as a bounding box on the screen. The cursor will be positioned on the object according to your specified placement point.
- ☐ Position the object on your drawing by moving the mouse and using a Dynamic Snap, if desired.
- ☐ Click in your drawing where you wish the object to be pasted.

Once you have established a placement point in the Paste Special dialog, the setting will remain until you change it. At Scale and Through Layers will also remain. You will not need to use Option key when activating Paste Special unless you want to change or verify a setting. The settings in the Paste Special options will become the default for Paste Special.



NOTE: The Fence Cut tool supports layers, drawing scale and scaling line weights when using Paste Special.



CAUTION: Objects which were created in a drawing on scale-assigned layers *will not* create new layers with the same attributes when pasted through layers into a new or existing drawing; therefore, if At Scale is also active, layer-scaled objects will be rescaled when pasted. If the new or existing drawing does contain layers identical in name and layer-assigned attributes as the object's origin, and Through Layers is active, the objects will retain their scale.

For more information on editing layer scale, see [Layer](#), “Advanced Layer Attributes” and “Layer Scale”.

Clear

Clear removes the currently selected object from the drawing area without placing a copy of it on the Clipboard.

To clear a selected object from the current layer of a drawing:

- ☐ Select the desired object from the drawing.
- ☐ Choose Clear from the Edit menu.
- ☐ The selected object will be removed from the drawing.

Clear has the effect of permanently removing the object from the currently selected layer of the drawing.

A selected object can also be removed by pressing the Delete or Backspace key.



NOTE: To recover from an inadvertent Clear, choose Undo from the Edit menu before going any further. The Undo command will replace the objects which were mistakenly cleared from the drawing.

Duplicate

Duplicate allows you to create exact duplicates of selected objects in a variety of locations, patterns or arrays.

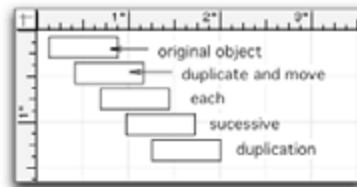
To use Duplicate:

- ❑ Select the object(s) to be duplicated.
- ❑ Choose Duplicate from the Edit menu.
- ❑ When you release the mouse button, the duplicate of the item will be placed directly on top of the original.



NOTE: The direct placement is a powerful feature allowing you to accurately move objects between layers.

Offsets for duplication may be set dynamically using the mouse. Each successive duplication will use the previous offsets. The offsets are reset to zero by selecting a new object.

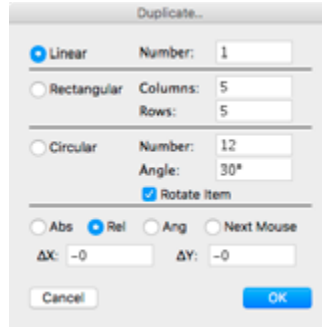


To dynamically specify an object's duplication offset:

- ❑ Draw a small rectangle or pick an existing object.
- ❑ Press the Option key and drag the object. Dynamic snapping may be used to specify a precise location if desired.
- ❑ The duplication cursor appears, release the mouse to duplicate the object in the new location.
- ❑ Press Command-D three more times, and you will have five equally spaced objects.



NOTE: Duplicate has three periods beside it in the Edit menu when you press the Option key. If you press the Option key and choose Duplicate, the following dialog will appear. You may place multiple duplicate objects in a linear, rectangular, or circular array.



Linear

Linear duplication will duplicate objects and space them evenly apart in a straight line. This line of objects can be set at any angle or in a flat plane.

To perform a Linear duplication:

- ☐ Select the object(s) to be duplicated.
- ☐ Press the Option key and choose Duplicate from the Edit menu.
- ☐ Select Linear in the dialog, and enter the number of duplications.
- ☐ Select a method of duplication:

Abs - Absolute allows you to specify an absolute location where the center of the first duplicate will be placed. The remainder of the objects will be placed according to the change in distance and angle from the original.

Rel - Relative allows you to specify the distance from the original, the first duplicate will be placed. The remainder of the objects will be placed according to the change in distance and angle from the original.

Ang - Angle allows you to specify where the first duplicate will be placed by specifying the distance and the angle/direction from the original. The remainder of the objects will be placed according to the change in distance and angle from the original.

Next Mouse - Allows you to specify where the first duplicate will be located by placing the cursor on the desired insertion point and clicking. The point specified refers to the upper left hand corner of the object. If you hold the mouse button when selecting the point, you will be able to see a dimmed object which will aid in placement. The point will not be established until the mouse button is released. The remainder of the objects will be placed according to the change in distance and angle from the original.

- ☐ When you click OK, the duplication will take place unless you selected Next Mouse.

A Rectangular duplication will place duplicated objects in rows and columns, evenly aligned vertically and horizontally. You specify the number of rows and columns needed.

To perform a Rectangular duplication:

- ☐ Select the object(s) to be duplicated.
- ☐ Press the Option key and choose Duplicate from the Edit menu.
- ☐ Select Rectangular in the dialog and enter the number of rows and columns.
- ☐ Select a method of duplication:

Abs - Absolute allows you to specify an absolute point where the center of an alignment object will be placed. The alignment object represents the object on the second row and the second column. This object will be diagonal from the original, but it specifies the spacing between columns and rows.

Rel - Relative allows you to specify the distance, in X,Y coordinates, from the original object's center to the center of the alignment object. The alignment object represents the object on the second row and the second column. This object will be diagonal from the original, but it specifies the spacing between columns and rows.

Ang - Angle allows you to specify the location of the center of the alignment object with respect to the center of the original. This point is determined by specifying the distance from the original object's center to the alignment object's center and the angle/direction of the move. The alignment object represents the object on the second row and the second column. This object will be diagonal from the original, and it specifies the spacing between columns and rows.

Next Mouse - Allows you to manually select a point which will represent the upper left hand corner of the alignment object. This point is specified by placing the cursor in the desired location and clicking. As soon as the location is selected, the rows and columns are created. The alignment object represents the object on the second row and the second column. The point will not be established until you release the mouse button. This object will be diagonal from the original, but it specifies the spacing between columns and rows.

- ☐ When you click OK, the duplication will take place unless you selected Next Mouse.

A Circular array will duplicate objects around the perimeter of a circle with their centers aligned with the circle's circumference line. You specify the number of objects and the angular increments that separate them.

To perform a Circular duplication:

- ☐ Select the object(s) to be duplicated.
- ☐ Press the Option key and choose Duplicate from the Edit menu.
- ☐ Select Circular in the dialog and enter the number of objects to be created.
- ☐ Enter the spacing you wish to have between the objects. If the array will cover 360° and you are not sure of the increments, enter 360°/the number of objects. The calculation will be done internally. A positive value will rotate the objects counterclockwise while a negative value will result in a clockwise rotation.
- ☐ Select a method of duplication:

Abs - Absolute allows you to specify a point on the screen, using X,Y coordinates, that represents the center of a circle where the object's center lies on the circle perimeter. The remainder of the objects will be placed counterclockwise according to the angle increments you specify unless the angle increments were negative, which would result in a clockwise rotation.

Rel - Relative allows you to specify the circle center by entering X,Y coordinates which represent the distance from the object's center to the circle center. The remainder of the objects will be placed counterclockwise for positive angles.

Ang - Angle allows you to specify the circle center by determining how far and at what angle/direction the circle center is from the object center. The remainder of the objects will be placed counterclockwise for positive angles.

Next Mouse - Allows you to specify the center of the circle by manually selecting with the cursor. If you hold the mouse button down after selection, you can move the mouse and see where the circle will be placed. The point will not be established until you release the mouse button. The remainder of the objects will be placed counterclockwise for positive angles.

- ☐ When you click OK, the duplication will take place unless you selected Next Mouse.
- ☐ There is a Rotate Item option, which determines if the object will be rotated as it is duplicated around the circle or if it will remain in its horizontal position as it is duplicated.

The settings will remain in effect until you change them.

Math in Dialog Boxes

Math can be performed in any dialog box which accepts keyboard-typed input.

To use math in the dialog boxes to create a Circular array:

- ☐ Select the object(s) you wish to duplicate.
- ☐ Press the Option key and choose Duplicate from the Edit menu.
- ☐ Select Circular and enter 15 in the number field.
- ☐ If you do not know the exact increments to place 15 objects evenly around a circle, enter an equation instead of angular increments: $360/15$.
- ☐ When you click OK, PowerCADD will calculate the equation and place the duplicate items in the correct location.

You may also enter equations having primary and secondary functions such as $(2+4)-(3*6)$ or items of different measurement systems such as: $34''/66\text{mm}$.

Here are some possible combinations of mathematical equations:

$$17/256 + 1/8$$

$$15''*(3''+24\text{mm})$$

$$2.549/12\text{mm}$$

$$13/254 + 16\text{mm} - .2568 / 56\text{mm}$$

The abbreviations are:

' = feet

" = inches

mm = millimeters

M = miles

cm = centimeters

m = meters

km = kilometers

Parentheses may be used to specify priority. Multiply and divide are evaluated before add and subtract.

Select All

Select All selects all objects contained in the current layer of a PowerCADD document. If Edit All Layers is active, all objects on every visible layer will be selected.

Select Special...

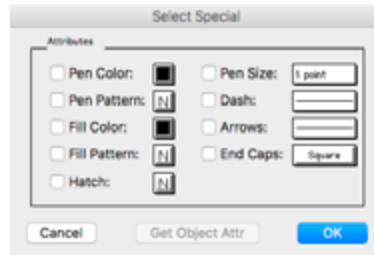
Select Special will automatically select all objects whose attributes match specified attributes or according to a selected model. Objects can be selected by pen color, pattern; fill color, pattern; or line weight, dash, and arrow attributes. You may also select objects by type.

Select Special will make selecting and editing objects quick and easy. By specifying attributes common to certain objects, you can locate and make global changes to many objects at once. You may specify object attributes individually or use a selected object as a model. Additionally, you may use the Selection Filter window to select only certain object types with specific attributes.

Select Special can be used with Edit All Layers On to search an entire drawing or Off to isolate the search to the active layer.

To use Select Special:

- ☐ Choose Select Special from the Edit menu. The following dialog will appear:



- ☐ Set the attributes by placing the cursor on pop-up menu beside the attributes and pressing. The appropriate palette will appear. Attributes which are not desired for a selection can be deactivated by clicking an active checkbox.
- ☐ Once the attributes are set, click OK. All matching objects will be selected.

Pen Color, Pen Pattern, Fill Color, Fill Pattern, Pen Size, Dash, Arrows, and End Caps allow you to specify attributes by which objects are selected.

To set an attribute, press and choose from the pop-up menus.

Get Object Attr sets the attribute options according to a selected object. This will allow you to use an object as a model.

You may also use a selected object as a model by which other objects are selected.

To use a selected object as a model:

- ☐ Select an object.
- ☐ While pressing the Option key, choose Select Special from the Edit menu. When the dialog appears, click Get Object Attr. The attribute options will reflect the selected object's attributes.
- ☐ Click OK. All objects whose attributes match the specified model will be selected.



NOTE: If more than one object is selected, the Get Object Attr button will appear dimmed and is unavailable.

*Selecting Objects by
Type*

Select Special may be used to select objects according to type by first checking the appropriate object type in the Selection Filter Window. This will allow you to be very specific; for example, you can find all red-filled, dashed rectangles. Select Special does not look inside of groups or symbols.

Select Adjoining selects a path along objects having touching endpoints. It works with lines, arcs, and polygons.

Select Adjoining

Select Adjoining is an efficient way to select the entire perimeter of a machine part or wall. You select one object, and Select Adjoining selects the rest.

Select Adjoining selects only one path. When it has a choice between more than one object, it selects the object that was created most recently.

Objects selected with Select Adjoining are ready for further manipulation. For instance, they can be grouped or attached.

To use Select Adjoining:

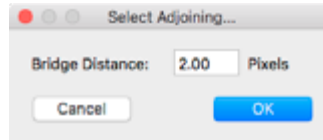
- ☐ Select a line, arc, or polygon.
- ☐ Choose Select Adjoining from the Edit menu. The adjoining objects are selected.

Select Adjoining Optional Settings

The optional settings for Select Adjoining allow you to specify what the maximum bridge distance between object endpoints is when performing a Select Adjoining operation. For example, a value of zero means the endpoints of the objects must be snapped together exactly. A value of two pixels means the endpoints can range from zero to two pixels apart and still be selected when choosing Select Adjoining.

To access the Select Adjoining optional settings:

- ☐ Press the Option key and choose Select Adjoining from the Edit menu. The following dialog will appear:



- ☐ Enter a numeric value in the Bridge Distance field and click OK.
- ☐ These settings remain in effect until you change them.

Select With Object

Select With Object selects all objects whose extents reside entirely within a selected object.

Select With Object is an efficient way to select objects using a fence or boundary. You select the boundary object and Select With Object selects all the other objects inside the boundary for you. This can be a very efficient way to remove objects from specific areas of a drawing.

To use Select With Object:

- ☐ Draw an object to be used as the selection boundary or select an existing object in the drawing.
- ☐ With the object selected, choose Select With Object from the Edit menu.
- ☐ All items inside the object will be selected and the original object used as the selection boundary will be deselected.

If Edit All Layers is On, all objects on visible layers inside the selection boundary will be selected. If Edit All Layers is Off, only objects on the active layer will be selected.

Select Inverse

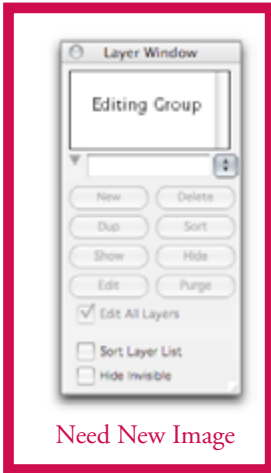
Choose Select Inverse to reverse the current selection set of objects. Objects that were selected will be deselected. Objects that were not selected will be selected.

Deselect All

Choose Deselect All to deselect all currently selected objects.

Edit Group

Exit Groups



Edit Group allows you to edit the individual objects within a group without disrupting the grouping characteristics.

Exit Groups allows you to back out of a single or nested group.

Pressing the Option key and choosing Exit Group will take you back to the first level without stepping through nested groups.

Edit Group makes it very easy and convenient to modify objects which are contained in a group because you do not have to upset the group's order or relationship. You can even edit objects which are nested several *levels* into a group.

Once you have entered a group, you can modify, add, or delete objects as needed. Any additional objects created while in the Edit Group mode will be added to the group when the editing session is complete.

Edit Groups will also allow you to move objects from one group to another group within a nesting. When editing a group, all other data in the drawing is temporarily hidden and the Layer Window will display 'Editing Group'.

To use Edit Groups:

- ☐ Select a group.
- ☐ From the Edit menu choose Edit Group. All other objects in the drawing will be hidden temporarily. This will reduce clutter while editing the selected objects.
- ☐ Edit the objects as needed. You may add or remove objects. Objects added will become part of the group.
- ☐ When finished, choose Exit Groups from the Edit menu. You are returned to the drawing window and all previously hidden objects are shown.

You may use all available tools and menu items while editing a group. You can even ungroup objects and create new groupings. This is useful when it is necessary to move an object from one group to another within a nesting.

To edit nested groups:

- ☐ Select a nested group.
- ☐ Choose Edit Group from the Edit menu. This will take you down into the nested group one level. Select another group inside this level and choose Edit Group again. Continue to select groups and the Edit Group command as needed to get to the desired object. All other objects will be hidden temporarily as you edit further into the nesting.
- ☐ Make changes as needed. When finished, you may step back to a previous level by choosing Exit Groups. Objects will reappear as you step back through the groups. Continue choosing Edit Groups to step back to see the entire drawing.

Pressing the Option key and choosing Exit Group will take you back to the first level without stepping through nested groups.

Edit...

Edit... opens a text edit window, a dimension editing window or a dialog where you can perform additional editing of the selected object.

To edit a dimension (or multiple dimensions):

- ☐ Select the dimension(s) to be edited.
- ☐ Choose Edit from the Edit menu.
- ☐ The Dimension Edit dialog will open.
- ☐ Make the desired changes and click OK.
- ☐ If a single dimension is selected, the change is applied to that specific object. If multiple dimensions were selected the change is applied to all selected objects.



For additional information on working with and editing dimensions see the [Tool chapter](#) and [Dimension Window](#) discussed later in this chapter.

To edit a text block:

- ☐ Select the text block or leader line object to be edited.
- ☐ Choose Edit from the Edit menu.
- ☐ A Text Edit window will open.
- ☐ Make the desired changes and close the text edit window.



For additional information on working with and editing text, see the Tool chapter, [Leader tool](#), [Text tool](#), and [Advanced Text tool](#).

Reshape

Reshape places edit handles on the selected object so the shape can be modified.

Reshape works directly on arcs, circles, lines, polygons, Béziers, and smoothed polygons (B-Splines). All other objects, such as rectangles, must first be Converted to Polygons before they can be reshaped.

For elliptical arcs, Reshape allows you to change the length of the arc, but will not allow you to change the radii.

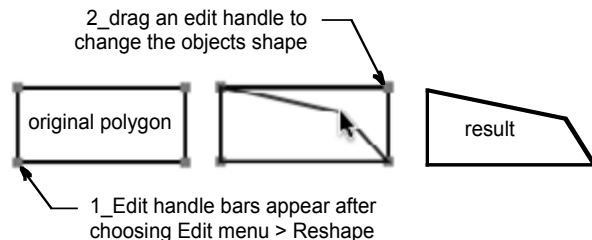
For circular arcs, both the radius and the length of the arc can be altered.

For a line, Reshape allows you to change the length of the line without changing its angle.

For a circle, Reshape allows you to change the radius.

To reshape an object:

- ☐ Select the object to be reshaped.
- ☐ Choose Reshape from the Edit menu.
- ☐ When you release the mouse, you will notice the original object now has edit handles at critical points which will allow you to redefine the shape of the object.
- ☐ Select one or more of the edit handles and drag them with the mouse in the appropriate direction to bring about the desired change.
- ☐ Exit reshape by clicking on the Tool palette or clicking in the drawing window or press the Enter key.



To reshape a line:

- ☐ Select the line you wish to reshape.
- ☐ Choose Reshape from the Edit menu.
- ☐ The line will display edit handles at each end.
- ☐ Select one of the edit handles and drag to reshape the line. The line will remain at its current angle. To remove the angular restraint, press the Shift key while dragging the edit handle.
- ☐ Exit reshape by clicking on the Tool palette, in the drawing, or pressing the Enter key.

To reshape a Bézier curve:

- ☐ Select the Bézier curve you wish to reshape.
- ☐ Choose Reshape from the Edit menu.
- ☐ The Bézier curve will display all control points and telescoping handles.
- ☐ You may move anchor points and control points by selecting them and dragging to a new location.
- ☐ You may independently change the control rod lengths by selecting each control point and moving it. Changing the length of the control rods changes the degree of pull on the curve. The longer control rod will have a stronger pull on the curve than a short control rod, therefore, the longer the control rod the more the curve will pull toward that side of the anchor point.
- ☐ You may remove anchor points by holding the Option key and selecting the anchor point you wish to delete.
- ☐ You may break a control rod by pressing the Command key and clicking on a control rod.
- ☐ You may add anchor points by holding the Option key and clicking on a curve segment. The anchor point will be inserted with the control rods set to zero.
- ☐ The curve may be visually observed during the modification process.
- ☐ You may set a control rods length to zero by holding the Option key and selecting the rod you wish to set to zero length.
- ☐ When you have finished the reshape process, press the Esc key to exit the Reshape command.

B-Spline curves can be reshaped in a similar manner to Bézier curves.

Polygon Reshape is a powerful feature. Missing points can be added to a shape, or points can be duplicated. Lines can be moved, and points can be lassoed and moved.

During polygon reshape, the following operations can be performed:

- ☐ Vertices can be added or deleted by holding the Option key while clicking on a handle to remove it or clicking on a line to add a vertex.
- ☐ Select All is available for selecting all vertices.
- ☐ Line segments may be individually selected for moving.
- ☐ Multiple handles may be selected by holding the Shift key.
- ☐ Multiple handles may be selected by pressing the Option key and lassoing.
- ☐ Multiple handles may be moved simulating complex mechanical movement. You might check for interference during movement of a single part in a complex design mechanism.
- ☐ Regular polygons may be turned into a parallelogram.

When you reshape a polygon or curves, you may move points or lines. If you choose to move a line, it will retain its present angle and length during the move, but the lines intersecting with either end will stretch to allow placement. If you select a vertex, you may stretch the two intersecting lines to any location. Option-click on a vertex to delete the vertex. Press the Option key and click on a line to add a vertex to the line.

To reshape an object, you select lines and vertices just as you select items in the drawing page. You may use a Selection Rectangle, a Lasso, or randomly select points by the Shift key and clicking.

Reshape allows you to change the diameter of circles. When you reshape a circle, a single handle will appear on the perimeter of the circle. If you select and drag the handle, you can change the diameter of the circle.

Reshape allows you to change the length and radius of an arc, up to the point where the arc becomes a circle or an ellipse. To change any characteristic other than the arc length, you must first convert the arc to a polygon.

Scale...

Scale is used to change the size of selected objects and/or text in a drawing.

Generally you would scale an object about its center.

An individual part, grouped objects, or a selection of objects will usually be scaled about their center or some other specific point.

Scaling about the next mouse is more intuitive if accurate placement is not needed.

Scaling can be used to mirror objects or groups.

Changing the scale in the Drawing Setup dialog does not automatically scale objects. The scale menu item physically changes the sizes and spatial relationships between objects in a drawing.

The entire drawing, or any individual part of it, can be scaled up or down, using the Scale command, including text and dimensions. When scaling an object and dimensions, the dimension is automatically recalculated to reflected the new scaled size.

A drawing can have details at different scales. Anything drawn or changed will be at the current scale. This is a very powerful feature, but it is also a case where you are not protected from yourself.

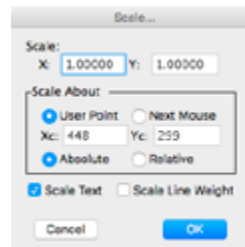
Scale is used to change all or part of a drawing. Scaling here is independent of the drawing scale. A ratio of 1/2 reduces the size by one half. A ratio of two makes the selected objects twice as large as the original. The X,Y coordinates shown are the center of the selected material.

Scaling with a minus-one on one axis will mirror the selection.

Scaling can be useful if you started a drawing on the incorrect size “paper”.

To scale an object:

- ☐ Select the object (or multiple objects) to be rescaled larger or smaller.
- ☐ Choose Scale from the Edit menu.
- ☐ The Scale dialog will appear.



Scale X: The amount to scale the data in the horizontal axis. Entering a value in this field automatically enters the same value in the Scale Y field.

Scale Y: The amount to scale the data in the vertical axis. The value entered in the Scale X field is automatically entered here. To scale a drawing with two different ratios, enter the unique Scale Y amount in the field and the Scale X value is not modified.

User Point: Lets you type in the coordinate defining the point about which to perform the scaling.

Next Mouse: Lets you use the next mouse click in the drawing area to define the point about which to perform the scaling. When scaling about the mouse, the target cursor is presented. Surface and intersection snapping are available. Scaling is performed when the mouse is released.

Absolute: The absolute point about which to perform the scaling. The initial values in the X,Y boxes are the distance from the current origin.

Relative: The relative offset with respect to the center of the selected object.

Scale Text: Allows you to scale text along with objects. If this item is checked, the text is rescaled. If this item is not checked the text font size remains the same.

Scale Line Weight: Allows you to scale the line weight attribute along with the drawing objects. For example, an object with a line weight of 2 points that is scaled 2 times larger will have a new line weight of 4 points. If Line Weights is not checked the line weight attribute is not changed.

Ratio numbers larger than one scale up; ratios smaller than one scale down. For example, a ratio number of two makes the object twice as large; a ratio of 0.5 makes object half as large. You may also enter mathematical equations in the dialog box. For example, entering $2/1.25$ in the Scale X field would scale the drawing by a factor of 1.6 times.

- ☐ Unless changed, this dialog will scale an object about the center of the object selected.
- ☐ Enter the appropriate values for each field based on the descriptions below and click OK.
- ☐ The selected objects are rescaled and remain selected.



To set the Drawing Scale, see [Drawing Setup](#) under the Layout menu.

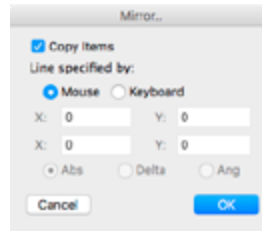
Mirror

Mirror allows you to create a mirror image of selected objects.

Mirror can create a duplicate mirror image of selected objects about a specified line or you may draw a mirror line or specify an existing one.

To Mirror about a mouse-drawn line.

- ☐ First, select the objects to be mirrored.
- ☐ Press the Option key while choosing Mirror from the Edit menu. The following dialog will appear:



- ☐ Select Copy Items if you want to mirror a duplicate set of the selected object. With Copy Items off, the selected objects will be moved to the mirrored position.
- ☐ Click OK and a target cursor will be presented, which you will use to draw the mirror line. You must draw a line or click on an existing line as the mirror line. Remember, the space between the mirror line and the selected objects will also be mirrored.
- ☐ The line will constrain to the preset angles. Pressing the Shift key will unconstrain the line.
- ☐ The selected object will be mirrored about the line drawn and the line will disappear.

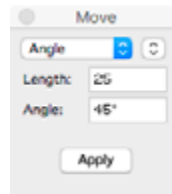
You may also select an existing line to act as a mirror line. When the target cursor appears, click on an existing line to be used as the mirror line. If the dimensions of the mirroring line are known in advance, they may be entered by the keyboard.

Move

Move allows you to relocate objects by typing in location coordinates, or dynamically by dragging the mouse in the drawing window.

One method of moving is to select the object to be moved and drag it to the desired location. The Move command is an alternative method of moving the selected object to an exact location.

Move allows you to place items in a drawing with great precision. You can move single items, or every item in the drawing, regardless of layers. To activate the Move dialog, press the Option key while selecting Move and the following dialog will appear.



If the Move Window External is loaded, choosing Move while pressing the Option key will present you with a floating Move window which floats on top of your drawing for ready access.



See the Window Menu section of this chapter for more information on the [Move Window External](#).

Move offers several methods of moving objects.

- ☐ **Absolute** allows you to place the center of the item on an exact X,Y coordinate with respect to the Drawing Origin.
- ☐ **Relative** allows you to move the object an exact X,Y distance with respect to its current location.
- ☐ **Angle** allows you to move items a specified distance at a specified angle. You enter the distance and angle.
- ☐ **By Mouse** allows you to determine the distance and angle moved by drawing a line with the mouse. You may use the snap keys at any time.
- ☐ **By Line** allows you determine the distance and angle moved by clicking on an existing line in the drawing. This move option is only available in the Move Floating Window (the Move External must be loaded).

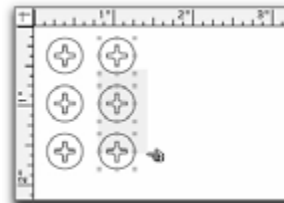
There are several ways to move objects with the By Mouse option. The movement can be an offset amount determined by the length of the line and angle at which it is drawn. You can use the By Mouse option when you want the center of one object to be moved to the center of another object. You might also want the vertex of one object to move to the center of a line.

The following illustration demonstrates two rows of screws, the right hand

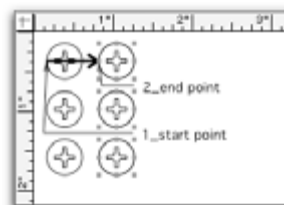
screws we want to duplicate and move the duplicated screws so all three sets of screw have an equal distance between them.



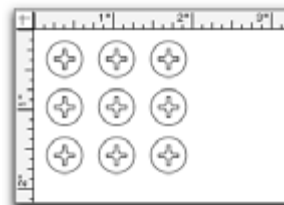
- ☐ Select the three right-hand screws.
- ☐ Choose Duplicate from the Edit menu. This will duplicate the objects on top of each other and look like the following illustration.



- ☐ Press the Option key and choose Move from the Edit menu.
- ☐ Choose By Mouse as your Move option and click OK.
- ☐ Move the cursor to the left-hand side of one of the screws in the first row.
- ☐ Drag to the same position on one of the screws in the second row, as the following illustration shows.



The duplicated objects will now be moved into the proper location. The three rows are equally distanced, as the following illustration shows.

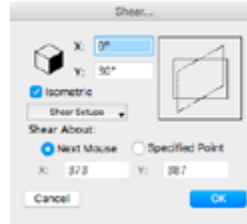


To move an entire drawing, you will need to ensure Edit All Layers is checked in the Layers window.

Shear...

Shear allows you to convert normal images into the distorted images used in isometric, diametric and oblique drawings; what most people call 'three-dimensional drawings'.

Select an object and choose Shear from the Edit menu. A dialog will appear.

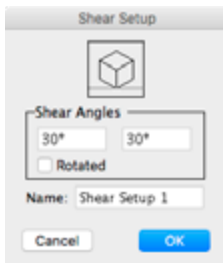


Shearing in X or Y skews the drawing. To observe this action, type an angle in either X or Y and notice the behavior of the rectangle in the example window to the right. The angles may be positive or negative, and shear has a limit of 80°—so objects cannot be pushed flat.

If you set the X shear angle to 30°, notice the vertical lines are tilted to the right, but the new polygon is still the same overall height as the original rectangle. This is a simple shear and causes the vertical lines of the rectangle to become longer. This is not correct for drawings which have a three-dimensional appearance. To keep tilted lines from becoming longer, click in the Isometric box, and notice the change in the example window.

Because shear in X tilts the Y-axis, many people find the computer-graphics terminology difficult. To protect you from this, PowerCADD provides a cube control. Click on one of the three sides of the cube to make the appropriate settings for X, Y and the Isometric box.

The axes of the three-dimensional presentation may be changed by choosing from the Shear Setups pop-up menu.



To create your own axis setup, choose New Setup from the Shear Setups menu, and a dialog will appear.

- Type the desired axis angles and notice the change in the cube in the example window. A diametric drawing has two axis angles which may be symmetrical or asymmetrical. A standard isometric is a diametric drawing in which both axis angles are 30°. In an oblique drawing, only one axis is tilted.



Isometric



Diametric



Oblique

- Type a name for the setup, then click OK.

To use Shear:

- ☐ Select the object you wish to shear.
- ☐ Choose Shear from the Edit menu. The shear dialog will appear.
- ☐ If the cube is not drawn in the desired axes, choose another setup from the Axis Setup pop-up menu or create a new axis setup.
- ☐ Click on the cube for the desired view.

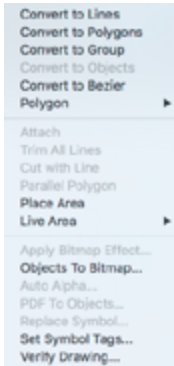
—or—

- ☐ Make the appropriate settings by typing the desired angles in X and Y and clicking on the Isometric box.
- ☐ Select Next Mouse or Specified Point. Shearing about the next mouse location may seem more intuitive, but in most cases the image must be moved after shearing, so it is normally not necessary. The initial X,Y values represent the geometric center of the selected object.
- ☐ Click OK. If Specified Point was selected, the object will be immediately sheared.
- ☐ If Next Mouse was selected, you will be presented with a target cursor. Click on the location to shear about.

To create views from below, think of the cube as two-walls-and-a-ceiling. The shear settings for the “ceiling” are the same as would be required for a floor.

Once created, three-dimensional views can be flipped horizontally, flipped vertically, scaled, mirrored and rotated like any other objects to create any desired presentation.

Tool Menu



The Tool menu consists of special operations which may be applied to selected objects in the drawing window.

Prior to using the Tool menu commands, the object must first be selected. If no items are selected before choosing a command from the Tool menu, the computer will issue a warning beep. Also, a warning will be sounded if too many items are selected for the command, or if the selected object is incompatible with the command.

When you press the Option key, you will notice the addition of three dots beside Convert to Lines, Convert to Polygons, Parallel Polygon, Objects to Bitmap and Place Area. By pressing the Option key when you choose the Tool menu these menu items will present a dialog with additional options.

Convert to Lines

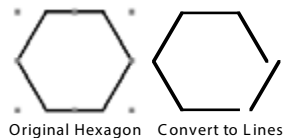
Convert to Lines converts objects into their individual line components.

This powerful tool allows you to create a complex polygon, for example, and subsequently “convert” — or break — it into its individual line components for further line-by-line modification.

To convert a polygon into its individual line components:

- ☐ Select the polygon you wish to convert. For example, here is a hexagon drawn with the Regular Polygon tool.
- ☐ To convert the hexagon into its individual line segments, choose Convert to Lines from the Tool menu.
- ☐ The selected object is now broken into its component line parts.
- ☐ Each line segment of the original object remains selected for further modification.

The following diagram shows a line segment moved from the original hexagon after it was converted to lines. This action would not be possible without the Convert to Lines command.

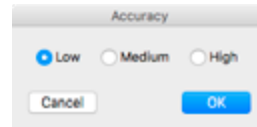


CAUTION: Exercise caution in using this command. Once an object is converted to lines and modified, the individual lines may be difficult to put back together again! Higher level geometry such as circles, once converted to lines, cannot be turned back into a circle. Attached the lines would create a polygon shaped like a circle.

Grouped objects must be ungrouped before converting to lines. Rectangles,

circles, Béziers, B-Splines, and ellipses can be converted to lines directly without being converted to polygons first.

You may find for some situations the number of line segments which result from Convert to Lines to be higher than needed. To adjust the conversion accuracy, press the Option key while choosing Convert to Lines. A dialog appears.



Click the desired setting and click OK. This setting will remain until changed. A Low accuracy settings uses the least number of line segments to recreate the object's shape. A setting of High uses the greatest number of line segments.

It is recommended you use the lowest resolution that will meet your needs. Curves, arcs, and circles which are converted at a high resolution will produce lots of short line segments and require more memory.



NOTE: Immediately after converting an object to lines, the entire set of lines remains selected for further modification. To select an individual line from the entire set, you must first deselect the entire set of line segments.

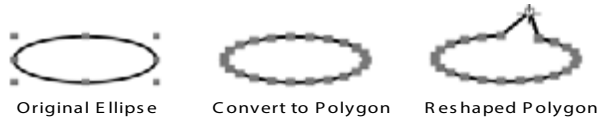
Convert to Polygon

Convert to Polygon turns an object into a polygon for subsequent reshaping.

You might want to convert an ellipse or circle, for example, to a polygon and then use Reshape for further object modification.

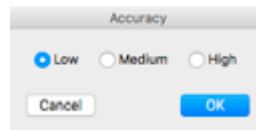
To convert an object to a polygon:

- ☐ Select the object to be converted into a polygon.
- ☐ Choose Convert to Polygon from the Tool menu.
- ☐ Next choose Reshape from the Edit menu. You will notice the original ellipse now has multiple selection handles available for use in reshaping.
- ☐ Selecting individual edit handles (or groups of handles while pressing the Shift key), allows modification of the newly created polygon.



Grouped objects must be ungrouped before converting to polygons.

Like Convert to Lines, the conversion accuracy settings will control the number of points or segments created with Convert to Polygon. To set the Accuracy, press the Option key while choosing Convert to Polygon. The same rules apply when setting the conversion accuracy for Convert to Polygons; use the lowest setting that will do the job.



NOTE: The Accuracy settings affect *both* Convert to Lines *and* Convert to Polygons. It is the same function accessible from two different locations. For example, if you set Accuracy to be Medium before using Convert to Lines, and then later use Convert to Polygons, the object will be converted to a polygon with Medium Accuracy setting.



CAUTION: Use Convert to Polygons with caution. Once an object is converted to a polygon, it no longer retains the characteristics of the original object. For example, a converted circle is no longer a circle; it is a multi-segmented polygon.

Convert to Objects

Convert to Objects converts placed images and hatches into editable objects.

Convert to Objects will allow you to convert placed images, dynamic hatches, and PICT files into PowerCADD objects which can be edited.

To convert to Objects:

- ☐ Select a placed image.
- ☐ From the Tool menu, choose Convert to Objects. The selected object will be converted into editable objects.

Vector-based images such as PowerCADD objects which were hatched will convert to vectors and may be edited using standard PowerCADD techniques. Use Convert to Bitmap instead of Convert to Objects for placed images such as JPEG, PhotoShop or other supported QuickTime file formats to edit the images using the Apply Bitmap Effects tools and commands.



NOTE: When Convert to Objects is used on some images, the converted objects may not be as accurate as the original images.



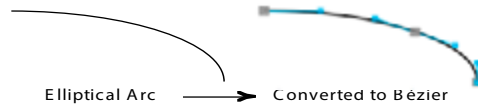
TIP: If you Placed a PDF file, EPSF, Illustrator or other Postscript-based file formats, use [PDF to Objects](#).

Convert to Bézier

Convert to Bézier will convert selected objects into a Bézier path. The objects selected may be lines, arcs, circular arcs, circular ellipses, ellipses, rotated ellipses, circles, rectangles, rotated rectangles, round rectangles, and open or closed polygons.

To use Convert to Bézier:

- ☐ Select an object in the drawing, for example an elliptical arc.
- ☐ From the Tool menu choose Convert to Bézier.
- ☐ The original object is converted to a Bézier whose path exactly matches the original object shape.
- ☐ The resulting Bézier path may be edited using standard Bézier reshaping and editing methods.



When used in conjunction with the Attach Tool or Attach menu item, Convert to Bézier can be used to create complex, smooth curve, paths from standard objects in your drawing.

To create a complex Bézier curve path from objects in the drawing:

- ☐ Draw path using the desired combination of standard object drawing tools or menu commands.
- ☐ Select all the objects in the path.
- ☐ From the Tool menu choose Convert to Bézier.
- ☐ Each object is converted to a unique Bézier path and automatically selected.
- ☐ With the objects still selected choose Attach from the Tool menu to create one contiguous path.



Using the Convert to Bézier and Attach menu items is just one method to create complex smooth curve paths. Another method is to use the [Attach Tool](#) with its optional settings set to Convert to Bézier. Additional methods also exist using a combination of tools and menu items to select, attach, convert and reshape a variety of objects types.



CAUTION: Use Convert to Bézier with caution. Once an object is converted to a Bézier, it no longer retains the characteristics of the original object. For example, a converted circle is no longer a circle; it is changed to a Bézier. If you need the original geometry, duplicate the objects to be converted first.

Polygon Menu



Polygon allows you to convert a polygon to a Bézier curve; you may also optimize a polygon by removing excess points or by distributing a polygon's points evenly along its path. Additionally, you may trace a bit image with polygon paths, automatically.

Polygons with many points require lots of memory when redrawing and printing, not to mention extra effort when editing. Examples of memory-intensive polygons include those created by smoothing curves, tracing scanned images, drawing maps, and so on. By removing excess points from, or distributing points, you improve performance and efficiency.

Converting a polygon to a Bézier curve is another way of optimizing. Not only does the resulting Bézier shape contain fewer points, but it can be reshaped using Bézier editing conventions.

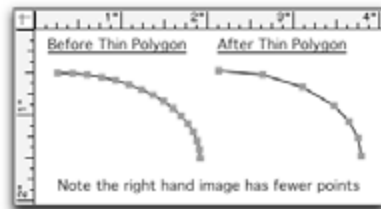
Thin Polygon

Thin Polygon allows you to remove excess points from a selected polygon.

Many times, a polygon will have more points than needed to define a particular shape, especially if the polygon is created by smoothing a path or by freehand tracing a scan. Thin Polygon will make polygons easier to edit on screen and less likely to cause memory problems.

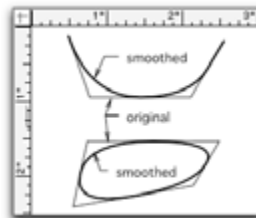
To use Thin Polygon:

- ☐ Select a polygon.
- ☐ From the Polygon submenu, choose Thin Polygon.



Smooth

Smoothing has the effect of rounding the corners of polygons into a shape known as a B-Spline — a type of mathematical curve, much like a French curve used in the manual drawing process. This curve generates curve sections which are continuous slopes so they may fit together smoothly.

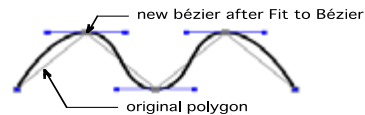


Fit Bézier

Fit Bézier applies a Bézier to the polygon points. Bézier curves are smoothed mathematical curves that go through the control points.

To use Fit Bézier to automatically construct a Bézier from a polygon:

- ☐ Select a polygon.
- ☐ From the Polygon submenu, choose Fit Bézier. The original polygon is replaced with a new Bézier curve. The resulting Bézier may be reshaped using normal Bézier editing conventions.



For more information on editing or adjusting Bézier curves, see [Bézier Tool](#).

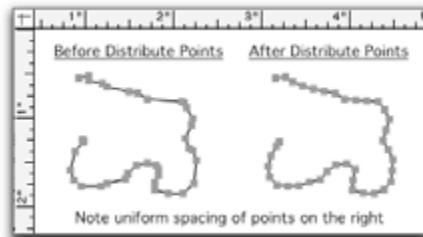
Distribute Points

Distribute Points allows you to arrange a polygon's points evenly along the entire length of the path.

This will provide a more uniform shape and make editing easier.

To use Distribute Points:

- ☐ Select a polygon.
- ☐ From the Polygon submenu, choose Distribute Points.

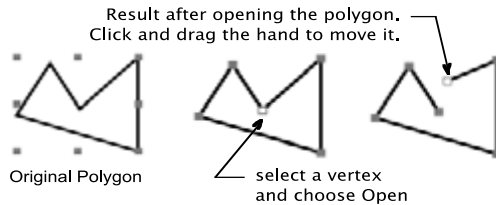


Open Polygon

Open Polygon puts a break at the selected point. Once opened, you may drag the segment's endpoint to create an open polygon path.

To use Open Polygon:

- ☐ Select a closed polygon.
- ☐ From Edit menu, choose Reshape or press Command-R.
- ☐ Click to select a vertex. The selected vertex will be the location where the polygon will be opened.
- ☐ From the Polygon submenu, choose Open.
- ☐ The polygon is broken at the selected vertex. Click and drag to open the polygon or press the Option key and click on the vertex to delete it.

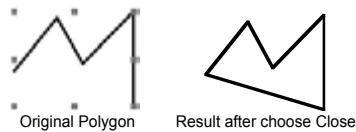


Close Polygon

Close Polygon places a line segment between the starting and ending points of an open polygon.

To use Close Polygon:

- ☐ Select an open polygon.
- ☐ From the Polygon submenu, choose Close.
- ☐ A new segment is added to close the polygon.

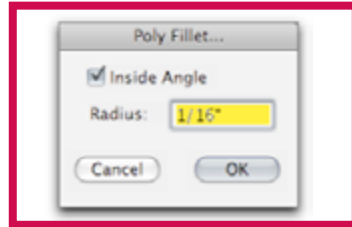


Fillet Polygon

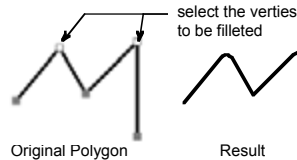
Fillet Polygon will insert a fillet (rounded corner) at a selected vertex, multiple selected vertices, or to all vertices simultaneously.

To use Fillet Polygon on a selected vertex or multiple selected vertices:

- ☐ Select an open or closed polygon.
- ☐ Choose Reshape from the Edit menu or press Command-R.
- ☐ Click to select the vertex to be filleted. Multiple points can be filleted at the same time by pressing the Shift key and selecting the additional vertices.
- ☐ Choose Fillet Polygon from the Polygon submenu and the Poly Fillet dialog will appear **DOES NOT WORK**.

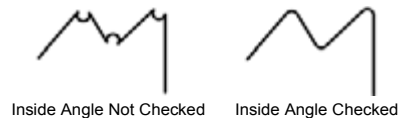


- ☐ Set the desired radius and click OK. When Inside Angle is active, a typical rounded corner will be inserted; when inactive, a rounded cutout will be placed.



To fillet all vertices on a polygon simultaneously:

- ☐ Select an open or closed polygon.
- ☐ Choose Fillet Polygon from the Polygon submenu and the Poly Fillet dialog will appear.
- ☐ Set the desired radius and click OK. When Inside Angle is active, a typical rounded corner will be inserted; when inactive, a rounded cutout will be placed.



NOTE: The current [Convert to Polygon](#) accuracy setting will affect the number of points which are used to create the fillet. Low settings will produce fewer points and result in a more jagged fillet. High settings will produce more points and a smoother curved appearance. The more points that are used, the more memory is required to store and edit the shape;

use the lowest setting which will produce the desired results. For more information on editing polygons and Conversion Accuracy, see [Convert to Lines](#), and [Convert to Polygons](#).

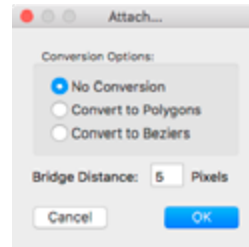
Attach

The Attach command allows you to join the end points of lines, arcs, and polygons to form a new, single polygon.

Attach joins separate objects to form a closed polygon or an open polygon for easier manipulation. You may also reattach objects previously converted to lines.

To change the settings for the Attach command:

- ❑ Press the Option key and choose Attach from the Tool menu. The following dialog will appear:



No Conversion: If the items to be attached include arcs, this setting will not change the original arc geometry. The results will be a Group containing the original objects and a new polygon (pen pattern: None) which is the bottom most object in the group. This setting has no affect when attaching lines or polygons; the result will always be a polygon.

Convert to Polygons: All objects to be attached are converted to a polygon; the original geometry is changed. For example, if the objects to be attached included a line and an arc, the original arc will be replaced with a polygon.

Convert to Béziers: All objects to be attached are converted to a Bézier curve; the original geometry is changed. For example, if the objects to be attached included a line and an arc, the original arc is replaced with a Bézier.

Bridge Distance: Allows you to specify the maximum distance between object end points in pixels. Objects which are farther apart than the specified distance are not attached.

- ❑ Make the desired changes and click OK.

Changes made in this dialog remain in effect until you change them.

To use the Attach command:

- ☐ Select all objects to be attached.
- ☐ Choose Attach from the Tool menu.
- ☐ The selected objects will be attached to form a single polygon or group.



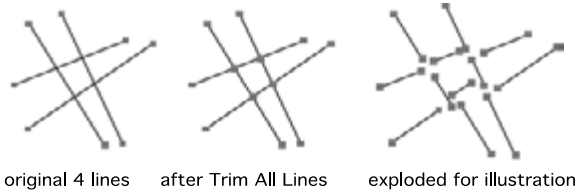
CAUTION: When Conversion Options are set to Convert to Polygons or Convert to Béziers, higher level geometry such as arcs and elliptical arcs are replaced. To retain the original higher level geometry, use No Conversion.

Trim All Lines

Trim All Lines allows you to cut two or more lines at their intersection points.

To use Trim All Lines:

- ☐ Select the lines you wish to trim.
- ☐ Choose Trim All Lines from the Tool menu.



Upon release of the mouse button, all lines selected will cut each other at their intersection points.

This feature will be helpful for preparation of data to be used in finite element stress analysis software.

Trim All Lines does not throw away any parts. All parts are retained for further modification.

Trim All Lines only works with lines. If you attempt to trim through polygons, groups, or fills, the command will only cut any single lines present and leave the other objects intact.

Cut with Line

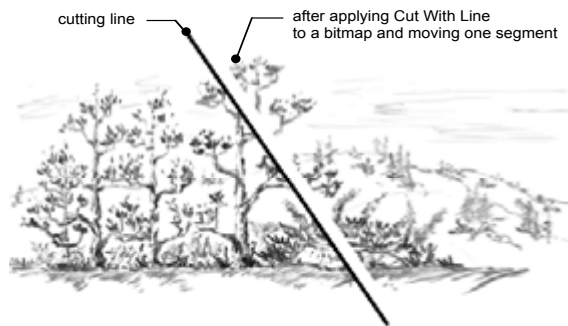
Cut with Line is a feature which allows one selected line to act like a knife to cut or trim every object below it on the same layer of the drawing. The width of the line is ignored; it cuts along its center. All of the cut parts are retained. If more than one line is selected, a warning beep is issued. The unwanted parts must be selected and deleted.

The following rules apply to Cut with Line.

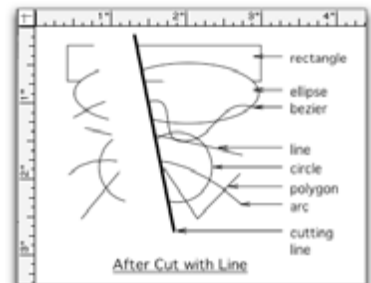
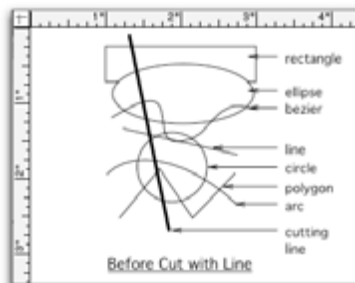
- ☐ Only the cutting line may be selected.
- ☐ The selected cutting line may be above or below the objects being cut.
- ☐ Cut with Line will not cut groups, symbols or text.
- ☐ With Edit All Layers On, all visible objects will be cut. Cut segments will remain on the original layers. To cut objects on the active layer only, turn Edit All Layers Off.
- ☐ Cut with Line will cut bitmaps but cannot cut Placed objects. To cut a Placed object, use Convert to Bitmap first.

To use Cut with Line:

- ☐ Select the cutting line. The line must pass entirely through the objects being cut.
- ☐ Choose Cut with Line from the Tool menu.



Circles are turned into arcs when cut. If the cutting line intersects a circle once, the circle turns into a 360° arc. If the cutting line intersects the circle twice, two arcs are created. Regular arcs and elliptical arcs may also be trimmed or cut with a line. Trimming a smoothed polygon first converts the smoothed shape to a normal polygon.



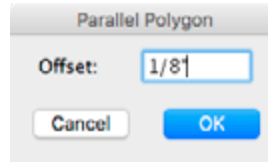
Parallel Polygon

Parallel Polygon allows you to offset a duplicate of any polygon to both sides of the original polygon.

This is helpful when constructing roads. You construct a polygon that represents the centerline of the road, and Parallel Polygon will place the outer boundaries of the road at the offset you specify.

To specify an offset:

Press the Option key while choosing the Parallel Polygon command from the Tool menu. The following dialog will appear:



- ☐ Type the desired offset. This setting will remain in effect until you change it.
- ☐ When you click OK, the offset will take place. There will be a duplicate offset placed on both sides of the original.

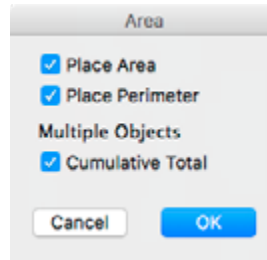
Place Area

Place Area calculates the area and perimeter of a polygon or other objects and inserts a text object in the drawing with the corresponding calculation.

Calculations will be given for area and perimeter based on your drawing units. Multiple objects can be calculated simultaneously with each objects unique area and perimeter calculation being placed at the center of the respective object. Area will calculate separate lines and arcs. It will also calculate polygons that have been created with attached lines and arcs.

To set the behavior of Place Area:

- ☐ Press Option and choose Place Area from the Tool menu, the Area dialog will appear:



Place Area inserts the area of selected objects into the drawing. Text is inserted at the center of the object. If multiple objects are selected and Cumulative Total is not checked, the area of each object is inserted.

Place Perimeter inserts the perimeter of selected objects into the drawing. Text is inserted at the center of the object. If multiple objects are selected and Cumulative Total is not checked, the perimeter of each object is inserted.

Cumulative Total allows you to insert the cumulative calculation for multiple objects into the drawing.

To find the area of a single object:

- ☐ Select the object to be calculated.
- ☐ Choose Place Area from the Tool menu.
- ☐ The area and perimeter of the selected object is inserted into the drawing.

To insert the area and perimeter of a single object or multiple objects into the drawing:

- ☐ Select the object(s) to be calculated.
- ☐ Choose Place Area from the Tool menu.
- ☐ The calculated values are inserted in the drawing using the text style assigned to the dimension tools.



CAUTION: Complex polygon areas will not be correct. A complex polygon is one which crosses back over itself.



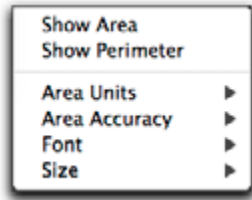
NOTE: For survey work, please verify the calculations to determine the degree of accuracy required by your application.

Live Area

Allows you to insert a smart text object in the drawing displaying the area and/or perimeter of an object. It appears as a contextual menu item.

To insert a Live Area smart text object in the drawing:

- ☐ Control- or Right-click on circle, rectangle, polygon, or Bézier in a drawing.
- ☐ The menu choices are:



Show Area: Inserts a text block at the center of the selected object displaying its area in the current drawing units.

Show Perimeter: Inserts a text block at the center of the selected object displaying its perimeter in the current drawing units.

When selected the area and/or perimeter of the object is inserted into the center of the object.

Resizing or reshaping the object will dynamically update the displayed value.

To change the optional settings of Live Area:

- ☐ Control- or right-click on circle, rectangle, polygon, or Bézier in a drawing.
- ☐ Select from the menu items as shown above left. The optional setting choices are:

Area Units: Allows you to specify the unit of measure used to display the area of the object. These units may be different than the units specified in Drawing Setup.

Area Accuracy: Allows you to specify the number of decimal points used to display the area of the object. This accuracy may be different than the units specified in Drawing Setup.

Font: Allows you to specify the font used for displaying area and perimeter.

Size: Allows you to specify the font size used for displaying area and perimeter.

The optional settings for the Live Area command are used until you change them.

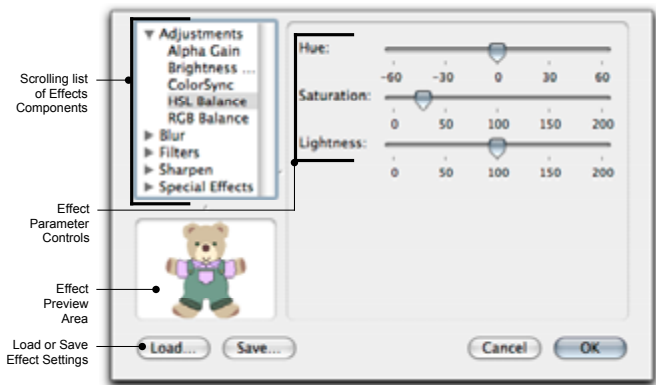
Apply Bitmap Effect

This allows you to apply a variety of filters to any bitmap object using QuickTime's built-in effects. To apply a bitmap effect to placed or normal PowerCADD object geometry, use Convert Object to Bitmaps first.

The ability to apply QuickTime special effects to bitmaps within PowerCADD allows you take technical drawings and renderings to a whole new level. Effects can be applied which create unique and interesting visual effects, sharpen a scanned image or provided feathered edges to digital photographs.

To use Apply Bitmap Effect:

- ☐ Select a bitmap or multiple bitmaps.
- ☐ Choose Apply Bitmap Effect from the Tool menu (if the menu item is dimmed then the objects you selected are not bitmaps).
- ☐ The Bitmap Effects dialog will open as illustrated below.



- ☐ Choose the desired effect and click OK.

The selected bitmaps are modified to reflect the specified settings. To apply multiple effects select the bitmap and choose Apply Bitmap Effect again. Using Undo will revert the bitmap to its previous state.

The Apply Bitmap Effects dialog is tapping directly into Apple's QuickTime technology. Much like the MacOS color panel, when you are seeing this dialog, you are working directly with the MacOS, in this case QuickTime.

Effects Components are listed in a scrolling list. Clicking on a reveal triangle displays a list of effects available in a particular effects category.

Effect Parameter Controls are displayed when you select a particular Effect Component. The controls to adjust a parameter may be a slider, check box, radio button, editable text field or any of several other interface elements depending on the type of Effect Component selected.

The **Effect Preview Area** provides real time feedback based on adjustments made to the Effect Parameter Controls and on the chosen Effect Component. This allows you to fine tune your effect before applying it.

Load allows you to Load a previously saved effect setting. This saves time if you have to apply the same effect to multiple objects or would like to use an effect created by a co-worker.

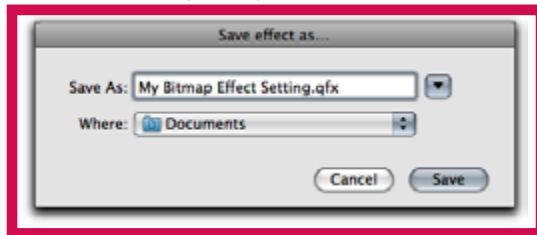
Save allows you to save the effect settings which can be loaded and applied to bitmaps again later. This saves time if you have to apply the same effect to multiple images or would like to share an effect with a co-worker.

Clicking **Cancel** aborts the apply bitmap effect operation without modifying the original bitmap. The effects dialog is closed, and you are returned to the drawing window.

Clicking OK applies the specified effect to the selected bitmap. The effects dialog is closed, and you are returned to the drawing window.

To save a bitmap effect setting:

- ☐ Select a bitmap or multiple bitmaps.
- ☐ Choose Apply Bitmap Effect from the Tool menu (if the menu item is dimmed then the objects you selected are not bitmaps).
- ☐ The Bitmap Effects dialog will open.
- ☐ Choose the effect and make the desired parameter adjustments.
- ☐ Click Save, the following dialog will appear.



- ☐ Name the file and specify a location (the .qfx is appended automatically).
- ☐ The effect settings are saved in the location you specified. Effects may be stored on your local hard drive or a network server to share with co-workers.

To load a previously saved bitmap effect:

- ☐ Select a bitmap or multiple bitmaps.
- ☐ Choose Apply Bitmap Effect from the Tool menu (if the menu item is dimmed then the objects you selected are not bitmaps).
- ☐ The Bitmap Effects dialog will open.
- ☐ Click Load, a dialog will appear.
- ☐ Navigate to the *.qfx file and click Open.

- ❑ The preview area updates to reflect the previously saved effect component and parameters.
- ❑ Click OK to apply the effect.



NOTE: To use Apply Bitmap Effects, a full installation of Apple's QuickTime software is required. For more information on QuickTime visit Apple's web site at <http://www.apple.com/quicktime>.

Object to Bitmap

Converts selected objects to Bitmaps. The source objects may be any type of PowerCADD object geometry, placed item or symbol.

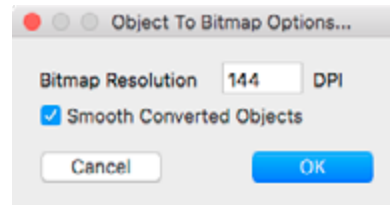
Using Object to Bitmap allows you to take advantage of various advanced PowerCADD bitmap tools and commands including Applying Bitmap Effects, using the Bitmap Transition tool or the Perspective Bitmap tool. Remember, any PowerCADD object can be converted to a bitmap to take advantage of these advanced features.



CAUTION: Use Object to Bitmap with caution as higher level PowerCADD object geometry is lost when the objects are converted to bitmaps. While you can Undo the Object to Bitmap operation, we recommend working on copies of your data.

To use Objects to Bitmap:

- ☐ Select any object or multiple objects.
- ☐ Choose Objects to Bitmap from the Tool menu (if the menu item is dimmed you have not selected any objects).
- ☐ The following dialog will appear:



- ☐ Your choices are:

Bitmap Resolution: Enter the number of dots per inch (DPI) you want the converted object to be.

Smooth Converted Objects: When checked anti-aliasing is applied to create a smoother, less jaggy, edge.

- ☐ Click OK. The selected object is converted to a bitmap.



You can now apply any PowerCADD Bitmap Effect menu item to the converted object. A bitmap may be cropped or uncropped.



For more on cropping, see [File menu, Place](#) earlier in this chapter.

Auto Alpha

Auto Alpha allows you to remove a specified color from a raster (bitmap) image. Once removed the previously colored pixels become transparent or 'see-through'.

To use Auto Alpha:

- ☐ Select a bitmap image in the drawing.
- ☐ Choose Auto Alpha from the Tool menu.

If the menu item is dimmed you have not selected a bitmap object, use Convert to Bitmap described earlier in this section).

- ☐ The MacOS Color Panel will appear.
- ☐ Click on the Magnifying Glass icon and move the cursor over the bitmap.
- ☐ Click the mouse when the cursor is over the desired color to be removed.
- ☐ The chosen color is displayed in the MacOS Color Panel.
- ☐ Click OK in the MacOS Color Panel.
- ☐ The specified color is removed and the previously colored pixels are now transparent or 'see-through'.



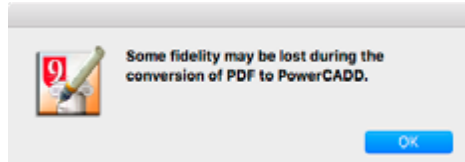
HINT: Auto Alpha may be applied multiple times to remove more than one color, or range of colors, from a single bitmap image.

PDF To Objects

Allows you to convert PDF files which have been Placed, Opened, or Imported into PowerCADD into editable bézier curves and text.

To use PDF To Objects:

- ☐ Select a PDF object in the drawing.
- ☐ Choose PDF To Objects from the Tool menu (if the menu item is dimmed you have not selected a PDF object). The following dialog will appear:



- ☐ Press OK.

The object is converted to editable bézier curve and text objects.



HINT: PDF To Objects can be used to convert object files based on the Postscript language to editable objects. Examples of Postscript-based files include EPSF, Illustrator, and Adobe Acrobat PDF.

Replace Symbol...

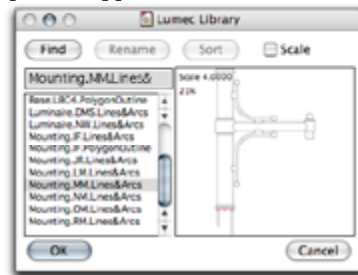
Replace Symbol replaces all occurrences of a symbol instance or group with another symbol.

With Replace Symbol, there is no need to change symbols manually. Replace Symbol finds and replaces symbols for you.

For example, all 3'-0" doors can be replaced with 2'-8" doors.

To use Replace Symbol:

- ☐ Open a symbol library or bring to the front the symbol library window you want to use.
- ☐ Click in the drawing window to make it active.
- ☐ Activate a layer containing a symbol to replace.
- ☐ Select one of the symbols to replace. This symbol must be on the active layer.
- ☐ Choose Replace Symbol from the Tool menu. A dialog like the following example will appear.



The dialog box represents the topmost library window.

- ☐ Select the replacing symbol from the list.
- ☐ Click OK, and the symbols will be replaced.

To find a symbol in the dialog box quickly, type a full or partial name in the name box and click Find.

To scale the symbol to your drawing scale, select the Scale checkbox. If a scaling factor was applied when the original symbol was inserted, the same scaling factor is applied to the replaced symbol.



NOTE: When Edit All Layers is On, Replace Symbol operates on all layers, including hidden layers. Otherwise, Replace Symbol replaces items on the active layer only. Symbols that are part of other symbols are not replaced.



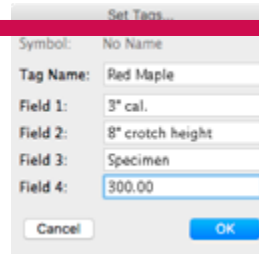
NOTE: If you have placed the same item as a group and as a symbol, Replace Symbol will only replace the common groups or the symbols and not both. To replace both, apply Replace Symbol twice, once to replace the symbols and once to replace the groups.

Set Symbol Tag

Set Symbol Tag allows you assign unique data attributes for up to five fields to a symbol or other PowerCADD object. Symbol Tags may be used to create bills of materials and schedules for a variety of design disciplines. Examples include door and window schedules, furniture schedules, plant material lists or lighting design schedules.

To assign a Symbol Tag to an object:

- ☐ Select an object in the drawing. The item selected may be a single object or a group.
- ☐ Choose Set Symbol Tag from the Tool Menu. The Set Tags... dialog appears.



- ☐ In the Tag Name field, type a name for this object tag.
- ☐ Press Tab to jump between fields and enter any additional information in the remaining fields, Field 1 through Field 4). In our example we created a tree symbol but the object and data could be for anything from a desk, door or pump component.
- ☐ Click OK.
- ☐ If the object you selected was a single object, it is automatically converted to a group with a non-printing point at its center.

The object may now be added to a Symbol Library. The next time the object is inserted into the drawing using the symbol tool, all the preset symbol tags will be present.

You may change an object's Symbol Tags at any time simply by selecting the object and choosing Set Symbol Tags from the Tool Menu.

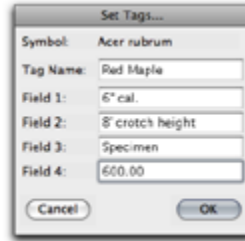
One to Many Relationships

The real power of Symbol Tags comes in being able to generate a 'one to many' relationship between all symbols which share the same Symbol Tags. A one to many relationship allows you change the Symbol Tag of one symbol and automatically update all related symbols in the drawing.

To create a one to many relationships between symbols and their symbol tags:

- ☐ Create a graphic element you'd like to use as a symbol.
- ☐ Set the Symbol Tags as previously described.
- ☐ Copy or cut the object and paste it into a new Symbol Library.

- ❑ Option-click the Symbol tool and select the Symbol radio button.
- ❑ Click the mouse button to place the Symbol tool in the drawing.
- ❑ With one of the symbols selected, choose Set Symbol Tags from the Tool menu. The Set Tags dialog will open.



- ❑ Notice the symbol field displays the name of our symbol as it appears in the Symbol Library and cannot be edited.
- ❑ Edit the data in any Field and Click OK. In our example we edited the data in field 1 to increase the value from 3" to 6" and increased the cost from 300.00 to 600.00.
- ❑ Select any of the other symbols, and you'll notice the values have automatically been updated.

Setting up a one-to-many relationship using Symbols and Symbol Tags is a fast and efficient method of updating drawings and cost estimates. Once the Symbol Tags have been updated in the drawing, the Save As Symbol Report feature may be used to create an updated report that may be imported into a database or spreadsheet application where additional calculations or analysis may be performed.



For more information about creating symbols, see [New Library](#). For more information on Saving Symbol Reports see 'Symbols'. For more information on inserting symbols, see [Symbol Tool](#) elsewhere in this manual.



NOTE: Only Symbol Tags, when used in conjunction with Symbols, can have a one-to-many relationships as described earlier. In all other cases, Symbol Tags are a one-to-one relationship; changing the data in a symbol tag field only updates the data for the selected item.

Verify Drawing...

Verify Drawing options allow you to globally delete objects defined by type and size. It also allows you to delete stacked multiple duplications and objects that are off the page.

Verify Drawing clears special objects that you define. You can specify objects by type and size. You can isolate lines and arcs in your specification along with empty text blocks. Lines and arcs can also be defined as general objects like polygons, circles, and rectangles. Objects and lines .015" or larger and arcs with an angle of one degree or more can be defined. Points and text are not included. You can search for objects on the current layer or all visible layers if Edit All Layers is On.

Verify Drawing will find and delete stacked multiple duplications. These are objects copied using the Duplicate command and left stacked directly over each other. Executing Verify Drawing will automatically search for and display the number of extra duplications on the current layer. If Edit All Layers is On, Verify Drawing will automatically search for and display the number of extra duplications on each individual layer.

Verify Drawing will locate and delete objects that have been accidentally thrown completely off the page. Executing Verify Drawing will automatically search for and display the number of objects that reside off the page on the current layer. If Edit All Layers is On, all layers will be searched.



NOTE: Verify Drawing works on visible layers only. Objects that fit specifications, but reside on hidden layers, will be ignored. Verify Drawing does not look inside of groups or symbols. To have Verify Drawing look inside of a group or symbols, use the Edit Groups menu item or Ungroup the items first.



NOTE: To apply Verify Drawing to all visible layers, make sure to turn Edit All Layers On. If Edit All Layers is Off, then Verify Drawing will be applied to the active layer only.

The Clear options provides a fast clean-up tool for removing small stray marks on your drawing before you plot, and it is a good way to clear odd objects from translated files. More importantly, it is an analytical tool that allows you to examine all cleared objects found before permanently deleting them. Each time a deletion is made, the deleted objects are pasted into a new drawing. If you realize an object has been found that should not be deleted, you may paste it back into the drawing.

Verify Drawing is a powerful tool. It is important to know how it calculates the objects you specify so that it removes only the objects you want removed. Below is an explanation of the calculations it executes and the relationship between Found and Total Found.

Suppose you have a drawing containing the following 25 objects:

10	Lines	1"
5	Circles	1.5" dia.
5	Squares	.75" x .75"
5	Arcs	40°, .5" x .5"

Under Type, in the dialog, there are four item specifications: Lines, Arcs, Empty Text Blocks, and Objects. If you select all of the objects and specify 1" lines to be deleted, clicking Calculate will display the following:

Found	Total
10	10

This indicates when you click OK, a total of 10 items will be deleted and that of those 10 items, all are lines 1" in length or shorter.

If, before clicking OK, you also specify 1" Objects to be deleted, clicking Calculate will indicate:

Found	Total
10	20

This indicates you have found 10 objects whose bounding box is 1" or less, and when you click OK, a total of 20 items will be deleted. Of those 20 items, 10 are lines 1" in length or shorter and 10 are objects whose bounding box is 1" or less, which represents the 5 squares and the 5 arcs.

Each time you specify a new type of object and click Calculate, Found refers to the last additional items specified. Total Found indicates the total number of objects that will be deleted when OK is clicked.

If you choose Objects first instead of Lines, and specify a size of 1", Calculate will include all lines, arcs that fit the specification along with squares, circles, polygons; it will treat all items as Objects. You will see:

Found	Total
20	20

If, before you click OK, you choose Lines and specify 1", Calculate will indicate:

Found	Total
10	20

This indicates that 10 lines with a length of 1" or less have been found among the 20 objects. Starting with Objects and moving up the list of Types will give you a breakdown of the number of each type of object fitting the specification.

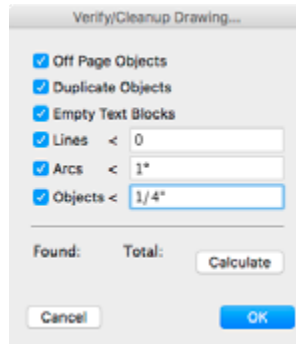
The above explanations indicate that choosing Lines will delete lines only. Choosing Objects will delete all objects, including lines and arcs, if size specifications fit.

Specifying Type

You may specify items to be deleted by selecting a Type. You may specify lines, arcs, empty text blocks, or objects. Objects includes all objects regardless of what they are—lines and arcs, as well as polygons, and circles or groups. An empty text block is defined as any text character that is invisible on the screen. This includes spaces, tabs, and carriage returns.

To delete or clear a specific type of item:

- ☐ Use a selection rectangle to define the search area or use the Select All command.
- ☐ From the Tool menu choose Verify Drawing. A dialog appears.



- ☐ Click the check box beside the desired item. A numerical field will appear to the right of Lines, Arcs, and Objects when each is selected.
- ☐ Enter the size of the item you want removed. You may specify lines or objects with a length of .015" or more and arcs with an angle of one degree or larger. For example, if you select Objects and enter .5", all objects whose largest dimension (height or width) is .5" or less will be deleted.
- ☐ Click Calculate. If any items fit your specifications, the number found will be displayed.

- ❑ Click OK. A new drawing titled Cleared Pieces will be created. The items deleted will be pasted into the new drawing and will remain selected.

Once the new drawing appears, you may examine each deleted piece individually. Click in the drawing to deselect the objects. Activate the Edit Window. Press the left bracket key ([) one time. One object will be selected and reported in the Edit Window. Using the bracket keys, you can step through the objects in the order they were created.



CAUTION: Verify Drawing abides by the active layer's scale. If your drawing has scaled layers, you should turn Edit All Layers Off and check layers one at a time. Failure to do this *will* result in the deletion of objects which are the correct size at their layer's scale, but which are smaller than the size specified at the active layer's scale.



NOTE: If you specify a type and size of object and then click Cancel, Verify Drawing will retain the specified information until you change it.

Off Page Objects

Off Page Objects will search for and delete all objects that reside completely off the page. A page is defined as the white drawing area seen on screen as defined in the Page Setup dialog and in the Drawing Setup, Size dialog.

To delete using the Off Page Objects function:

- ❑ Double-click in the drawing area to deselect everything.
- ❑ From the Tool menu choose Verify Drawing. A dialog appears.
- ❑ Click the Off Page Objects checkbox and click Calculate. The number of off-page objects found will be displayed.
- ❑ Click OK. A new drawing titled Cleared Pieces will be created. The items deleted will be pasted into the new drawing and will remain selected.

Duplicate Objects will search for and delete all stacked multiple duplications.

Stacked multiple duplications are objects that have been duplicated on one layer using the Duplicate command, but not distributed. They will appear as one item because they are stacked directly over each other.

To delete using the Duplicate Objects function:

- ☐ Double-click in the drawing area to deselect everything.
- ☐ From the Tool menu choose Verify Drawing. A dialog appears.
- ☐ Select the Duplicate Objects check box and click Calculate. The number of duplicate objects will be displayed. The original object that was duplicated will not be deleted.
- ☐ Click OK. A new drawing titled Cleared Pieces will be created. The items deleted will be pasted into the new drawing and will remain selected.

Objects that are drawn directly above each other, but on separate layers, are not considered multiple duplications. For example, if you draw a square, duplicate it and then cut and paste it onto another layer directly above the original object, Verify Drawing will consider these as single objects on separate layers and not multiple duplications.

Empty Text Blocks will search for and delete all text blocks in the drawing which do not contain any characters.

Empty text blocks are blocks of text which do not contain any characters and are most commonly seen when importing DWG or DXF files due to how many other drawing applications handle text. Empty text blocks can create unexpected printing results and serve no practical purpose in the drawing.

To delete using Empty Text Blocks:

- ☐ Double-click in the drawing area to deselect everything.
- ☐ From the Tool menu choose Verify Drawing. A dialog appears.
- ☐ Select the Empty Text Blocks check box and click Calculate. The number of empty text blocks will be displayed.
- ☐ Click OK. A new drawing titled Cleared Pieces will be created. The items deleted will be pasted into the new drawing and will remain selected.

Lines

Lines will search for and delete all lines in a drawing smaller than a specified length.

To delete using the Lines function:

- ☐ Double-click in the drawing area to deselect everything.
- ☐ From the Tool menu choose Verify Drawing. A dialog appears.
- ☐ Select the Lines checkbox, enter a line length in the Line field and click Calculate. The units used to specify line length match the Drawing Setup, Units.
- ☐ The number of lines smaller than the specified line length will be displayed.
- ☐ Click OK. A new drawing titled Cleared Pieces will be created. The items deleted will be pasted into the new drawing and will remain selected.

Arcs

Arcs will search for and delete all arcs in a drawing smaller than a specified arc angle.

To delete using the Arcs function:

- ☐ Double-click in the drawing area to deselect everything.
- ☐ From the Tool menu choose Verify Drawing. A dialog appears.
- ☐ Select the Arcs checkbox, enter an arc angle in the Arcs field and click Calculate. The units used to specify arc angle match the Drawing Setup, Units.
- ☐ The number of arcs smaller than the specified arc angle will be displayed.
- ☐ Click OK. A new drawing titled Cleared Pieces will be created. The items deleted will be pasted into the new drawing and will remain selected.

The Arcs function only finds arcs smaller than a specified arc angle. To search for arcs smaller than a specific radius use the Objects function.

Objects

Objects will search for and delete all objects in a drawing smaller than a specified size.

The Objects function will search for all object types including groups, symbols, rectangles, circles, arcs, lines and Béziars which are smaller than a specified size.

To delete using the Objects function:

- ☐ Double-click in the drawing area to deselect everything.
- ☐ From the Tool menu choose Verify Drawing. A dialog appears.
- ☐ Select the Objects checkbox, enter a size in the Arcs field and click Calculate. The units used to specify size match the Drawing Setup, Units.
- ☐ The number of objects smaller than the specified arc angle will be displayed.
- ☐ Click OK. A new drawing titled Cleared Pieces will be created. The items deleted will be pasted into the new drawing and will remain selected.

Arrange Menu

Bring to Front	⇧⌘]
Bring Forward	⇧⌘]
Send Backward	⇧⌘[
Send to Back	⇧⌘[
Move To Layer	▶
Group	⇧⌘G
Ungroup	⇧⌘U
Lock	⇧⌘L
Unlock	⇧⌘⌘L
Hide	
Show All	
Align	▶
Align...	
Rotate	R
Rotate by Centers	
Rotate to	▶
Flip Horizontal	
Flip Vertical	

The Arrange menu is used to arrange objects in relation to each other and the drawing itself. From this menu, you can move objects from within levels of the current layer of the drawing and multiple selected objects can be grouped and treated as one object.

Once drawn, objects can be protected from further modification by locking them.

In addition, other commands give you the ability to align objects with respect to themselves or specified points on the drawing.

Finally, objects can be rotated with great precision and ease.

The following section describes each of the Arrange menu commands in detail.

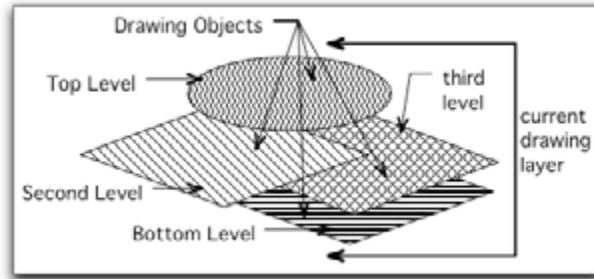
PowerCADD remembers the order in which objects were drawn, essentially stacking objects one upon the other as they are drawn on the active layer. Objects drawn first are considered to be on the bottom-most level (bottom of the stacking order). Objects drawn last (most recent) are considered to be on the top level (top of the stacking order). The object stacking order is unique for each layer in the drawing.

The commands Bring to Front, Send to Back, Bring Forward, and Send Backward give you the capability to move objects within levels of the currently selected layer of the drawing. For example, in a drawing, the most recently drawn object will, on occasion, cover other previously constructed shapes. This may require shifting the covered objects to a higher level so they can be modified.

When objects are moved up or down a level you are adjusting their stacking order.

When choosing Bring Forward or Send Backward, an object moves up or down the object stacking order, one position at a time, relative to other objects on the same layer. When choosing Bring to Front or Send to Back, an object moves to the top-most or bottom-most position in the layer stack relative to objects on the same layer.

The following diagram illustrates the relationship between levels and layers in a drawing.



TIP: The following items in the Arrange menu also appear in the main drawing window toolbar: Bring to Front, Bring Forward, Send to Back, Send Backward, Group, Ungroup, Lock, Unlock, and Show/Hide Shadows.



NOTE: It is important to understand the difference between drawing layers (as they appear in the Layer Window) and how objects are layered relative to each other as they are drawn. Objects may be moved forward or backward relative to other objects on the same layer. Layers may be moved up or down relative to other layers in the drawing.

Bring to Front

Bring to Front brings the selected object to the top level of the active layer.

The most recently drawn object is always in the top level of the current layer of the drawing. Each object subsequently drawn replaces the original position of the objects drawn earlier, thus forcing them down a level in the object stacking order.

To manually bring objects to the top level of the active drawing layer:

- ☐ Select the object to be brought to the front (top) level of the active drawing layer.
- ☐ Choose Bring to Front from the Arrange menu.
- ☐ The selected object will now be placed 'on top' of previously drawn objects, becoming the top-most object in the stacking order on that layer.

If you bring more than one object to the front, those objects will retain the order with respect to each other, but they will be moved to the top of the current layer.

Bring Forward Send Backward

Bring Forward/Send Backward allows you to change the stacking order of objects up or down one level at a time relative to other objects on the same layer.

To use Bring Forward/Send Backward:

- ☐ Select the object you wish to rearrange.
- ☐ Choose Bring Forward or Send Backward from the Arrange menu.
- ☐ The selected object will be moved up or down the object stacking order depending on which command you select.

Send to Back

Send to Back puts the selected item at the bottom level of the active layer.

To send an object to the back (bottom) level of the current layer:

- ☐ Select the object to be sent to the back level of the drawing.
- ☐ Choose Send to Back from the Arrange menu.
- ☐ The selected object will now be placed 'behind' all previously drawn objects.

When you send items behind a large filled object, the object sent to back will no longer be seen. Therefore, if you need to bring it back to the front later, you must move the obstructing objects or not fill them.

If you need to send an object to the back that you know must be brought forward later, draw a small object to the side of the drawing and group it with the object you wish to move and send the group to the back. To bring the object back to the front, select the object you grouped with it and choose Bring to Front. This way, you are able to retrieve the object without disrupting the drawing.

Move To Layer

Allows you to select which visible drawing layer an object should be moved to. The layer the object is currently on, and hidden layers are dimmed (unavailable) in the list. The list of layers presented is from the active drawing. Moving an object removes it from its current layer onto the selected layer in the same X,Y location.

To send an object to a specific drawing layer:

- ☐ Select the object to be moved to a new layer.
- ☐ Choose Move to Layer from the Arrange.
- ☐ Select a layer name from the list of layers.

The selected object will now be moved to the new layer in their original X, Y location. If multiple objects were selected, they are all moved to the same chosen layer. If Edit All Layers is on, and the objects selected were on different layers, they are all moved to the chosen layer.



TIP: Move to Layer is also available from a contextual menu when you Control- or right-click on an object. The contextual menu also includes a special menu item, Go To Layer, which allows you to change the active drawing layer. See Contextual Menus earlier in this chapter for information.

Group

Group takes several selected objects and groups them into one object.

Grouping objects allows you to manipulate more than one object at a time as a single item.



NOTE: This command is not the same as using the Combine tool.

Group collects items together in the PowerCADD drawing and locks their relationship together. Grouped items cannot be individually changed until first ungrouped or by using the Edit Groups menu item. Grouped items are moved and stretched together. Stretching modifies all of the items in the group proportionally to each other. Groups may be stretched in both the X and Y direction in unequal proportions. Stretching X and Y does not have to be the same amount.

To group multiple selected objects:

- ☐ Select the objects to be grouped.
- ☐ Choose Group from the Arrange menu.
- ☐ The objects will now be grouped and considered as one object.

There is no limit to the number of objects that can be grouped. Groups may also be nested within groups. If Edit All Layers is On, only selected objects on the same layer will be grouped.

Grouped objects may be ungrouped with the Ungroup command.

When objects are grouped, the resulting group takes the level position of the uppermost object. For example, if the top-level object is grouped with the bottom level object, the resulting group will be present on the top level on the current drawing layer. Undo does not work with Group. Therefore once the objects have been grouped, and the level placement is lost, you must return them to the correct level individually if level placement is critical.



NOTE: Attributes such as line weight and dashes for grouped objects are not specified by checkmarks in the menus. Although all objects in the group may be modified in unison (color, line weight, etc.), you cannot select a group and look at the group attribute, because each object in the group maintains its own unique attributes.

Ungroup

Ungroup separates previously grouped objects so they can be treated as individual objects again.

To Ungroup previously grouped objects:

- ☐ Select the object to be ungrouped.
 - ☐ Choose Ungroup from the Arrange menu.
 - ☐ The object will now be ungrouped into its original objects.
 - ☐ If group contained multiple nested groups, continue applying the Ungroup menu item until you hear a beep, indicating all items have been ungrouped.
-

Lock/Unlock

Lock allows you to lock objects so they cannot be altered until you Unlock them.

To use Lock:

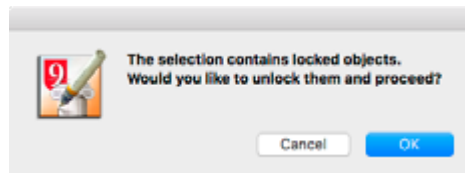
- ☐ Select an object to be locked.
- ☐ Choose Lock from the Arrange menu.
- ☐ The object can no longer be resized or moved.

To Unlock:

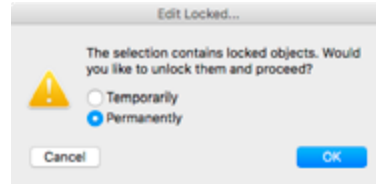
- ☐ Select an object to be unlocked.
 - ☐ Choose Unlock from the Arrange menu.
 - ☐ The object has now been returned to a normal state.
-

Moving Locked Objects

If you attempt to move, rotate, scale or other wise manipulate locked objects, the following dialog box will appear asking you to confirm the operation. Clicking OK allows you to complete the operation and the objects are automatically locked again when the operation is complete.



Modifying Locked Objects If you attempt to modify a locked object (examples: change attributes, scale, or rotate an object), the following dialog box will appear asking you to confirm the Unlock operation.



Temporarily: Checking this box will unlock the objects, clicking OK will complete the current operation. The objects are then locked again when the operation is complete. For example, if you attempt to move multiple objects, one of which is locked, Temporarily will unlock the objects to complete the move then lock it again in its new location.

Permanently: Checking this box will unlock the objects, clicking OK will complete the operation. The objects remain unlocked when the operation is complete.

Cancel: Click to abort the current operation, any locked objects will remain locked.

This capability is helpful when working under crowded conditions. If you have trouble selecting an object because of objects in close proximity, lock those objects. Once locked, these objects cannot be moved until you unlock them.

Hide

Hide allows you to hide selected objects in a drawing. This can be helpful if you are working in a crowded drawing and need to temporarily hide objects to speed up the drawing or editing process.

To use Hide:

- ☐ Select the objects.
- ☐ Choose Hide from the Arrange menu.
- ☐ The selected objects are not drawn on screen.

Objects remain hidden until you choose Show All from the Arrange menu.

Show All

Show All allows you to show all hidden objects in a drawing.

To use Show All:

- ☐ Choose Show All from the Arrange menu.
- ☐ All objects hidden using Hide from the Arrange menu are displayed on screen.

If there are no hidden objects in a drawing this menu item is dimmed.



To globally Show or Hide only Shadows, see [View Menu, Hide/Show Shadows](#) discussed later in this chapter.



NOTE: Choosing Print from the File menu when objects are hidden will bring up a warning dialog. Clicking OK will continue the print operation and the hidden objects will not be printed. To print the hidden objects, click Cancel to abort the print operation. Choose Show All from the Arrange menu and print the drawing.



CAUTION: The hidden state of objects is remembered when a drawing is saved and closed. The next time you open the drawing, the objects will be still be hidden. If you think information is missing in your drawing missing don't panic! Check the Arrange menu and choose Show All to see all previously hidden objects.

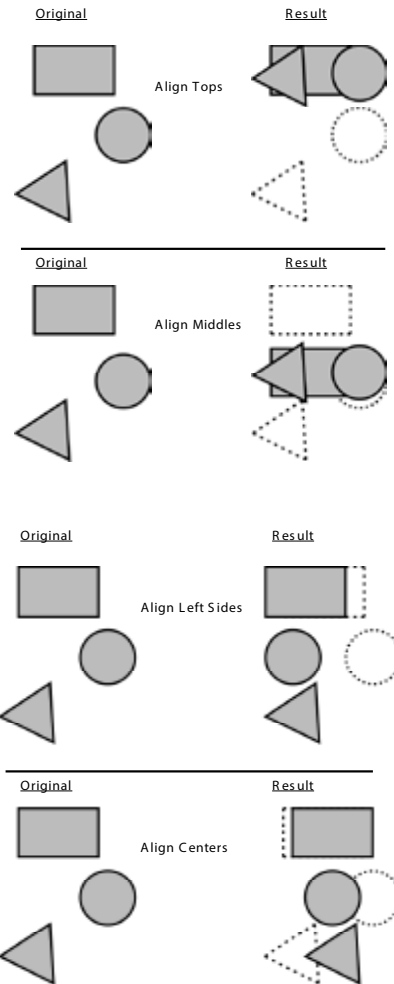
Align



Align allows you to line up two or more drawing objects with respect to their centers, sides, bottoms, and tops.

To align two or more selected objects:

- ☐ Select the objects you wish to align.
- ☐ Choose Align from the Arrange menu. The align submenu appears as shown on the left.
- ☐ Choose the alignment you desire and release the mouse button.
- ☐ The selected objects will be redrawn according to their new alignment orientation.



TIP: Locked objects do not move during an align operation. Selected objects are realigned relative to the anchored (locked) object.

Align to Grid

Align to Grid snaps all selected objects to the grid, when the grid is turned on.

To align objects to the grid:

- ☐ Select the object(s) to be aligned to the current grid.
- ☐ Choose Align to Grid.

See [Drawing Setup, Snap](#) in the Layout menu to turn Grid snap Off and On.

CAUTION: Align to Grid aligns all corners (vertices) of the selected object to the grid.

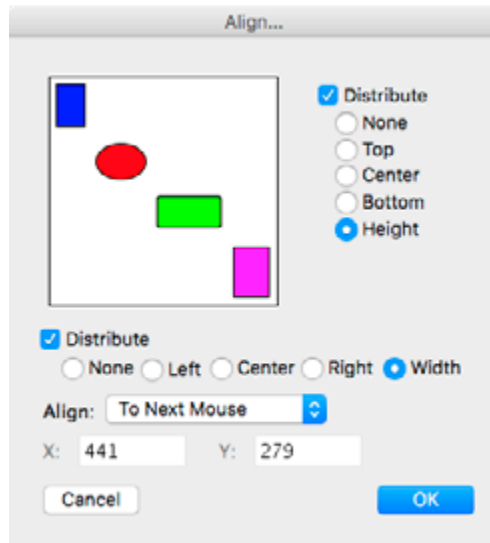


Align...

Align gives you a precise method of lining up and distributing objects.

To use Align:

- ☐ Select the objects you wish to align.
- ☐ Choose Align from the Arrange menu and the following dialog will appear:

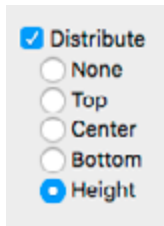


- ☐ Choose the desired alignment by selecting the appropriate buttons, then click OK.
- ☐ Align will shift the objects into place.

In the dialog box you have many options from which to choose. You may align objects horizontally, vertically, or a combination of the two. The objects can also be aligned to the grid, to an object, to a point, or to the next mouse click. The example window provides immediate visual feedback based on your specified options.

The vertical alignment radio buttons are located along the right side of the example window. These buttons allow you to distribute, cancel vertical alignment, or align tops, centers, and bottoms. These options apply only to the vertical alignment but can be used in conjunction with the horizontal alignment.

Distribute allows you to place the alignment points an equal distance apart along the vertical axis. For example, if you specify the objects are to be aligned by tops, the tops of the objects will be aligned in such a manner they are equally spaced. An easy way to view the function of Distribute is to select four objects. Align the objects and Distribute tops. Measure top to top of each object starting at the top-most object. You will find the tops of all objects are spaced evenly apart.



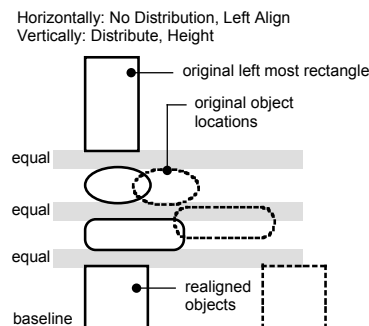
None allows you to cancel an alignment. This is helpful when you wish to align only one axis.

Top allows you to align the tops of all selected objects. If you are aligning multiple objects to the top, all objects will be aligned with the upper-most selected object. This upper object will not move and all other objects will snap into alignment with it.

Center allows you to align the object's top/bottom centers. All selected items will form a bounding rectangle. PowerCADD will calculate the centerline of this bounding box and align all selected items to the centerline.

Bottom allows you to align the bottoms of all selected objects. If you are aligning multiple objects to the bottom, all objects will be aligned with the lower most selected object. This lower object will not move and all other objects will snap into alignment with it.

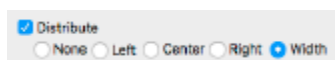
Height allows you to align objects so the free space between the selected objects is equalized. When aligning multiple objects, the top-most and bottom-most objects do not move vertically. In the example below notice the free space between the original object locations are not the same. The free space between the realigned objects is equal.



The horizontal alignment radio buttons are located along the bottom of the examples window. These buttons allow you to distribute, cancel alignment, or align left, centers, or right sides. These options apply only to the horizontal alignment, but can be used in conjunction with the vertical alignment.

Distribute allows you to place the alignment points an equal distance apart. For example, if you specify the objects are to be aligned by left, the objects will be distributed in such a manner the left side is equally spaced. An easy way to view the function of distribute is to select four objects. Align the objects and Distribute left. Measure the left to left of each object, starting with the left-most object on the page. You will find the lefts are spaced evenly apart.

None allows you to cancel an alignment. This is helpful when you wish to align only one axis.

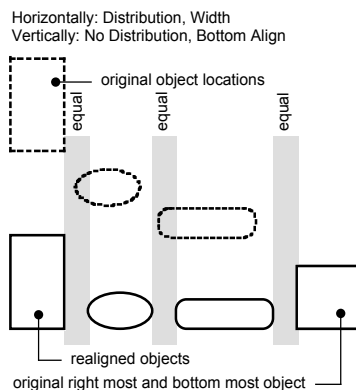


Left allows you to align the left sides of the selected objects.

Center allows you to align the objects' left/right centers of the selected objects. All of the selected items will form a bounding rectangle. PowerCADD will calculate the vertical centerline of this bounding box and align all selected items to the centerline.

Right allows you to align the right sides of all selected objects. If you are aligning multiple objects to the right, all objects will be aligned with the object located to the extreme right. This object will not move and all other objects will snap into alignment with it.

Width allows you to align objects so the free space between the selected objects is equalized. When aligning multiple objects, the left most and right most objects do not move horizontally. In the example below notice the free space between the original object locations are not the same. The free space between the realigned objects is equal.



When you align objects to center on both axes, PowerCADD will calculate a bounding rectangle encompassing all selected objects. The exact center of the bounding rectangle will be found and the centers of all objects will be aligned to the center of the bounding rectangle.

When you are aligning objects to a specific point, all objects will be aligned to that point. For example, if you align all objects to the top left point, all objects will be aligned with the upper left most object on the screen. This upper left most object will not move and all selected objects will snap into alignment with the specified point on that object.



The Example Window

The Example Window allows you to view the type of alignment before it is executed. If you disable one axis and select the different alignment buttons, you can get a better idea of how the items are aligned. For example, if you disable the horizontal alignment and select the left align, all example objects will move to the left. Center and right work the same way as the buttons on the horizontal alignment, if the case is reversed. If both axes are activated, all of the example objects align to the specified point on both the vertical and horizontal axis.

Align Menu



To Each Other will align the objects as specified by the location table in the dialog box.

To Next Mouse works the same as point alignment except instead of entering X,Y coordinates, you specify the point of alignment by clicking on the drawing in the location of desired alignment.

To Specified Point will align the objects as specified by the location table, but you specify the exact location of the alignment point by entering X,Y coordinates

To Grid allows you to align all selected objects to the grid. To align to grid, specify the alignment point of the objects using the buttons below and to the right of the example window. The point you specify, such as bottom right, will snap to the nearest grid division intersection. For example, if the grid is set to one inch, there will be a grid division intersection at every one-inch increment on the screen.

Objects are aligned by their bounding rectangle. When you align an object to a single axis, you are aligning either a side of the bounding rectangle or a line running through the center of the bounding rectangle on the perspective alignment axis. (Center aligns to the centerline.) When you align an object to both the vertical and the horizontal axes, you are aligning either specific handles of the bounding rectangle or the center point of the bounding rectangle. This is most apparent when you align the top left

points of a circle and a rectangle. The exact corners of the objects are not aligned, but the top left handles of their bounding rectangles are.

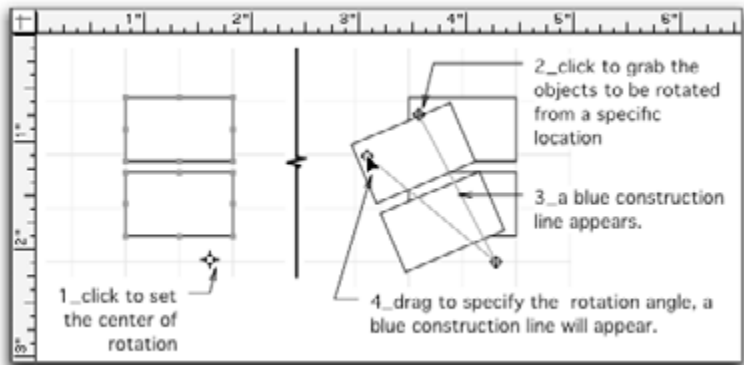
When you activate Distribute on both the vertical and horizontal axes, the specified alignment points will be spaced evenly apart both vertically and horizontally.

Rotate

Rotate permits the rotation of selected objects by dragging the rotation cursor in the desired direction.

To rotate a single object or grouped objects:

- ☐ Select the object to be rotated.
- ☐ Choose Rotate from the Arrange menu or press R.
- ☐ When you see the rotation target cursor in the drawing area, click to set the center of rotation.
- ☐ Click to grab the objects to be rotated from a specified point.
- ☐ Drag the mouse to rotate the items. The original objects are also shown for context.



- ☐ When object is positioned as desired, release the mouse. The object will be rotated to the new position.
- ☐ Pressing the Tab key when dragging to rotate the objects will take you into the Edit Window. Release the mouse, type the desired angle or rotation and press the Enter key. The objects are rotated the angle specified in the Edit Window.

Dynamic snapping is in effect for all portions of the rotation operation. You may use the snap command to specify the center of rotation, where you grab an object for rotation or to complete the rotation operation. This makes it very easy to dynamically rotate objects to match existing drawing geometry without having to calculate or know the rotation angle.

The following keyboard commands can modify the degree of rotation:

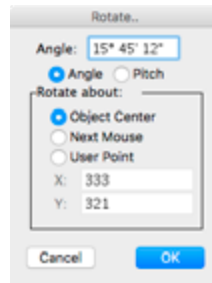
- ☐ The Shift key constrains the rotation to the snap angles defined in Drawing Setup.
- ☐ Command performs a free rotation with no preset angle increments.
- ☐ If no key is pressed the rotation automatically steps in one degree increments.
- ☐ The Tab key suspends the rotation operation and enters a specific value in the Edit Window as described above.

Rotate...

Rotate permits the rotation of selected objects by specifying a precise angle using a dialog box. You can rotate objects about their centers, next mouse click or a known x,y coordinate point in the drawing.

To specify a rotation angle:

- ☐ Select the object to be rotated.
- ☐ Press the Option key while choosing Rotate from the Arrange menu.
- ☐ The Rotate dialog will appear.



- ☐ Type the number of degrees, minutes, and seconds you wish the object to be rotated. A positive value rotates objects counterclockwise; negative values produce a clockwise rotation.
- ☐ In addition, specify the point about which you prefer the object to rotate, such as the center. If you wish to specify a specific X,Y coordinate, click User Point and enter values in the X and Y fields.

Degrees, minutes, and seconds may be entered as an angle, even though unit accuracy is set to decimal degrees. For example, these are all valid entries:

12° 30' 30"

12 30 30

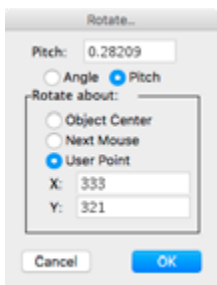
12 30'

12 30 30"

360/5

Decimal degrees may be entered when unit accuracy is set to degrees, minutes, and seconds. Any dialog box expecting angular dimensions may

be used as an angle conversion calculator, for example in a drawing setup for degrees, minutes and seconds, you can enter angles in decimal degrees. When the dialog box is opened again the angle will be converted to the degrees, minutes and seconds.



- ☐ Type in the X,Y values you wish to be the center of rotation.
- ☐ Type in the rotation angle. The value of the angle may be entered in either decimal degrees or degrees, minutes, and seconds, depending upon your preference. Objects can be rotated by angle or by pitch. Pitch is rise over run (slope) and is used in the construction trades.
- ☐ You may rotate the object about the next mouse click in the drawing area. Snap commands may be used to specify a precise location for the center of rotation.
- ☐ Upon specifying the requested values, press the Return key or click OK. The object will be redrawn according to the specifications you provided in the dialog.

For rotating multiple items, follow the directions above, but first; select all the objects to be rotated. Rotating multiple items about object center will rotate all items about the center of their extended bounding box (i.e. it rotates all objects as if they were grouped). A new bounding box center is calculated each time an item is rotated. Successive rotation can appear to be about a different center depending upon the object.



TIP: To have objects retain their position relative to each other and rotate about their individual centers, use 'Rotate by Centers' discussed later in this chapter.

Pitch is a common term in the construction industry. Pitch represents the slope (X/Y) and is usually expressed as a ratio (5/12), but may be expressed as the decimal equivalent.

When working with rotated objects, you may use the Edit Window to realign the objects.

To align objects through the Edit Window:

- ☐ Make sure the Edit Window is open.
- ☐ Select the object you wish to rotate.
- ☐ Press the Tab key to activate an edit field. The selected cell is indicated by a border. The Tab key will move you through the cells.
- ☐ Tab to the Angle field.
- ☐ Enter an angle of 45°.
- ☐ Press the Return or Enter key and the object will snap to 45°.

This method can be used to rotate objects to any angle. The angle you enter will correlate with the currently active measurement system: Cartesian, Gradient, etc. For example, if you have a rectangle rotated to 314°, and you edit-rotate it 315°, there would only be a change of one degree. (The object is not rotated from its present location; it is rotated from zero degrees.) This makes rotating objects very simple because you do not have to figure out how many degrees, positive or negative, you need to move to reach the desired point.

Rotate by Centers

Rotate by Centers permits the rotation of multiple objects about their unique object center points. Rotate by Centers is only available if two or more objects are selected. To rotate a single object about its center use the Rotate or Rotate To menu items.

To rotate multiple objects about their unique center points:

- ☐ Select the objects to be rotated and choose Rotate by Centers from the Arrange menu.
- ☐ The cursor changes to a target cursor.
- ☐ Click the mouse anywhere in the drawing window and drag to dynamically specify a rotation angle. The Edit Window will display the current rotation angle. Click to complete the rotation operation.
- ☐ To constrain the rotation to preset snap angles, press the Shift key when dragging to specify the rotation angle.
- ☐ To specify an exact rotation angle, press the Tab key when dragging to specify the rotation angle. The Edit Window will activate. Type a rotation angle and press the Return key.

Rotate to

Rotate to is a submenu for rotating to 0° or 90°, left, and right.

Rotate To Zero

Rotate to Zero rotates rectangles, arcs, ellipse, and text to zero degrees.

Rotate to Zero
Rotate Left
Rotate Right
Rotate 180

To rotate a rectangle, arc, ellipse, or text to zero degrees:

- ☐ Select the object(s) to be rotated.
- ☐ Choose Rotate to Zero from the Arrange menu.
- ☐ The object's orientation will place it at zero degrees.

A polygon, for example, cannot be rotated back to zero degrees. You can use Undo to counteract an unintended rotation.



Rotate Left

Rotate Left rotates the currently selected object 90° to the left—counterclockwise.



To rotate an object 90° to the left:

- ☐ Select the object(s) to be rotated.
- ☐ Choose Rotate Left from the Arrange menu.

Rotate Right

Rotate Right rotates the currently selected object 90° to the right—clockwise.



To rotate an object 90° to the right:

- ☐ Select the object(s) to be rotated.
- ☐ Choose Rotate Right from the Arrange menu.

Menus

Flip Horizontal



Original Object



Flip Horizontal

Flip Horizontal reverses drawn objects horizontally.

To flip an object horizontally:

- ☐ Select the object(s) to be flipped.
- ☐ Choose Flip Horizontal from the Arrange menu.

Flip Vertical



Original Object



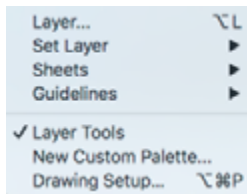
Flip Vertical

Flip Vertical reverses drawn objects vertically.

To flip an object vertically:

- ☐ Select the object(s) to be flipped.
- ☐ Choose Flip Vertical from the Arrange menu.

Layout Menu



The Layout menu permits the setup and modification of your drawing environment, including:

- ☐ Layers, Sheets and other drawing layer management options
- ☐ Guidelines
- ☐ Drawing Units
- ☐ Custom tool palettes
- ☐ Drawing Setup parameters including scale, units, drawing size and others

Layer...

Layer allows you to create, arrange, and edit layers. You can specify a variety of layer parameters including changing layer status (hidden, visible, dimmed) and layer attributes (scale, color, locking, transparency).

Layers are the electronic equivalent of overlays or acetates used in manual drafting. Traditionally, when using these transparent or semi-transparent sheets, the draftsperson would trace objects from previously drawn pages onto a new page and then add additional information to the tracing. For example, a set of architectural floor plans generally consists of a series of overlays of the basic floor plan. Once the floor plan is established, the walls are traced and used as a base for plumbing, electrical, heating, ceiling, foundation plans, and so on.

While manually tracing was traditionally more convenient than measuring and redrawing, an efficient use of layers in PowerCADD will far surpass all manual drawing overlay conventions.

Layers should be used for two important reasons:

- ☐ **Speed.** Organizing the elements of a drawing with layers allows you to hide layers that are not being used. Since only the visible layers will redraw on the screen as changes are made, the extra time, which would normally needed to redraw everything, can be significantly reduced. Planning for and using layers requires a bit more thought in the short term, but will be very beneficial as your drawings get larger and more complicated.
- ☐ **Drawing Management.** Using a separate layer for elements like, walls, dimensions, doors, windows, notes, and so on will allow you to hide or dim elements which might otherwise be in your way while drawing. Using separate layers, Sheet Setups can be saved which remember your layer order and status, allowing you to print only specific layers at a time; an entire set of working drawings can be stored in a single drawing file but separated using many layers.

A single PowerCADD file can have almost unlimited layers, depending upon drawing complexity and the memory capacity of the Macintosh computer.

Objects can only be created on the active layer. Only one layer may be active at one time. The active layer's name will be displayed in the drawing window's title bar beside the drawing's name.

Editing objects on layers other than the active layer can only take place when Edit All Layers is On.



For more information on Edit All Layers, see [Edit All Layers](#).

NOTE: To access and modify layers, you must have the Layer Window External loaded. If you do not, Layers will not appear under the Layout menu.

Creating and Re-naming Layers



To create a layer:

- ☐ Choose Layers from the Layout menu and the floating Layers window will appear.
- ☐ Click New. A new layer will appear in the list. The new layer's name will appear in the name immediately below the layer list.
- ☐ Rename the layer as desired by typing the layer name you wish to use. As you enter the layer name, the modification appears in the layer list.

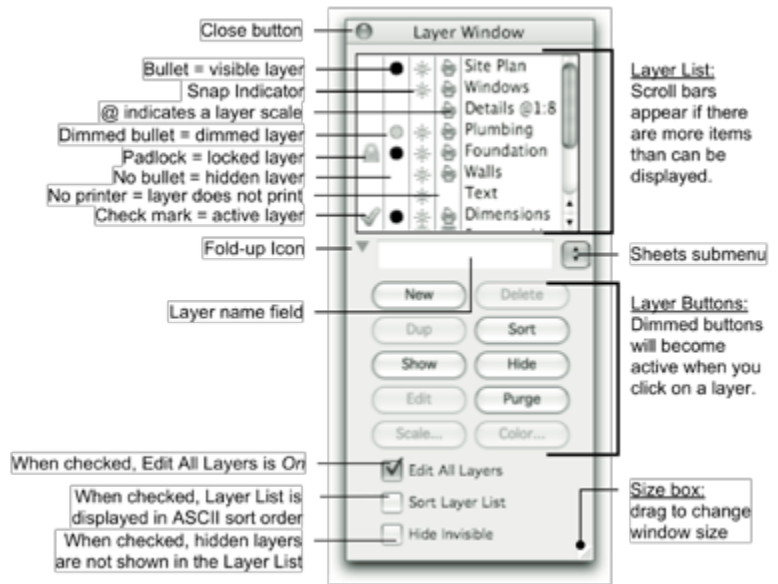
To add more layers, continue to click New and name them. New layers are always appended to the top of the layer stack and made the active drawing layer, but can be moved to any position as described later in this section.

To change the name of an existing layer:

- ☐ Select the layer name in the Layers window.
- ☐ Use the mouse to highlight the text in the layer name field and type the new text.
- ☐ The changes are reflected instantly in the Layers window.

Locked layer names cannot be edited until unlocked. See the Layer Attributes section of this chapter for more on setting layer attributes and locking/unlocking layers.

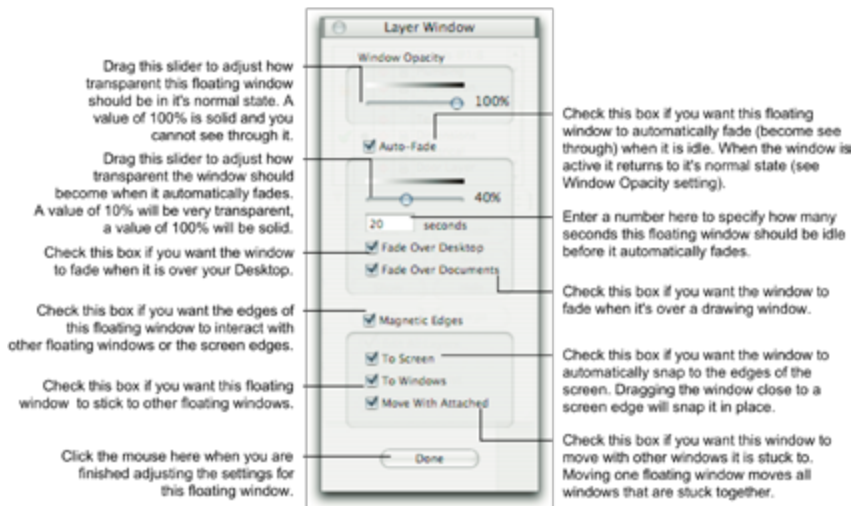
When you choose Layers from the Layout menu or select Layers from the Windows submenu of the File menu, the following window will appear:



Auto-Fade

The Layers window floats on top of your drawing and can be moved and resized.

To adjust the Auto-Fade settings for the Layers window, Control- or right-click anywhere in the body of the Layer window except the Layer List. The following sheet will be appended to the Layer Window.



Make the desired changes and click Done.

Layer List

The layer list displays the order and attributes of the layers. Layers can be completely visible, hidden, or dimmed. Many layers can be visible at one time. Layers may be rearranged, locked, assigned colors, or assigned scales.

The order of the layer list indicates the top-to-bottom stacking order in the drawing. This is more obvious when adding a new layer, because it goes to the top of the list. Similarly, objects drawn on the top layer will appear in front of the objects on layers below.



NOTE: If your mouse has a scroll wheel, you may use the scroll wheel to move up and down the layer list if it is too long to display in the current Layer Window. If you do not have a scroll wheel mouse use the scroll arrows or drag the scroll bar to move up and down the layer list.

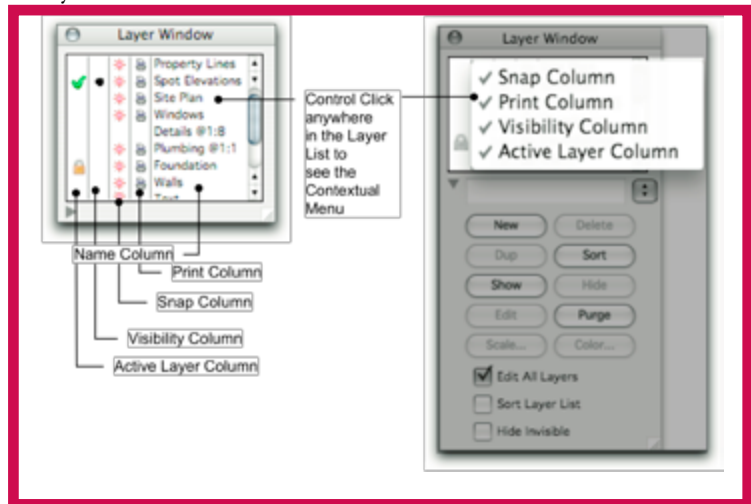
Layer List Column Display

You can customize which columns are displayed in the Layer List.

To select which columns are visible:

- ☐ Control- or right-click in the layer list.
- ☐ Choose the name of the column from the contextual menu to show or hide a column. Checked items in the list are displayed.

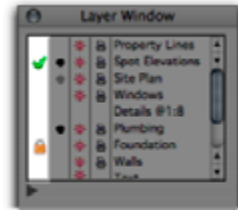
The layer Name Column cannot be hidden



The active layer is indicated by a checkmark in the Active Layer Column.

To set the active layer:

- ❑ Click in the Active Layer Column beside the desired layer.

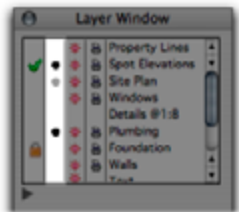


Other ways to change the active layer is are to click on the Layer Up or Layer Down arrows in the lower left corner of the document window, or use their keystroke equivalents, selecting the layer name from Layers in the Set Layer, Layout menu or from the Layer list located at the bottom left of the drawing window.

A padlock appears in the Active Layer Column for layers with a Locked attribute setting.



TIP: When setting the Command keys, you might want to consider the convenience of the L key, the Comma key and the Period key, which are located together, for calling up the Layer window and toggling the active layer.



Visibility Column

The Visibility Column contains bullets indicating a layers status as visible, dimmed, or hidden. A visible layer is indicated by a bullet. A dimmed or dotted bullet indicates a dimmed layer; if no bullet is displayed, the layer is hidden or invisible. Repeatedly clicking these bullets will cycle through visible, dimmed, and hidden settings. A hidden layer will become visible automatically if it is made active. Sheet setups remember a layers visibility status.

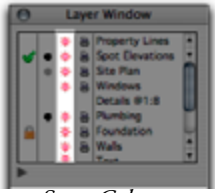
To modify a layer's visibility status:

- ❑ Choose Layers from the Layout menu and the floating Layers window will appear.
- ❑ Set the status by clicking in the Visibility Column next to the layer name.

To set the visibility status of multiple layers at the same time Shift-click to select sequential layers, or Command-click to select multiple nonsequential layers, and click in the visibility column next to a selected layer name.



NOTE: Filled objects on dimmed layers will appear transparent.

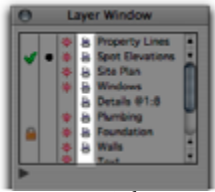


Snap Column

The Snap Column allows you to specify whether you can snap to objects on the specified layer. If snap indicator icon next to a layer name indicates you can snap to objects on that layer. No icon indicates objects on that layer cannot be snapped to. By default all new layers have the snap status turned on. Sheet setups remember a layers snap status.

To set the snap status of multiple layers at the same time:

- ☐ Shift-click to select sequential layers, or Command-click to select multiple nonsequential layers, and click in the snap column next to a selected layer name.



Print Column

The Print Column allows you to specify whether information on a specific layer should be printed. A printer icon next to a layer name indicates the layer will print. No icon indicates the layer will not print. By default all new layers have their print status turned on. Sheet setups remember a layers print status.

To set the print status of multiple layers at the same time:

- ☐ Shift-click to select sequential layers, or Command-click to select multiple nonsequential layers, and click in the print column next to a selected layer name.



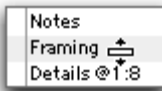
Name Column

The Name Column displays the name of a layer as entered in the Layer Name field. Layer scale or color attribute settings are reflected in the displayed layer name.

To set the Layer Attributes for multiple layers at the same time:

- ☐ Shift-click to select sequential layers, or Command-click to select multiple nonsequential layers, and double click on a selected layer name.

Moving Layers



To move a layer to a different position in the stack, place the cursor on the layer's name and press. When the cursor changes drag the layer to the new position in the list.

Moving layers up and down in the stack is just like setting the overlay order in manual overlay drafting. Sheet setups can be used to save the layer order allowing you to use a single file for more than one drawing. See Sheets later in this chapter for more information.

Modifying Layer Attributes

The advanced layer attributes setting including Layer Locking, Layer Color, and Layer Scale are intended to let you set certain parameters for a drawing before you begin the drawing process. You should plan ahead and thoroughly understand the advanced features. Take time to consider how these layer attributes will be used in your office and how they will affect others who may receive your drawings. Establish standards and inform *everyone*.



TIP: To set the Layer Attributes for multiple layers at the same time Shift-click to select sequential layers, or Command-click to select multiple nonsequential layers, and double-click on a selected layer name.



TIP: Layer Attribute settings, with the exception of Layer Scale, are remembered with Sheet Setups. For example a layer could be locked, 50% opacity with Sheet Setup 1 and be unlocked, 100% opacity with Sheet Setup 2.

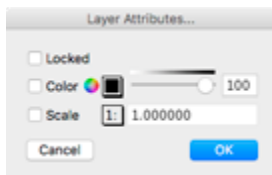


NOTE: If you have been using the Smart Layers External in previous versions of PowerCADD to control Layer Pen Color and Layer Scale in your drawings, you can quickly reset the drawings for use with Layer Attributes. Simply follow the instructions for setting Layer Pen Color and Layer Scale for the appropriate layers in your existing drawings as needed.



CAUTION: When using the advanced layer attributes, they will control such things as pen color and scale—automatically. Therefore, you should be alert to the conditions that will exist when drawing in this smart environment. We have indicated, where applicable, some of the conditions you should be aware of when using these powerful layer attributes. Because we cannot begin to address all of the possible conditions, which could exist in your drawing environment, it is up to you, the user, to be aware of the settings of the layer attributes when drawing and editing, especially pen color and scale.

Editing Layer Attributes



To edit a layer's attributes:

- ☐ Choose Layer Window from the Window menu and the Layers floating window will appear.
- ☐ Select the layer name and click Edit (or double-click the layer name) and the Layer Attributes dialog will appear.
- ☐ Set the desired layer attributes and click OK.

The name column in the Layer window will update to reflect the attribute setting. If a layer color was specified, the drawing window will update to reflect the color change.

Locking/Unlocking Layers

Locked will lock all objects on the selected layer. When a layer is locked, you will not be able to add, delete, or modify its objects in any way. However, you will be able to snap to objects on a locked layer when drawing on other layers.

To lock the selected layer:

- ☐ Click Locked. A checkmark will appear in the checkbox indicating the layer is locked.
- ☐ Click OK.

A padlock icon will be displayed beside the layer's name in the layer list. Also, an "L" will be displayed beside the layer's name when the layer list is accessed from the Set Layer menu. The status of a locked layer may be changed to visible, dimmed or hidden by clicking in the status column next to the layer name.



To unlock a locked layer:

- ☐ Select the locked layer and click Edit.
- ☐ Uncheck the Locked box and click OK.



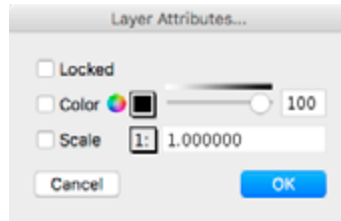
NOTE: The active layer cannot be locked. When the active layer is selected, the Locked attribute will be unavailable.



Color will assign a specified pen color to a layer. Once a layer's pen color is established, objects drawn on the layer will assume the layer's color attribute automatically. The Layer Visibility Status indicator will display in the selected color. To also display the Layer Name in the selected color check [Show Layer Color in Name](#).

NOTE: Layer color attributes override tool attribute settings for pen color, fill colors are not modified. If you can't seem to change an object's pen color, check to see if a layer color has been assigned to the layer the object is on.

Individual layer color can be helpful for large drawings where color-coding for clarity is necessary, like wiring diagrams, facility plans, and so on.



To assign a color to the selected layer:

- ☐ Click Color.
- ☐ Place the cursor on the **square color swatch** to the right of Color and click. The current PowerCADD color palette will appear.
- ☐ Select a color and click OK.

Layer Color Swatch



CAUTION: Colors displayed in the PowerCADD color palette are affected by [Preferences, Global Attributes](#).

To assign a color to the selected layer from the MacOS color panel:

- ☐ Select Color.
- ☐ Select the colored circle next to Color. The MacOS color panel will appear.
- ☐ Select a color and click OK.

You may also change the existing pen color of a layer; for example, if a layer has been given a color of red, you may change the layer to blue by using the same process for setting the color. When you return to the drawing window, all objects which exist on the layer in the previous color will change to the new layer color automatically.



NOTE: Pen color attributes are saved with the object. Turning a Layer Color Attribute off will revert object pen colors to their original attributes prior to assigning a Layer Color Attribute. For example, Layer 1 has a rectangle with a blue pen color and circle with a black pen color. Assigning

a Layer Color Attribute of red to Layer 1 will change the pen color for both the circle and square to red. Unchecking the Layer Color Attribute for Layer 1 will restore the original pen color attributes of blue to the rectangle and black to the circle.

Copy/Pasting Objects

Objects can be copied or cut from one color-assigned layer and pasted into a another color-assigned layer of a different color. Once pasted, the object will assume its new layer's color.

Layer Scale



CAUTION: The DXF and DWG Translators will abide by the drawing scale set in Drawing Setup. Layer Scale information will be ignored. See “DXF and Layer Scale” and “Externals” for more information.

Once a layer's scale is set, all objects constructed on the layer will assume its scale. When layers are changed, the drawing scale will change automatically.

Different layer scales can make complex, multi-detail drawings much easier to manage; you can give each detail its own layer. Because changing layers automatically changes the scale, you can work with confidence knowing your details are scaled correctly.

You may assign a different scale to each layer or to only a single layer. Layers which do not have a scale assigned will assume the drawing's global scale set in the Drawing Setup dialog.

To assign a scale to the selected layer:

- ☐ Click Scale.
- ☐ Drag to choose a predefined scale, or press Tab and type the desired scale in the numerical field.
- ☐ Click OK.



NOTE: When a layer with no layer scale assigned is activated, the drawing scale set in the Drawing Setup dialog will be used.

Edit All Layers and Layer Scale

The Edit Window will ALWAYS reflect the scale of the layer of the selected item. The Show Location Bar will ALWAYS reflect the scale of the active layer.

Therefore, you must remain alert when working on scaled layers with Edit All Layers On.

For example, if your active layer is at 1:1 scale, and you select a 4' x 4' square object on a layer with 1:48 scale, the Edit Window will indicate the object is 4' x 4'.



STOP & THINK: Because the Edit Window and Show Location Bar will ALWAYS report according to the active layer, you must be completely aware when working with Edit All Layers On. Make sure you are working on the correct layer, especially when creating and editing objects.

Dimensions and Layer Scale

Drawing Dimensions. Dimension layers *must* have the same scale as the layer objects being dimensioned. For example, if your floor plan is drawn at $1/4" = 1' 0"$ scale, the layer containing the dimensions for the floor plan should also be set to $1/4" = 1' 0"$ scale.

Editing Dimensions. When stretching dimensioned objects, the dimension will recalculate based on its layer's scale, regardless of which layer is active. If no layer scale is specified the drawings default scale specified in Drawing Setup is used.

Rescaling Dimensions. If you wish to rescale a dimensioned object, but not change its drawn dimension, use the Paste Special External's Paste At Scale option.

You may also change an object's drawn dimensions *without* changing the physical size of the object. Suppose you are creating an architectural model of a house plan drawn at $1/4" = 1' 0"$ scale, and you need full size patterns for cutting the foam core and balsa materials for the model. Simply copy the dimensions and paste them onto a layer scaled at 1:1. While the pasted dimensions are still selected, choose Edit from the Edit menu. When the dialog appears, just click OK. You will end up with full scale patterns for a 1:48 model without having to measure again.



For more information see [Paste Special](#) in the Edit menu section.

The DXF and DWG translator will abide by the current drawing scale, *instead* of each layer's scale. For example, if objects are drawn at a layer scale of 1:48, and translated with a drawing scale of 1:1, the resulting file will have to be scaled up by a factor of 48.



STOP & THINK: Because of the potential for confusion, using layer scale in drawings which will be converted to DXF or DWG is not recommended, especially drawings with dimensions.

If you must communicate via DXF or DWG:

- ☐ Hide all layers except those with the same scale.
- ☐ Set the Drawing Scale to match the visible layers' scale.
- ☐ Use the option for saving only visible layers found in the DXF or DWG translator and create a DXF or DWG file.
- ☐ Repeat this process for all the layer scales in the drawing. This will result in multiple DXF or DWG files for the same drawing.

—or—

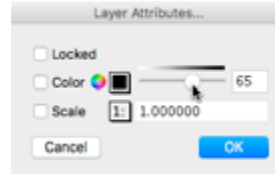
- ☐ If the drawing is a sheet of separate details or elevations, indicate the different scales on the drawing with each detail.
- ☐ Set the Drawing Scale to be 1:1, or to match the layer scale used most frequently in the drawing.
- ☐ Let the person receiving the file know which details will be correctly scaled, and which will need a scale factor applied to be true full scale.

Opacity

Opacity allows you to specify to what degree information is hidden or revealed below the layer. For example, a layer opacity of 100% hides all information below the layer. A layer set to 10% opacity would appear almost invisible, allowing you to see through to objects below. Adjusting layer opacity affects all objects on a layer, areas with no objects or objects with no fill or pen pattern are not affected.

To adjust the opacity attribute for layer:

- ❑ Drag the opacity slider bar shown below or enter a number in the opacity field.

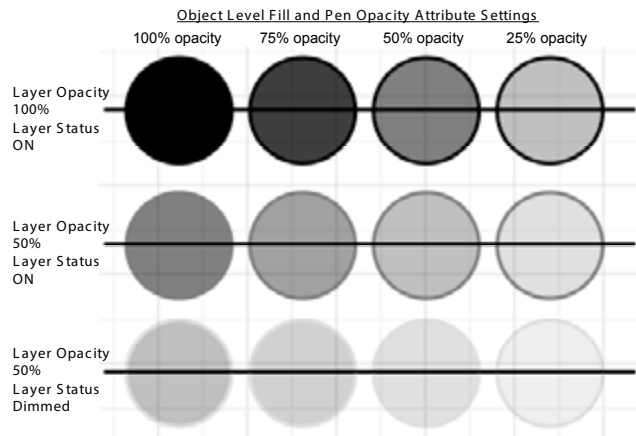


- ❑ Click OK.



NOTE: The effects of adjusting both an objects opacity attribute and its layers opacity are cumulative.

In the example shown below, compare the cumulative results when object level opacity settings are changed in conjunction with layer opacity and layer status. The circles are on the top-most layer and the horizontal line is on the bottom most layer with a layer opacity of 100% and pen and fill opacity attributes of 100%. The circle layer has objects with pen and fill opacity attributes as shown and the layer opacity and layer status are adjusted. Notice how the horizontal line becomes more or less visible as opacity attributes are combined.



Layer Buttons

New

There are eight buttons which allow you to create, modify, and delete layers.

New allows you to create a new layer. When a new layer is added, it will be selected so that a name can be immediately typed in without any further selection.

Delete

Delete allows you to remove an existing layer. If you select Delete, the selected layer will be removed and will be unrecoverable.

The last remaining layer cannot be deleted. However, if you need to remove all items from the last layer, choose Select All from the Edit menu and then choose Clear from the Edit menu.

Show

Show makes all layers in the drawing visible. The active layer is not changed.

Hide

Hide makes all layers but the active layer hidden.

Edit

Edit allows you to edit the attributes of a selected layer. You may set pen color, scale, locking attributes and transparency.

Purge

Purge deletes all empty layers, except the active layer.

Dup

Dup creates a new layer which copies the layer attributes of the currently selected layer. Objects on the selected layer are ignored. The new layer name has 'copy' appended to the end.

Sort

Sort changes the physical stacking order of all layers to display all layers in the drawing layer list in an A-Z type order. The sorting is not case sensitive. Numbers appear before letters and special characters such as '•' or ':-' or spaces appear before numbers. This operation cannot be undone; we recommend saving the unsorted layer list as a Sheet Setup prior to pressing Sort. The Sort operation cannot be undone; if the current layer order is important save a Sheet Setup before sorting.

Scale...

Scale allows you to assign a specific scale to a layer as described in Layer Scale earlier in this section.

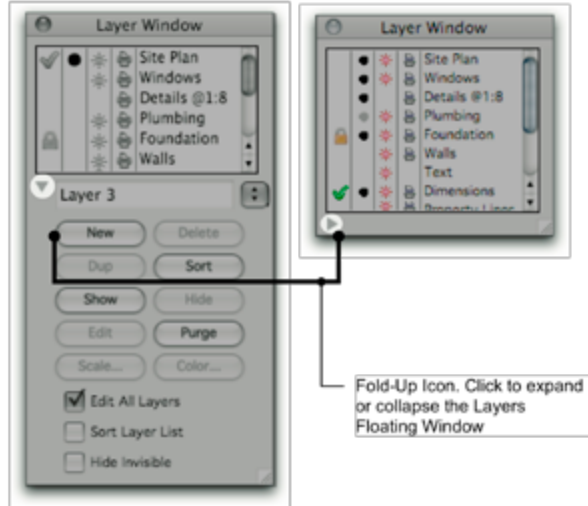
Color...

Color allows you to assign a specific color to a layer as described in Layer Pen Color earlier in this section.

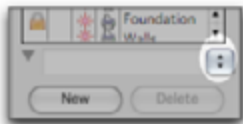
Fold-Up Icon



The fold-up icon allows you to fold or unfold the Layers Window. When the window is folded, the layer buttons and layer name field disappear. Click the icon to fold or unfold the window.



Sheets Submenu



The Sheets submenu may be used to perform the various layer management tasks of Saving, Deleting or Resetting Sheets. Picking a saved sheet from the list will adjust the layer stacking order and layer status based on their state when the Sheet was created.

For more information on managing sheets see the Sheets section later in this chapter.



Select the Edit All Layers checkbox to select or deselect the Edit All Layers option.

Edit All Layers allows all layers to be active at one time if checked. Edit All Layers boosts speed and productivity by giving you an efficient way to modify objects without switching layers.

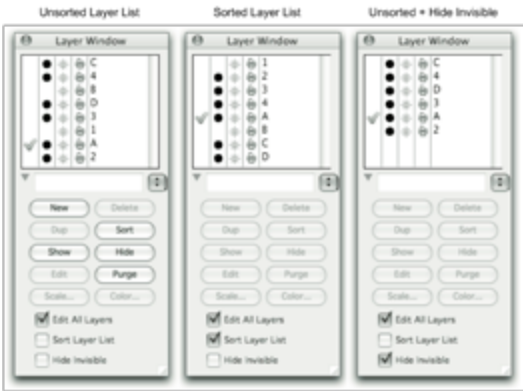
Move, Delete, Edit, Reshape, and Mirror are just a sample of the editing capabilities available to objects on multiple layers. It also provides an easy method of working on a drawing with which you are not familiar. With Edit All Layers, modifying an older drawing, where layer set-ups may have been forgotten, is less frustrating and more efficient. When Edit All Layers is On, objects on all visible layers can be selected. It can be used with all tools and menu commands.

The following rules apply:

- ☐ The Show Location Bar will ALWAYS reflect the scale of the active layer.
- ☐ All drawing occurs on the active layer.
- ☐ Modification of an object occurs on the object's layer.
- ☐ Objects created by using objects that are on the same layer appear on the objects' layer.
- ☐ Objects created by using objects on different layers appear on the active layer.
- ☐ Attach moves all of the objects to the active layer and attaches them.
- ☐ Select All and then Cut and Paste will place all objects on the active layer.

Sort Layer List

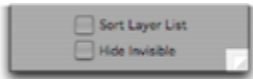
Sort Layer List changes the order in which layer names are displayed in the Layer List window. The physical stacking order of the layers are not changed (see Sort button). This list is displayed in an A-Z type order. The sorting is not case sensitive. Numbers appear before letters and special characters such as ‘•’ or ‘.’ or spaces appear before numbers. To undo this operation, uncheck the Sort Layer List box; the layer list is restored to the previous state.



Hide Invisible

Hide Invisible removes all layers with a hidden status from the Layer List. The layers are not deleted from the drawing. To restore the full list, uncheck the Hide Invisible box.

Size Box



The Layer Window can be stretched both horizontally and vertically with the use of the size box. Stretching horizontally allows larger layer names to be viewed without truncation. Stretching vertically allows a greater range of layer names to be viewed at a glance. Although stretching takes up more screen space, the “roll-up” box can easily be used to roll up the window, preserving working screen space.

Please see [Floating Windows](#) in the PowerCADD Tutorial for a complete description of features special to floating windows.



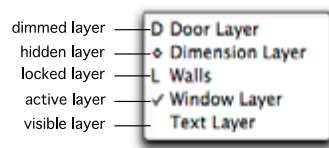
For best results, use the following Layering Tips:

- ☐ **USE LAYERS!** While you may have to force yourself in the beginning to remember to change layers while you work, you will benefit greatly from having and using layers. Eventually, using layers will become a natural extension of your design process.
- ☐ Hide layers which are not in use. Hidden layers do not redraw which shortens redraw time.
- ☐ Dim layers which do not need to be fully visible. Dimmed layers display as grayed outlines. This is useful when a layer needs to be visible but not in the way. The currently active layer can be dimmed. Additionally, layers which are dimmed will print gray.

Set Layer

Set Layer allows you to change the active layer without opening the layer window. You may also set the display attributes.

Choosing Set Layer will activate a secondary menu, listing all layers in the drawing.



To make a layer active, drag to the layer's name and release. If you select a hidden layer, it will become visible when selected.

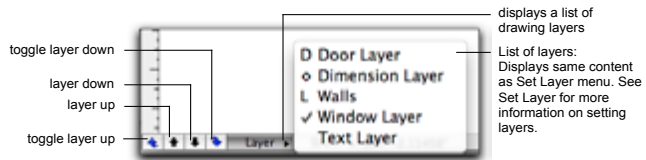
To hide a visible or dimmed layer, press the Command key while choosing the layer.

To dim a hidden or visible layer, press the Option key while choosing the layer.

In addition to picking layer names from the Set Layer menu, you have several other methods of layer navigation.

Layer Window: As discussed in the Layer menu item section of the manual, you can change the active drawing layer or change a layer status by clicking in the appropriate column in the layer window.

Layer Navigation Arrows: Clicking on the layer navigation arrows in the bottom left of the drawing window (or assigning key commands to those items) can be a very fast way to navigate through layers or change the active drawing layer. The Toggle Layer Up and Down arrows can be a very effective way to review drawings prepared by others or imported via DXF or DWG to determine their organizational structure or for presentation purposes such as slide shows.



Toggle Layer Down moves you down the layer stack and hides the previously viewed layer. Locked layers with a hidden status are skipped when toggling through layers.

Toggle Layer Up moves you up the layer stack and hides the previously viewed layer. Locked layers with a hidden status are skipped when toggling through layers.

Layer Down moves you down the layer stack and the previously viewed layer's status is not changed from visible or dimmed. Hidden layers are skipped.

Layer Up moves you up the layer stack and the previously viewed layer's status is not changed from visible or dimmed. Hidden layers are skipped.

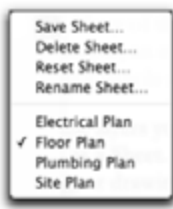
Layer Sheets

Sheets allows you to organize a drawing into different arrangements of layers and save each arrangement as a Sheet.

To understand the Sheets concept think about the contents of a set of working drawings for a house. A plan set may include a plot plan, floor plans, foundation, floor and roof framing, electrical, plumbing, and so on. Each “plan” is a sheet of paper which includes elements of the project needed for each stage of construction. Many of the sheets usually contain some of the same information. For example, an electrical plan will usually include the walls as well as the electrical symbols and legends.

Or think of it as a layer arrangement, like a flower arrangement, where the position of the roses, daffodils and tulips (the layers), stay in the same location but vary in color, line weight or opacity.

Switching Sheets



With Sheets, you simply make visible the layers you wish to have on a sheet, then name and save the settings as a Sheet. You can use the sheets to organize a project for printing, or drawing.

To switch sheets, choose Sheets from the Layout menu, then drag to a sheet's name in the submenu and release. The status and stacking order of the layers will change according to the selected sheet. The active sheet will be indicated with a checkmark to its left. If the layer settings do not conform to any sheet settings, no sheet will have a checkmark.

The active sheet will be indicated with a checkmark to its left. If the layer settings do not conform to any sheet settings, no sheet will have a checkmark.

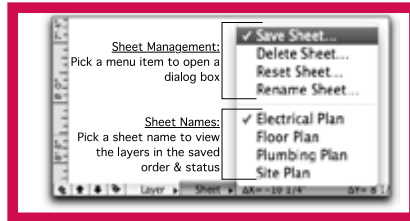
Managing Sheets

In addition to the Sheets menu, you have several other methods of accessing sheet management features.

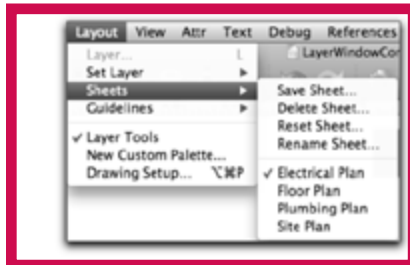
Layer Window: Select the Sheet pop-up to the right of the layer name field to access sheet management features as shown here.



Drawing Window: Select the Sheet pop-up in the bottom left corner of the drawing window to access the sheet management features as shown here.

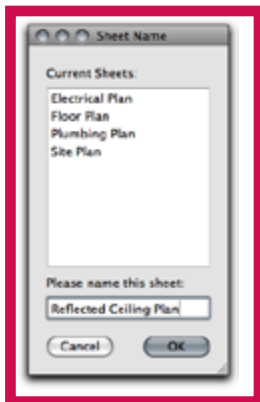


Layout Menu, Sheets submenu: Choose Sheet from the Layout menu to access sheet management features.



Each sheet management feature, Save, Delete, Rename, or Reset Sheets provides access to the same features whether accessed the Sheets menu item, the Layer Window or the drawing window Sheet pop-up. Review the sections on Save Sheet, Delete Sheet, Rename Sheet, and Reset sheet for more information on sheet management using those menu items.

Save Sheet...

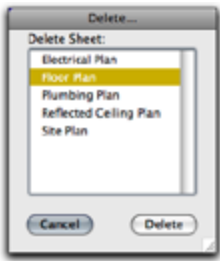


To save a sheet:

- ☐ Set each layer to the desired display status (visible, dimmed, or hidden), layer attribute settings, and to the correct top-to-bottom order for the new sheet in the Layer Window.
- ☐ Choose Sheets from the Layout menu.
- ☐ Choose Save Sheet from the Sheets pop-up menu and a dialog appears.
- ☐ Type a name for the sheet and click OK. The sheet name will be added to the Sheets pop-up menu. Once set, the sheet can be chosen from the Sheets pop-up menu.

The active drawing layer, layer stacking order, layer status, and layer attributes including scale, color, locked status and opacity are all saved with each sheet setup. For example, the 'Door Layer' could have a blue color and locked attributes for the 'Plumbing Plan' sheet and then have a green color and unlocked attributes for the 'Floor Plan' sheet.

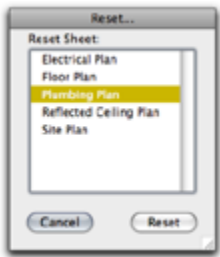
Delete Sheet...



To delete a sheet:

- ☐ Choose Delete Sheet from the Sheets pop-up menu. A dialog appears.
- ☐ Select the sheet to be deleted. Click Delete. To delete multiple sheets at the same time Shift-click to select sequential sheets, or Command-click to select multiple nonsequential sheets.
- ☐ A dialog will appear requesting you to confirm the Delete operation. Click OK to delete the selected sheets. Click Cancel to keep the sheets and cancel the operation.

Reset Sheet...



Existing sheets can also be modified. This is useful, especially on drawings which have several layers and you only need to make a couple of changes.

To modify an existing sheet:

- ☐ Select the sheet to be modified from the Sheets pop-up menu. Alternately, you may start from your present layer arrangement if it is closer to the desired sheet setting.
- ☐ Make the desired changes to the layers. You may hide, dim, or make visible the layers as needed, or rearrange their order.
- ☐ Choose Reset Sheet from the Sheets pop-up menu. A dialog will appear.
- ☐ Select the sheet to be modified.
- ☐ Click Reset. A message will appear, to make sure you have selected the correct sheet.
- ☐ Click OK. The sheet will adopt the new settings.

Layers and Resetting Sheets

When new layers are added to a drawing they are always appended to the top of the layer stack with a visible status. New layers are never appended automatically to existing sheets.

To append new layers to an existing sheet, simply drag the layers to their desired location in the layer stack and use Reset Sheet as described earlier.

Rename Sheet...



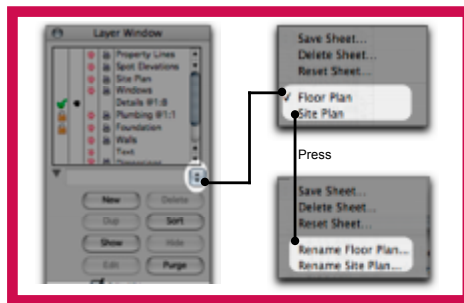
To rename an existing sheet:

- ☐ Select Rename Sheet from any of the three standard locations noted earlier.
- ☐ Select the sheet to be renamed.
- ☐ Type the new name in the Please-name-this field at the bottom of the dialog box.
- ☐ Select the sheet to be modified.
- ☐ Click OK. The name of the sheet will be updated in all locations.

Renaming Sheets Options

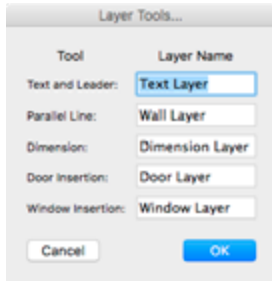
To rename an existing sheet:

- ☐ Click on the Sheet pop-up menu. A list of current sheet names is displayed.
- ☐ Press the Option key and Rename is added to the start of each existing sheet name.



- ☐ Select the sheet name to be renamed. The Rename dialog appears.
- ☐ Enter the new name and click OK.

Layer Tools



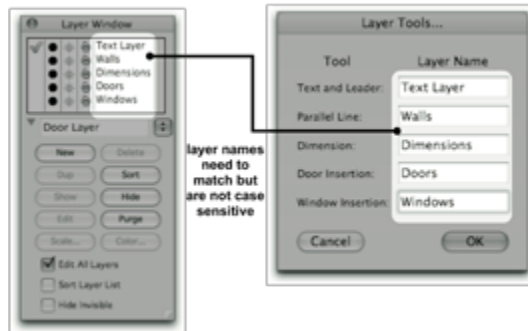
Layer Tools will allow you to activate specific predefined layers by choosing a tool. All objects created with the tool will be placed on the correct predefined layer.

When Layer Tools is active, specific tools will automatically place objects drawn by them onto predefined layers regardless of which layer is active when the tools are chosen. If the layer does not exist, or if Layer Tools is deactivated, the tool will place objects on the active layer.

To specify the layers to be used when Layer Tools is active:

- ☐ Press the Option key while choosing Layer Tools from the Layout menu. The dialog on the left will appear.
- ☐ Enter a Layer Name to be activated for each Tool listed and click OK.

When setting up drawing layers, the layer name in the Layer Window must match the name entered in the Layer Tools dialog. Layer names are not case sensitive (*MyLayer* is the same as *mylayer*) and spaces count (*MyLayerName* is not the same as *My Layer Name*).



Setting up Layers

To set up layers:

- ☐ Choose Layer from the Layout menu and the Layers Window will open.
- ☐ Click the New button in the Layers Window.
- ☐ Enter the appropriate layer name or names.

To use Layer Tools:

- ☐ Verify Layer Tools is activated in the Layout menu. A checkmark appears next to its name when active.
- ☐ Select a tool from the Tool palette that is associated with a layer. For example, if you have set up Text Layer, select the Text tool.
- ☐ The associated layer will automatically become the active layer.



NOTE: If a layer name field is blank in the Layer Tools setting dialog, then the currently active layer will be used by the selected tool. For example, if Dimension tool layer name field is blank then selecting a dimension tool will not change drawing layers and the dimensions will be inserted on the active drawing layer.



NOTE: In order to see the layer change, set a layer other than a tool-associated layer to be active.



NOTE: The Survey Dimension tool will not activate the Dimension Layer automatically. If you wish to put survey dimensions on the Dimension Layer, you will have to activate the Dimension Layer before drawing them.

Guidelines

Guidelines menu allows you to create a custom grid system and allows you to globally delete existing Guides.

The Guidelines command allows you to set orthographic (horizontal and vertical) or axonometric (angular) grids, which appear on the screen as Guides. Once your custom grid is set, Grid snap can be used to snap your objects to the custom grid. You may also drag single Guides from the rulers and place them to your custom grid.

The following general rules apply to guidelines:

- ☐ Snap commands may be used to position guidelines or to snap objects to a guideline.
- ☐ Guidelines do not print.
- ☐ Guidelines will display with rules turned on or off.
- ☐ To turn guidelines on or off use Drawing Setup, Show and uncheck Guidelines.
- ☐ Guidelines may be moved or deleted at any time without affecting the objects in the drawing.
- ☐ Guidelines may set to any angle, be any color and be displayed in front or behind objects.

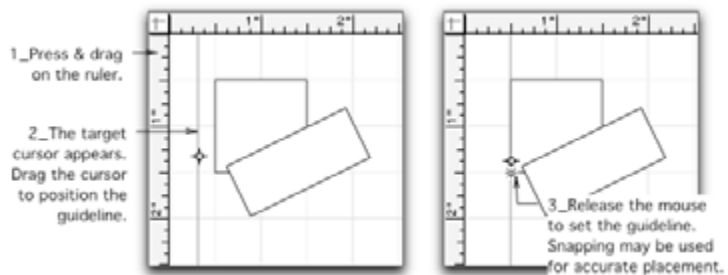
Adding a Guideline



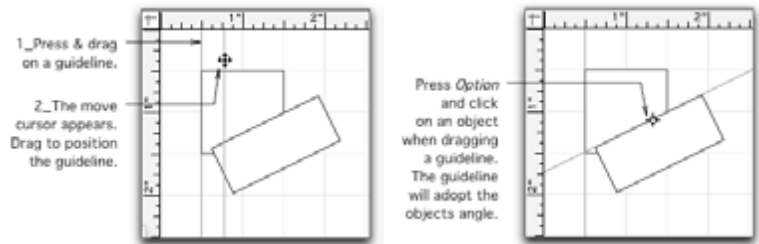
Guidelines may be added one at a time by dragging them off the ruler and positioning them anywhere in the drawing. Horizontal guidelines are added by dragging off the horizontal ruler. Vertical guidelines are added by dragging off the vertical ruler.

To add a guideline to your drawing:

- ☐ Position the mouse over the ruler. The cursor will change to a guideline cursor.
- ☐ Drag into the drawing window. The cursor changes to a target cursor.
- ☐ Drag the mouse to the desired location and release to insert the guideline.



Guidelines may be moved at any time by clicking and dragging on the guideline. To have a guideline match the angle of an object in the drawing, Option-click on the object when dragging the guideline.



Removing Individual Guidelines There are two methods to remove an individual guideline from a drawing.

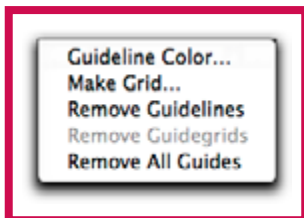
Remove Guideline
Hide Guidelines

By dragging:

- ☐ Drag the guideline from the drawing to the vertical or horizontal drawing ruler.
- ☐ The guideline is removed.

By the Contextual menu:

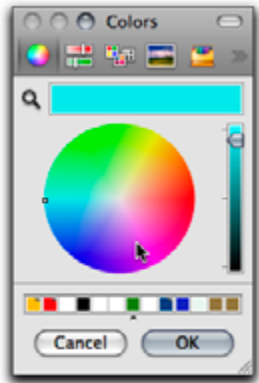
- ☐ Place the cursor on the guideline to be removed.
- ☐ Control- or right-click on the guideline.
- ☐ Choose Remove Guideline from the Contextual menu.



To remove all guidelines with a single action choose Remove Guidelines from the Guidelines submenu.

The Guidelines submenu provides a variety of advanced methods for managing guidelines and creating guideline grid patterns.

Guideline Color

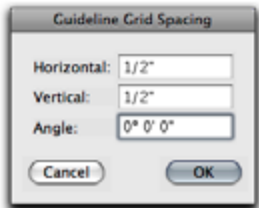


To change the color of a guideline:

- ☐ Choose Guideline Color... from the Guidelines submenu and the MacOS Color Panel appears.
- ☐ Choose a color and press OK.
- ☐ The selected color is now used for guidelines.

If the guideline colors do not immediately change to reflect your new color, hide, then show, the guidelines to force them to refresh. This setting also affects the display of Construction Line colors. See [Construction Line Color](#).

Making Grids



To create an orthographic grid:

- ☐ Choose Make Grid from the Guideline submenu and the Guideline Grid Spacing dialog will appear.
- ☐ Set the horizontal and vertical spacing as desired. Spacing is tied to the units and scale specified in the Drawing Setup, Units tab.
- ☐ Click OK and your custom grid will appear.

To eliminate confusion when snapping, you may wish to deactivate the Show Grid function under Drawing Setup. This will turn off the standard PowerCADD grid and leave only your custom grid.

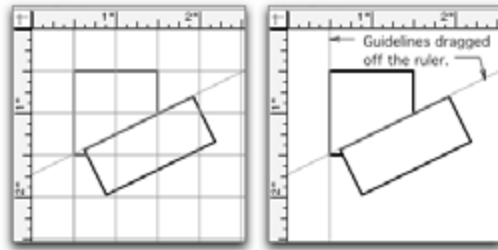
To create an axonometric or 3D grid:

- ☐ Set the horizontal and vertical spacing as desired.
- ☐ Set the desired angle. Entering an angle of 30 degrees, for example, will produce a standard isometric grid.

If a color has been set for the guides, your grid will assume the attributes you have chosen.

Removing Guides

The Guidelines submenu allows you to remove guides globally. You may remove only the Guidelines, which are pulled from the rulers, **only the Guidegrids, which is your custom grid, or you may remove all guides.** To remove guides, choose the desired menu command.



Results of using Make Grid. The grid lines are added to any existing guidelines in the drawing.

Results of using Remove GuideGrids. Notice manually placed guidelines are not removed.

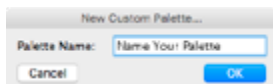
Undo can be used to replace removed guides. This will allow you to momentarily see your drawing without the gridlines, and replace them to continue drawing.

New Custom Palette

New Custom Palette provides a method for you to customize a tool palette, specific to your own requirements.

A new custom tool palette may contain any tool in the main PowerCADD Tool palette, tools from external tool palettes and external tools residing in the main Tool palette.

Create Custom Palette



Custom palettes are designed to look and perform like any PowerCADD tool palette. Custom palettes can be created for use as office standard tool sets or to create custom palettes for special design or drawing situations.

To create a custom palette:

- ☐ Choose New Custom Palette from the Layout menu and the New Custom Palette dialog box will appear.
- ☐ Enter the name for your palette and click OK.

Your new empty palette will appear in the upper left hand corner of the document window. You may now add tools to the custom palette.

Add Tools to Custom Palette

To add tools to your custom tool palette:

- ☐ Press the Command key and select the tool you wish to add.
- ☐ Drag the tool to the custom palette and drop it over the blank space.

Additional tools can be added to the same row of tools by dragging and dropping additional tools directly on the row. All tools added to a row are placed in the front of the row. Plan ahead for your most convenient set of tools.

Additional rows can be added by dragging the tool to the empty slot. New rows can be inserted between existing rows by dragging the tool to the row and pressing the Control key. When you press the Control key, you will see the rows drop down to their new location.

Delete Tools from Custom Palette

To delete tools from your custom tool palette:

- ☐ Press the Command key and select the tool you wish to delete.
- ☐ Drag the tool from the custom palette and release the mouse button.

The cursor will change to a trashcan as you move the tool from the palette. You can release the mouse when you see the trash can icon. The tool is now deleted from the custom palette. Remember, it only deletes the tool from the custom tool palette.

Attributes can be assigned to these tools just like other tools. Multiple copies of a tool can be added to the palette and each instance of the tool assigned a

different attribute. For example, multiple instances of the Line tool can be assigned different attributes; line weights, color, dash, and arrows.



NOTE: Dialog box settings for tools cannot be independently assigned. Dialog box settings are assigned to all occurrences of the tool.

Command keys can be assigned to tools in the custom palettes; however, they are assigned according to the row and column position of the tool. If you modify your tool palette after setting the Command keys, the Command keys will be incorrect and will need to be reset. After building or modifying your custom tool palette, it is a good idea to quit and restart PowerCADD before assigning Command keys.



NOTE: Assigning Command keys should be done after all modifications to the palette have been completed.



NOTE: Command keys are set to the position of the tool in the custom palette, not the tool itself.

When you quit PowerCADD, custom palettes are stored in the Custom Tool Palettes folder in your PowerCADD Preferences folder. The PowerCADD preferences folder is located in your user domain, Library folder, Preferences.

Custom palettes containing tools from External Tool palettes require the Externals to be loaded when you launch PowerCADD. If your custom palette contains a tool from an External Tool palette not loaded, the custom palette will be unavailable until PowerCADD is restarted with the tool available. An alert dialog will appear detailing the name of the custom tool palette affected by the missing Externals.

Custom palettes containing tools from External tools, which reside in the PowerCADD main Tool palette, also require the Externals to be loaded when you launch PowerCADD. These External tools include Survey Dimension, Blend, Arc Functions, and Trim/Extend tool (Wall tools).

When you include these Externals in your custom palette, your custom palette will open, but if the External tools are not loaded, they will be unavailable from the custom palette.

To reload an External tool into your custom palette:

- ☐ Choose Preferences from the PowerCADD menu and select the Loader tab.
- ☐ Select the External tool you need for your custom palette and click OK.
- ☐ Choose Quit from the PowerCADD menu.
- ☐ The next time PowerCADD is opened the External tool will appear in the custom palette.

Since generally you work with the same set of tools, reloading an External tool will not be used often, but as custom tool palettes can be created and shared with others, this method can keep all custom palettes the same.

Drawing Setup

Drawing Setup allows you to set specific drawing parameters particular to your drawing and working style. Drawing Setup parameters are saved with the particular drawing, they do not affect overall application behavior. To set overall application preferences, see [Preferences](#).

Within the Drawing Setup tabbed dialog box, you are able to set and control drawing specific items ranging from sheet size, units, scale, and object snapping to dimension preferences and the drawing log.

To use Drawing Setup:

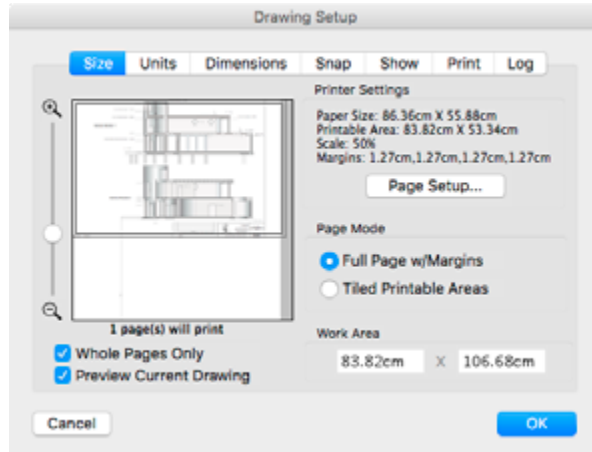
- ☐ Choose Drawing Setup from the Layout menu and a dialog box will appear.
- ☐ Each tab contains groups of options you can set. Click a tab to see the different options.
- ☐ Check a box or specify a particular value for the desired settings and click OK.
- ☐ The Drawing Setup parameters are set for the specific drawing.



NOTE: To save time, after you've set all your drawing setup parameter, consider adding your standard title block or layer structure to the drawing. Then, save the customized drawing as a Stationery Pad. Each time you open a stationery pad drawing, all your custom parameters are set. You can even have PowerCADD use your custom stationery as the default drawings opened when using New from the File menu. See the section on Preferences for more information on [how to have PowerCADD automatically open your own custom stationery](#).

Drawing Size

The Size parameters tab allows you to specify the orientation and size of your drawing area.



Drawing Size parameters reflect the page size you specified using the Page Setup dialog. While PowerCADD supports page sizes up to 200 x 200 inches, the actual printable area is controlled by your printer driver and limited to the physical size of paper the output device can accept. The Size tab displays the options available for specifying the drawing page area and provides several features which allow you accurately setup and preview your drawing for any output device.

PowerCADD always displays the actual printable area based on the page size you chose in Page Setup. For example, the printable area on an 11x17 sheet of paper is only 16.5 inches on some printers and a full 17 inches on others. As long as everything you draw is on the white page, it will fit when printed on the chosen printer.

Preview Pane

The Preview Pane provides you with feedback based on the printer and page size specified. Each page break line represents one sheet of paper or tile in the size specified during the Page Setup. Select one tile to select a single sheet, drag on multiple tiles to select multiple sheets of paper of the specified size. The preview pane also provides you with a thumbnail view of your drawing to help you determine where it will be on the printed page.

Zoom Slider

Dragging the Zoom Slider up or down allows you to zoom in and out to see more or fewer tiles. This is much like zooming in or out in your drawing to see more drawing area. To see the maximum page size of 200 x 200 inches, drag the slider down to the maximum zoom out position at the bottom of the zoom slider. If Preview Current Drawing is checked, adjusting the zoom level will also redraw the preview.

Whole Pages Only

When Whole Pages Only is checked (the default setting) clicking in the Preview Pane will select full page sizes. For example, if Page Mode is set to Tiled Printable Areas, clicking on a tile in the Preview Pane selects only whole page tiles. If Whole Pages Only is not checked you can click and drag to select a random page size.

Preview Current Drawings

Preview Current Drawing allows you to turn the drawing preview on or off in Preview Pane. When checked (the default setting) a preview of the current drawing is displayed in the Preview Pane. Adjusting the Zoom Slider up (zooming in) will display a larger preview.

Printer Settings

Printer Settings provides a summary of the drawing paper size based on the options selected during the Page Setup operation. To change any of these options, including page orientation, click Page Setup.

Page Mode

Page Mode options allow you to draw on Tiled Printable Areas of multiple sheets of paper, or on a single sheet of paper displayed as a Full Page with Margins. The Preview Pane provides confirmation of these radio button options.

Work Area

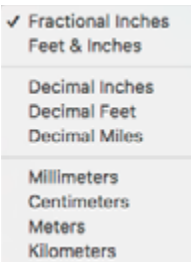
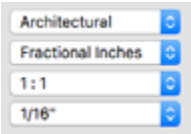
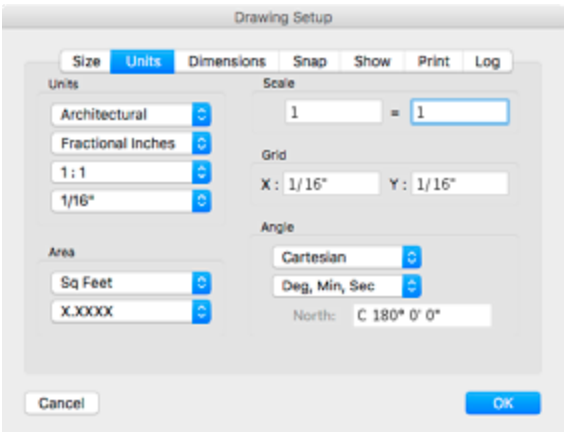
Work Area indicates the physical drawing area you have available to work on. This value will change depending on the Page Mode selection and on the information displayed in Printer Settings. To change the Work Area select a different page mode or click Page Setup to change page size options.



Please see [How to Get Started](#) in the PowerCADD Tutorial for a complete description of how Printer Settings, Page Mode settings and Work Area work in together.

Units Preferences

The Units parameters tab allows you to set the drawing scale and units used when creating objects in your drawing.



Units parameters are just like the triangular scales used in manual drafting. The settings you make here determine what scale your drawing will be when creating objects and when printed at 100%. The drawing rulers will update to reflect your scale and units settings.

Architectural contains selections for Engineering or Landscape. Choosing a particular pop-up automatically loads typical presets for the most commonly used scales in that design field. You may also set your user defined values for any item regardless of the preset you selected.

Units contains selections allowing you to choose the type of drawing units. Choosing a particular pop-up automatically loads typical presets for unit accuracy. For example, choosing feet & inches loads fractions for accuracy, choosing meters loads decimals for accuracy.

The units specified here are also automatically applied to the dimension tools and to the sheet size. For example selecting millimeters as units will display the drawing sheet sizes in centimeters.

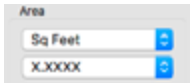
Although dimension units are set to automatically match drawing units, you may override the dimension tool settings. This would allow you to perform dual dimensioning or to draw in one set of units and dimension in another. See the Tools Chapter, dimensions, for additional information.

Predefined Scales contains selections of predefined scales based on your units and setting of Architectural, Engineering or Landscape. To set your drawing scale, choose an item from the list. The value is displayed the Scale fields. User defined scales are also permitted, see Scale for more information.

Unit Accuracy contains selections allowing you to choose the smallest value to be displayed. The unit accuracy is related to the type of units specified (e.g. settings units to decimal inches yields decimal accuracy choices to set the number of decimal points).

Area

Area parameters allow you to specify units of measure and accuracy for area calculations separate from your drawing units. The units and accuracy specified here are displayed in the Edit Window and used by the Area menu item to insert area and perimeter calculations into a drawing.



Units contains selections allowing you to choose the type of area units. You may specify area units which are different than drawing units. For example, if drawing units are set to feet & inches, area units may be set to acres or square meters.

Area Accuracy contains selections allowing you to choose the smallest area value to be displayed. Area accuracy is independent of drawing unit accuracy.


Scale

Scale parameters indicate the values chose in predefined scales or you may specify a user defined scale if an appropriate preset scale is not available. Scale may be entered in plain language matching traditional drawing scales (e.g. $1/4'' = 1'$) or may be expressed as a ratio (e.g. $1 = 48$).

Grid Spacing

Grid spacing allows you specify how fine or coarse the invisible drawing grid will be. For example, with a drawing scale of $1/4'' = 1'$, a grid spacing of 3" will cause an object to grow in 3" increments as it is drawn.

Angle Preferences



✓ Cartesian
Azimuth
Bearing
Gradian

Angle parameters allow you specify units of measure and accuracy for angular drawing. The units and accuracy specified here are displayed in the Edit Window.

Cartesian contains additional pop up selections allowing you specify the type of angular measurement.

Cartesian allows you to construct angles in a plane where 0° is East (right), 90° is North (up), 180° is West (left), 270° is South (down), and 360° is East (right). Positive angles are counterclockwise.

Azimuth allows you to construct angles in a plane where 0° is North (up), 90° is East (right), 180° is South (down), 270° is West (left), and 360° is North (up). Positive angles are clockwise.

Bearing allows you to construct angles with respect to direction. Up is North and down is South. The angle is specified by degrees North or South. You may reset the North orientation on the drawing when working in Bearing and Distance. To change the direction of North, type a new value. The value you type is based upon Cartesian coordinates. This will allow you to specify North any degree to the right or left of the default North location.

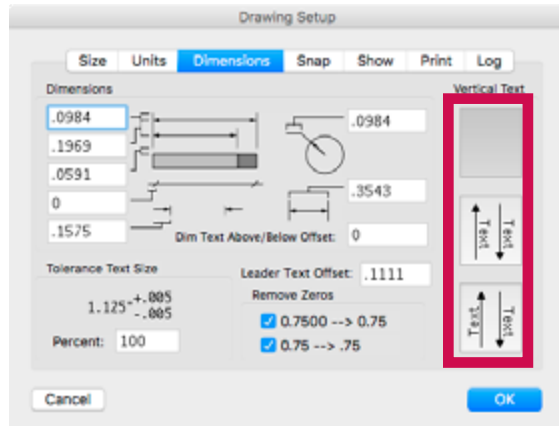
Gradian allows you to create angles in a plane where a circle is broken into 400 parts instead of 360° . The increments of the gradian circle begin and end at the extreme right hand side. Positive increments are counterclockwise.

Angular Unit Accuracy allows you to specify the smallest displayed measurement. These options can be set by selecting the desired items from the pop-up menu.

North allows you specify a particular north angle bearing. To set the north angle choose bearing then type the value in the north field.

Dimensions Preferences

The Dimensions parameters tab allows you to alter the way dimension lines are placed in the drawing. This option allows you to specify the length of leader lines, extension line offset, stack dimension line offset, text rotation, text location, and so on. All values entered are at normal print size (i.e. 100% or 1:1).



You may force dimension line extensions to be short by entering a negative extension line offset. When set, dimension tools which draw extension lines will create short extension line segments, regardless of the distance between the object being dimensioned and the dimension line itself.

Dim Text Above/Below Offset: 0.000

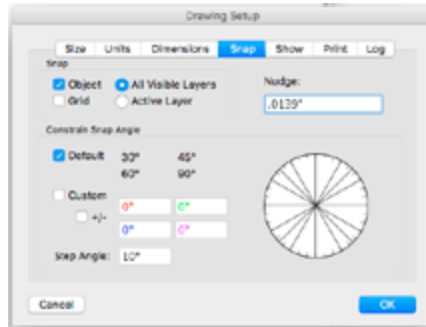
Dim Text Above/Below Offset allows you to specify the amount of space between the bottom of the dimension text and the dimension line for dimension text set to Above/Below only. A positive value moves the dimension text farther (higher) from the line. A negative value moves it closer (lower) to the line.

Remove Zeros
☒ 0.7500 --> 0.75
☒ 0.75 --> .75

Remove Zeros allows you to specify how leading and trailing zeros are displayed when dimensioning in decimal units.

Snap Preferences

The Snap parameters tab allows you to specify various dynamic snap settings in your drawing.



Snap governs the following snapping preferences.

Object allows you to find basic snap points on objects more easily. The basic snap points are: circle and ellipse centers, arc radius points, line midpoints and endpoints, and all polygon and freehand vertices. When you drag an existing object, or start to draw a new object near a snap point, the object will act much like a magnet; it will pull away from the cursor and attach itself to the point.

Grid allows you to turn the normal grid snap On or Off. With Grid Snap On, the object being created or moved will snap to the grid setting rather than the screen location. With grid snap On, pressing the Command key will temporarily disable grid snap in the drawing.

All Visible Layers allows you to utilize the Snap functions on all layers that are visible rather than just the active layer.

Active Layer Only allows you to restrict the snapping function to just the currently active layer.



CAUTION: Snapping can also be turned off for a specific layer using the Layer Window, Snap Column. Turning off snapping at the layer level overrides the choice to snap to All Visible Layers or Active Layer.

Nudge Settings



Nudge Settings allow you to specify the distance selected objects will be moved right, left, up or down when the arrow keys are pressed. A total of ten nudge amounts are remembered. Nine may be custom values, one value (1 pixel or 0.01389") is always included in the nudge amount list. When more than eight custom nudge amounts are used, the oldest amount (the one entered first) is removed from the bottom of the list and the newest (most recently entered) amount is added to the top of the list.

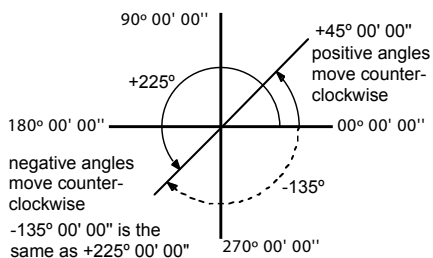
TIP: Nudge settings automatically adjust for layer or drawing scale settings.

The accuracy of the nudge setting is independent of the drawing setup accuracy. For example, if the drawing setup accuracy is set to two decimal points (X.XX) or to fractional units, you may enter a nudge value to any number of desired decimal points (X.XXXX). The units of a nudge setting are also independent of the drawing setup units. For example, if the drawing setup units are fractional inches, you can enter nudge values in centimeters and to any desired accuracy. Changing the drawing setup units or accuracy will not modify the units or accuracy of previously entered nudge values.

Constrained Angles

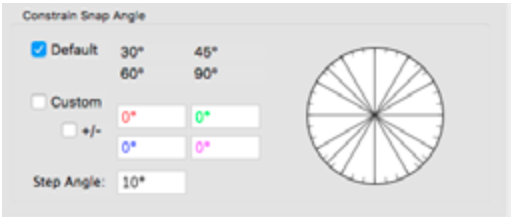
Constrained Angles allows you to specify the angles to which objects will snap while drawing.

For example, when you draw a line with the Multi-Angle Line tool, you will notice the line seems to jump to certain angles while you drag the line to length. These jumps, which are normally the standard drafting angles of 30, 45, and 60 degrees, can be set. PowerCADD measures positive angles in a counterclockwise direction and negative angles in a clockwise direction as illustrated here.



Default Snap Angles

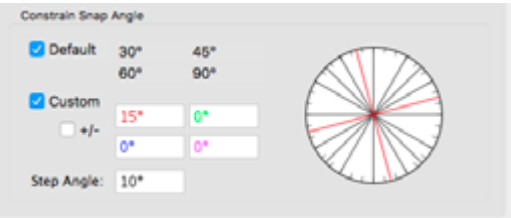
Default: Checking this box allows you to snap to the preset snap angles. Each angle is available in all four quadrants of a coordinate system. For example, a setting of 30 degrees snaps to 30, 120, 210 and 300 degrees measured counter clockwise from horizontal (zero degrees). Default snap angles may be used by themselves or in conjunction with Custom and Step snap angles.



When checked, the snap angle preview panel shows a solid black line for each quadrant a default snap angle appears in as shown here:

Custom Snap Angles

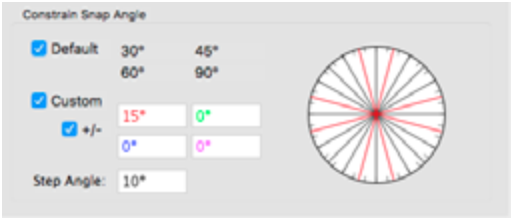
Custom: Checking this box allows you to snap to custom snap angles specified in the Custom Snap Angle Fields. Custom snap angles may be used by themselves or in conjunction with the Default and Step snap angles. The angle specified is added to the root angles of 0, 90, 180 and 270 degrees. For example, a custom angle of 15 degrees would snap to 15, 105, 195, and 285 degrees.



If +/- is not checked, the custom angle is added to the base angles of 0, 90, 180 and 270 degrees.



+/-: Checking this box allows the custom snap angles to be both added *and* subtracted from the root snap angles of 0, 90, 180, and 270 degrees. For example, when checked, a custom snap angle of 15 degrees would snap to 15, 75, 105, 165, 195, 255, and 285 degrees as illustrated below.

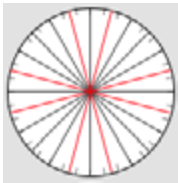


Custom Snap Angle Fields: You may specify a maximum of four custom snap angles. Each field is a unique color with a matching color displayed in the preview panel. Each angle field calculates four complementary snap angles. When used in conjunction with the +/- checkbox, each field calculates a total of eight snap angles.

Custom snap angles can be useful in adding additional snap angle increments to the Default angles. They can also be very helpful any time you need to draw relative to an arbitrary baseline angle. For example, many site planning projects require a site plan to be drawn relative to a property line which is typically not at zero (horizontal) or 90 degrees (vertical). By using custom snap angles you maintain the ability to quickly and easily draw geometry which is parallel and perpendicular to the arbitrary baseline angle.



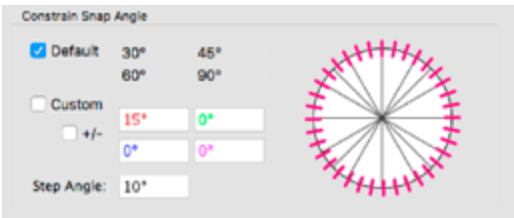
TIP: When entering custom snap angles it is not necessary to enter the degree, minute, and second marks. For example a custom angle of 22 degrees, 32 minutes, 48 seconds, could be entered as 22 32 48 (simply insert a space between each value).



Preview Panel

Preview Panel: The preview panel provides instant feedback for all Default, Custom and Step snap angle settings. Default angles are black lines, each custom angle is rendered in a separate color, and the step angle appears as a tick mark around the perimeter of the circle.

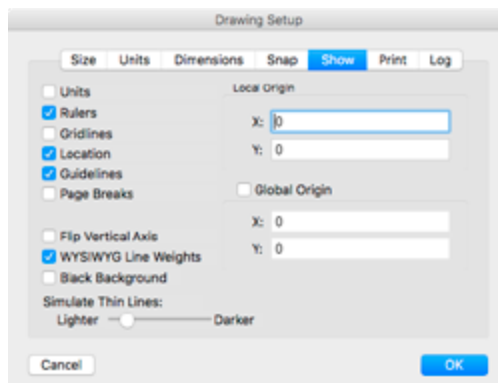
Step Angle: Allows you to set an angle which works incrementally around a 360 degree circle (an 'angle grid' if you like). Step snap angles may be used in combination with Default and Custom snap angles.



Step Angle

Show Preferences

The Show preferences tab allows you to customize the look and feel of your drawing environment.



Units, if checked, the drawing units are displayed in the Edit Window. When this is not checked, drawing units are not displayed for any decimal or metric drawing units. If show units was not checked, setting the drawing units for fractional inches or feet & inches will automatically check show units.

Rulers, if checked, shows the rulers in the drawing window. Rulers must be turned on to drag guidelines off the ruler. When rulers are not checked, no rulers are displayed in the drawing window but any guidelines will still be displayed. This item may also be turned On or Off using the contextual menu, which appears when you Control- or right-click in the drawing window.

Gridlines, if checked, shows a dimmed reference grid in the drawing window. The grid may be displayed in front or behind objects using the settings in the Preferences, View tab.

Location displays the exact location of the cursor on the drawing page with respect to the Drawing Origin and allows you to measure both linearly and angularly from one point to another.

The Show Location Bar at the bottom of the document window shows the location of the mouse in three methods: absolute, relative and angle-and-length.

When the absolute method is used, the mouse location is given as "X=... Y=..." where the values for X and Y are from the origin of the drawing.

When the relative method is used, the mouse location is given as " $\Delta X=...$ $\Delta Y=...$ " where the location is relative to the last mouse click, thus you can

easily measure coordinate change from one point to another by simply clicking on a point and moving the mouse to the next point.

To measure distance angularly and linearly, place the cursor at a starting point and click. this will set the linear display to zero and the angular display to 90°. Move the cursor to the second point and read the display. To toggle between the three methods, simply click in the Show Location Bar.

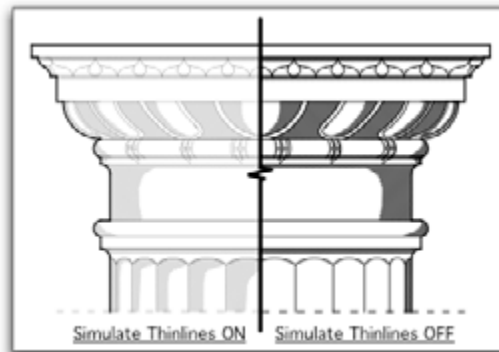
Guidelines, if checked, can be created by dragging from the horizontal and vertical ruler. Unchecking Show Guidelines will hide all drawing guidelines. Guidelines are not deleted and can be shown again by checking show guidelines.

Page Breaks will place grayed lines on the screen to indicate where page breaks would occur if a large drawing were printed on standard sheets of paper. These lines will not print or plot. They are a visual aid and are visible only on the screen. You can choose to display objects in front or behind page breaks by setting the appropriate settings for [Grid Behind Objects](#).

Flip Vertical Axis changes the direction of the positive-Y coordinates from positive up to positive down.

WYSIWYG Line Weights, if checked, displays objects on screen with the line weight attributes you specified. When zoomed in more than 6,400 percent, WYSIWYG line weights is temporarily disabled. Zooming out will automatically enable WYSIWYG line weights again.

Simulate Thin Lines improves the screen rendering quality to better simulate thin line weights in your drawings. Moving the slider bar left renders the thin lines light on screen. Moving it right renders them darker. Moving the slider bar to the right-most position turns Simulate Thin Lines Off. The Lighter/Darker slider does not affect printing.



Black Background, if checked, changes the drawing color background to black. The black background doesn't print.



CAUTION: If black background is On, and you have chosen Black and White Printing in the Drawing Setup, Print tab, the white lines will be printed white and your printed sheet will look empty.

Local Origin

Local Origin allows you to specify X, Y coordinates as the drawing origin. The coordinate units are the same as the drawing units and may be entered as positive or negative values. You may also specify a local drawing origin dynamically by dragging the origin from the top left corner of the rulers in the drawing window.

This is the normal way of drawing for almost all situations.

Global Origin

Global Origin allows you to establish a common set of coordinates between different drawings.

For example, if you have a proposal for a campus, which consists of a drawing describing the campus and individual drawings for each building on the campus, Origins allows you to set an origin common to each drawing.

Since each drawing in a project may show a different section of the whole project from the origin, you can set the offset for each drawing.

To set the offset of the origin check the Global Origin box and enter the desired X and Y coordinates. You may enter positive or negative coordinate values.

Once the offset has been established, you may find the origin of any point in your drawing. Essentially, the origin offset is the origin of the place you selected. It takes the drawing's coordinates and adds the offset amount. On your campus project, for example (which has north as up), if the origin of the building you are drawing is 500 feet south and 1500 feet west of the main drawing's origin, you would set the origin offset as $X=1500'$ and $Y=500'$.

When Show Coordinates is checked, the information in the Edit Window will reflect the global values. The Show Location Bar will reflect the global values if the absolute method of location is used. Otherwise, the local values will be used in the Edit Window and the Show Location Bar.



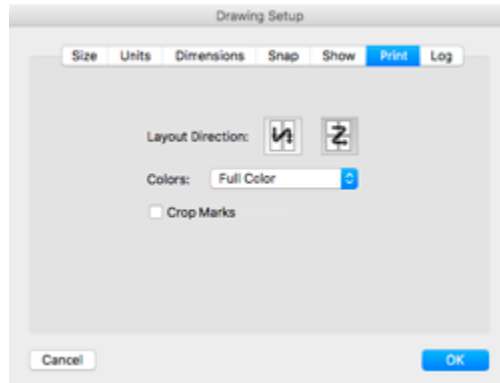
NOTE: The origin offset is a physical point on your drawing and is independent of scale.



REALITY CHECK: While there is no question that a global origin is vitally essential when working over large areas and also that it is an important addition to PowerCADD, it is also extremely difficult to calculate. The industry standard way of doing this is to identify a point in the drawing as location XXXXX,YYYYY and have the original calculated back from that. There are tools in Paolo Rossi's Topography Tools and in TopoTools in WildTools that make this very easy.

Print Preferences

The Print parameters tab allows you to specify some of the printing options for your drawing.



Layout Direction will allow you to specify the print order when printing a large drawing to a small printer. You may arrange the pages to print from left to right or from top to bottom. Note the numbers in the page sequence icons. To set the page sequence, simply click the appropriate icon.

Crop Marks places crop marks on the four corners which define the document size and adds a 1/4" margin around your document. For example, if your original document is 8.5"x11", your printed document will be 9"x11.5". If your available paper size is only 8.5"x11", four pages will be required for printing.

Full Color allows you to print in full color.



Grayscale allows you to print in grayscale only.

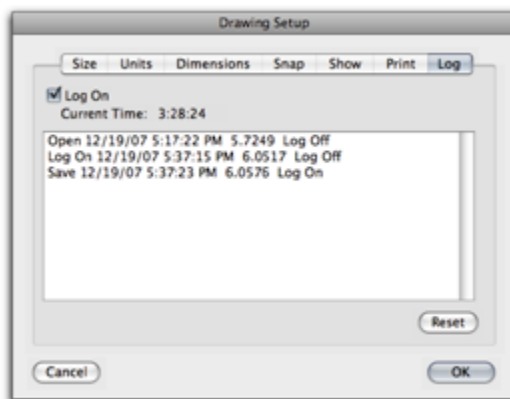
Black and White allows you to print in black and white only.



See the section on the File Menu, [Print](#) and [Page Setup](#) menu items for information regarding additional printing options. See [How to Get Started](#) in the PowerCADD Tutorial for additional information on working with page sizes and margins.

Log Preferences

The Log will keep a cumulative tally of the time the drawing is open.



The Log is a useful option and can be used by architects, drafters, and other professionals who charge by the hour for their work. When turned on, the log runs continuously even when the drawing is in the background so care should be taken if you are using the log for time-billing purposes.

The log file is written to disk in the same location as the drawing file which logs all drawing open and save operations. This file can be opened into an Excel spreadsheet by double-clicking on the Log file. The log is a tab-delimited text file and can also be read into most word processing, spreadsheet or database programs. The log file will be updated each time the drawing is opened or saved.

To use the log:

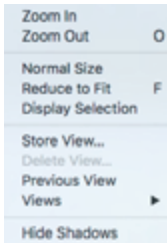
- ☐ The Log On checkbox allows you to turn the Log On or Off.
- ☐ Current Time tells you exactly how long the drawing has been open since it was originally created.
- ☐ The Reset button allows you to set the log to zero.

The following is an example of the log file after opening in Microsoft Excel.

The screenshot shows the 'DrawingSetup1.LOG' file opened in Microsoft Excel. The data is as follows:

	A	B	C	D	E	F	G
1	Open	2/20/03	01:40:40 PM		0.0124		Log off
2	Save	2/20/03	01:50:20 PM		0.1721		Log on
3	Save	2/20/03	01:54:58 PM		0.2493		Log on
4	Save	2/20/03	01:59:28 PM		0.3244		Log on
5	Save	2/20/03	03:43:45 PM		2.0624		Log on
6	Save	2/20/03	04:28:31 PM		2.8083		Log on
7	Save	2/20/03	04:34:42 PM		2.9114		Log on
8	Save	2/20/03	05:04:54 PM		3.4147		Log on
9	Save	2/20/03	05:14:12 PM		3.5699		Log on
10	Save	2/20/03	07:32:23 PM		4.5697		Log on
11	Save	2/20/03	07:32:50 PM		4.5773		Log on
12							

View Menu



The View menu allows you to choose zooms and views. Frequently used views and zooms may also be stored and recalled from the Views menu.

In addition to the zoom and view features offered in the View menu, deep zooming while drawing and zooming and navigation using the Overview Window are also available.

For more information on deep zooming see Tools chapter [Deep Zoom & Zoom on the Fly While Drawing](#).

For information on using the [Overview Window](#) see the Windows section, Overview later in this chapter.

Zoom In and Zoom Out allow you to magnify part of the drawing for greater detail or show a greater area of the drawing on the Macintosh screen.

Zoom In

To use the Zoom In command:

- ☐ Choose Zoom In from the View menu.
- ☐ The drawing area will be zoomed in by a factor of two.

With each execution of the Zoom In command, you magnify the drawing by two.

Select the Arrow Selection tool to cancel Zoom In.

The Zoom tool located in the Tool palette performs the same operation as Zoom In except you specify the area that will be shown on the screen. This tool may save time by eliminating screen redraws when zooming in on small objects from a full drawing view.

Zoom Out

To use the Zoom Out command:

- ☐ Choose Zoom Out from the View Menu.
- ☐ Upon selection of Zoom Out, the drawing area will be zoomed out by a factor of two. The center of the current view area will be the center of the zoom-out operation.



NOTE: Zoom In and Zoom Out also work inside the Edit Text Window. See the Tool chapter, [Text tool](#), [Edit Text Window](#) for more information.

Normal Size

Normal Size returns your drawing zoom to normal size (100%).

When zoomed in, Normal Size displays the drawing in the center of the screen at the scale you have chosen to be the drawing scale. If you have already chosen the Reduce To Fit command, Normal Size will take you to the center of the viewing area.

Reduce To Fit

Reduce To Fit reduces the entire drawing page size to fit in the Macintosh window.

This brings the drawing from any zoom status to fit within the current drawing window.

Display Selection

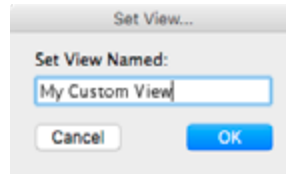
Display Selection takes you to a zoom that displays the currently selected object regardless of your location or zoom on the drawing page.

Display selection brings the selected material very close to the center of the screen. The selected material can be larger than the screen at the current zoom. The zoom is adjusted so the selected material takes up a reasonable portion of the screen.

Store View

To use Store View:

- ☐ Position the drawing window location and zoom level to the view you wish to store. This view might be an actual drawing view such as “right side”, a detail such as “framing detail”, or simply a location that will need much detail drawing.
- ☐ Choose Store View from the View menu. The Set View dialog will appear.



- ☐ You may use your cursor and keyboard to change the name. For example, you might want to call “View 1” the “Front View” to be more descriptive.
- ☐ Press the Return key to enter a name or click Cancel to go back to the drawing.

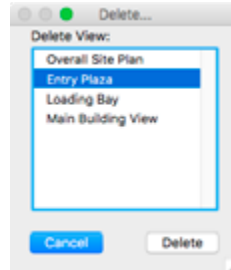
Also, before any views are stored, selecting a view will put you at home view (the upper left corner of the page and at normal size).

Delete View

Delete View allows you to delete views that are no longer needed.

To delete a previously stored view:

- ❑ Choose Delete View from the View menu. The Delete View dialog box will open.



- ❑ Select which view you wish to delete. Click Delete to remove the view or Cancel to return to the drawing. To delete multiple views at the same time, Shift-click to select sequential views or Command-click to select multiple nonsequential views.

Views

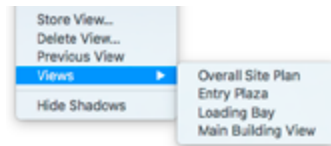
Views allows you to recall a saved view.

A saved view is like a bookmark. Just as a bookmark lets you quickly return to a place in a book, a view lets you return to a place in a drawing. Views are created with the Store View command.

When you choose Views, a submenu appears listing the views that have been set for the drawing. Views are listed in the order they were saved, the first saved view is at the top of the list, the last saved view at the bottom. The submenu lets you recall a particular view.

To use Views:

- ❑ Choose Views in the View menu. The Views submenu appears.



- ❑ Choose a view in the submenu and release the mouse button. The selected view will appear in the drawing window.

Previous View is always available, even if no views have been set for the drawing. Previous View returns you to your last location in the drawing. The previous zoom will also be recalled.

Hide/Show Shadows

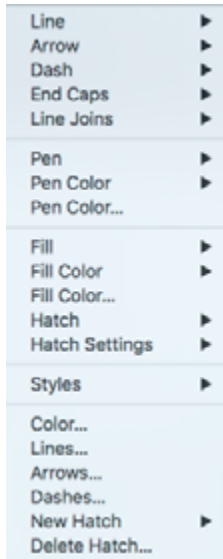
Hide Shadows: Choose this menu item to temporarily turn off all shadow attributes in a drawing.

Choosing Show Shadows will turn all shadow attributes back on.



CAUTION: Hidden Shadows do not print.

Attribute Menu



The Attribute menu allows choosing the attributes for line, fill, fill color, pen and pen color.

Patterns may be modified or added.

A variety of pen widths may be specified including several weights of hairlines.

Colors may be edited or added.

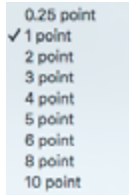
Arrow types and sizes may be customized and dashes may be set-up to suit your individual drawing style.

Items in the Attribute menu are also available using contextual menus. Control- or right-clicking on an object in the drawing will bring up the Attribute menu items.

The Defaults Window or the Attributes Window may also be used to access items in the Attribute menu.

CAUTION: If Global Attributes has been turned On, the choices displayed for Pen (line weights), Color (fill and pen), Hatch patterns, Fill Patterns, Arrows, Dashes, and Attribute Styles are affected. See [Global Attributes](#) for additional information.

Line Submenu



Line submenu allows you to change the line weights of objects.

To set the default of the Line menu:

- ☐ Select the Line tool by double-clicking on the Tool palette. Clicking twice insures no objects in the drawing are selected.
- ☐ Choose Line from the Attribute menu. The Line menu will appear.
- ☐ Choose the desired line weight.
- ☐ Place the cursor on the drawing area and create a line. The line will have the weight you just specified.

These changes will remain in effect in the menu until the tool is changed.

If you want to set the default of objects with a specialized line weight, follow the same steps shown above, except select the object's tool you want to set.

To change existing lines:

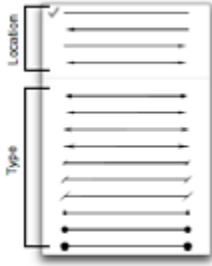
- ☐ Select the line or object you wish to change.
- ☐ Choose the line weight from the Line menu you want.

When changing the appearance of an object, such as changing line weights in existing objects, the Tool palette setting is not changed. You may change several objects at a time by selecting all of them and then making the change.

You may change the line thicknesses listed in your Line submenu by choosing Lines[...] under the Attribute menu or press the Option key when choosing a line weight.

Changing the available line thicknesses does not change any existing objects in the drawing.

Arrow Menu



Arrow allows you to add arrows, nodes, and backslashes on either or both ends of a line.

To add arrows to existing lines:

- ☐ Select the line to which you want to add arrows by double-clicking on the Line tool.
- ☐ Choose the location *and* type of arrow you want from the Arrow menu.
- ☐ When you release the mouse button, the arrows will be placed on the line as part of the object.

When changing the appearance of an object, such as adding arrows to existing objects, the tool defaults are not changed. You may change several objects at a time by selecting them and then making the change.

To set a tool default to Arrows:

- ☐ Select the Line tool from the Tool palette.
- ☐ Choose Arrow from the Attribute menu. A menu will appear.
- ☐ Select the location of the arrow by choosing one of the top four lines. These lines specify if the arrow will appear at the start of the line, the end of the line, or on both ends of the line.
- ☐ Select the type and size of arrow you wish to use.
- ☐ When you release the mouse button, the arrow will be placed on any line you draw until you change it.

When changing Arrow, remember the arrow changes are additions to the Line menu changes made earlier. If the line is set to a 10-point dashed line, and you set the Arrow for any reason, the outcome will be a 10-point dashed line with the original size arrows. The end-markers are a fixed size, but you may design additional end-markers to fit the 10-point lines.

You may change the arrows available in the Arrow submenu by choosing Arrows[...] under the Attribute menu or press the Option key when choosing a line weight.

Dash Menu



Dash allows you to change the line forms (dashes) of lines or objects.

To set a tool default to dashes:

- ☐ Select the Line tool from the Tool palette by double-clicking on the Line tool.
- ☐ Choose Dashes from the Attribute menu. A dashes menu will appear.
- ☐ Choose the desired line type.
- ☐ Place the cursor on the drawing area and draw a line. The line will have the dash type you just specified.

These changes will remain in effect in the menu until you change them again.

If you want to set an object's default line to dashes, follow the same steps shown above, except select the object you want to set instead of the Line tool.

To change existing lines:

- ☐ Select the line or object you wish to change.
- ☐ Choose a line type from the Dashes menu.
- ☐ When you release the mouse button, the change will occur.

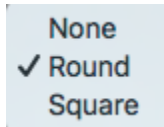
When changing the appearance of an object, such as changing line types in existing objects, the change is made to the object. Defaults are not changed and must be reselected for each item changed. You may change several objects at a time by selecting all of them and then implementing the change.

You may change the line types available in your Dash submenu by choosing Dashes[...] under the Attribute menu or press the Option key when choosing a dash pattern.

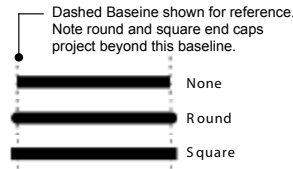


TIP: The Line, Arrow and Dash menu items will automatically scroll if the list is too long to be displayed on screen.

End Caps Menu



End Caps allows you to specify what style of end cap is applied to open end objects such as lines, arcs, polygons and Bézier curves. You may select from None, Round or Square whose differences are noted in this diagram.



To set the default end cap for a tool:

- ☐ Double-click on a tool in the Tool palette (e.g. the Line Tool). Clicking twice insures no objects in the drawing are selected.
- ☐ Choose End Cap from the Attribute menu. The End Cap menu will appear.
- ☐ Choose the desired End Cap.
- ☐ Place the cursor on the drawing area and create a line. The line will have the end cap you just specified.

To change end caps of existing lines:

- ☐ Select the line or object you wish to change.
- ☐ Choose an end cap type from the End Cap menu.
- ☐ When you release the mouse button, the change will occur.

Fill Menu



The Fill menu is used to choose a fill pattern for the various objects that define an area as described earlier.

The standard default fill is white. When white is assigned to an object, it acts as a blocking object. When plotted, the white object blocks out objects below it in the drawing. Its outline is plotted, but the white fill will remain the color of the paper with every object behind it eclipsed. The fill pattern, “None”, is used for drawing an object outline, and its center will be treated as transparent. None is different from white in that items behind the object are visible through a None fill where they are not through a white fill.

When you choose a fill pattern, first select the object to be filled. From the Fill menu, choose the desired fill.

To assign a fill to a drawing tool for repeated use:

- ☐ Select a drawing tool from the Tool palette.
- ☐ Choose Fill from the Attribute menu and a submenu will appear.
- ☐ Choose the desired fill and release the mouse.

Each time you create an object with that tool, it will be filled with the fill pattern you specified until you change it.

We recommend that you do not use objects filled with a white pattern to hide unwanted sections in the drawing unnecessarily. Tools such as Break, Clip, and Cut with Line are much better alternatives for eliminating drawing sections.

Fill patterns are excellent means to provide a variety of textures in your drawings while maintaining a very small file size when compared to using hatch patterns.

You may change the fill patterns available in your Fill submenu by choosing Pattern[...] under the Attribute menu or by pressing the Option key when choosing a fill pattern. The first nine fill pattern positions in the first column, starting from None and going down, are reserved and cannot be edited. Any changes made to the Fill Pattern palette will also affect the Pen Pattern palette.



TIP: Editing an existing PowerCADD fill pattern will not affect the fill patterns of objects in the drawings to which the previous pattern had been applied. To update the old objects, use Select Special to select the objects with the old fill pattern then choose the edited (new) fill pattern from the Pen Pattern menu.



TIP: When Global Attributes is Off, editing a fill pattern position affects the PowerCADD pen and fill pattern palettes for the specific drawing. If

Global Attributes is On, editing a fill pattern position affects all drawing pen and fill palettes.



TIP: Fill patterns are saved with the object. Copying and Pasting an object with a custom pattern from drawing 1 into drawing 2 will retain the objects pattern and the custom pattern is added to the PowerCADD Pen and Fill pattern palettes.



NOTE: Fill patterns are unique to the MacOS. As such, if you save your drawing as a DXF or DWG file, the fill patterns are not included, objects will have a fill of None when translated. If it is important to have your patterns included with a DXF or DWG file, consider using hatch patterns instead of fill patterns.

Hatch Menu



A hatch is a vector-based pattern rather than a bit image fill. Originally they were fills used by plotters.

Hatch patterns are assigned to objects in the same manner as Fill patterns.

To hatch an object:

- ☐ Select the object you wish to assign a hatch pattern. You may select the individual lines or arcs which make up a complex shape, or you may select polygons.
- ☐ Choose Hatch from the Attribute menu (or Control- or right-click on an object) and select the desired pattern.
- ☐ Upon selection of the desired pattern, release the mouse. The hatch pattern will be assigned to the selected object.

Hatch patterns are associative and behave much like fill patterns. If you resize or reshape a hatched object, the hatch pattern automatically recalculates to reflect the new shape.

Hatch patterns are different from fill patterns in that they are composed of true object geometry such as lines. As such, hatch patterns translate correctly via DWG or DXF while fill patterns do not. However, hatch patterns are similar to fill patterns in that they are associative. If you resize or reshape a hatched object, the extents of the hatch are also be recalculated similar to how a fill patterns behave.



NOTE: Hatch patterns may be made from any object in the drawing including pixel based (raster) images which behave like Fill patterns. **Fill patterns are unique to the Apple Macintosh computing platform. As such, if you save your drawing as a DXF or DWG file, the fill patterns are not included, objects will have a fill of None when translated. If it is important to have the patterns included with a DXF or DWG file, consider using object (vector) based hatch patterns instead of raster based hatch patterns.**



TIP: When Global Attributes is Off, copying an object with a custom hatch pattern from drawing one and pasting it into drawing two will add the custom hatch to the hatch palette in drawing 2. If Global Attributes is On, a custom hatch pattern added to drawing 1 automatically appears in all drawings.



TIP: You may assign both a Hatch and Fill attribute to an object. The hatch attribute will be layered above the fill attribute. Adjusting the fill opacity attribute affects the hatch and fill patterns equally.



TIP: Hatch patterns are not drawn to screen when the zoom exceeds 6400 times. Zoom out to less than 6400 times and the hatches are drawn to screen.

It is possible to 'harden' a hatch after it has been applied to an object. Hardening a hatch pattern breaks the associative link between the hatch pattern and the object and allows attributes to be assigned (e.g. changing pen color, line weight, etc.). A hardened hatch pattern behaves like hatch patterns in early versions of PowerCADD and PowerDraw.

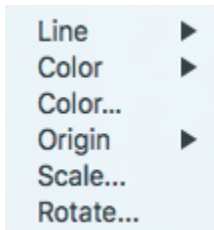
To harden a hatched object:

- ☐ Select the hatched object in the drawing.
- ☐ Choose Convert to Object from the Tool Menu.
- ☐ The association between the object and the hatch pattern is broken. A separate hatch object is created in the drawing.
- ☐ Change the attributes of the hatch pattern by choosing the desired attributes from the Attribute menu for pen color, line weight, etc.

Hatch Settings Menu

Hatch Settings allow you to control various hatch attributes of associative hatch patterns. Attributes may be adjusted without having to first harden the hatch pattern.

The Hatch Settings submenu contains the following choices which may be applied to a hatched object:



Line allows you to change the line weight of a hatch pattern.

Color allows you to select a pen color for a hatch pattern from the PowerCADD color palette.

Color... allows you to select a pen color for a hatch pattern using the MacOS color panel.

Origin allows you to change the starting point (origin) of a hatch pattern.

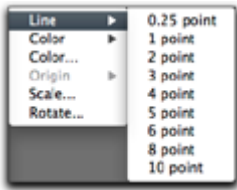
Scale... allows you to specify a scale factor to enlarge or reduce the size of a hatch pattern.

Rotate... allows you to specify a rotation angle of a hatch pattern.



CAUTION: If Global Attributes has been turned On, the choices displayed for Line (line weights), and Color in the Hatch Settings menu are affected. See [Global Attributes](#).

Hatch Line Weight

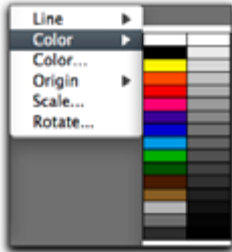


To set the line weight of a hatch for an object:

- ☐ Select the hatched object in the drawing.
- ☐ Choose Hatch Settings, Line, from the Attribute menu.
- ☐ Choose the desired line weight from the list.

The line weights in the list match all other Line menu choices in the drawing.

Hatch Pen Color



To set the pen color of a hatch for an object from the PowerCADD color palette:

- ☐ Select the hatched object in the drawing.
- ☐ Choose Hatch Settings, Color, from the Attribute menu.
- ☐ Choose the desired hatch pen color from the list.

The color choices in the list are from the PowerCADD color palette and match all other pen and fill color menu choices in the drawing. Hatch pen color settings are independent of the object pen and fill color settings.

Set Hatch Pen Color.



To set the pen color of a hatch for an object from a MacOS system color:

- ☐ Select the hatched object in the drawing.
- ☐ Choose Hatch Settings, Color... from the Attribute menu.
- ☐ Choose the desired hatch pen color from the MacOS color panel and click OK.

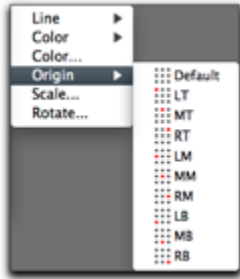
You have complete access to all MacOS color selection options and management features. Hatch pen color settings are independent of the object pen and fill color settings.

Select Hatch Pen Color

To select a hatch pen color that matches an existing color in the drawing:

- ☐ Select the hatched object in the drawing.
- ☐ Choose Hatch Settings, Color... from the Attribute menu.
- ☐ Click on the magnifying glass in the MacOS color panel.
- ☐ Move the mouse over the color you want to match and click. The color is displayed in the color panel. Click OK

Hatch Origin



Many hatch patterns such as brick or tile, often require their origin or starting point to be in a specific location to accurately represent the intended material in the drawing.

To set the origin of a hatch for an object:

- ☐ Control- or right-click on the object.
- ☐ Choose Hatch Origin from the contextual menu or choose Hatch Settings, Origin from the Attribute menu.
- ☐ Choose the desired origin from the list.

When you apply a hatch it will start at the specified origin. If no origin is specified, the default origin is used. Each origin (LT, MT, etc.) is represented by a red square in the list of choices.

Hatch Scale



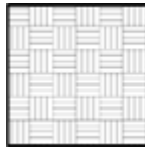
Depending on the scale of the your drawing, some hatch patterns may be too large or too small to represent the selected material. Rescaling a hatch pattern is a quick way to correct that condition.

To change the scale of a hatch for an object:

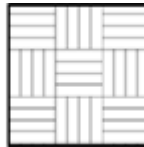
- ☐ Select the hatched object in the drawing.
- ☐ Choose Hatch Setting, Scale[...] from the Attribute menu.
- ☐ Enter a numeric value in the Scale field and click OK.

When entering scaling factors, a value of 1.0 will not rescale the hatch. A value of 0.5 will make the hatch half as large. A value of 2.0 will make the hatch twice as large. You may enter any whole or decimal number in the Scale field.

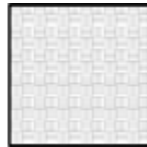
Original Hatch Scale



Result of applying a 2.0 scale factor



Result of applying a 0.5 scale factor



Hatch Rotation Angle

To change the rotation angle of a hatch for an object:

- ☐ Select the hatched object (or multiple hatched objects) in the drawing.
- ☐ Choose Hatch Setting, Rotate[...] from the Attribute menu.
- ☐ The Rotation settings dialog will open.
- ☐ Enter a numeric value in the Rotation field and click OK.

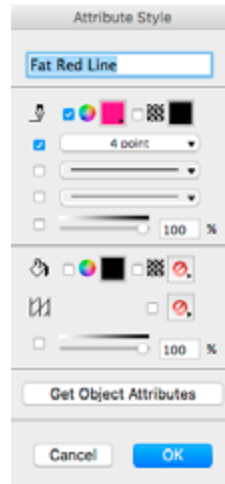
The hatch pattern for the selected items is rotated. To return a hatch pattern to its original rotation either reverse the rotation angle or simply assign the original hatch to the object.

Attribute Styles

Styles is a time-saving feature allowing you to save specific attribute setting combinations as an Attribute style. These Attribute styles can then be applied by selecting them from the Styles menu in the Attribute menu or from a contextual menu when you Control- or right-click on an object.

To create a new attribute style:

- ☐ Select an object in the drawing which has the attributes you want to use to create the style.
- ☐ Choose Styles, New[...] from the Attribute menu — or Control- or right-click on the object and choose New Style from the Styles contextual menu. The Attribute Style dialog will appear.



- ☐ Enter a name for the new attribute style in the name field.
- ☐ Click Get Object Attributes to set the line, fill and hatch attribute settings to match the selected object in the drawing.
- ☐ Select the checkbox next to each attribute you want saved with the new style. You may also make any adjustments to an attribute choices by selecting a new attribute from any pop-up menu.
- ☐ Click OK. The style is saved and appears in the Styles submenu.

Attribute Styles are saved with the drawing. If you want to save an attribute style and have it appear in all drawings, enable Global Attributes in the Preferences menu, General tab before creating your attribute style.



CAUTION: If Global Attributes has been turned On the choices displayed for selecting previously defined Attribute Styles is affected. See [Global Attributes](#).



TIP: When Global Attributes is Off, adding a new Attribute Style to drawing 1 will only affect the menu display in that drawing. If Global

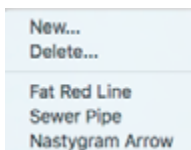
Attributes is On, a new Attribute Style added to drawing 1 automatically appears in all drawings.



See [Attributes Window](#) in the Window section of this chapter for additional information.

Applying an Attribute Style

Once you have created a new attribute style, applying it as simple as choosing it from the Style menu.



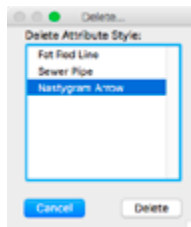
To apply an attribute style to an object:

- ☐ Select the object(s) in the drawing.
- ☐ Choose your custom attribute style name from Styles in the Attribute menu or Control- or right-click on an object and choose the style name from the Styles contextual menu.
- ☐ The attributes associated with the saved style are applied to the selected object(s).

Deleting an Attribute Style

To delete an attribute style:

- ☐ Choose Styles, Delete[...] from the Attribute menu. The Delete Attribute Style dialog will appear.

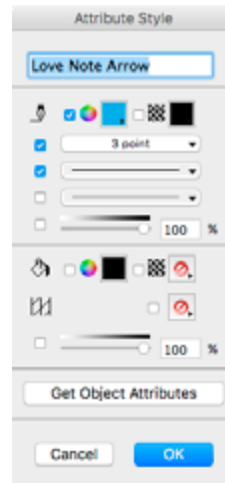


- ☐ Choose the style from the list and click Delete.

Deleting an attribute style only removes it from the Styles list; objects in the drawing are not affected. To delete multiple styles at the same time Shift-click to select sequential styles, or Command-click to select multiple nonsequential styles.

To rename or edit an attribute of a saved attribute style:

- ☐ Press the Option key while choosing Styles from the Attribute menu.
- ☐ The menu display changes, *Edit* is added to the start of each existing style name.
- ☐ Choose the style name to be renamed or edited. The Attribute Style dialog appears.



- ☐ Enter a new name in the Style Name field, make any other attribute changes, and click OK.
- ☐ The new name and any associated attributes are saved. The new name will appear in the Styles menu.

Fill Color Menu



The Fill Color menu is used to add color or gray scales to regular fills, hatched fills, and objects. Colors selected reside in the PowerCADD color palette, managed in the Color Edit (...) dialog box.

To change the color or gray scale of a fill:

- ☐ Select an object in the drawing.
- ☐ Assign a fill pattern to the object as explained in Fill (???).
- ☐ Choose the Fill Color name from the Attribute menu.
- ☐ Choose a color or gray scale from the Fill Color menu and release the mouse button.
- ☐ Upon release of the mouse button, the fill pattern (what?) will be applied to the color or gray scale specified.

You may also fill an object with a solid color by selecting an object and then choosing a color.

For black-and-white Macintoshes, color can be determined by selecting the object from the drawing area, pulling down the Attribute menu, and observing which color is marked “✓”. This permits the use of color plotting and printing devices with all Macintoshes. In addition, color drawings created on black-and-white Macintoshes can be transported to the color Macintoshes for correct color screen rendition.



If you set the fill color to white in this menu, the object will appear as an outlined, opaque object on the screen.

You may change the fill colors available in your Fill Color submenu by choosing Color[...] under the Attribute menu. To change the color in a particular position in the palette press the Option key when choosing a fill color and select the new color from the MacOS color panel. Any change made to the Fill Color palette will also affect the Pen Color palette.



TIP: Editing an existing PowerCADD fill color will not affect the colors of objects in the drawings to which the previous color had been applied. To update the old objects, use Select Special to select the objects with the old fill color then choose the edited (new) fill color from the fill color palette.



TIP: When Global Attributes is Off, editing a fill color position affects the PowerCADD pen and fill color palettes for the specific drawing. If Global Attributes is On, editing a fill color position in the PowerCADD color palette affects the fill and pen color palettes in all drawings.



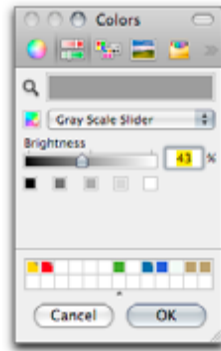
TIP: Fill colors in the PowerCADD color palette are saved with the object. Copying and Pasting an object with a custom fill color from drawing 1 into drawing 2 will retain the object's color attribute but the PowerCADD color palette is not modified to show the new color.

Fill Color...

Fill Color... provides direct access to the MacOS Color Panel allowing you to assign color or gray scales fill colors to objects. Colors accessed directly via the Color Panel are not stored in the PowerCADD color palettes.

To change the color or gray scale of a fill:

- ☐ Select an object (or multiple objects) in the drawing.
- ☐ Choose Fill Color[...] from the Attribute menu — or Control- or right-click on an object — to open a submenu. The MacOS Color Panel will appear.



- ☐ Choose a color or gray scale value from any of the MacOS color palettes and click OK.
- ☐ Upon release of the mouse button, the fill color will change to the color or gray scale specified.

The color assigned to the object does not appear in the standard PowerCADD color palettes. To use the same fill color on a different object you can use the Eyedropper tool, manually add the color to the PowerCADD color palette or use any of the MacOS-level color management features to save and recall your favorite colors.

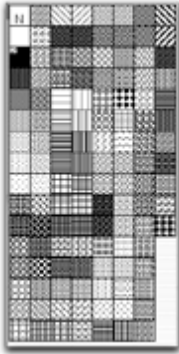


TIP: Editing an existing system fill color will not affect the colors of objects in the drawings to which the previous color had been applied.



TIP: Objects to which a system level fill color has been assigned using the MacOS Color Panel are saved with the object. Copying and Pasting an object with a system level color from drawing 1 into drawing 2 will retain the object's color attribute and the object's system color is appended to the PowerCADD pen and fill color palettes.

Pen Pattern Menu



Pen allows you to assign a pattern to lines other than solid black.

Pen pattern allows you to create objects appearing to have no border. If you fill an object with a diagonal crossed pattern, and you fill the pen with the same diagonal crossed pattern, the object will appear as a fill pattern because it has no border.

To change the pen pattern:

- ☐ Select the object(s) you wish to change.
- ☐ Choose Pen from the Attributes menu.
- ☐ Choose the desired pattern from the Lines menu.

The lines or border of the object you selected will change to the specified pattern.

If you save your drawing as DXF file, pen patterns are automatically converted to solid in the saved file. The original pen patterns in your drawing are not modified.

To change the pen pattern for continuous use:

- ☐ Deselect all items in the drawing area **or the change will affect only that item.**
- ☐ Select a drawing tool from the Tool palette.
- ☐ Choose Pen from the Attributes menu and choose a pen pattern as explained above.

From this point on, whenever the tool is selected, the pen will have the specified pen pattern.

When patterns are assigned to lines, thin lines will appear as unstructured dashes. If you assign a pattern to a heavier line, the pattern will be more apparent.

We recommend that you do not use lines filled with a white pattern to hide unwanted lines. Tools such as Break, Clip, and Cut with Line are much better alternatives for eliminating line segments.

You may change the pen patterns available in your Pen submenu by choosing Pattern[...] under the Attribute menu or pressing the Option key when choosing a pen pattern. The first nine pen pattern positions in the first column starting from None and going down, are reserved and cannot be edited. Any changes made to the Pen Pattern palette will also affect the Fill Pattern palette.



TIP: Editing an existing PowerCADD pen pattern will not affect the pen patterns of objects in the drawings to which the previous pattern had been applied. To update the old objects, use Select Special to select the objects with the old pen pattern, then choose the edited (new) pen pattern from the pen pattern menu.



TIP: When Global Attributes is Off, editing a pen pattern position affects the PowerCADD pen and fill pattern palettes for the specific drawing. If Global Attributes is On, editing a pen pattern position affects all drawing pen and fill palettes.



TIP: Pen patterns are saved with the object. Copying and Pasting an object with a custom pattern from drawing 1 into drawing 2 will retain the object's pattern and a the custom pattern is added to the PowerCADD Pen and Fill pattern palettes.

Pen Color Menu



The Pen Color menu is used to add color or gray scales to lines and borders that outline objects. Colors selected reside in the PowerCADD color palette managed in the Color Edit (...) dialog box.

To change the color or gray scale of a pen:

- ☐ Select an object on the drawing (or multiple objects).
- ☐ Choose Pen Color from the Attribute menu to open a submenu.
- ☐ Choose a color or gray scale from the submenu and release the mouse button.
- ☐ Upon release of the mouse button, the pen color will change to the color or gray scale specified.

You may also fill a line with a solid color by selecting an object and then choosing a color.



For black-and-white monitors, color can be determined by selecting the object from the drawing area, pulling down the Attribute menu, and observing which color is marked “✓”. This permits the use of color plotting and printing devices with all Macintoshes. In addition, color drawings created on black-and-white Macintoshes can be transported to those with color monitors and display the correct color settings.

If you set the pen color to white in this menu, the object will appear as an object with no border.

If you change the color of the pen or the pen pattern to white and print it through a laser printer, the color will revert to black. This is because the laser printer does not recognize colors and any color assigned will print as black. Some newer laser printer devices recognize colors and properly print the color white.

You may change the pen colors available in your Pen Color submenu by choosing Color[...] under the Attribute menu. To change the color in a particular position in the palette press the Option key when choosing a pen color. Any change made to the Pen Color palette will also affect the Fill Color palette.



TIP: Editing an existing PowerCADD pen color will not affect the colors of objects in the drawings to which the previous color had been applied. To update the old objects, use Select Special to select the objects with the old pen color then choose the edited (new) pen color from the pen color palette.



TIP: When Global Attributes is Off, editing a pen color position affects the PowerCADD pen and fill color palettes for the specific drawing. If Global

Attributes is On, editing a pen color position in the PowerCADD color palette affects the fill and pen color palettes in all drawings.



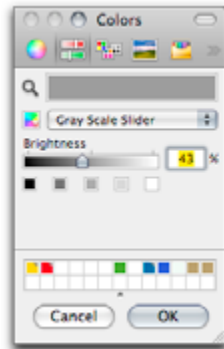
TIP: Pen colors in the PowerCADD color palette are saved with the object. Copying and Pasting an object with a custom pen color from drawing 1 into drawing 2 will retain the object's color attribute but the PowerCADD color palette is not modified to show the new color.

Pen Color...

Pen Color... provides direct access to the MacOS Color Panel allowing you to assign color or gray scales to lines and borders that outline objects. Colors accessed directly via the Color Panel are not stored in the PowerCADD color palettes.

To change the color or gray scale of a pen:

- ☐ Select an object(s) in the drawing.
- ☐ Choose Fill Color[...] from the Attribute menu (or Control- or right-click on an object) to open a submenu. The MacOS Color Panel will appear.



- ☐ Choose a color or gray scale value from any of the MacOS color palettes and click OK.
- ☐ Upon release of the mouse button, the fill color will change to the color or gray scale specified.

The color assigned to the object does not appear in the standard PowerCADD color palettes. To use the same pen color on a different object you can use the Eyedropper tool, manually add the color to the PowerCADD color palette or use any of the MacOS-level color management features to save and recall your favorite colors.



TIP: Editing an existing system pen color will not affect the colors of objects in the drawings to which the previous color had been applied.



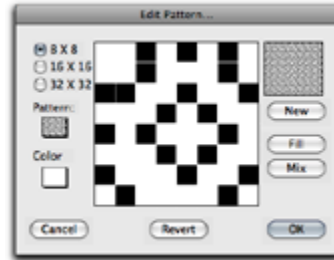
TIP: Objects to which a system level pen color has been assigned using the MacOS Color Panel are saved with the object. Copying and pasting an object with a system level color from drawing 1 into drawing 2 will retain the object's color attribute and the object's system color is appended to the PowerCADD pen and fill color palettes.

Pattern Menu

Pattern allows you to create your own fill patterns. Specialized fills can be added to the Fill palette.

To edit or create a pattern:

- ❑ Choose Patterns from the Attribute menu.



- ❑ To change an existing pattern, click on the Pattern box and hold the mouse button. The **Fill menu** will appear. Choose a fill from the menu you wish to change. Note the fill you select is the fill that will be changed in the fill selection table. The Editing Window in the center of the dialog box and the box in the upper right corner will display the selected fill pattern.
- ❑ To create a new pattern, select the New button. A default pattern will be displayed in the Editing Window. The pattern displayed will be a pattern from the Fill palette which will act as a starting point, but because you clicked New the original fill will not be altered and a new fill will be created with its own space in the Fill menu.
- ❑ Place the cursor inside the central Editing Window and click. A black or a white square will appear. By clicking on these squares to make them black or white, you will be able to create a new fill.
- ❑ To add color, click on the Color box. A table will appear containing the color or gray selections. To choose a color, drag the cursor onto the Color menu. When the cursor is over the desired color and the box is highlighted, release the mouse button. Upon release of the mouse, the Color palette will disappear and the Color box will change to the color you selected. You may then place that color in the fill pattern in the same manner as the black-and-white squares were placed.



NOTE: The box in the upper right hand corner will change as you add or delete a square. This box shows you what the fill will look like when used in a drawing.

- ❑ When you have created a suitable pattern, click OK. This will add the edited fill pattern to the fill table or replace the old fill pattern with the new pattern. If a new fill pattern was created it will be displayed in a new column to the right in the Fill menu. A new pattern will be added to the color table in a new column. An edited color will replace the

originally selected pattern. Note when the new fill pattern is entered, the box entitled Pattern will change to the newly created fill.

- ❑ Click OK and the new fill pattern will be placed in the Fill menu under the Attribute menu. The new fills will also be accessible through Hatch and Pen.



CAUTION: If [Global Attributes](#) has been turned on, changes made to the fill pattern palette affect all drawings. If Global Attributes is Off, changes made affect the active drawing only.

Other options in the Pattern dialog box:

- ❑ Fill allows you to begin creating with a black or colored background instead of white. This is helpful when creating dark or colored fills. If you are filling with a color, simply click on Color and select a color or a gray scale. When you select Fill, the central editing window will fill with that color or gray scale.
- ❑ Mix lets you begin to create a fill starting with a color or gray scale checkerboard central editing window. To mix colors or gray scales, first select a color and fill the central editing window. Then select another color and click on Mix, a checkerboard effect will appear in the central editing window. If you select another color and mix, the original fill will remain and the two colors you selected to mix will switch.
- ❑ Revert lets you switch back to a solid white background provided the New command has not been executed.
- ❑ New serves two purposes. First, it specifies if the fill to be created is new or edited from an existing fill pattern. Second, it specifies the fill is finished and is ready to be placed in the Fill palette.
- ❑ 8 X 8 provides a coarse grid by which to edit. The central editing window is broken down into 64 editable squares.
- ❑ 16 X 16 provides a medium grid by which to edit. The central editing window is broken down into 256 editable squares which allows for better resolution in the fill pattern.
- ❑ 32 X 32 provides a very fine grid by which to edit. The central editing window is broken down into 1024 editable squares which provides optimum resolution and color in the fill pattern.



NOTE: The fills you create are stored with the individual document, not with the PowerCADD program.

If you intend to use the new or modified fill patterns in any drawing other than the current drawing, you must save the drawing as stationery. You will want all of your customized settings in Untitled stationery in the PowerCADD folder.

You may also copy an object with a custom fill pattern from one drawing and paste it into another. The pattern will be added to the existing palette in the new drawing.

Color Menu

Color allows you to edit individual colors in a PowerCADD color palette to suit your needs. You may create hues and different brightness and variations of color to add to the Color palette. Colors saved, added or edited here are separate from colors created or applied using the Pen Color(...) or Fill Color(...) menu items.

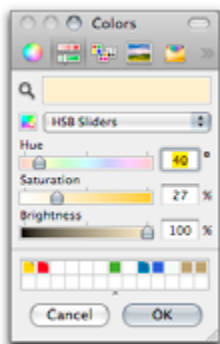
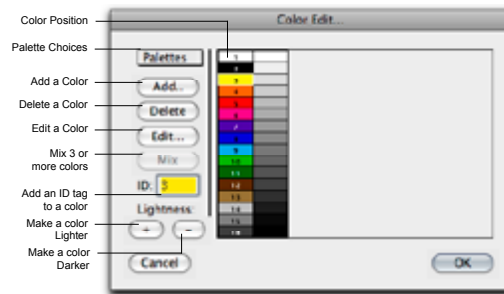
You may also create custom color palettes and assign identification numbers or text to specific colors. You can create your own collections of color palettes and easily switch between collections. Your office can standardize their color palettes. Since you can apply identification to the colors, you can create a numbering/naming system and be able to discuss the subtle shades between colors, i.e., gray #6 rather than a slightly darker gray.



CAUTION: If Global Attributes has been turned On, changes made to the color palette affect all drawings. If Global Attributes is Off, changes made affect the active drawing only. See [Global Attributes](#).

When you select Color from the Attribute menu, the following Color Edit dialog appears:

To add a color to the Color palette:



- ☐ Choose Color(...) from the Attribute menu and the Color Edit dialog box will appear, containing all the colors present in the PowerCADD Color palette.
- ☐ Select Add from the dialog box. The MacOS color panel appears. You may use any of the MacOS color pickers available in the color panel, this example illustrates the HSB (Hue, Saturation and Brightness) color picker.
- ☐ Drag the Hue, Saturation, or Brightness sliders or type in a value in any of their respective fields. The new color is shown in the color indicator at the top of the window.
- ☐ When you have created the color you need, click OK and the first dialog box will appear with the new color present. Click OK on the first dialog box and the color is added to the Color palette.

Hue shows the location angle of the color you have selected. As you adjust the Hue slider, the color indicator will preview the change.

Saturation allows you to set the tone of a particular color. As you adjust the Saturation slider, the color indicator will preview the change.

Brightness allows you to set the intensity of the color. As you adjust the Brightness slider, the color indicator will preview the change.



NOTE: You cannot add a color to full palettes, like 16Hues, EarthTones, Pastels or Rainbow. Instead, edit existing colors or delete unwanted colors.

To delete a color from the Color palette:

- ☐ Choose Color... from the Attribute menu.
- ☐ Select a color. When selected, the color will be highlighted.
- ☐ Click Delete. The color will be removed from the palette.
- ☐ Click OK to leave the dialog box.



NOTE: You cannot undo deleting a color. If you delete the wrong color or did not want to delete the color, click Cancel to leave the dialog box without saving any changes.

When a color is removed from a palette, all colors following it will shift their place in the palette by one position. Each object in the drawing remembers its specific color value. As such, editing, adding or deleting color positions in the color palette has no effect on objects already in the drawing.

To edit an existing color:

- ☐ Choose Color from the Attribute menu. A dialog box will appear.
- ☐ Select a color. When selected, the color will be highlighted.
- ☐ Click Edit. A second dialog box will appear.
- ☐ You may now edit the color in the manner described above.
- ☐ When the desired color has been created, click OK. The color you selected to edit will be replaced by the color you created. Click OK to leave the first dialog box.

If you select three or more colors, the Mix button will become active. If the Mix button is selected, the colors selected will change to a scale of the first color selected to the last color selected. For example, if you choose white first and drag to blue, Mix will result in a scale from white to blue with the number of colors in between governed by the number of color boxes selected. If five boxes were selected, then two would be blue and white and three would be shades of the blue and white.

Using the ID field, you can give each color its own ID number. Just select

a color—the ID field will become active, and you can type in a number or text up to four characters long. If you use this color in another palette, the number or text carries over to the other palette. The IDs that were entered are saved in a file called Color Palettes.

Lightness allows you to modify the saturation without entering the Color picker. First select a color from the palette. Then click on the Plus button to lighten the color or the minus button to darken the color. The highlighted color will lighten or darken as you click. If you select multiple colors at once, the Lightness buttons will change them simultaneously.

Colors also allows you to use several pre-created palettes which are located to the left hand side of the color dialog box. Pressing on the pop-up menu named Palettes opens a submenu.

To use one of the palettes:

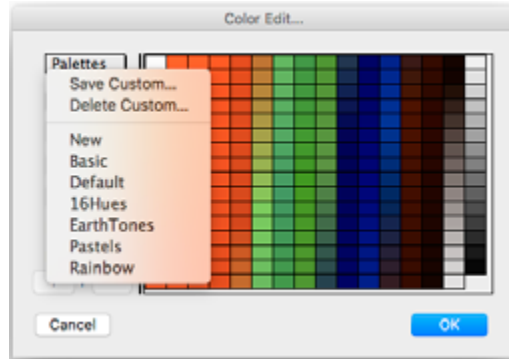
- ☐ Choose Palettes and drag to the desired palette and release.
- ☐ Upon release of the mouse, the color table to the right will change to the selected palette. This palette will also show when you are using the Color palette under the Attribute menu as well as the Defaults Window or through contextual menus.

Save Custom Palette

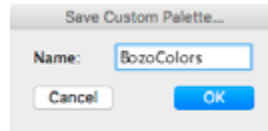
Save Custom allows you to save new custom color palettes.

To save a custom color palette:

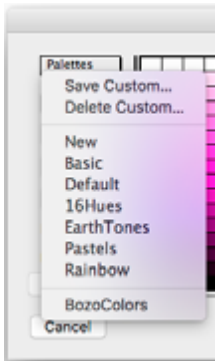
- ❑ Choose Save Custom from the Palettes pop-up menu.



- ❑ The Save Custom Palette dialog appears.



- ❑ Enter the name of the palette and click OK.



The new palette will appear at the bottom of the list.

All custom color palettes you create in a single session of PowerCADD, as well as all IDs you assign to colors, will be stored in a single file called Color Palettes. The Color Palettes file is stored in the PowerCADD folder in your User directory, Library folder, Preferences folder. Any customized color palettes you create can be saved by copying the Color Palettes file into another folder, or by renaming the file. If you rename the file, PowerCADD will create a new file called Color Palettes the next time you assign IDs to colors or make new palettes.

This makes color palettes easy to save, transport, and interchange between different users. You can make customized color palettes for a project, and save them with the project in the same folder. If you create a special color scheme or numbering system that a colleague wishes to use, you can send them the Color Palettes file. Later, if you wish to use this color scheme with another project, you can copy the file back into your PowerCADD folder and rename it to Color Palettes of the list.

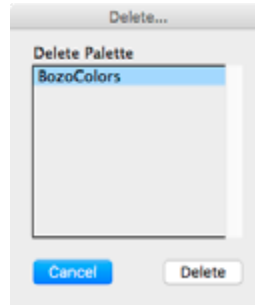
Delete Custom Palette

Delete Custom allows you to delete customized color palettes.

Only custom color palettes may be deleted.

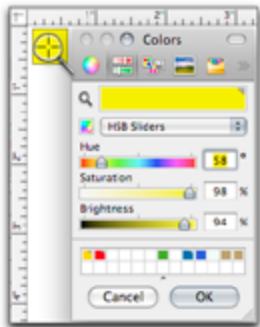
To delete a custom color palette:

- ☐ Choose Delete Custom from the Palettes pop-up menu. The Delete Custom Palette dialog appears.



- ☐ Choose the name of the palette so it highlights and click Delete. A warning message will appear, to verify that you really want to delete the palette. To delete multiple custom palettes at the same time Shift-click to select sequential palettes, or Command-click to select multiple nonsequential palettes.
- ☐ Click OK.

If you intend to use the new or modified colors in any drawing other than the current drawing, Save As stationery or Save the colors in a custom palette. Otherwise, you will have to paste an object with the desired color from one drawing to another, or create the special colors in each drawing as needed.



Custom colors pasted from one drawing to another are not appended to the color palette. However, since the color value is saved with the object, the object will retain its color.

To add the pasted object's color to the new drawing color palette:

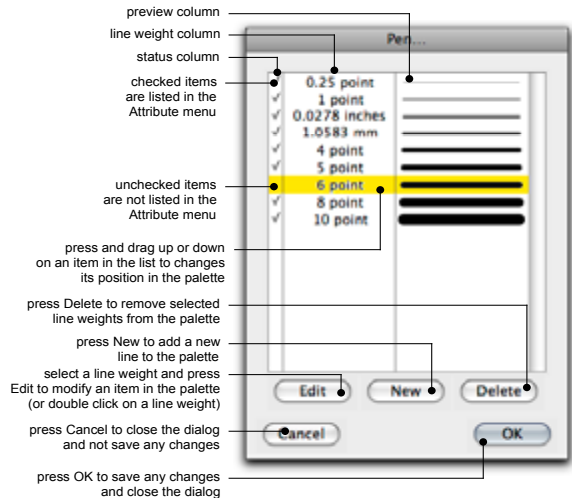
- ☐ Choose Color(...) from the Attribute menu. The Color Edit dialog will appear. Click the Add button and the MacOS Color Panel will appear.
- ☐ Click on the magnifying glass next to the color well.
- ☐ Drag the magnifying glass cursor over the object in the drawing containing the color you want to add and click the mouse.
- ☐ The color will be displayed in the MacOS color well, click OK.
- ☐ The color will be appended to the PowerCADD color palette.

Attribute Line Weight

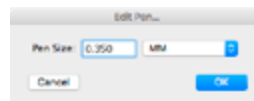
Lines allows you to enter customized line weights. These weights can be entered in inches, points, or millimeters.

To customize the Line Weight palette:

- ☐ Choose Lines from Attribute menu.
- ☐ The following dialog will appear:



- ☐ The dialog presents a list of current line weights in the drawing and which items are currently displayed in the Attribute menu. You have complete control over how many line weights are displayed in the Attribute menu by checking or unchecking an item in the status column.
- ☐ Double-click any line in the preview pane or select a line and click Edit. The Edit Pen dialog will appear.



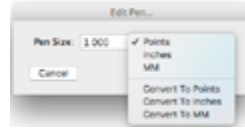
- ☐ The current pen size is listed in the Pen Size field. You may type in the desired line weight and select the desired units (Points, Inches or Millimeters) from the pop-up list.
- ☐ After you have made all the changes necessary, click OK to place the line weights in the Line Weight palette.
- ☐ The line weight will remain in effect until you change them. They are stored with the document and are not made available in other documents.

If you intend to use the new line weights in any drawing other than the current drawing, you must Save As stationery. If you do not Save As stationery, it will be necessary to create the special line weights in each drawing they are needed. You will want all of your customized settings in Untitled stationery in the PowerCADD folder.

New line weights may be added to the palette at any time.

To add a line weight:

- ☐ Choose Lines[...] from the Attribute menu.
- ☐ Click New in the Pen... dialog box. The Edit Pen dialog will appear.



- ☐ Type the desired line weight in the Pen Size field and choose the desired units from the pop-up menu.
- ☐ After you have made all the changes, click OK. The new line is appended to the bottom of the palette.
- ☐ To change the position of the new line weight, click and drag on the line in the preview pane. Release the mouse when you have positioned the line in the desired location.

Line weights are appended to the palette when pasting information from different drawings or inserting data from symbol libraries. If you find the menu contains too many items or the items need to be reordered, simply turn items On or Off or change their order as previously described.



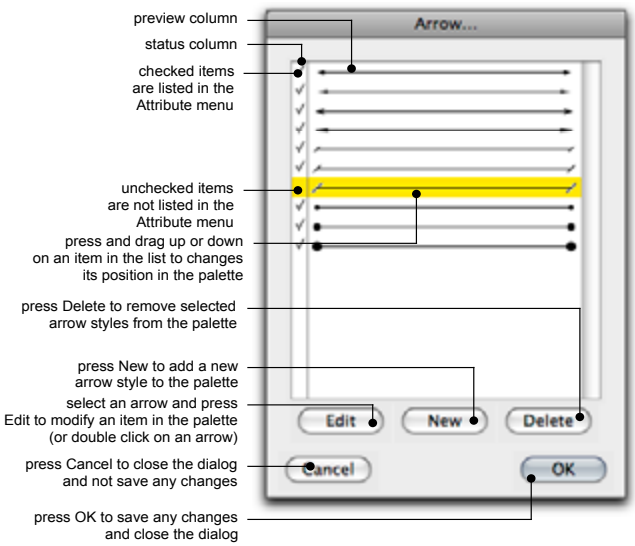
CAUTION: If Global Attributes has been turned On, changes made to the line weight palette affect all drawings. If Global Attributes is Off, changes made affect the active drawing only. See [Global Attributes](#).

Attribute Arrows Menu

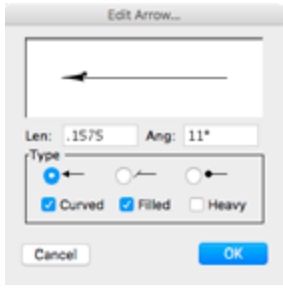
Arrows allows you to create custom arrows to use with a particular drawing. The arrows created are stored with the document or the stationery, not with the PowerCADD program.

To change the selection in the Arrows menu:

- ❑ Choose Arrows from the Attribute menu.
- ❑ The following dialog will appear:



- ❑ This dialog shows a list of the current arrow styles in the document, their position in the menu and whether they are visible or not visible in the Arrow menu.
- ❑ Double-click any arrow in the preview pane or select an arrow style and click Edit. The following dialog will appear:



Not all options are available for all arrow types. If an option is unavailable, it will be dimmed, and you will be unable to select it.

Arrowheads can be constructed of straight lines, or curved lines for a more artistic look. They can also be filled, have a heavier line for the arrowhead, or have any combination of the three options.

Heavy is the only option for the tick mark arrow, and it will use a larger point size for the tick mark.

Filled is the only option for the node arrow, and it will fill the node.

- ☐ Select the desired type and the arrow will be placed in the Editing box.
- ☐ Place the cursor on the handle of the arrowhead in the Editing box.
- ☐ Drag the cursor in any direction. Note how the arrowhead changes shape. A second way to create an arrow is to type a length and angle. The arrow will snap to the parameters immediately.
- ☐ When you have created the arrow to your satisfaction, click OK and the arrow will be inserted in the menu box of the arrows.

If the arrow was edited from an existing arrow and the same arrow appears in both windows, the OK button will place it in both the Available box and the Menu box.

If the arrow was edited from an existing arrow and the arrow did not exist in both boxes, the edited version will only be placed in the Available box.

If a new arrow was created, the OK button will place the arrow in the Available box, and you must transfer it to the Menu box.

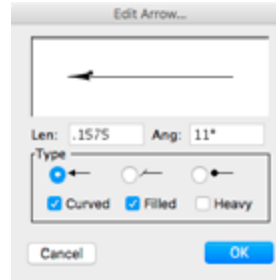
If you intend to use the new or modified arrows in any drawing other than the current drawing, you must save as stationery. If you do not save as stationery, you must create the special arrows in each drawing they are needed. You will want all of your customized settings in Untitled stationery in the PowerCADD folder.

Attribute Arrow Styles

New arrow styles may be added to the palette at any time.

To add an arrow style:

- ☐ Choose Arrows[...] from the Attribute menu.
- ☐ Click New in the Arrow... dialog box. The following dialog will appear.



- ☐ Specify the desired options as described earlier for editing arrows.
- ☐ After you have made all the changes, click OK. The new arrow style is appended to the bottom of the palette.
- ☐ To change the position of the new arrow style, click and drag on the arrow in the preview pane. Release the mouse when you have positioned the arrow in the desired location.

Arrow styles are appended to the palette when pasting information from different drawings or inserting data from symbol libraries. If you find the menu contains too many items or the items need to be reordered, simply turn items On or Off or change their order as previously described.



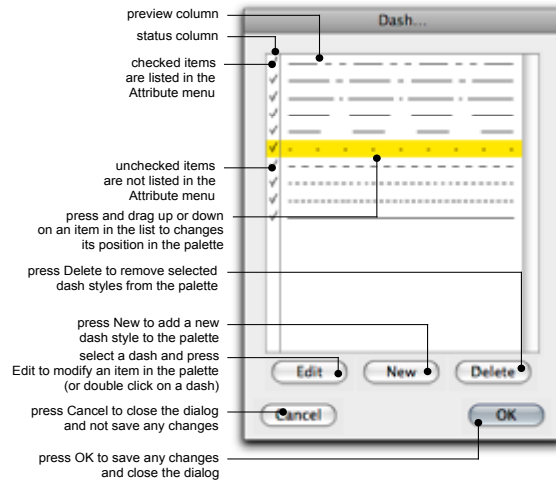
CAUTION: If Global Attributes has been turned On, changes made to the arrow style palette affect all drawings. If Global Attributes is Off, changes made affect the active drawing only. See [Global Attributes](#).

Attribute Dash Menu

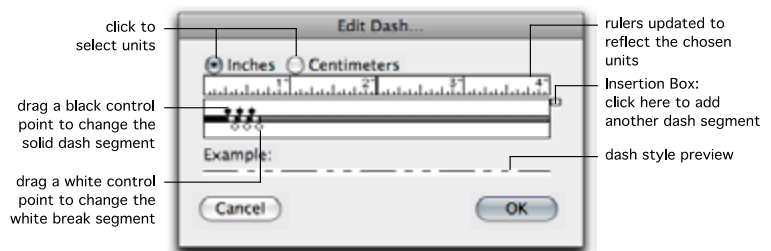
Dashes allows you to create customized dashed lines. These dashed lines are stored with the document or the stationery.

To change the dashed line forms in the Dash menu.

- ☐ Choose Dashes from the Attributes menu.
- ☐ The following dialog box will appear:



- ☐ This dialog shows a list of the current dash styles in the document, their position in the menu and whether they are visible or not visible in the Dash menu.
- ☐ Double-click any dash in the preview pane or select a dash style and click Edit or New. The following dialog will appear:



In this box is a ruler to set up the dashes and breaks, a choice of inches or centimeters, a gauge line to construct the dashes and breaks, an Insertion box which adds dash segments and breaks, and an example line to display the appearance of the line.

- ☐ Click the small Insertion box on the right. A line segment and a line break will appear. The black handle on top of the gauge line corresponds to the dash segment while the white handle below the gauge line corresponds to the break segment.

- ❑ Select one of the diamond-shaped handles and drag the cursor in either direction. The example line moves with the line segment change. The ruler allows you to create dashed lines of a specific size. When the mouse button is released, the line break/dash will be set until you change it. For additional dashes or breaks, click the Insertion box at the right hand side of the gauge line. If you place too many breaks in the line, select and drag the segment handle as far as possible to the right. The segment will only stretch a certain distance, you must drag until the cursor is off of the gauge line to the right. This will remove extra line dashes and breaks.
- ❑ When you have created the dashed line to your satisfaction, click OK and the line will be inserted at the bottom of the list in the dialog window.

The line forms that are created in a drawing are stored with that document. The dashed lines that were created will not be available in other drawings.

If you intend to use the new or modified dashes in any drawing other than the current drawing, you must save as stationery. If you do not Save As stationery, it will be necessary to create the special dashes in each drawing they are needed. You will want all of your customized settings in Untitled stationery in the PowerCADD folder.

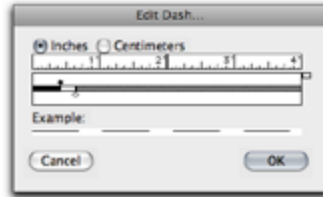


CAUTION: If Global Attributes has been turned On, changes made to the dash pattern palette affect all drawings. If Global Attributes is Off, changes made affect the active drawing only. See [Global Attributes](#).

New dash styles may be added to the palette at any time.

To add a new dash style:

- ☐ Choose Dashes[...] from the Attribute menu.
- ☐ Click New in the Dash... dialog box. The following dialog will appear.



- ☐ Specify the desired options as described earlier for editing dashes.
- ☐ After you have made all the changes, click OK. The new dash style is appended to the bottom of the palette.
- ☐ To change the position of the new dash style, click and drag on the dash style in the preview pane. Release the mouse when you have positioned the dash in the desired location.

Dash styles are appended to the palette when pasting information from different drawings or inserting data from symbol libraries. If you find the menu contains too many items or the items need to be reordered, simply turn items On or Off or change their order as previously described.

New Hatch

New Hatch allows you to make your own custom hatch patterns. Custom hatches from selected objects in the drawing or using the Line Hatch editing dialog. Custom hatches are appended to the main hatch menu and are created to scale based on the current drawing or layer scale.



CAUTION: If Global Attributes has been turned On, changes made to the hatch pattern palette affect all drawings. If Global Attributes is Off, changes made affect the active drawing only. See [Global Attributes](#).

From Selection

To create a custom hatch pattern from selected objects in the drawing:

- ☐ Set up your drawing scale and units to reflect the scale you want the hatch pattern created. For example, if you want to create a custom tile or brick hatch appropriate for drawings at 1:48, setup the drawing scale and units accordingly.
- ☐ Draw the objects representing the hatch pattern and assign any desired attributes for color, line weight, fill, etc. The hatch pattern may contain any type of PowerCADD object.
- ☐ Select all the objects to be included in the hatch and choose New Hatch, From Selection, from the Attribute menu.
- ☐ The custom hatch is appended to the end of the hatch palette.

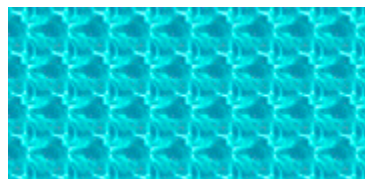
There is no method to directly edit a hatch pattern once it has been added to the palette. As such, it is recommended you save a PowerCADD drawing with the core objects used to create your custom hatch patterns. Having a saved drawing makes it fast and easy to go back and edit the pattern or share the patterns with other PowerCADD users.



TIP: To lock the pen color and line weight of a hatch pattern, group the objects before choosing New Hatch, From Selection. If the objects are not grouped you will be able to change the pen color and line weight after the hatch is applied to an object using the Hatch Settings, Line and Color menu items.



TIP: You can create a hatch from a bitmap image for anything. A good example is shown for water below. Of course, you could also have grass, fabric, gravel, etc.



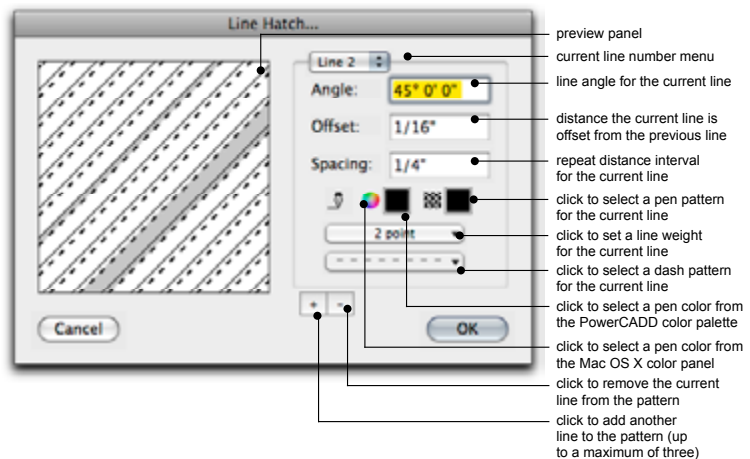
The Line Hatch editor allows you create line-based hatch patterns which redraw seamlessly when an object they are applied to is reshaped or resized.

The following general rules apply to new hatches created with the Line Hatch menu item:

- ☐ A new line hatch pattern cannot be edited using the Line Hatch dialog once it is added to the hatch palette.
- ☐ If the new line hatch contains multiple lines, using the Hatch Settings menu to change the hatch pen color or line weight of a pattern applied to an object will change all lines in the pattern.
- ☐ The origin of a Line Hatch pattern cannot be changed.
- ☐ The scale of a Line Hatch pattern cannot be changed.
- ☐ A line hatch may contain up to five lines. The attributes and angle for each line may be set independently.

To create a new line hatch pattern:

- ☐ Choose Hatch, Line Hatch[...] from the Attribute menu, the Line Hatch editor will appear.



- ☐ Selected the desired options from the dialog box and click OK. The new line hatch is appended to the hatch pattern palette.

The Preview Panel provides instant feedback as changes are made to other line hatch settings.

The Current Line Number pop-up menu allows you to select a particular line number for editing. The values and options displayed for angle, offset, spacing, and object attributes may be adjusted separately for each line number. The display will update when switching between different line numbers from the Current Line Number pop-up menu.

The Angle field allows you specify a line angle for the currently selected line number. Angles may be entered in positive or negative values, decimal degrees, or degrees minutes and seconds. For example an angle of 45.0 or an angle of -45.0 or an angle of 135.0 all produce the same result. You may also enter math in the angle dialog box, for example 45-15 would draw the line at 30 degrees.

The Offset field affects multi-line hatch patterns, allowing you to specify the offset distance from Line 1 to Line 2, or from Line 1 to Line 3. The offset distance is always measured relative to Line 1, and each line may have a different offset distance. The units and scale for values entered here are taken from the Drawing Setup, Scale tab settings.

The Spacing field allows you specify the interval before a line repeats itself. Spacing is measured relative to a current line number, and each line may have a different spacing interval. The units and scale for values entered here are taken from the Drawing Setup, Scale tab settings.

- ☐ Click on the MacOS Color Wheel to choose a pen color for the current line using the MacOS color panel. Each line may have a different pen color.
- ☐ Click on the PowerCADD Color Swatch to choose a pen color for the current line from the PowerCADD color palette. Each line may have a different pen color. The values in this palette are the same as those available in the Attribute menu.
- ☐ Click on the Pen Pattern Swatch to choose a pen pattern for the current line from the Pen Pattern palette. Each line may have a different pen pattern. The values in this palette are the same as those available in the Attribute menu.
- ☐ Click on the Line Weight pop-up menu to choose a line weight for the current line. Each line may have a different line weight. The values in this palette are the same as those available in the Attribute menu.
- ☐ Click on the Dash Pattern pop-up menu to choose a dash pattern for the current line. Each line may have a different dash pattern. The values in this palette are the same as those available in the Attribute menu.



CAUTION: When Global Attributes is On, values displayed for Pen Pattern, Line Weight, and the PowerCADD Color Palette are affected. See [Global Attributes](#).

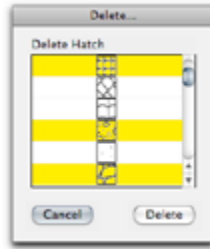
- ☐ Click the plus button to add a new line to the hatch pattern. The new line number is displayed in the Current Line Number pop-up menu.
- ☐ Click the minus button to remove the current line number displayed in the pop-up menu at the top of the dialog from the hatch pattern. This operation cannot be undone.

Delete Hatch

Delete Hatch allows you remove unwanted patterns in the hatch menu.

To delete a hatch pattern:

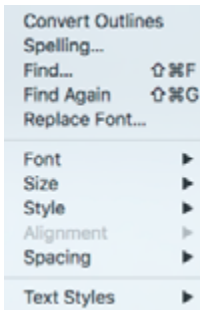
- ☐ Choose Delete Hatch from the Attribute menu. The Delete Hatch dialog will appear.



- ☐ Select the hatch pattern to be deleted and click Delete. To delete consecutive hatches, Shift-click on the patterns. To delete non-consecutive hatches, Command-click on the patterns.
- ☐ Click OK to close the dialog box.

The hatch pattern will be removed from the Hatch menu.

Text Menu



Text can be entered at any zoom, but it is best entered at normal scale. When text is zoomed too small to see, it becomes a rectangle the approximate size of the text.

Unusual character spacing problems can occur when you send a drawing to another office across the country and the MacOS the drawing is viewed on does not have the same fonts you have.

Convert Outlines

Convert Outlines produces Bézier curves tracing the outlines of your TrueType fonts.

Convert Outlines is useful, as Bézier curves can be manipulated in ways not possible with text.

These outlines can be scaled, reshaped, mirrored, or sheared. Non-proportional scaling or resizing creates expanded or condensed text. Sheared outlines can be used to place text in isometric drawings or can be used in desktop publishing applications. Mirrored outlines can be used to produce plans to be used on light boards.

To convert text to outlines:

- ☐ Select the text to be converted.
- ☐ Choose Convert Outlines from the Text menu. The original text is replaced with Bézier outlines.



NOTE: Convert Outlines converts TrueType and OpenType fonts only.

Spelling...

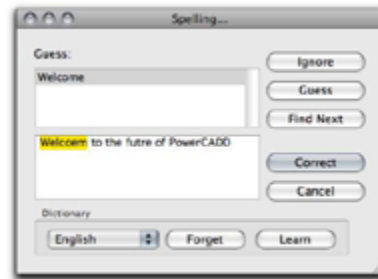


Spelling finds words in your drawing having no match in the dictionary and makes suggestions for correction. Spelling can be checked for the entire document or a selection of text.

NOTE: PowerCADD uses the standard spell checker and dictionaries used in other MacOS programs such as TextEdit or Mail. This allows you to have one common dictionary for all MacOS applications.

To spell check your entire document:

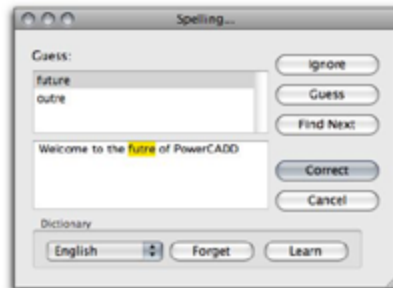
- ☐ Choose Deselect All from the Edit menu.
- ☐ Choose Spelling from the Text menu.
- ☐ When a word having no match in the dictionary is encountered, the Spelling dialog will appear:



The lower pane of the dialog selects the misspelled word and displays it as it is used in the text block. The upper Guess pane of the dialog lists any suggested spelling corrections and selects the recommended change.

If the selected suggestion is correct, click Correct, or press the Return key, or double-click on the suggested word to apply the new spelling. If the selected word is in fact correctly spelled but simply not in the dictionary, click Learn. If a word was incorrectly added to the dictionary, select the word and click Forget. You can change languages by clicking on the popup menu and choosing a preferred language dictionary from the list.

You can jump to the next misspelled word by pressing Find Next. The misspelled will be selected and any suggested spelling changes are listed as shown here:



You can go to the next word without any change by clicking Ignore. To see a word suggestion for any word, click on the word in the lower portion of the dialog and a list of suggested words will appear in the Guess pane.

To spell check selected text in a document:

- ☐ Select text block or multiple text blocks to be spell-checked.
- ☐ Choose Spelling from the Text menu.
- ☐ When a word having no match in the dictionary is encountered, the Spelling dialog will appear. Follow the steps described earlier to complete the spell-check operation.

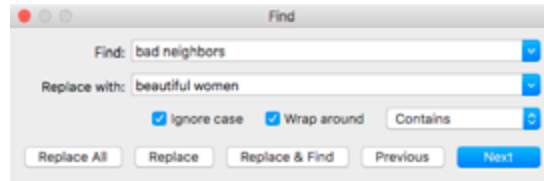
Find

Find searches the document for the text you specify. It does not search in dimensions or symbols.

Find and Replace works very similarly to your word processing applications. You can find a single word or a combination of words, then replace one occurrence, replace all of them, or just find the next one. Find and Replace can be a very efficient way to change references to detail keys in a drawing, change page number references or update part numbers, plant names or spot elevations in a drawing

To find a word or phrase:

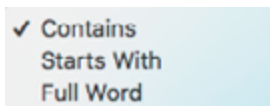
- ❑ Select Find from the Text menu. The Find/Replace dialog will appear.



- ❑ Type the text you are searching for in the Find box and any text you wish to replace it with in the Replace box. You can also select text within your document, copy the selection and paste it into the Find box.
- ❑ Click Next. The first instance of the phrase will be highlighted. Alternately, you may replace just that occurrence by clicking Replace, or replace every occurrence by clicking Replace All.

Repeat these steps to find or replace the next occurrence.

Case Sensitive searches for instances of the word which has the same case spelling, so you can look for instances of *SOAR* and ignore instances of *soar*.



To change the type of search click **Contains** and choose the desired option from the popup list. **Contains** searches for instances where the phrase is part of a word. For example, if you are trying to find *cat* and **Contains** is active, it will consider *cataract* a valid choice. If **Starts with** is active, only words which start with the specified phrase are found. **Full Word** will search for instances where the phrase is a whole word (words separated by spaces), and will exclude instances where the phrase is part of a word. For example, if you are trying to find *cat* and **Full Word** is active, it will not consider *cataract* a valid choice

Find goes to the first instance of the phrase and highlights the phrase in the text block. Replace goes to that instance, substitutes in the phrase from the Replace field, and highlights the replacement in the text block. Replace All performs a replacement for all valid instances in the document.

Find Again

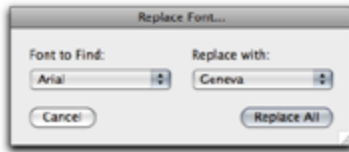
Find Again finds the next occurrence of the given characters, highlighting them for editing. After finding the last occurrence, it will start over, choosing the first occurrence again.

Replace Font

Replace Font allows you globally change one font family to another. For example, you can change all text blocks in your drawing from Geneva to Times.

To use Replace Font:

- ☐ Choose Replace Font from the Text menu.
- ☐ The Replace Font dialog will appear.



- ☐ The Font-to-Find list contains a list of all fonts currently used in your document. Click on the first font name in that popup list and select the desired font that is going to be replaced.
- ☐ The Replace-with list contains a list of all fonts available in MacOS. Click on the first font name in that popup list and select the desired font to be used as a replacement.
- ☐ Click Replace All and all fonts are updated in the drawing.

Font



Font allows you to select the font that you want to have in your text.

To select a font for continuous use:

- ☐ Select the Text tool from the Tool palette by double-clicking on it.
- ☐ Choose Font from the Text menu.
- ☐ Choose the Font you wish to use from the submenu.
- ☐ The font you selected is active and will remain active until you change it.

To change a font momentarily:

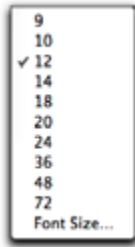
- ☐ With the Text tool selected, click where you want to place the next piece of text.
- ☐ Choose Font from the Text menu as explained before.
- ☐ Enter the text with the new font characteristics. When you click on the next point to enter text, the font will revert to the font type that was active before the change.

This font change is temporary and will only remain in effect until you add another piece of text.

To change the font of existing text:

- ☐ Select the text you wish to change.
- ☐ Choose the correct font type from the Font menu.
- ☐ Upon selection of the font, the selected text will change to the specified font type.
- ☐ Alternately, select in the Text and highlight the text to be changed. Multiple styles and fonts may be used in a single piece of text.

Font Size



Size allows you to select the font size you wish to use.

To select a font size for continuous use:

- ☐ Select the Text tool from the Tool palette by double-clicking.
- ☐ Choose Size from the Text menu, press and hold the mouse button.
- ☐ Choose the size you wish to use.
- ☐ The size you selected is active and will remain active until you change it.

To change a font size momentarily:

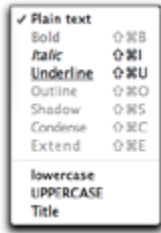
- ☐ With the Text tool selected, click where you want to place the next piece of text.
- ☐ Choose Size from the Text menu and choose a font size as explained before.
- ☐ Enter the text with the new font size characteristics. When you click on the next point to enter text, the font size will revert to the font size that was active before the change.

This size change is temporary and will only remain in effect until you attempt to add another piece of text. The Text tool defaults are remembered and change only when the Text tool is selected.

To change the font size of existing text:

- ☐ Select the text you wish to change.
- ☐ Choose the correct font size from the Size menu.
- ☐ Upon selection of the font size, the selected text will change to the specified font size.

Font Style



Style allows you to select the font styles you wish to use.

To select a font Style for continuous use:

- ☐ Select the Text tool from the Tool palette by double-clicking.
- ☐ Choose Style from the Text menu, press and hold the mouse button.
- ☐ Choose the style you wish to use.
- ☐ The style you selected is active and will remain active until you change it.

To change a font Style momentarily:

- ☐ With the Text tool selected, click where you want to place the next piece of text.
- ☐ Choose Style from the Text menu and choose a font style as explained before.
- ☐ Enter the text with the new font style characteristics. When you click on the next point to enter text, the font style will revert to the font style that was active before the change.

This style change is temporary and will only remain in effect until you attempt to add another piece of text.

To change the font Style of existing text:

- ☐ Select the text block you wish to change. Selection handles will appear to indicate the text has been selected.
- ☐ Choose the correct font style from the Style menu.
- ☐ Upon selection of the font style, the selected text will change to the specified font style.

Lowercase/Uppercase

Lowercase or Uppercase allows you to change the case of existing text.

To change case in portions of existing text:

- ☐ Select the Text tool from the Tool palette.
- ☐ Select the text you want to change.
- ☐ Select the portion of the text you wish to change by highlighting.
- ☐ Choose Lowercase or Uppercase from the Text menu.
- ☐ The text will change to the characteristics you specified.

To change the case of entire portions of existing text:

- ☐ Select the text block you wish to change. Selection handles will appear to indicate the text has been selected.
- ☐ Choose Lowercase or Uppercase from the Text menu.

Title Text

Title allows you to change existing text to title text. You are able to select a piece of text and change the first letter of each word from lowercase to uppercase.

To change portions of existing text to Title text:

- ☐ Select the Text tool from the Tool palette.
- ☐ Select the text block you wish to change.
- ☐ Select the words you want to change by highlighting.
- ☐ Choose Title from the Text menu.
- ☐ The first letter of the words you selected will be changed to uppercase letters.

To change entire portions of existing text to Title text:

- ☐ Select the text block you wish to change. Selection handles will appear to indicate the text has been selected.
- ☐ Choose Title from the Text menu.

Text Alignment



Left/Center/Right/Full

Alignment allows you to select the alignment you wish to use.

To select a font Style for continuous use:

- ☐ Select the Text tool from the Tool palette by double-clicking.
- ☐ Choose Alignment from the Text menu, press and hold the mouse button.
- ☐ Choose the alignment you wish to use.
- ☐ The alignment you selected is active and will remain active until you change it.

Left, Center, Right and Full allows you to select how you want the text to be aligned.

To select a text alignment for continuous use:

- ☐ Select the Text tool from the Tool palette by double-clicking.
- ☐ Choose Left, Center, Right or Full from the Text menu.
- ☐ The text alignment you selected is active and will remain active until you change it.

To change a text alignment momentarily:

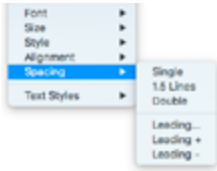
- ☐ With the Text tool selected, click where you want to place the next piece of text.
- ☐ Choose a different text alignment from the Text menu.
- ☐ Enter the text with the new text alignment characteristics. When you click on the next point to enter text, the text alignment will revert to the alignment active before the change.

This alignment change is temporary and will remain in effect only for this piece of text.

To change the text alignment of existing text:

- ☐ Select the text you wish to change.
- ☐ Choose Left, Center, Right or Full from the Text menu.
- ☐ Upon selection of the text alignment, the selected text will be aligned according to specifications.

Text Spacing



The Spacing commands in the Text menu allow you to control the distance between lines of text.

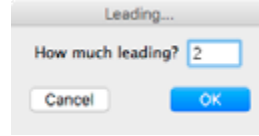
Spacing between lines can be chosen as in any word processing program, by choosing single spacing, 1-1/2 spacing or double spacing.

Text Leading

Leading allows you to choose the exact amount of leading required in point size.

To use Leading:

- ☐ Select the text to which you wish to apply a specific leading.
- ☐ The following dialog will appear:



- ☐ Type the desired amount and click OK.

Leading Plus

Leading+ adds space between lines. If more leading is desired, you may choose Leading+ more than once for the same piece of text.

Leading Minus

Leading- reduces space between lines. As with Leading+, you may choose Leading- more than once for the same piece of text.

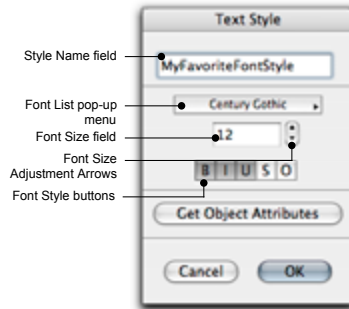
Add Text Style

Add Style allows you to add a style to the Text Style menu.

If you have a particular set of text attributes you use regularly, creating a favorite is an excellent way to streamline your workflow.

To create a new favorite:

- ☐ Select the text block in the drawing which has the correctly assigned text attributes.
- ☐ Click on the Favorites pop-up in the Text Attributes window and choose Add Style. The following dialog will appear:

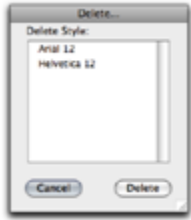


- ☐ Enter a name for the new favorite in the Style Name field.
- ☐ Click the Get-Object-Attributes button to acquire the attributes of the selected item in the drawing.
- ☐ Make any desired changes by adjusting the Font Size or Style or select a different font from the Font List pop-up.
- ☐ Click OK and a preview of the new style is appended to the Text Styles menu and to the Favorites menu in the Text Attribute Window.



NOTE: Pen color, pen pattern, fill color and fill pattern are not saved with a text favorite.

Delete Text Style



Delete Style allows you to remove any text style from the Text Style menu.

To delete a style from the Text Style menu:

- ☐ Choose Delete Style from the Text menu and the Delete Style dialog box will appear.
- ☐ Choose the style you wish to remove from the Text Style menu and select the Delete button. To delete multiple Styles at the same time, Shift-click to select sequential Styles, or Command-click to select multiple nonsequential Styles.

Text Styles Menu

Text Styles allows you to select a style from a menu of styles you have created for your drawing.

To use Text Styles:

- ☐ Select a block of text.
- ☐ Choose Text Styles from the Text menu and a submenu will appear.
- ☐ Choose the desired text style.
- ☐ The text style you just specified will be assigned to the selected text in the drawing.

If no text is selected, the selected tool's default will change.

If both the Text tool and text in the drawing are selected, the drawing text will be changed.

Text in the drawing takes precedence and the Text tool defaults remain unchanged.

References Menu



The References menu contains items pertaining to referencing other drawings, allowing you to add a reference (other drawing files) to your master drawing. Any PowerCADD drawing or any file type PowerCADD can open (e.g. a PhotoShop file may be referenced if the Graphics Export External is loaded) may be referenced.

You can place references in any drawing, including multiple references of the same drawing.

When a drawing is added as a reference, it reloads the referenced drawing from the original as needed. References allow you to work with other drawings without permanently adding them to the master drawing.

Referencing Key Concepts

The biggest advantage to using references is having the most current information without having to redraw content for use in another file or having to manually update drawings sharing the same content via copy and paste or other techniques.

To fully understand how referencing works and how it might benefit your workflow we can break it down into two fundamental components:

- ❑ Master Drawing: A destination into which we put content. In this case the content may be objects we draw from scratch or objects which come from a linked, outside source.
- ❑ Reference Drawing: A source of content which is placed into a master drawing, thereby establishing a dynamic link between the Master (destination) and Reference (source) drawings.

A master drawing can be composed of many referenced drawings. When a referenced drawing is modified and saved, the master drawing is automatically updated with the new content.

In a simple example, a reference drawing might be a project title block drawing containing standard elements such as project name, your company name, and other standard graphic elements.

A Master Drawing might be an electrical plan, and another master drawing may be a reflected ceiling or site plan. Each Master Drawing can reference the same title block drawing. Using References, a change made to the title block referenced drawing (e.g. adding a client logo to the title block) will automatically appear in each Master Drawing.

In a more complex example, a master drawing for a motel might be composed of many referenced drawings representing the entire project ranging from different suites, or perhaps different drawings representing

plumbing, electrical, and site development. There may be many people working on the project, each with different directives.

By using referencing, you can reference any or all of their drawings and consolidate the information into one master drawing. When any of the original referenced drawings are modified, the master drawing will also be modified reflecting any changes.

Place Reference

Place Reference allows you to place a reference to another file in your master drawing which can be updated automatically without redrawing.

To place a reference drawing in your master drawing:

- ☐ Choose Place Reference from the References menu. The standard MacOS Open dialog appears allowing you to navigate and locate your drawing.
- ☐ Select the drawing you wish to use as a reference.
- ☐ Click Open and the following dialog box appears:



- ☐ Apply the desired setting choices and click Place. Each choice is described later in this section.
- ☐ A preview of the referenced drawing appears along with a move cursor in the top left-hand corner. By default the preview matches the extents of the data in the referenced drawing. The displayed extents may be modified by masking in the master drawing after it is placed.
- ☐ Drag to position the top left corner and click the mouse to place the referenced drawing.
- ☐ Snapping may be used to insert the top left corner in a precise location.
- ☐ A new layer titled “References” is automatically created.



CAUTION: Changing the layer status of the “References” layer to hidden (Off) will also automatically hide all referenced objects on all other layers.



TIP: If you are placing multiple references into the same master drawing, the first reference placed sets the master reference origin. Placing additional referenced drawings will automatically use the master reference origin. Each referenced drawing may be moved independently after it is placed if different origins are required.

The **Scale to Destination Drawing Scale** checkbox will automatically scale the reference drawing to the scale of the master drawing. If left unchecked, the reference drawing will retain its scale as set in the original drawing.



CAUTION: The Layer Attributes settings for Opacity, Color, and Scale are not transferred when referencing drawings into a master drawing. Line weights, dash patterns, and text sizes are not scaled.

Reference Layer Settings

Layer Settings control how the layers on the reference drawing are handled by your master drawing.

Referenced Layers

All Layers will place all layers from the reference file into the new drawing. The layers will be placed as they are set up in the master drawing. If there are more layers in the reference drawing than in the master drawing, additional layers will be added to the master drawing as needed. As an example, if you have ten layers in your master drawing and ten layers in your reference drawing, as long as the name of the layer is the same in both drawings, each layer will be merged into a single layer.

If the names of the layers are different, new layers will be added to your master drawing to accommodate the reference drawing's layers.

Visible Only will place only visible layers from the reference drawing into the master drawing. If the layer names match in the master and referenced drawings, the content is merged on to the common layer. If layer names are different new layers are added to the master drawing.

Layer Name Prefix

No Prefix inserts new layers into the master drawing using the same name as appeared in the referenced drawing. If the layer name in the reference and master drawings are the same, the data from both drawings is combined onto the same layer; referenced data on a common layer cannot be selected.

“Referenced Drawing 01.PC9” is a dynamic field where the name of this field will change to match the name of the drawing file you are referencing. Choosing this radio button will append the name of the file as it appears in the dialog to the start of every layer referenced into the master drawing.

Custom allows you specify what prefix should be appended to layer names from the referenced drawing into the master drawing. If the layer name in

the reference and master drawings are the same, the data from the referenced drawing appears on its own layer with the custom prefix appended to the start of the layer name.



TIP: If you have common layer names in the referenced and master drawing, and you want the data to appear on different layers when referenced, add the drawing name or a custom layer name prefix. The data will be referenced into an appropriately named new layer.

Update All

Update All will reload all of the reference drawing content. If you know the references in your drawing have changed, or you want to ensure you have the most recent content, choose Update All from the References menu.

Update Selected

Update Selected allows you to specify which referenced drawing should be updated.

To update a specific Referenced Drawing:

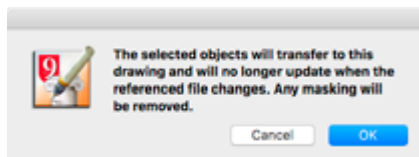
- ☐ Select the Referenced drawing as described in Working With References later in this section.
- ☐ Choose Update Selected from the References menu.
- ☐ The contents of the selected reference are updated in the master drawing.

Bind To Drawing

Bind To Drawing allows you to permanently add (bind) a referenced drawing to your master drawing. When you bind a reference drawing, you are disconnecting the reference and you will no longer be able to update the master drawing when you change the reference drawing. When saving your drawing as .dwg or DXF, the references are automatically bound.

To bind a reference:

- ☐ Select the reference drawing as described in Working With References later in this section.
- ☐ Choose Bind To Drawing from the References menu.
- ☐ The following dialog will appear:



- ☐ Click OK and the reference is simply disconnected and the objects are placed into the master drawing permanently.

When saving drawings as DXF or DWG, binding is carried out automatically for you.

Masks

Rectangle and Polygon Masks allow you to hide (mask) part of a Referenced Drawing after it has been placed in the Master Drawing. Masking in effect provides a window, or view port, into just the portion of the referenced drawing you want displayed.



For information on Selecting References see [Working With References](#) described later in this section.

Rectangle Mask

To create a rectangular mask that hides part of a referenced drawing:

- ☐ Select the reference drawing; a bounding box appears around the selected reference.
- ☐ Choose Rectangle Mask from the Reference menu.
- ☐ The cursor changes to a cropping cursor.
- ☐ Click and drag inside the bounding box of the selected reference.
- ☐ When you release the mouse, the view port you defined by clicking and dragging is displayed in the drawing window.

To modify a rectangular mask hiding part of a referenced drawing:

- ☐ Select the reference drawing; a bounding box appears around the selected reference.
- ☐ Choose Rectangle Mask from the Reference menu.
- ☐ The cursor changes to a cropping cursor and the extents of the referenced drawing are displayed.
- ☐ Click and drag inside the bounding box of the selected reference to define the new view port. To see the entire referenced drawing (no masking) click anywhere outside the bounding box.

To create a polygon mask that hides part of a referenced drawing:

- ☐ Select the reference drawing; a bounding box appears around the selected reference.
- ☐ Choose Polygon Mask from the Reference menu.
- ☐ The cursor changes to a cropping cursor.
- ☐ Click and drag inside the bounding box of the selected reference to define the first segment of the polygon mask.
- ☐ Move the cursor and click to define each subsequent segment of the polygon mask. Double-click when complete.
- ☐ Click anywhere outside the bounding box to complete the operation.

To modify a polygon mask hiding part of a referenced drawing:

- ☐ Select the reference drawing; a bounding box appears around the selected reference.
- ☐ Choose Polygon Mask from the Reference menu.
- ☐ The cursor changes to a cropping cursor and the extents of the referenced drawing are displayed.
- ☐ Click and drag inside the bounding box of the selected reference to define the new view port as described earlier. To see the entire referenced drawing (no masking) click anywhere outside the bounding box.

Working With References

Referenced Drawings behave like typical PowerCADD objects as they must first be selected before they can be manipulated. Once selected a reference may be manipulated using features found in the PowerCADD menus. For example, referenced drawings may be rotated, moved, deleted, grouped, or mirrored just like any other PowerCADD object.

Objects in a referenced drawing cannot be edited but may be used as references for snapping to create new objects. To make changes to the referenced objects simply open the original referenced drawing and make the desired changes.

To select a Referenced Drawing:

- ☐ In the Layer Window, ensure the References Layer has a layer status as the active layer.
- ☐ Click on one of the objects in the Referenced Drawing to select that reference.



TIP: If the References Layer is not the active layer, make sure Edit All Layers is On. Then click on an object in the Referenced Drawing to select the reference.

To hide a Referenced Drawing:

- ☐ Turn off the References Layer or turn off all layers associated with that reference.

Manipulating References After a reference drawing is placed in a master drawing it may be manipulated using menu items just like any other PowerCADD object. Typical manipulations of a reference drawing include moving, scaling, or rotating a reference after it has been placed in a master drawing.

To manipulate a referenced drawing:

- ☐ Select the reference drawing; a bounding box appears around the selected reference.
- ☐ Choose the desired menu item. For example to rotate a reference choose Rotate from the Arrange menu.
- ☐ Complete the steps required for the chosen menu.

Removing a Reference

You may wish to permanently remove a reference drawing from the master drawing.

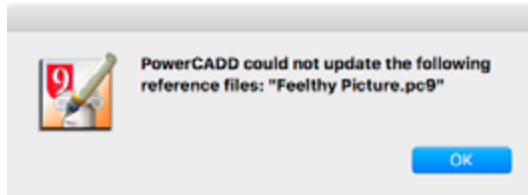
To remove a reference:

- ☐ Select the reference you wish to remove.
- ☐ Press the Delete key or choose Clear from the Edit menu.
- ☐ The reference drawing will be deleted.
- ☐ To remove any empty layers associated with the deleted referenced drawing press Purge in the Layers Window.

See References Window (WTF?) in the Window section of this chapter for additional information on managing references.

Missing References

While PowerCADD always attempts to locate referenced drawings for you, there may be times when a referenced drawing cannot be automatically located. If a referenced drawing can't be found you will be prompted when the master drawing is loaded for the name of the missing file:



In most cases, provided the referenced drawing is on the same hard drive (be it a local hard drive or a server), PowerCADD will typically find a referenced drawing even it has been moved or renamed. However, in those rare circumstances where a referenced file can't be found you will need to manually reset the link to the reference drawing.



TIP: If you are in a network environment where drawings are saved and opened off a server and across the network it is important all your drawings be on the server BEFORE placing your references.

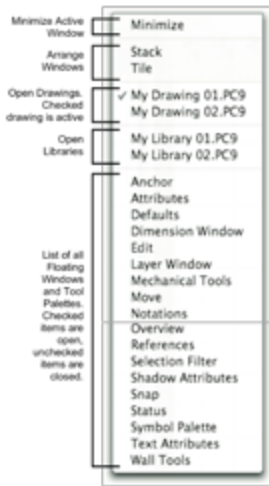
To reset a link to a missing reference:

- ☐ Select the reference you wish to remove.
- ☐ Press the Delete key or choose Clear from the Edit menu.

The reference drawing will be deleted and its name will be removed from the Reference Window (WTF?). As an optional step, you can press Purge in the Layers Window to remove all empty layers before placing the reference again.

- ☐ Place the reference again using the steps described earlier.

Window Menu



Minimize/Stack/Tile

The Window menu gives you a method of managing multiple open documents, along with the ability to stack or tile your documents. This option will allow you to bring any document currently open to the front in an active state.

The Window menu is also used to bring Symbol Library windows to the front or turn any loaded floating window or tool palette On or Off.

If you have many documents or Symbol Libraries open at once, each will be represented by name in Window menu. To bring a document to the front, choose Window and choose the document you wish to view. When the selection is made, that document will be brought to the front in an active state and a checkmark will appear next to its name in the Window menu.

Minimize collapses the active drawing window to the MacOS Dock.

Stack will display all open drawing windows in a staggered arrangement so the title bar of each drawing window is visible. This allows you to very quickly locate and select a drawing which is behind other drawings.

Tile will arrange the open drawing windows so a portion of each drawing is visible, simultaneously. If two drawings are open, the screen will be divided horizontally by two; if three drawings are open, the screen will be divided by three, and so on. The scroll bars for each window are also visible so you can adjust the view of each drawing. This arrangement will make copying and pasting objects from one drawing to another a quick process because you can see all the drawings involved at the same time.

An open floating window or tool palette will have a checkmark next to its name. To open or close a floating window or tool palette, choose Window and choose the tool palette or floating window name from the list. These items may also be opened or closed using the Anchor Window or clicking on a palette or window close button.

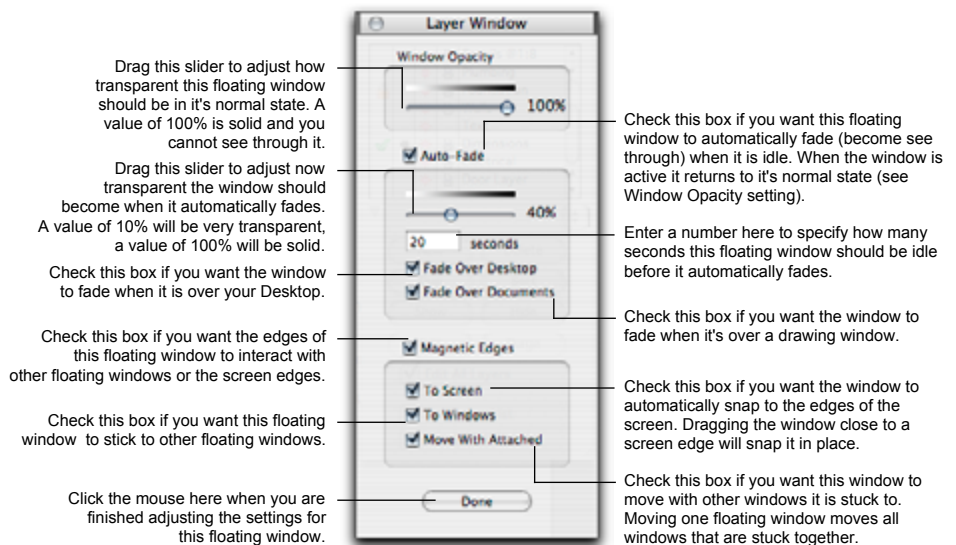
Floating Windows and Tool Palettes

Floating Windows and Tool Palettes provide direct access to various features and are managed from either the Anchor Window or selecting an item from the Window menu.

When you activate a floating window or tool palette, either by clicking on a window name in the Anchor window or selecting it from the Window menu, it is opened in the last saved position. When a floating window or tool palette is closed its name appears in the Anchor Window. When open, floating windows and tool palettes are not listed in the Anchor Window.

To customize the settings of a floating window:

- ❑ Press the Control key and click anywhere in a blank area of a floating window. The following dialog will appear:



- ❑ Make the desired setting changes and click Done.



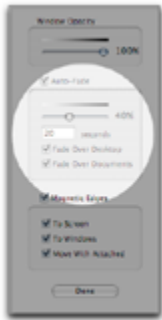
TIP: The settings for tool palettes may also be customized with the same options as described above for floating windows. To customize the settings of a tool palette, Control- or right-click on the tool palette, and choose Palette Settings from the contextual menu. The settings options for the specific tool palette will appear.

The settings for each floating window and tool palette may be adjusted independently.



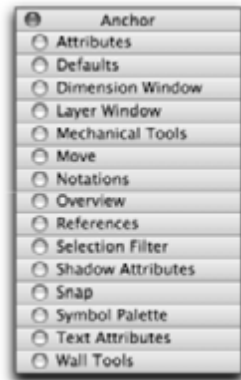
NOTE: The normal state for a tool palette is determined by the Window Opacity setting. A tool palette that has faded on screen will automatically return to its normal state when you click in the palette to activate it.

The ability to turn Auto-Fade On or Off for a tool palette or floating window is controlled by the Window Fading Preference settings. If the Auto-Fade settings are dimmed as shown on the left:



- ☐ Choose Preferences from the PowerCADD menu and select the General tab.
- ☐ Check the Window Fading box to turn it on and click OK.
- ☐ Set the Auto-Fade settings for the floating window or tool palette again as described earlier.

Anchor Window



The Anchor Window contains a list of all closed tool palettes or floating windows and may be positioned anywhere on screen.

To open a floating window or tool palette from the Anchor Window, choose Anchor from the Window menu and select the button next to the items name or click on the items name.

When a floating window or tool palette is open, its name is removed from the Anchor Window list.

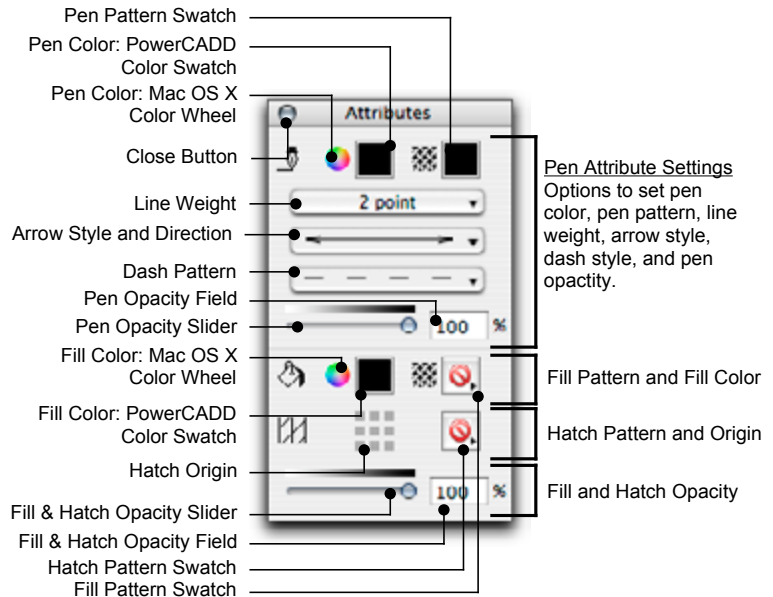
To close the Anchor Window, click the button in the top left corner or choose Anchor from the Window menu. Opening or closing the Anchor Window does not affect any open floating windows or tool palettes.

Attributes Window

The Attributes Window is a floating window which allows you to change the drawing attributes of specific tools or all tools available in the Tool palette quickly and easily. You may also use the Attributes Window to change the characteristics of existing objects just as you do using the Attribute menu.

To activate the Attributes Window:

- ❑ Choose Attributes from the Window menu or select Attributes in the Anchor Window. The following floating window will appear:



The Attributes Window will display the current defaults of any selected tool or any selected object in the drawing. As you change tools, you will also notice the Attributes Window display will change to the selected tool defaults. You may change the defaults, if they are not correct. A checkmark appears next to the currently selected attribute setting in a given list. The Attribute menu, Contextual menu, and Attribute window all share the same attribute options.



CAUTION! When Global Attributes is On, values displayed for Pen Pattern, Line Weight, Pen Color, Fill Color and Fill Pattern in the Attributes Window are affected. See [Global Attributes](#).

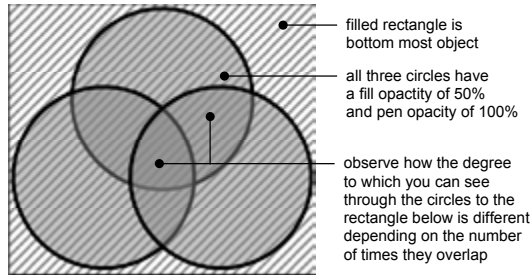
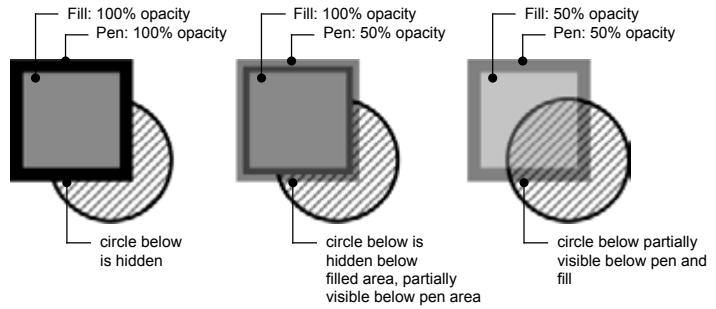
- ❑ Click on the MacOS Color Wheel to choose a color using the MacOS color panel. Pen Color and Fill Color may be adjusted independently by clicking on their respective color wheel.
- ❑ Click on the PowerCADD Color Swatch to choose a color from the PowerCADD color palette. Pen Color and Fill Color may be adjusted

independently by clicking on their respective color wheel.

- ❑ Click on the Pen Pattern Swatch to choose a pen pattern from the Pen Pattern palette.
- ❑ Click on the Line Weight pop-up menu to choose a line weight from the Line Weight palette.
- ❑ Click on the Arrow Style and Direction pop-up menu to choose an Arrow Style and Direction.
- ❑ Click on the Dash Pattern pop-up menu to choose a dash pattern.
- ❑ Click and drag on the Pen Opacity Slider to adjust how transparent ('see-through') an the Line Weight and Pen Pattern should be. As you drag the slider, the value in the Pen Opacity Field will change. You may also type a specific whole number ranging from 0 to 100. Press Tab to cycle between the Pen and Fill Opacity fields.
- ❑ Click and drag on the Fill & Hatch Opacity Slider to adjust how transparent (see through) an objects Fill or Hatch Pattern should be. As you drag the slider, the value in the Fill & Hatch Opacity Field will change. You may also type a specific whole number ranging from 0 to 100. Press Tab to cycle between the Pen and Fill Opacity fields. Fill and Hatch opacity cannot not be independently adjusted.
- ❑ Adjusting an Object's Pen or Fill opacity determines to what degree information is hidden or revealed below the Object. For example, an Object opacity of 100% hides all information below the Object. An Object with 10% opacity would appear almost transparent, allowing you to see through to objects below. The effects of overlapping two Objects with adjusted opacity is cumulative. An object's Pen Opacity and Fill Opacity may be adjusted independently. When an object has a Shadow, adjusting the Fill opacity of the Object will allow you to see through to the Shadow and other objects below.



NOTE: The Pen and Fill opacity sliders are linked if the selected object is a Group. Adjusting Fill opacity also adjusts Pen opacity. To adjust the Pen or Fill opacity of a group independently, use the Edit Groups command and then make the desired adjustments.



- ❑ Click on the Fill Pattern Swatch to choose a pattern from the Fill Pattern palette. Open and closed objects may have fill patterns assigned. Assigning a fill pattern to a line will have no change on the lines attributes, use a Pen Pattern instead.
- ❑ Click on the Hatch Pattern Swatch to choose a pattern from the Hatch Pattern palette. Open and closed objects may have hatch patterns assigned. An object may have both a hatch pattern and a fill pattern assigned; the hatch pattern will be on top of the fill pattern.
- ❑ Click on the Hatch Origin icon to select the starting point (origin) for a hatch pattern. Each square in the icon is a hot point corresponding the hatch origin location. For example, clicking on the center square would set the hatch origin to Middle (MM). Setting the hatch origin with the Hatch Origin icon is the same as choosing Hatch, Hatch Origin from the Attribute menu.



See [Hatch Settings in the Attribute menu](#) for information on how to adjust the line weight, pen color, scale and origin of hatch patterns.

Setting Tool Attributes

You may assign specific attributes for a tool prior to drawing an object. All objects will have the same attributes until the tool attributes are changed or you edit the attributes of an existing object in the drawing.

To set the attributes for a drawing tool:

- ☐ Double-click on the tool to deselect all objects in the drawing.
- ☐ Select an Attribute Window item and choose an item from the associated list. For example, select the PowerCADD Pen Color Swatch and choose a color from PowerCADD Color Palette list.
- ☐ Repeat this process for each tool attribute you want to set prior to drawing an object.
- ☐ Start the drawing operation. The tool will create an object with the selected attributes.



NOTE: Layer Color Attribute settings will override a tool's Pen Color setting. For example, drawing on a layer with a red Layer Color Attribute will draw all objects with red lines regardless of the pen color specified in the Attributes Window.



TIP: Pressing the Shift key as you set a tool's attributes will set the attributes for all tools in the palette at the same time.

Changing an Object's Attributes

The attributes of any object may be changed after it has been drawn.

To change the attributes of an existing object:

- ☐ Select the object in the drawing whose attributes you want to change.
- ☐ Select the Attribute Window item you want to change and choose a new attribute from the associated list. For example, select the Fill Pattern Swatch and choose a new pattern from the list.
- ☐ The Attribute for the selected object is changed.

If the selected object was a group, then all objects in the group will take on the new attributes. The attributes for items inserted from Symbol Libraries as Symbols cannot be changed after they are inserted into a drawing. To change the attributes of a Symbol, create or edit a new symbol and use Replace Symbol in the Tool menu to update your drawing.



TIP: Use the same process to change the attributes for multiple objects at the same time. Simply select all the objects you want to change before making a selection in the Attributes Window.

See the [Attribute menu](#) for information on working with object and tool attributes.

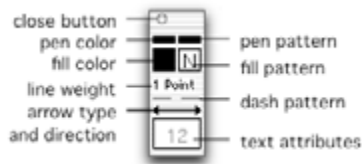
Defaults Window

The Defaults Window allow you to change the drawing defaults of specific tools or all tools available in the Tool palette quickly and easily. You may also use the Defaults Window to change the characteristics of existing objects just as you do using the Attribute menu.

The Defaults Window will allow you to change pen color, pen pattern, fill color, fill pattern, line weight, line dash type, line arrow type, and text defaults without entering the menus.

The Defaults Window is a floating window which may be moved and turned off just as the Edit Window and the Tool palette are.

To activate the Defaults Window, choose Defaults from the Window menu or select Defaults in the Anchor Window. The following window will appear:



To change pen defaults:

- ❑ Select the tool you wish to change by double-clicking on it to insure no objects are selected. If objects are selected, the objects will take on the changes instead of the tool.
- ❑ The two small rectangles at the top of the Defaults Window govern the pen defaults. The left rectangle specifies the color of the pen while the right rectangle specifies the pen pattern.
- ❑ Press the left rectangle and a color palette will pop-out allowing you to choose a suitable color. With the color palette open, drag to the desired color and release. The color you select will appear in the Defaults Window.
- ❑ Press the right rectangle and a pen pattern palette will pop-out allowing you to choose a suitable pen pattern. With the pen pattern palette open, choose the desired pen pattern and release. The pattern you select will be displayed in the Defaults Window.

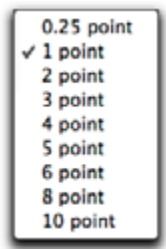
The pen default rectangles will display the current defaults of any tool selected. As you change tools, you will also notice these rectangles will change to the selected tool defaults. You may change the defaults, if they are not correct.

To change fill defaults:

- ☐ Select the tool you wish to change by double-clicking on it to insure no objects are selected. If objects are selected, the objects will take on the changes instead of the tool.
- ☐ The two squares located just below the pen default rectangles govern fill pattern defaults. The left square specifies the color of the fill, while the right square specifies the fill pattern.
- ☐ Press the left square and a color palette will pop-out allowing you to choose a suitable fill color. With the color palette open, select the desired color and release. The color you select will appear in the Defaults Window.
- ☐ Press the right square and a fill pattern palette will pop-out allowing you to choose a suitable fill pattern. With the fill pattern palette open, select the desired pattern and release. The pattern you select will be displayed in the Defaults Window.
- ☐ The fill defaults have now been modified and these changes will remain in effect until you change them again.

The fill default squares will display the current defaults of any tool selected. As you change tools, you will also notice these squares will change to display the selected tool defaults. You may change the defaults if they are not correct.

On black and white machines, the fill default color will be specified in the Defaults Window by a number. This number corresponds to the location of the selected color in the fill color palette.



To change the line weights:

- ☐ Select the tool you wish to change by double-clicking on it to insure no objects are selected. If objects are selected, the objects will take on the changes instead of the tool.
- ☐ The current line weight is displayed just below the fill default squares.
- ☐ Press the text below the fill defaults and the menu on the left will pop-up.
- ☐ Choose the correct line weight and release.
- ☐ The line weight you select will appear in the Defaults Window and will remain in effect until you change it.

The line weight default will display the default settings for the tool currently selected. If two tools have different defaults, the line weight display will change as you select between the tools.



Dash menu

To change dashed line types:

- ☐ Select the tool you wish to change by double-clicking on it to insure no objects are selected. If objects are selected, the objects will take on the changes instead of the tool.
- ☐ The current dash default is displayed just below the line weight display.
- ☐ Press the default line below the line weight default and the dash menu will pop-up.
- ☐ Choose the correct dash type and release.
- ☐ The dash selected will appear in the Defaults Window and will remain in effect until you change it.

The dash default line will display the default settings for the tool currently selected. If two tools have different dashing, the dash line will change as you select between the tools.

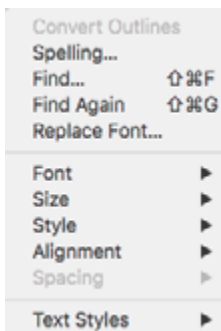


Arrow menu

To change arrow line types:

- ☐ Select the tool you wish to change by double-clicking on it to insure no objects are selected. If objects are selected, the objects will take on the changes instead of the tool.
- ☐ The current arrow default is displayed just below the dash default line display as a default line.
- ☐ Press the default line below the dash default line and the Arrow menu will pop-up.
- ☐ Drag to the correct arrow line and release. The four arrow placement lines at the top of the menu specify if the arrow will be placed at the start of the line, the end of the line, or on both ends of the line.
- ☐ Return to the arrow menu, drag to the desired arrow type, and release.
- ☐ The arrow you select will appear in the Defaults Window and will remain in effect until you change it.

The arrow default line will display the default settings for the tool currently selected. If two tools have different defaults, the arrow line will change as you select between the tools.



Text menu

To change text defaults:

- ☐ Select the tool you wish to change by double-clicking on it to insure no objects are selected. If objects are selected, the objects will take on the changes instead of the tool.
- ☐ The current text defaults are displayed in the square at the bottom of the Defaults Window. Inside the box will be a number corresponding to the text size. This number will be displayed in the current font setting and will contain all style settings such as italic, outline, or bold.
- ☐ Press the text default square and the Text menu will pop out.
- ☐ With the menu open, choose Font and a second pop-up menu will appear. Choose the desired font and release. The selected font will be displayed in the text default square.

- ❑ Return to the menu by pressing the text default square and drag to Size and a second menu will appear. Drag to the desired text size and release. The number you selected under Size will be displayed in the text default square.
- ❑ Continue returning to the text default menu and making default changes until all of the desired settings have been made. These settings will remain in effect until you change them again.

Tools with different default settings will display different defaults as they are chosen.

The Defaults Window may be used to change existing objects as well. To change an existing object, select the object and make default changes, as explained before. Since the objects are selected, the default changes will affect the objects instead of the active tool.

Defaults for all tools in the Tool palette may be changed at one time.

To change the defaults of all tools:

- ❑ Select any tool by double-clicking on it to insure no objects in the drawing are selected. If objects are selected, the objects will take on the changes instead of the tools.
- ❑ Press Shift and make any changes necessary.

The changes you make while pressing the Shift key will affect all tools in the Tool palette. Every tool in the Tool palette will now have identical default settings.

Dimension Window

The Dimension window allows you to access dimension options from a floating window. Options for Dimensioning tools and existing dimensions can be set in this window.

To activate the Dimension window, choose Dimension from the Window menu or select Dimension in the Anchor Window. The following window will appear:



Text Field allows you to place a prefix or suffix in the dimension text, and to edit any text in this field, including dimension text.



Text Placement allows you to change text placement relative to the dimension line.



Leader Position (for circular dimensions and existing leaders with text) allows you to change the direction of the final leader.



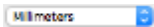
Show Units allows you to turn unit abbreviation On or Off.



Arrow Position allows you to set arrows inside or outside of extension lines.



Rotate Text allows you to set the dimension text to remain horizontal regardless of the dimension line's angle or to be parallel to the dimension line.



Units allow you to specify the standard of metric or imperial units of measure.



Accuracy allows you to specify the smallest unit of measure.



Tolerance allows you to specify if a dimension will have a dimension only, a dimension with a single plus or minus tolerance, a dimension with a pair of tolerances, or a pair of limiting dimensions.



Draw Horizontal Bar Fractions allows you to specify how fractional dimensions are displayed.



NOTE: The Dimension Window External must be loaded with PowerCADD to be active. See [Loader Preferences](#).

If you wish to set the options for a Dimensioning tool itself, double-click on the tool before changing any option. Selecting an existing dimension first will change only the selected dimension.

Editing Dimension Text

To add characters to dimension text in the Dimension window:

- ☐ Select an existing dimension.
- ☐ Click the Text field and enter additional characters before or after the up-carets(^). Any text associated with the dimension will appear exactly as you type it. If the carets are removed and text is typed in, the typed text will be displayed in place of the dimension.
- ☐ Press the Enter key or click in the drawing area to enter the text.

You can replace a numerical dimension with a label like *width* or *size*, etc. Select the existing dimension and delete all of the dimension text characters in the text field. Enter the label and press the Enter key.

Text Placement

To set the Text Placement in the Dimension window:

- ☐ Select a linear dimensioning tool or an existing linear dimension.
- ☐ Click Text Placement button to cycle through the placement options. The “x” in the icon will indicate the text position each time the button is clicked. Text may be placed either centered above, below, or on a dimension line or to the right, or left of the dimension lines.

If an existing dimension is selected before the text’s placement is edited, the dimension text will reflect the changes interactively as the Text Placement button is clicked.

Leader Position

To set the Leader Position in the Dimension window.

- ☐ Select a Circular Dimension tool, an existing circular dimension, or an existing leader with text.
- ☐ Click the Leader Position button to cycle to the desired option.

The icon will reflect the change, either left or right.

If an existing circular dimension or leader with text is selected, the leader position will reflect the changes interactively as the button is clicked.

Show Units

To set Show Units using the Dimension window:

- ☐ Select a Dimension tool or an existing dimension.
- ☐ Click the Show Units button to cycle to the desired option.

The icon will interactively reflect the change.

If an existing dimension is selected, the unit abbreviation will appear or disappear as the button is clicked.

The Show Units option will only be highlighted when the appropriate dimension units are being used. Units based on feet and inches or fractional inches will automatically display the foot (') and inch (") marks. Cartesian units will automatically display degrees (°).

Arrow Position

To set the Arrow Position option using the Dimension window:

- ☐ Select a Dimension tool or an existing dimension.
- ☐ Click the Arrow Position button to cycle to the desired option.

The icon will reflect the change.

If an existing dimension is selected, the arrow position will change interactively as the button is clicked.

Rotate Text

To set the Rotate Text option using the Dimension window:

- ☐ Select a Dimension tool or an existing dimension.
- ☐ Click the Rotate Text button to cycle to the desired option.

The icon will reflect the change.

If an existing dimension is selected, the text rotation will change interactively as the button is clicked.

Units

To set the Units option using the Dimension window:

- ☐ Select a Dimension tool or an existing dimension.
- ☐ Choose the desired unit of measurement from the Units pop-up menu.

The menu will display the units.

If an existing dimension is selected, the unit of measure will be reflected in the dimension.

Accuracy

To set the Accuracy option using the Dimension window:

- ☐ Select a Dimension tool or an existing dimension.
- ☐ Choose the desired degree of accuracy from the Accuracy menu. The accuracy will display as fractions or decimal places depending on how units are set.

The Accuracy menu will display the accuracy.

If an existing dimension is selected, the degree of accuracy will be interactively reflected in the dimension.

Tolerance

To set the Tolerances using the Dimension window:

- ☐ Select a Dimension tool or an existing dimension.
- ☐ Choose the desired tolerance from the Tolerance menu. If One is selected, one numerical field will appear at the bottom of the palette. Selecting Two or Limits will provide two numerical fields.
- ☐ Click a numerical field and enter the desired tolerance.
- ☐ Press the Return key or click in the drawing window.

If an existing dimension is selected, the tolerance changes will be reflected in the dimension when the Enter key is pressed.

Choosing None will retract all tolerance numerical fields. Clicking the toggle switch in the top right corner of the Dimension window's title bar will display or hide the Tolerance numerical fields.

Fractions

To set how fractions are displayed:

- ☐ Select a Dimension tool or an existing dimension.
- ☐ Select the Horizontal Bar Fractions Icon. You may toggle between the two display options for fractions by clicking on the icon again.



Example when Horizontal Bar Fractions not checked.



Example when Horizontal Bar Fractions is checked on.

Edit Window

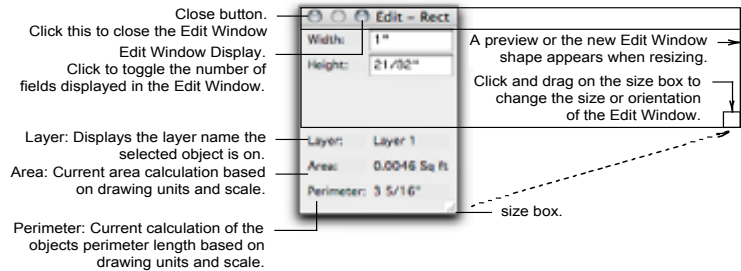


The Edit Window allows you to specify an object's dimensions and provides real-time display of an objects area and perimeter as it is being drawn.

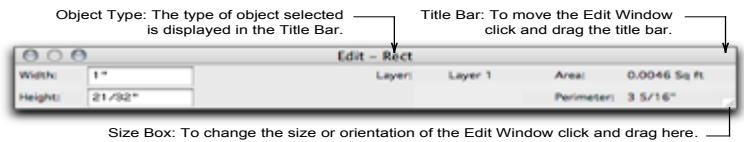
NOTE: When a bitmap image is selected the Edit Window will display the DPI (dots per inch) and bit depth (number of colors).

To activate the Edit Window,

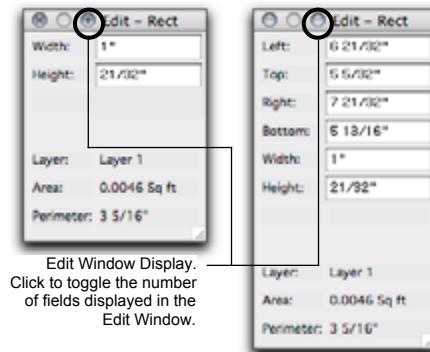
- ❑ Choose Edit from the Window menu or select Edit Window in the Anchor Window. The following dialog will appear.



The Edit Window may be resized by dragging its size box in the lower right hand corner.



The Edit Window Display can be modified to change the number of fields shown. The specific fields will vary with the type of object selected.



To change the size of an existing object using the Edit Window:

- ☐ Select the object you wish to edit.
- ☐ Press the Tab key to cycle through the edit fields or click the mouse in the desired field.
- ☐ Enter the specified value or enter a mathematical equation.
- ☐ Click the mouse in the drawing window or press the Return key.

The object will reflect the new values.

To specify the size of an object while drawing using the Edit Window:

- ☐ Select a drawing tool and begin to draw an object.
- ☐ Before you complete the draw operation, press the Tab key.

The drawing operation will be suspended and the Edit Window will activate.

- ☐ Enter the specified value or enter a mathematical equation in the desired Edit Window field.
- ☐ Click the mouse in the drawing window or press the Return key.

The object is drawn to the specified values.

Layers Window

The Layers Window allows you to access layer options from a floating window.

When the Layers Window External is loaded, it can be accessed by the Layer command in the Layout menu or through the Anchor window. You can assign a Command key to the menu item, the default key command assigned is 'L'.

The complete description is located in the Layout menu earlier in this chapter.

We recommend that you always load the Layers Window External with PowerCADD.

Mechanical Tools

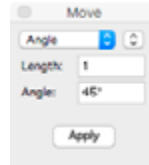
The Mechanical Tool palette provides specialized tools for mechanical drafting and design, extending the capability of PowerCADD for mechanical engineers.

The complete description is located in the Tool menu section of this manual.

To activate the Mechanical Tools Palette, choose Mechanical Tools from the Window menu or select Mechanical Tools in the Anchor Window.

Move Window

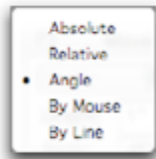
The Move Window provides an alternate Move dialog, which floats on your drawing for convenience while providing more defined control for your relative moves.



NOTE: The Move Window External must be loaded with PowerCADD to be active. See [Loader Preferences](#).

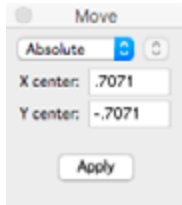
The Move Window can be put away by clicking on the close button in the upper left-hand corner. It will reappear the next time you choose Move from the Edit menu while pressing the Option key.

The Move Window provides a total of five move methods. In each case, select the objects to be moved, then choose the appropriate type of move from the pop-up menu.



- ☐ Absolute allows you to specify the absolute center X,Y coordinate location of an object.
- ☐ Relative moves objects the specified amount relative to their current X,Y location.
- ☐ Angle moves objects the specified length and angle.
- ☐ By Mouse moves objects dynamically using the mouse and dragging on screen.
- ☐ By Line moves objects based on the length and angle of a line you click in the drawing.

Absolute Moves



To move selected objects to an absolute position based on the center X and Y coordinates:

- ☐ Select the object(s) to be moved.
- ☐ Choose Move from the Edit or Window menu.
- ☐ Choose Absolute from the pop-up menu and enter a value in the X center and Y center fields.

You may enter mixed units (metric, or imperial), mathematical equations, and positive or negative values.

- ☐ Press the Return key or click Apply.

The objects are moved from their previous X, Y center coordinates to the new X,Y center coordinates.

The Move window gives you more control and more option with relative moves. Not only can you specify up (north), down (south), left (west) or right (east), you can also specify northwest, northeast, southwest, or southeast by clicking on the appropriate Direction buttons.

Relative Moves



Direction Button



previous move values

To move selected objects a relative amount:

- ☐ Select the objects to be moved.
- ☐ Choose Move from the Edit or Window menu.
- ☐ Choose Relative from the pop-up menu and enter a value in the Delta X, Delta Y fields.
- ☐ Click the Direction button corresponding to the direction you wish to move.

The objects will move the specified Delta X, Delta Y distances. Negative numbers do not modify the direction of the move.

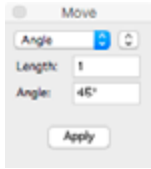
If you only have a move amount specified for Delta X, clicking any of the right- or left-hand Direction buttons will move your object horizontally, right or left.

If you only have a move amount specified for Delta Y, any of the top or bottom Direction buttons will move your object vertically, up or down.

The last eleven relative move amounts are remembered in the Previous Moves pop-up menu.

To reapply a saved relative move, select it from the Previous Moves pop-up menu. When more than eleven relative move amounts are used, the oldest amount (the one entered first) is removed from the bottom of the list and the newest (most recently entered) amount is added to the top of the list.

Angle & Length Moves



To move selected objects to a specified length and angle (direction):

- ☐ Select the objects to be moved.
- ☐ Choose Move from the Edit or Window menu.
- ☐ Choose Angle from the pop-up menu and enter a value in the Length and Angle fields.

You may enter mixed units (metric, or imperial), mathematical equations, or positive or negative values for lengths.

Angles may be entered in decimal degrees, or degrees minutes and seconds in either positive or negative values.

- ☐ Press the Return key or click Apply.

The objects are moved the specified length and angle.



TIP: When entering angular values in degrees, minutes and seconds, use spaces to separate the values to speed up data entry. For example, to specify an angle of 30° 0' 25" simply enter 30 0 25 (insert a space between each value).



previous move values

The last eleven length and angle move amounts are remembered in the Previous Moves pop-up menu.

To reapply a saved angle and length move, select it from the Previous Moves pop-up menu.

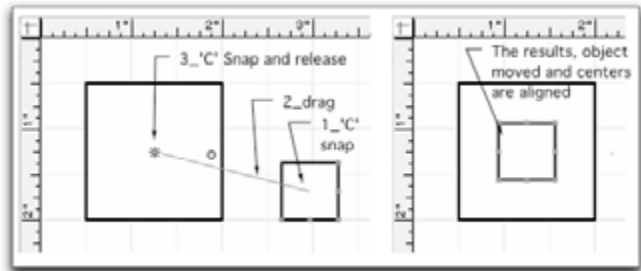
When more than eleven length and angle move amounts are used, the oldest amount (the one entered first) is removed from the bottom of the list and the newest (most recently entered) amount is added to the top of the list.

Move by Mouse



You can also use the mouse to specify the distance and direction of your move with the Move Window. When using this method of movement, you may use the snap keys to aid in the precision of the move.

In the illustration below, there are two rectangles. We want to move the center of the smaller rectangle to the center of the larger rectangle.



- ☐ Select the smaller rectangle.
- ☐ Choose Move from the Edit menu.
- ☐ Choose By Mouse from the pop-up menu.
- ☐ Click Apply and as you move to your drawing window, the cursor changes into a target cursor.
- ☐ Move your cursor to the small rectangle and press the C key for a center snap.
- ☐ Drag the cursor to the larger rectangle and press the C key for a center snap.
- ☐ Release the mouse button and the center of the smaller rectangle will move to the center of the larger rectangle as illustrated below.

Move by Line



To move selected objects based on the length and angle of a line in the drawing:

- ☐ Select the objects to be moved.
- ☐ Choose Move from the Edit or Window menu.
- ☐ Choose By Line from the pop-up menu.
- ☐ Move the mouse into the drawing window and the target cursor appears.
- ☐ Click on an existing line in the drawing.

The objects are moved to match the length and angle of that line.

TIP: Remember the length and angle of the move is determined by the length and angle of the line you click on. The angle of a line is determined by the direction you dragged when drawing it. Turning on [Hollow Line Handles](#) will show you the start and end point (direction) of a line when it is selected. The Edit Window will also display the length and angle of a selected line.

Notations Tool Palette

The Notation Tool palette provides specialized provides specialized tools for inserting various styles of bubble notes into a drawing.

The complete description is located in the Tool menu section of this manual. To activate the Notations Tools Palette, choose Notations from the Window menu or select Notations in the Anchor Window. The Notations External must be loaded, see [Loader Preferences](#).

Overview Window

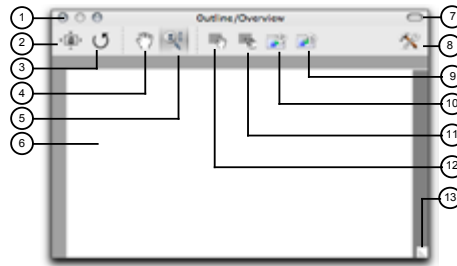
The Overview Window provides additional methods of navigating in the active drawing. Changes made in the main drawing window are immediately updated in the Overview Window.

The Overview Window renders the entire drawing, or portion of the drawing, in a drawing view. This high-performance drawing model makes it very efficient to navigate large drawings, or when switching between multiple open drawings.

Positioning the Overview window on a second monitor can make drawing navigation even more efficient.

To activate the Overview floating window

- ❑ Choose Overview from the Window menu or select Overview in the Anchor Window. The Overview Window will open and display the active drawing:



(1) Window Buttons: Click the left button to close the Overview window. Click the right-most button to maximize the size of the window to fit your screen. When a window is maximized clicking the same button will minimize the window to its previous size. The center button is always inactive.

(2) Fit Drawing: Click to perform a reduce to fit operation. The display in the Overview window is updated to show the entire drawing.

(3) Redraw: Click to force the contents of the Overview window to redraw.

(4) Pan Tool: Click to activate the Pan tool and the cursor in the Overview window changes to a hand. Press and drag in the Overview window to see a different area of the drawing. Pressing the Spacebar will also allow for immediate interactive panning regardless of which toolbar item is currently selected. The pan operation only affects the content displayed in the Overview Window, the main drawing view is not affected.

(5) Live Zoom: Click to activate the Live Zoom tool and the cursor in the Overview Window changes to a magnifying glass. Press and drag up to zoom in, press and drag down to zoom out. If you have a scroll wheel mouse, or a laptop touch pad, Live Zoom is always available regardless of which toolbar item is currently selected. The pan operation only affects the content displayed in the Overview Window, the main drawing view is not affected.

(6) Drawing Area: The primary Overview window drawing area displaying the currently active drawing. Switching between open drawings automatically updates the content in the drawing area.

(7) Toolbar Button: Click this button to show or hide the Overview Toolbar.

(8) Customize: Click to display a drop-down panel containing items to customize the toolbar. You may add or remove items, change their location, or change the display size of icons in the toolbar. Pressing Control and clicking on the toolbar will display a contextual menu to change the Window Opacity, Auto-Fade and Magnetic Edge properties of the window.

(9) Send View: Click to set the view in the current active drawing to match the current view in the Overview Window.

(10) Get View: Click to set the view in the Overview Window to match the current view in the active drawing.

(11) Draw View: Click to activate the Draw View tool and the cursor in the Overview Window changes to a crosshair. Click and drag in the Overview Window to define the area to be displayed in the main drawing window.

(12) Pan View: Click to activate the Pan View tool and the cursor in the Overview Window changes to a move cursor. Click and drag on the Draw View rectangle in the Overview Window to pan; the active drawing area will update in real time as the Pan View tool is used in the Overview window.

(13) Resize box: Click and drag to make the window larger or smaller.

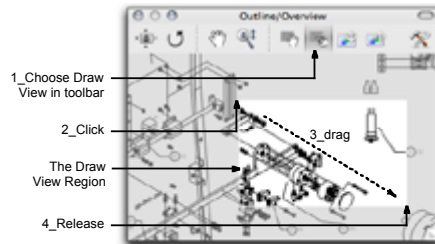


TIP: To see short tool tips, let your mouse hover over a toolbar icon. To see extended tool tips, press the Command key after the short tool tip has appeared.

Zooming in a Drawing

To zoom in your drawing using the Overview window.

- ☐ Click on the Draw View icon in the toolbar.
- ☐ Move the cursor into the Overview Window, the cursor changes to a crosshair cursor.
- ☐ Press and drag to define an area in the Overview Window.
- ☐ The main drawing window will zoom in and pan to display the defined area.



Zooming in the Overview Window

To change the zoom level displayed in the Overview Window:

- ☐ Click on the Live View zoom icon in the toolbar.
- ☐ Move the cursor into the Overview Window, the cursor changes to a zoom cursor.
- ☐ Click and drag up to zoom in, or click and drag down to zoom out.
- ☐ The area of the drawing displayed in the Overview will change interactively.

Changing the zoom level in the Overview Window does not change the view displayed in the main drawing window. If you have a scroll wheel mouse, or use a laptop touch pad with Trackpad Gestures enabled, live zoom is always available when the cursor is in the Overview Window.

Panning in a Drawing

To pan in your drawing using the Overview Window:

- ☐ Click on the Pan View tool in the toolbar.
- ☐ Move the cursor into the Overview Window, the cursor changes to a Move cursor.
- ☐ Press and drag on the Draw View region to change the view in the active drawing window.

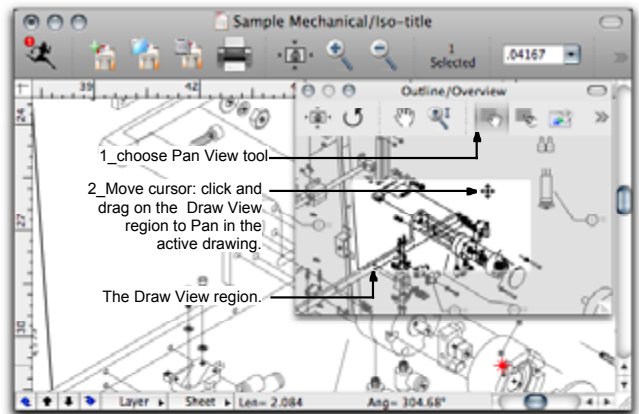
Using the Overview Window to pan the drawing view will change only the drawing window view. If you can't see enough drawing area in the Overview Window to pan to the desired drawing area, resize the Overview Window or pan in just the Overview Window (described below) to see more of the drawing.

Panning in the Overview Window

To pan in the view shown in the Overview Window:

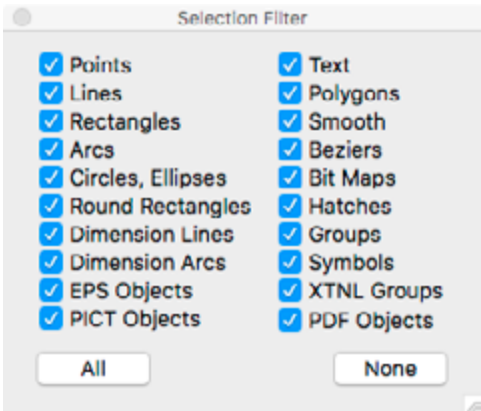
- ☐ Click on the Pan tool in the toolbar or press the Spacebar.
- ☐ Move the cursor into the Overview Window, the cursor changes to a Hand cursor.
- ☐ Press and drag on the Overview Window to change the view.

Panning the Overview Window does not change the view displayed in the main drawing window.



Selection Filter

Selection Filter allows you to specify the type of objects that can be selected. This function will be very helpful when you are working in a very busy area of the drawing.



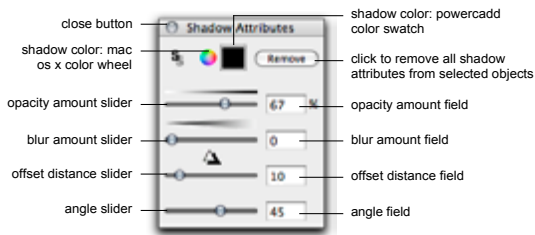
The [complete description](#) is located in the Tool menu section of this manual. To activate the Selection Filter floating window, choose Selection Filter from the Window menu or select Selection Filter in the Anchor Window.

Shadow Attributes Window

The Shadow Attributes Window is a floating window which allows you to assign and manage shadows attributes to selected objects.

To activate the Shadow Attributes Window:

- ❑ Choose Shadow Attributes from the Window menu or select Shadow Attributes in the Anchor Window. The following floating window will appear:



The Shadow Attribute Window will display the current defaults of any selected objects which have a shadow attribute. As different objects are selected, you will also notice the Attribute Window display will change to the selected objects shadow attributes. If an object does not have a shadow attribute applied the default settings are displayed as shown in the diagram above.



CAUTION: When Global Attributes is On, values displayed for Shadow Color choices when clicking on the PowerCADD Color Swatch are affected. See [Global Attributes](#).

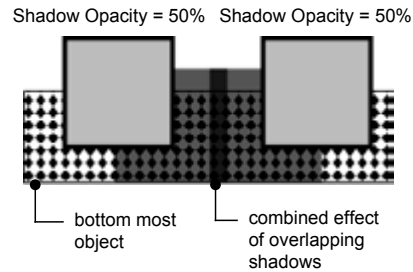
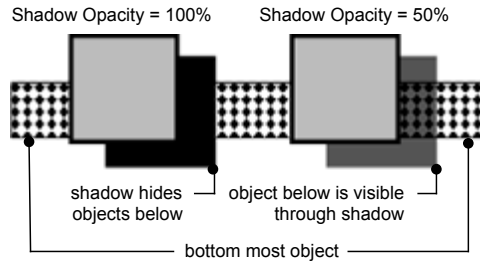


NOTE: At zoom levels equal to or greater than 1600 times, all shadows are drawn to screen with no blur effect. Zooming out to less than 1600 times draws the shadows to screen with their specified blur settings.

- ❑ Click on the MacOS Color Wheel to choose a shadow color using the MacOS color panel.
- ❑ Click on the PowerCADD Color Swatch to choose a shadow color from the PowerCADD color palette.

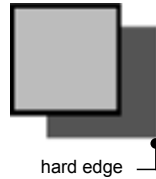


NOTE: An object's shadow color may be adjusted independently from its fill or pen color (see [Attributes Window](#) for more on fill color).

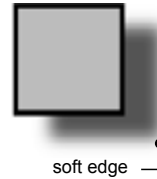


- ❑ Drag on the Opacity Slider to change how transparent (see-through) a shadow is. As you drag the slider, the Opacity Amount Field will update. You may also enter a specific amount in the opacity amount field.
- ❑ Adjusting an objects shadow opacity determines to what degree information is hidden or revealed below the shadow. For example, a shadow opacity of 100% hides all information below the shadow. A shadow opacity of 10% would appear almost transparent, allowing you to see through to objects below. The effects of transparent overlapping shadows are cumulative.

Shadow Blur = zero

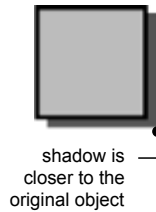


Shadow Blur = fifteen

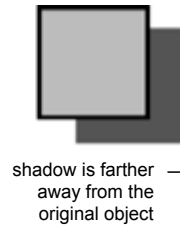


- ❑ Drag on the Blur Slider to change how soft, or fuzzy, the edges of a shadow should be. As you drag the slider, the Blur Amount Field will update. You may also enter a specific amount in the blur amount field. A value of zero (minimum) blur creates a shadow with a sharp edge. A value of fifteen (maximum) blur creates a shadow with a very soft edge.

Shadow Offset = ten



Shadow Offset = thirty



- ❑ Drag on the Offset Slider to set how far away from the source object the shadow should be. As you drag the slider, the Offset Distance Field will update. You may also enter a specific amount in the offset distance field. An offset value of zero (minimum) creates a shadow which is directly below the original object and therefore the shadow cannot be seen. An offset value of ninety-nine (maximum) creates a shadow which is farther away from the original object.

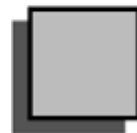
Shadow Angle = - 30



Shadow Angle = 30



Shadow Angle = - 140



Shadow Angle = 140

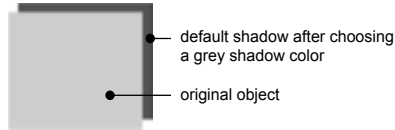


- ❑ Drag on the Angle Slider to set the shadow casting angle relative to the source object. As you drag the slider, the Angle Field will update. You may also enter a specific amount in the angle field. The angle of the shadow may be positive or negative.

Assigning a Shadow Attribute

To assign a shadow attribute to an object:

- ❑ Select the object in the drawing to which you want to assign a shadow attribute.
- ❑ Choose a color for the shadow by clicking on the PowerCADD Color Swatch or on the MacOS Color Wheel. The object will have a default shadow in the selected color applied as shown below. Adjusting any of the slider bars, instead of choosing a color, will assign a default color of black to the shadow which may be changed later.



- ❑ Drag on any of the slider bars to modify the associated shadow attribute. The changes are previewed on screen in real time as you adjust a slider. Release the mouse when the desired effect is reached.
- ❑ Repeat this process for each of the shadow attributes.

If multiple objects are selected, the same shadow attributes are applied to all selected objects.



TIP: The Eyedropper tool may be used to pick up and apply shadow attributes from one object to another.

Editing Shadows

To edit the attributes of an existing object with a shadow:

- ☐ Select the object in the drawing you want to edit.
- ☐ Make the desired adjustments to any of the options in the Shadow Attributes Window. The changes are previewed on screen in real time.
- ☐ Repeat this process for each of the shadow attributes you want to edit.

If multiple objects are selected, the changes will be applied to all selected objects.



TIP: Experiment! The number of possible visual effects you can create by combining the various Shadow Attribute settings in combination with Layer Opacity Attributes and object opacity attributes for pen and fill are virtually endless.

Removing Shadows

To remove a shadow attribute from an object:

- ☐ Select the object in the drawing that has a shadow to be removed.
- ☐ Click Remove in the Shadow Attribute Window.

The shadow is removed from the selected object.

If multiple objects are selected, clicking Remove will delete the shadow attribute from all selected objects.

Dynamic Snap Indicator The Dynamic Snap Indicator is a floating window that provides additional visual feedback when a dynamic snap is applied when drawing, moving or editing objects.



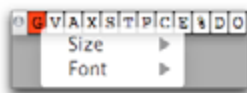
NOTE: The Snap Indicator External must be loaded with PowerCADD to be active. See [Loader Preferences](#).

To activate the Dynamic Snap Indicator

- ☐ Choose Snap from the Window menu or select Snap in the Anchor Window.

To change the font or size,

- ☐ Control- or right-click on the Dynamic Snap Indicator. The following dialog will appear.



- ☐ Choose the desired font and size from the contextual menu.

When a particular snap is active, the corresponding letter will highlight in red in the Dynamic Snap Indicator. For example, when Center snap is active, the 'C' will highlight in the Dynamic Snap Indicator.

Status Window

The Status Window allows you to access many commands and functions easily from a floating palette which can be placed anywhere on the drawing.

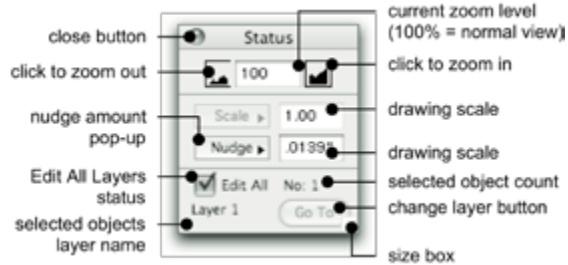
The Status Window imports functions from various drawing setup preferences and other functions into a single floating window. This window can be placed right where you need it at all times, which will allow you to be more efficient and therefore, more productive.



NOTE: The Status Window External must be loaded with PowerCADD to be active. See [Loader Preferences](#).

To activate the Status floating window

- ☐ Choose Status from the Window menu or select Status in the Anchor Window. The following floating window will appear:



Zoom Control consists of a small mountain icon on the left representing Zoom Out, a rectangular numerical field in the center displaying the Current View, and a large mountain icon on the right representing Zoom In. A specific view can be set by typing a percentage in the Current View field. The default is 100%, which represents Normal Size under the View menu.

Scale Status consists of a Scale field displaying the current drawing scale (as specified in Drawing Setup) or the current layer scale (as specified in Layer Attributes). This is display value only, it cannot be modified from the Status Window.

Nudge Control consists of a Nudge pop-up menu and a numerical field (also available in Drawing Setup and in Drawing Window Toolbar) allowing you to enter a Nudge Amount.

Edit All shows the current status of Edit All Layers and allows you to toggle it On or Off. An Object Count of selected objects is displayed beside the Edit All checkbox. The Layer Name field shows a selected object's layer and lets you activate that layer by clicking on the Go To button. If the selected objects are on multiple layers, no layer name is displayed.

Zoom In, Zoom Out, Current Zoom

Zoom In and Zoom Out allow you to magnify part of the drawing for greater detail or show a greater area of the drawing on the Macintosh screen. The Zoom Magnification enables you to set specific Zoom In/Out increments. Current Zoom allows you to set a specific zoomed view.

To use the Zoom In command from the Status Window:

- ☐ Click in the drawing at the location you wish to zoom.
- ☐ Click the Zoom In icon.
- ☐ Zoom will occur with the last mouse click in the center.

To use the Zoom Out command from the Status Window:

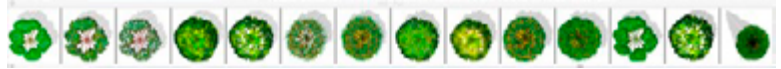
- ☐ Click the Zoom Out icon.
- ☐ This will execute the Zoom Out command.

To set a specific Current View:

- ☐ Double-click the Current View numerical field to highlight it.
- ☐ Enter a percentage and press the Return key. Normal view is 100%.

Symbol Palette Window

The Symbol Palette floating window provides an alternative method of seeing and placing symbols saved in Symbol Libraries. The Symbol Palette displays your library as a row of thumbnail views.



NOTE: The Symbol Palette External must be loaded with PowerCADD to be active. See [Loader Preferences](#).



For more information on Symbol Palette, see [Symbol Tool](#) in the Tool chapter.

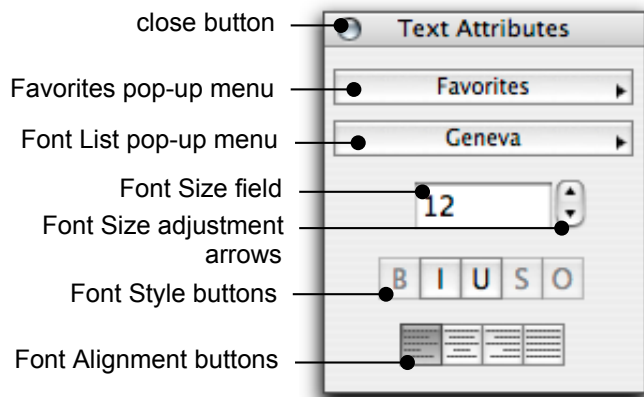
Text Attributes Window

The Text Attributes Window is a floating window allowing you to change the text defaults for any tool which supports text, or for editing existing text in a drawing.

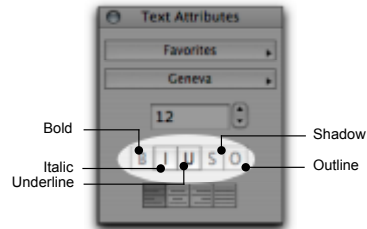
Choices available in the Text Attributes Window are also available in the Text menu, Text, or from contextual menus when Control- or right-clicking on an object.

To activate the Text Attributes window:

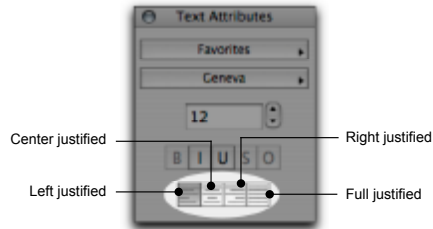
- ❑ Choose Text Attributes from the Window menu or select Text Attributes in the Anchor Window. The following floating window will appear:



- ❑ As you change tools, you will also notice the Text Attributes window display will change to the selected text defaults. As you change tools, you will also notice the Text Attributes window display will change to the selected text defaults.
- ❑ You may change the defaults, if they are not correct.
- ❑ When an object is selected, the Text Attributes window will display the text attributes of the selected object.
- ❑ When choosing items from the Favorites pop-up menu, select a favorite font style you have or to manage your saved font styles. The choices here are the same as choosing Text Styles from the Text menu.
- ❑ When choosing items from the Font List pop-up menu, select the font you want to use. PowerCADD will display all font families and font styles you have installed in MacOS and enabled using Apple's Font Book application. The choices here are the same as choosing Font from the Text menu.
- ❑ In the Font Size field, type a value, in points, for the size you want the text to be. You may also adjust the font size up or down one point size at a time by clicking on the Font Size Adjustment Arrows.



- ☐ Select a Font Style button to apply the associated style. The choices here are the same as choosing Styles from the Text menu. If a button is dimmed then that particular font style has not been installed or is not available for the specific font family.



- ☐ Select a Font Alignment button to apply the associated text justification. The choices here are the same as choosing Alignment from the Text menu.

Setting Text Tool Attributes

You may assign specific text attributes for a text related tool prior to drawing the object or entering text. All objects will have the same text attributes until the tool's text attributes are changed or you edit the text attributes of an existing object in the drawing. Tools which accept text attributes are the Text tool, Leader tool, Table tool, Date and Time Stamp tool, all Dimensioning tool, and all Notation tools.

To set the text attributes for a tool:

- ☐ Double-click on a text related tool to deselect all objects in the drawing.
- ☐ Select a setting in the Text Attribute Window and choose an item from the associated list.
- ☐ Repeat this process for each text attribute you want to set prior to drawing an object.
- ☐ Start the drawing operation. Text will be created with the specified attributes.



TIP: Pressing the Shift key as you set a tool's text attributes will set the text attributes for all tools in the palette at the same time.

Changing an Object's Text Attributes

The attributes of existing text in a drawing may be changed.

To change the text attributes of an existing object:

- ☐ Select the object in the drawing whose text attributes you want to change.
- ☐ Select the Text Attribute Window item you want to change and choose a new text attribute from the associated list. For example, select the Right Justified button to change the alignment.

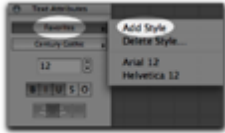
The Attribute for the selected text is changed.

If the selected text was a group, then all text objects in the group will take on the new text attributes. If multiple objects were selected, the text attributes for all objects are changed.



See the [Text menu](#), in this chapter, for information on working with text styles and working with fonts and text.

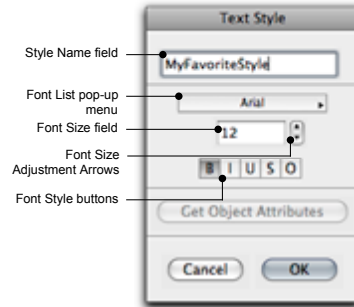
Creating Text Favorites



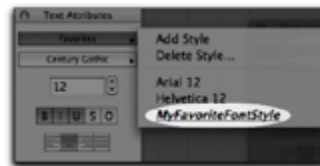
If you have a particular set of text attributes you use regularly, creating a favorite is an excellent way to streamline your workflow.

To create a new favorite:

- ☐ Select the text block in the drawing which has the correctly assigned text attributes.
- ☐ Select the Favorites pop-up in the Text Attributes Window and choose Add Style. The following dialog will appear:



- ☐ Enter a name for the new favorite in the Style Name field.
- ☐ Click the Get Object Attributes button to acquire the attributes of the selected item in the drawing.
- ☐ Make any desired changes by adjusting the Font Size or Style or select a different font from the Font List pop-up.
- ☐ Click OK and a preview of the new style is appended to the Favorites menu and to the Text menu, Text Styles submenu.



NOTE: Pen color, pen pattern, fill color and fill pattern are not saved with a text favorite.

To apply a favorite font style to text in the drawing:

- ☐ Select the text block in the drawing.
- ☐ Click on the Favorites pop-up in the Text Attributes Window.
- ☐ Choose the desired font favorite from the list.



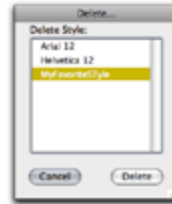
TIP: You can assign a custom key command to a text favorite

Deleting Text Favorites



To delete a favorite font style from the list:

- ☐ Choose Delete Style from the Favorites pop-up in the Text Attributes Window and the Delete dialog will appear.



- ☐ Select the text style favorite to be deleted and click Delete.

To delete consecutive items, press the Shift key and click on the items in the list.

To delete non-consecutive items, press the Command key and click on the items.

The style is removed from the favorites list.

Tool Palette

The Tool palette contains all the PowerCADD tools for creating, dimensioning and modifying geometry in your drawings.

The complete description is located in the Tool menu section of this manual.

To activate the Tools palette

- ☐ Choose Tools from the Window menu or select Tools in the Anchor Window.

Wall Tools Palette

The Wall Tools palette provides specialized tools for walls in architectural drafting and design. These tools extend the capability of PowerCADD for architects.

The complete description is located in the Tool menu section of this manual.

To activate the Wall Tools Palette

- ☐ Choose Wall Tools from the Window menu or select Wall Tools in the Anchor Window.

Help Menu



The Help menu contains help for both the novice and expert.

What's New opens an Adobe Acrobat PDF file with a list and general description of changes for this version of PowerCADD.

Tutorial opens an Adobe Acrobat PDF file with information to help you get started using PowerCADD.

PowerCADD Manual opens the entire PowerCADD manual in Adobe Acrobat PDF format on your computer. The manual is fully searchable, enabling you to quickly find information on any given PowerCADD related tool or menu item.

PowerCADD Website takes you directly to the Engineered Software website homepage.

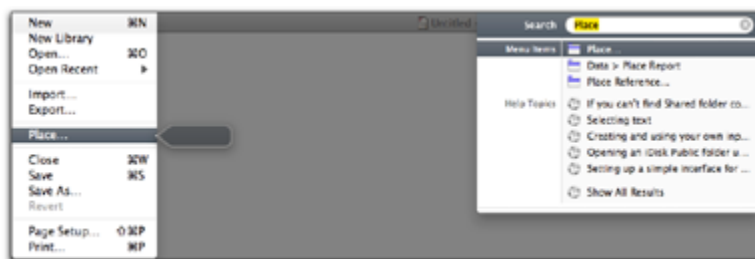
PowerCADD Help Web Page takes you to the Engineered Software web site help pages.

Updates... allows you to update your version of PowerCADD when you receive an email notification from Engineered Software with detailed instructions and a description of what's new or changed. An internet connection is required to perform all updates. If you are using a personal or network firewall, the default SQL port 3306 must be open.

Search: Users can use the Help Search feature to quickly locate help on various MacOS features or to locate PowerCADD menu items. The Help Search results are broken into two parts. The upper portion of the window contains results for PowerCADD Menu Items. The lower portion of the window contains results from the MacOS Help Topics.

To search for PowerCADD menu items:

- ☐ Choose Help and click in the Search field.
- ☐ Type the desired menu item you are searching for (for example: Place).
- ☐ A list of results appears. Click on an item to see the menu location activate. Press Return when a search results item is selected to activate the menu command.



Safe Mode Instructions: Safe mode helps you troubleshoot technical difficulties with PowerCADD. Choose this menu to see detailed instructions.

Submit Trouble Shooting Report: Choose this menu item to open up a form and submit details about your version of PowerCADD and any additional comments to Engineered Software technical support. An active email account and internet access is required to submit the report.

Reset PowerCADD

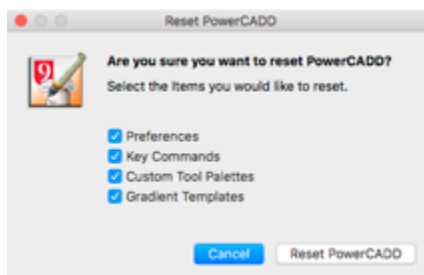


Reset PowerCADD: If you are experiencing an unforeseen or unexpected problem with PowerCADD resetting a particular preference file can be a quick way to troubleshoot or fix the problem.

CAUTION! Resetting a selected item will permanently delete it from the PowerCADD Preferences folder. We recommend making a backup of items in your PowerCADD Preferences folder prior to choosing Reset PowerCADD. See [Chapter 1](#) for additional information on what files and folders (??? can't find so created list below) are saved in the PowerCADD Preferences folder.

To reset PowerCADD:

- ☐ Choose Reset PowerCADD from the Help menu. The Reset PowerCADD dialog will open:
- ☐ Check the box next to the desired preference to be reset. [For a full list](#)



- [of Preference](#) files see Chapter 1. (??? can't find)
- ☐ Click Reset PowerCADD. PowerCADD will quit and the checked preference items will be deleted.

The next time you launch PowerCADD a new, factory default, version of the selected preference is automatically created.

PowerCADD Preferences Files

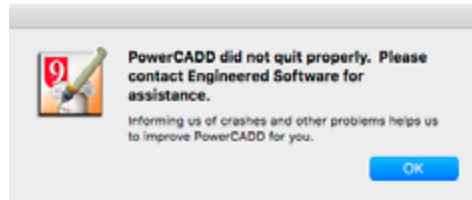
The following preferences files are located in

User/Library/Preferences
com.engsw.ES Update Priv.plist
com.engsw.PowerCADD9.plist

PowerCADD
PowerCADD 9 Key Commands

Even with extensive testing and quality control efforts, it is virtually impossible to replicate the myriad of combinations of Apple hardware, MacOS system software, and third party applications and hardware PowerCADD users may be using. To help us understand the nature of any problems and to help address them as quickly as possible we have implemented an automated crash log reporting system.

If you experience an unexpected crash or situation where you are required to use the MacOS feature to Force Quit PowerCADD, the next time you launch PowerCADD you will see the following dialog:



Clicking OK automatically submits details of the PowerCADD crash log to Engineered Software for review and analysis.

The information received is confidential and used exclusively by Engineered Software for ongoing quality control issues related to PowerCADD.

If you have any additional information you'd like to share regarding the crash log, or to share a copy of the drawing file to assist in trouble shooting, please contact us directly by calling 1-336-299-4843 or emailing support@engsw.com

Technical Tips, Hints & Shortcuts

Tips, Hints & Shortcuts

This chapter contains information that may save you time and energy trying to discover things on your own. Many of these hints were discovered during the development and testing PowerCADD and are passed along to you for your benefit.

Networking

High performance CADD requires high performance tools. Dedicated local hard disk drives provide significant performance and productivity.

It is advisable for both license and copyright laws and practical technical reasons, not to double-launch PowerCADD. PowerCADD will allow users to open the same drawing more than once concurrently to allow *before and after* referencing, different views on multiple monitors and so on.



For more information on opening files stored on a network server, see [Network Servers](#).

Uses of the Option Key

Option Palette

Pressing Option while selecting certain tool palette items will present a dialog box allowing you to change the tool's defaults. Any tool that has two dots (..) in the tool icon will present a dialog box, including dimension tools, parallel line tool, round corner rectangles, regular polygons and others. Some menu items will display three dots (...) when you press the Option key and select a menu item. Pressing the Option key when selecting the menu items will present a dialog box.

Option Duplicate

Choosing the Duplicate menu item with the Option key depressed presents a dialog box and allows multiple duplications and the selection of the offset.

This can be used for placing the duplicate at a specific location and for setting the number of copies.

Rectangular and circular arrays can be used.

You can also duplicate an object while you drag it by pressing the Option key.

Option Reshape

During Reshape, pressing the Option key while selecting a point deletes the point.

Pressing the Option key while clicking on a line adds a point to the polygon, Bézier curve or B-Spline.

Only a few control points are necessary with Béziers. You will find more points make the curve much harder to manipulate.

Option Editing

Pressing the Option key while selecting an attribute, fill pattern, hatch line type, arrow, dash, or color will bring up a dialog and allow editing of the attribute.

Move Points

After selecting Move Points, the Option key will allow a lasso selection of the area to be moved.

Option Guidelines

Pressing the Option key while dragging a guideline from the rulers onto an existing angled object, like a rectangle or line, will force the guideline to assume the object's angle.

Uses of the Shift Key

The Shift key allows you to constrain and to release constraint while drawing and editing.

Shift as a Constrain Key

When drag-moving an object, the Shift key will constrain its movement to vertical and horizontal.

The Shift key unconstrains a dimension line so any angle dimension line can be drawn.

Pressing the Shift key while completing a dimension line placement will create a parallel dimension.

The Shift key unconstrains the Parallel Line tool to draw at any angle.

Pressing the Shift key after a rectangle has been started will constrain the rectangle to a square and an ellipse to a circle. The Shift key constrains the diagonal to any of the preset angles in the Drawing Setup under the Layout Menu.

Pressing the Shift key after line stretching has started causes the line to maintain its original angle while being lengthened or shortened.

Pressing the Shift key while using Reshape on a line will release the angular constraint on the line.

Pressing the Shift key while stretching a corner handle causes rectangles, circles, and ellipses to be stretched proportionately.

The Three-Point Arc tool is constrained to the preset angles by pressing the Shift key after drawing has started. The constraint should make it easy to construct regular arcs starting and ending on North, East, South, or West.

Other Uses of the Shift Key

The Shift key also allows deselecting an individual object in a selection. When making multiple selections by pressing the Shift key, you may deselect a selected item by continuing to press the Shift key and clicking on the object.

Setting Tool Defaults

Pressing the Shift key while setting a tool attribute sets this attribute as the default in all of the tools if the default is applicable.

Uses of the Space Bar Key

Flipping Orientation

While drawing an elliptical arc, an arc by its diameter, tapping the Space Bar to flip its horizontal or vertical orientation.

Draw by Center

While drawing a rectangle, rounded rectangle or line, tapping the Space Bar will draw the object about its center point.

Panning

With any tool selected, pressing the Space Bar will enable interactive panning.

Drag the mouse while pressing the Space Bar to pan, release the mouse to return to the previously selected tool.

Parallel Lines

While drawing the first segment of parallel lines, pressing the Space Bar will toggle the offset between left, center, and right. Each keypress toggles the offset.

Uses of the Command Key

The Command key releases all snapping while drawing. Also, by pressing Command while moving an object will allow the object to be moved off the page.

Command Period (.) and Command-Spacebar

Using Command-Period and Command-Spacebar

- ☐ Stops screen updating. Eliminates slow redraws.
- ☐ Command-Spacebar and Command-Period will stop a redraw in process.

Killing the redraw is very valuable with huge drawings. If no redraw is forced, the computer will catch up when it gets a chance. It is more likely you will perform another screen move after scrolling, such as Zooming-to-final-location for drawing. After zooming, the update will be correct.

Selection Arrow

Selections can be made, by drawing a selection marquee touching the desired objects.

By pressing the Command key, all objects inside the selection rectangle are selected.

Pressing the Shift key allows cumulative selection or deselection of objects.

Dialog Shortcuts

You may use keyboard shortcuts in a variety of dialog boxes for specific items.

Pressing Command-Period is the same as clicking Cancel.

Pressing the Return or Enter key is the same as clicking OK.

Pressing Command-D is the same as clicking the Don't Save button.

Copy and Pasting

Copying from one drawing and pasting to another drawing places the pasted material exactly in the location copied unless you are using another pasting option.

If the drawing you are pasting to is smaller (page size) than the original, the material may be pasted off-page.

Use Paste Special, At Mouse to move content from one file to another and control where it is pasted.

For more information on pasting at mouse, see [Paste Special](#).



Other Tips

Trim

If the trim defaults are set to delete the line segments, the line segments specified with the bridge line will be retained and the line segments extending past the intersection will be deleted.

Tool Settings

For the individual tool settings, you have the best of both worlds.

In order to make a permanent change in a tool's defaults, you cannot have any item selected.

You may double-click on the desired tool, which will deselect any items in the drawing.

Choose the preferred characteristics from the menus, such as fills or colors. The defaults you specified are assigned to the tool until you change them.

To change the defaults momentarily, select a tool with defaults assigned to it. Draw an object and while the object is still selected, change its characteristics using the menus. When you draw the next item with that tool, you will find it has returned to its original defaults.

Editing Rotated Text

Rotated text may be edited by selecting the text tool and clicking on the rotated text. The text will temporarily return to zero rotation or horizontal for editing. When you have finished editing, the text will return to its original rotation angle.

Measuring

Measuring can be accomplished at any time by using the Show Location Bar, which is displayed in the lower left corner of the drawing window.

To activate the Show Location bar, select Show Location from Show Tab in Drawing Setup chosen from the Layout menu.

By clicking directly on the bar, you can toggle the measurement system through Absolute, Relative, or Length-and-Angle.

By activating the Length-and-Angle option, you can measure from point to point. Start by clicking where you wish to begin measuring. The values in the Show Location Bar will reset. As you move the cursor away from the starting point, the Show Location Bar will display the distance and angle or direction of cursor travel. Therefore, by clicking to define the start of the measure, and placing the cursor on the end point of the measure, you can read the dimension and angle in the Show Location Bar. You will find dynamic snapping is invaluable during this process.

Accuracy

The drawing database is accurate to twelve decimal places at normal scale, and sometimes more.

The manual drawing process is at best accurate to .005".

Paper stretch and shrinkage can create another 1/32" error.

Basically, this means the computer is 5 to 30 times more accurate than a manual process, but not infinitely accurate.

It is absolutely mandatory to test any computer operation for reasonableness; check your application of the computer tools against the desired results.

Calculations requiring accuracy greater than the nearest .001", at 1:1 scale, should be checked by other means.

Working at Angles

Rotating an entire drawing or a layer will allow you to work at a fixed angle.

The layer should be rotated about the drawing origin. Work on the section desired until complete and then rotate back to the original angle.

This method has been used on very large drawings with good results. The concept is not much different than the way some of us worked when there were only T-squares and parallel bars.

Alternatively, use Custom Snap angles to set a baseline, which matches the rotation angle of the data. This allows you to snap to those angles without having to rotate the drawing to be perfectly vertical or horizontal.

Of all the tasks performed during computer drawing production, the ability to easily zoom in or out (that is to change the on screen view by enlarging or reducing its size) may be at the core of efficient drawing navigation. Since none of us work in exactly the same way and drawing circumstances vary widely, multiple methods of zooming are available in PowerCADD to address those nuances. While we can summarize the available methods, there is no ‘best method’. Rather, each one has its own unique place in the workflow and with time they can all be mastered and used seamlessly.

Zooming method can be accessed using the mouse (for example to select a tool or menu item) and many of these methods can also be access using a preassigned key command, or you can add your own custom key commands.

Zoom Tool: Located in the first row of the tool palette, this tool allows you to press and drag in the drawing window to dynamically specify how far to zoom in. Pressing the Option key with the Zoom Tool selected will zoom out.

View Menu: The view menu contains commands for zooming in and out of a drawing, normal size viewing and reduced to fit. All of these may be accessed using key command shortcuts. In addition, you may save frequently used views and assign your own custom key commands to access those views. All of the view menu items work together to create a suite of time saving navigation features.

Overview Window: Click and drag in the Overview Window to specify which area of the main drawing you’d like to see. Drag a small marquee to zoom in, drag a large marquee to zoom out; the drawing window updates in real time to reflect the drag operation in the Overview Window.

Zoom In/Out/Previous ‘now’ (on the fly zooming): This feature is unique in that it allows you to modify the zoom ratio even when a drawing tool is selected. The default key commands are (1) to zoom in , (2) to zoom out, and (3) to toggle between the current and previous zoom levels. If you’d like to modify these key commands, choose Preferences from the PowerCADD menu and click on the Commands tab. Look for Zoom In Now, Zoom Out Now and Previous Zoom in the list of items.

Scroll Wheel Zooming: If you have a multi-button mouse with a center scroll wheel, turning the wheel can zoom in or out ‘on the fly’ while a drawing tool is selected. This feature is Off by default and needs to be enabled by checking Mouse Wheel Zooms in the PowerCADD Preferences, View tab.

Drawing Window Toolbar: Icons in the drawing window toolbar also allow you to zoom in or out.

Points

Points are temporary markers used to locate other items.

Points represent an excellent method to mark locations to be snapped. They may be left in the drawing indefinitely; however, we recommend you occasionally delete the points and other construction lines which are no longer necessary.

Also, as a particularly crowded area gets busier, the snap-to will become more intense and slightly slower. In any event, delete the points when they are no longer needed.

A quick way to remove these points is to use the Selection Filter window.

Bit Image Symbols

Bit images as symbols can sometimes take much less space in the drawing than their vector counterpart. For example a 1,990-object industrial stove takes a lot of space in each drawing in which it is used.

To experiment:

- ☐ Draw the item as you would normally as a vectored object in PowerCADD.
- ☐ When finished, place the symbol in an otherwise empty drawing or separate layer.
- ☐ Check About PowerCADD to determine the count and file size.
- ☐ Select the objects and choose Objects to Bitmap from the Tool menu. The object has now been converted to bits.
- ☐ Compare the new file size and object count.

You will probably want both the vector and bit image copy of the symbol. As always, use the most efficient technique to accomplish the job.

Lines, Text and Color Settings in the Menu

As you have already observed, when you pull down a menu or sub-menu, some items are marked with a checkmark (✓). To determine an object's attributes, select it and choose the appropriate menu; the object's attributes will have a checkmark next to the named menu item. To determine a tool's attributes, select the tool and choose the appropriate menu.



CAUTION: Having both an object and a palette tool selected can be confusing. To avoid confusion, double-click on the palette tool before checking the tool's attributes. Double-clicking assures only the palette tool is selected. To avoid confusion, select the Selection Arrow tool and then select the object. This assures only the object is selected. The last item touched is the one for which the menu checkmarks apply.

Line Types

The repeat length of the dashing is accounted for when turning the corner of polygons, rectangles, and other objects. Polygons and smoothed polygons will be dashed when printed as you would draw them manually.

Snap Angles

Snap angles may be set in the Snap tab of the Drawing Setup dialog. The snap angles may be any angle you desire, including fractional angles. Snap angles apply to virtually all drawing tools, move, and editing operations.

Pressing the Shift key will override preset snap angles for a tool, which is always constrained (e.g. the Multi-Angle Line Tool).

Pressing the Shift key will force a preset snap angle on a tool, which is always unconstrained (e.g. the Line tool).

Multiple Scales

Think of PowerCADD scaling the same as flipping your triangle scale over as you work on a document—same drawing—just measured differently.

Drawing Walls

To start a wall, select the Parallel Line tool, press and drag the mouse to define the first segment of the wall. Once a wall is started, click at each intersection point of each wall. The wall can be drawn on centerline or to the outside or inside of the wall. The centerline is a construction line, which will disappear when finished drawing.

Press the Option key while selecting the palette to bring up a dialog to change the left and right offsets of the wall. The settings will be retained until changed.

Press the Space Bar while drawing to toggle between left, right, and center offsets on the fly.

Double-clicking ends the drawing of this item. Clicking close to the starting wall point closes the wall section. Double-clicking rapidly while drawing, or clicking twice in the same place, ends the polygon. If the end of the polygon joins itself at the beginning of the polygon (as in a building), a special wall closure occurs.

Once drawn, parallel line polygons can be filled or modified in the normal manner. Reshaping allows adding points, moving lines, deleting points, and many other changes.

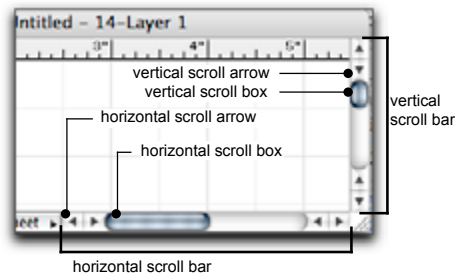
Parallel line walls can be stretched without changing the wall thickness. Choose Reshape and lasso all the points to be moved. Pick one of the points, and drag the entire wall end to its new location. Reshaping allows moving the wall without changing its thickness. These polygons can be combined with other polygons.

PowerCADD offers various methods for panning and scrolling in a drawing. Much like the various methods of zooming, taking the time to understand and master each one can help create a more efficient workflow.

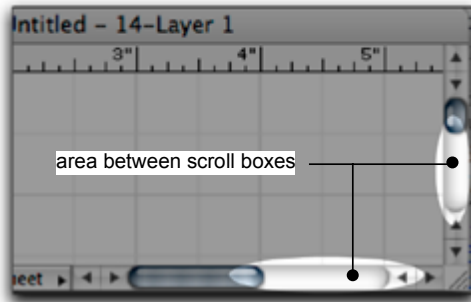
Pan Tool: In the first row of the tool palette, choose the Pan Tool then drag in the drawing window to ‘move the paper’ so you can see the desired area.

Interactive Pan: Similar to the Pan Tool only it goes to the next logical level. With any drawing tool selected, press the Space Bar. The current tool or cursor will change to the grab hand; drag to pan. Release the Space Bar and your drawing tool is still active.

Scroll Arrows: Clicking on a scroll arrow will move you up, down, left or right one line depending on which scroll arrow you press. Pressing on a scroll arrow causes continuous scrolling. This is a MacOS standard, and it is easy to overlook, especially for new users of Windows/AutoCAD converts. Of course PowerCADD offers a creative twist. Choose Preferences from the PowerCADD menu and select the View tab. Specify a smaller or larger number in the Line Scroll field to adjust how far you move each time you click on a scroll arrow.



Scroll Box: Drag a scroll bar (vertical or horizontal) to reveal a different portion of the drawing area. The position of the scroll bar in the scroll bar indicates the position of what is in the drawing relative to the entire drawing window. Again, this is a MacOS standard, which can be customized in PowerCADD to help you optimize performance for your particular hardware configuration. Select the View tab in the PowerCADD Preferences and specify a smaller or larger number in the Page Scroll field.



Area Between Scroll Bars and Scroll Arrows: Here is yet another MacOS golden oldie method of navigation. If you click in the grey area between a scroll bar and a scroll arrow, you will advance the height or width of the drawing window. This can be a great way to surf around a drawing, one drawing window tile at a time. Think of this as moving one page at a time where a page is defined as what you can see in the drawing window. Remember, the drawing window can be any size, and this feature will behave consistently with respect to that size.

Scroll Up/Down/Left/Right: You can assign custom key commands which perform the same task as clicking on an up, down, left, or right scroll arrow in the drawing window. Select Commands tab from the PowerCADD Preferences dialog box. Select the Miscellaneous pop-up menu and assign key commands to Scroll Up, Scroll Down, Scroll Left or Scroll Right. Now you can scroll in any direction simply by selecting the command key instead of moving your mouse.

Page Up/Down/Left/Right arrows: There are key commands which are also available to move one drawing window at a time in the direction specified by the arrow. This has the same effect as clicking in the grey area between scroll boxes and scroll arrows described earlier. The default key commands assigned are Shift key + [the arrow key] where [the arrow key] is left, right, up or down as shown on the keyboard. Like all other PowerCADD key commands, you may also assign your own custom values in the Preferences dialog, Commands tab, Miscellaneous pop-up menu and choosing Page Up, Page Down, Page Left or Page Right.

Auto Scroll: When dragging to draw or move an object, the screen will automatically scroll when your mouse collides with an invisible barrier on all sides of the drawing window.

Overview Window: Drag the red zoom rectangle in the Overview Window to move it. As the red zoom rectangle moves the main drawing window is updated.

Concentric Circles

Concentric circles can easily be drawn by leaving object snap on. The radius will snap to the point in the center of the circle for each new circle. Alternately, you can draw concentric circles rather sloppily at their approximate location. Later lock the smallest of the circles, select all of the circles and arrange by aligning centers and middles. You might also place a point at the circle center, and let the center snap to the point. Another method of drawing concentric circles is using the Parallel Offset tool.

Layer Info

In the Layer window, multiple selection across a range of layers is possible with use of the Shift key. Multiple selection of random layers is possible with the use of the Command key. Multiple selections should make it much easier to change many layers to dimmed or invisible at once. Remember the current layer cannot be hidden or deleted.

Grid Snap

Grid snap is a toggle setting—On or Off. We highly recommend you always leave the grid snap on. When required, you can release your drawing from the grid snap by pressing the Command key.

Bit Images

Bit images may be resized by dragging their corner handles. The bit image retains its original accuracy when reduced or enlarged. Enlarging may produce unusual printed results. Shrinking or reducing the size will generally enhance the printed appearance. Try drawing the bit images twice as large as needed and Scale by .5 when in PowerCADD. Any PowerCADD object may be converted to a bit image using the Object to Bitmaps menu item in the Tool menu. Bit images may also be cropped by pressing the Command key when dragging an edit handle.

Log

The Log is a method of keeping a cumulative record of the amount of time a drawing is open or in use. Some caution is in order here since two drawings can be open and accumulate time together. Also, the drawing will be accumulating time while you are on the phone. In some circles this is known as “double dipping”. This is also the method some professionals use to determine billing. The log file can be opened directly in Microsoft Excel or other spread-sheet applications. Double-click on the file. The log file can be read by most word processors. To align things, the tab settings will need to be made in most word processors.

Bounding Box

A bounding box is the smallest rectangle that will completely enclose an object or multiple selected objects. The sides of the bounding rectangle are the horizontal and vertical axis. In the Cartesian coordinate system the “X” is always the horizontal axis and the “Y” axis is always the vertical axis.

Cursor Arrow Control

Screen scrolling is connected to the cursor arrows in addition to paging in the scroll bar and line scroll in the scroll arrow. Cursor control is approximately one-half page. This is intuitive and reacts much like a spreadsheet and word processor program.

Spot Color Separation

Using a layer for each color will allow easy spot color separation. When you are finished with your drawing, print each color (layer) individually.

Printed Circuit Traces

The Parallel Line tool represents a good method for drawing printed circuit traces. The resulting trace would normally be filled black. Using a wide line is probably not a good method of representing a printed circuit board trace. Parallel Line Polygons can be scaled up and down. Also, the pad to trace clearance will be correct on Zooming.

Printed Circui

Some of our customers have used PowerCADD to create chip circuitry. One-mil-wide traces are commonplace. Normal silicon design techniques are used to maintain accuracy.

Self-Healing Walls

Walls constructed as polygons or rectangles can always be self-healing. Construct the doors and windows with opaque fills. When the window is placed over the wall, the wall lines disappear. When the window is removed or its location changes, the wall lines reappear. Printing, screen appearance, and plotting will be correct. The “permanent self healing” is a most powerful feature and is unique to the MacOS. You will find many other uses for “permanent self healing” in PowerCADD.



NOTE: The use of opaque or white-filled objects to mask other areas should be avoided on drawings destined for DXF or DWG translation.

Digitizing

First, scan the maps or other drawings. Use either the scanning software or a paint program to clean up the stray pixels. Save the scanned image in any of the many raster formats that PowerCADD can Open, Place, or Import. Place the scanned and cleaned-up material into a blank PowerCADD layer. Add a layer to PowerCADD and draw over the top of the scanned material to trace the image. The screen digitizing process works best when one of the layers is a different color.

Entering Keyboard Data

You will find the process of entering keyboard data to be much faster if you first draw a rough outline of an object with the mouse and then edit the drawn shape to exact dimensions with the Edit Window and the keyboard. This is also true for polygon survey boundaries.

Screening a Bitmap

If you have a bitmap, which you would like to use as a faded or screened backdrop, place it on its own layer. Then use the Layer Attributes, Fill Opacity setting to fade the bitmap. Reduce the Fill Opacity to about 35% and you can draw easily on top of the bitmap.

Alternatively, you can also use the Apply Bitmap Effects menu item and apply a color tint to the bitmap, which can simulate the same effect. Using the Layer Opacity technique is typically faster, easier and non-destructive but does not allow you to specify a color tint. Color tints may also be applied by assigning a pen color to a black and white bitmap.

Database Choices

What are my choices when using PowerCADD to create schedules, lists, and build a database of text data for drawing, project and facility management?

Unfortunately there is no one-click answer and a sound understanding of the fundamentals is critical to success in this type of situation. The trade off is some time and effort is required on your part to learn the available methods and how they can be applied to your specific needs.

The key to remember is PowerCADD is a graphic interface to the drawing database. Every object in the drawing already contains a variety of information (example: its x,y location and size among many other attributes). You can also assign your own unique text attributes to special objects using Symbol Tags. This additional data becomes part of the PowerCADD database and can be exported for use in other applications to perform value added analysis.

We can't really tell you what to do as it depends on your specific drawing and data management needs. However, we can give you an overview of your available options, which may help get you started. In all cases, it is important to clearly understand and plan out in advance just what your data management needs will be, then compare those needs to your available choices. There is rarely a single 'auto-magic' solution to a particular data management situation. Rather, you may need to look at using several techniques to meet your project needs.

Here is a basic breakdown of the current features and what they do to help guide your decision-making process. Each feature has a unique and valuable place in the database workflow. In the case where a report can be saved, the resulting exported files are in plain text format, which may be opened in a variety of applications to value added analysis.

Symbol Libraries: Libraries may contain any PowerCADD object or collection of objects. A symbol may be as simple as a snap point containing text attribute data or as complex as a master construction detail, site plan or wall assembly.

Behavior / Qualities:

- ☐ Very easy to use
- ☐ Library is not linked to a specific drawing file
- ☐ Object drawing data (but not associated database attributes) will export via DWG or DXF once in the drawing (not including raster based data).
- ☐ Can contain any valid PowerCADD object(s)
- ☐ Can be applied at any time in the drawing process
- ☐ Automatic scaling of objects when inserted into drawings
- ☐ May contain additional database attribute data in addition to 'name'
- ☐ Preplanning required prior to using to maximize its power
- ☐ Symbol library object may contain special user defined text assigned using Set Symbol Tags command.

Symbol Tool: This tool contains two optional settings and works in conjunction with Symbol Libraries and the Set Symbol Tags Command.

Behavior / Qualities:

- ☐ Very easy to use
- ☐ Option-click to set tool for inserting objects as groups or instances
- ☐ Symbol instance creates a one to many relationship for updating Set Symbol Fields
- ☐ Ability to use Replace Symbol to globally replace symbols inserted into a drawing with this tool speeds up drawing management/increased productivity
- ☐ Ability to Set Data Fields (as opposed to Set Symbol Tags) when objects inserted into the drawing.

Set Symbol Tags: This command permits data tags to be set for any existing object in a drawing or for objects inserted from Symbol Libraries. Data tags can be reported and opened in other applications for value added editing and analysis.

Behavior / Qualities:

- ☐ Moderate ease of use
- ☐ Requires preplanning to make best of use of this feature
- ☐ Use Save As, Symbol Report, to generate a report of Symbols in the drawing and their related database attribute tags
- ☐ Up to four unique data fields in addition to one name field
- ☐ When objects are inserted as Symbol Instances, the data fields have a

one to many relationship (edit one data field will update all data fields of symbol instances with the same name)

- ☐ Best applied to items inserted from Symbol Libraries but may also be applied to individual objects in a drawing
- ☐ Reporting feature can discriminate to report all symbols in drawing or only selected symbols
- ☐ Data does not print or export unless included as a table in the drawing

Replace Symbol: This commands permits global replacement of symbols in a single drawing.

Behavior / Qualities:

- ☐ Very easy to use
- ☐ Works across all layers, whether layer status is on, off, grayed or locked
- ☐ Remembers scale and automatically re-scales objects to correct % during replacement
- ☐ Works with symbols inserted as groups or instances (keys on group name)
- ☐ Very fast method to update an entire drawing with new objects that may contain different or updated database attributes

Summary: There are several components to database creation and management in PowerCADD. Each feature will have pros and cons depending on your particular needs. Remember, each feature fits a specific niche in the work flow, and they can be used alone or in combination with other tools and commands to solve a variety of reporting needs. The learning curve can be steep in some respects but time and costs invested in self study and training in the short term can pay off significant dividends in the long run.

DXF & DWG Translators

DXF & DWG Translators

This chapter provides information on the .dxf and .dwg Translator Externals that allows you to open various non-PowerCADD file formats or to save your PowerCADD drawings into other file formats.

Translator Externals

The file suffix “.dwg” is a generic reference to a “drawing” file used in the Windows environment.

The Translator Externals provide an integrated interface to translate drawings to or from PowerCADD to any application that uses the AutoCAD® DWG or DXF native file format.

The Translators are Externals. As such, they must be activated in the PowerCADD Preferences, Loader tab.

Using the example files **included on your PowerCADD Master CD-ROM**, we'll explore the various features and methods for translating drawings. We recommend having the examples files on hand to work through each

Since the process for reading or writing .dwg or .dxf file names to or from PowerCADD is the same, we will use the .dwg translation process for illustration and demonstration purposes. Any translation process, which differs for .dxf will be noted.

The .dwg and .dxf Translator External contains the following sections:

- ☐ Terminology used in the section.
- ☐ What you should know before translating.
- ☐ Translating your PowerCADD drawings to .dwg or .dxf.
- ☐ Translating .dwg or .dxf files to PowerCADD.
- ☐ Reference Guide.
- ☐ Frequently Asked Questions.



NOTE: The Translator reads AutoCAD (ACAD) files, release 10 through release 2010.



NOTE: The Translator writes files in AutoCAD release 2000 through 2010. Newer versions can read older versions. For example, if you are sending a .dwg file to someone using version 2006 they will be able to open a file saved as version 2000.



NOTE: Binary .dxf files cannot be opened by the Translator and .dxf files must be saved in ASCII (American Standard Code for Information Interchange).



NOTE: To open any .dwg or .dxf file, the filename must have the appropriate .dwg or .dxf suffix. Files without the correct filename suffix will appear dimmed in the Open dialog.

Terminology

The following terms and definitions are used when describing the .dwg or .dxf translation processes:

- ☐ **Translate a drawing:** The act of converting the drawing data from one file type to another. For example a PowerCADD drawing may be translated to a .dwg or .dxf file. The language used in the file, for example English or French, is not affected.
- ☐ **.dwg:** The internal drawing file format and extension used by AutoDesk for their AutoCAD drawing application running in the Windows computer operating system.
- ☐ **DXF or .dxf:** The Drawing Interchange Format and file extension created by AutoDesk (authors of AutoCAD). DXF files can be created by almost all CAD applications.
- ☐ **Read:** To Open a .dwg or .dxf file into PowerCADD. The result is a new PowerCADD drawing.
- ☐ **Write:** To Save a PowerCADD drawing as a .dwg or .dwg file. The result is new drawing saved to disk which is independent of your PowerCADD drawing.

What You Should Know Before Translating

Introduction

When opening a .dwg or .dxf file, the translator will detect which release is being used and adjust accordingly.

When sending a file to another computer on a CD or DVD, you need to consider the type of computer when formatting a disk to hold your file.

To send a file to a Windows machine, it should be formatted accordingly.

When sending a file to another Macintosh, no special formatting of the disk is required.

While every effort has been made to insure your data arrives correctly, there are many capabilities of the MacOS we all take for granted, that are simply not available or difficult to accomplish in other applications on different operating systems. Fill patterns, bit images, line weights, line patterns and many different fonts are standard in the Macintosh environment; this is not the case in the Windows world. Although add-ons can be purchased to do some of the same types of things, they are very expensive and generally not owned by all users.

Although the ability to create line weights exists through polylines (and some recent versions have started to support line weights in the traditional PowerCADD sense), it is not as easy to accomplish and therefore, used less often. Line weights are commonly identified by color. Red might indicate a 2-point line, so all lines that need to be 2 points are assigned a red color. The Translator will maintain the pen colors when reading the file, but all line weights in PowerCADD will have a default value of 1 point.

Since fills, patterns, and pen patterns are bit images, these bit images do not translate. If you are constructing a drawing you know will be used in Windows, there are certain things you can do to ensure your drawing will have the same characteristics when used there.

In the Windows world, there basically is no scale as PowerCADD knows it, i.e. one mile equals one mile. They deal in generic units or unitless drawings. PowerCADD, on the other hand, provides a scaling factor, in much the same way as the drawing board, in fact, every layer can have its own scale. PowerCADD scales with inches, feet, decimals, meters, centimeters, etc., and thus, $1/4"=1'$ is 1:48. This can create some confusion if you are not aware of the scaling conventions.



CAUTION: You should not use Layer Scales in PowerCADD if your drawing is going to be translated to Windows.

In a 2D program such as PowerCADD, the “Z” dimension is not used. All 3D entities are read as flat. This includes 3D lines, 3D polylines, 3D faces and polyface meshes. 3D meshes are ignored.

A PowerCADD drawing saved as a .dwg file will be considerably smaller than a corresponding .dxf file, making it easier to send and receive files on CD-ROM, over the network, or over the Internet. Generally, the .dwg file format is preferred, but if you are not sure which type of file to write when translating your PowerCADD drawing, contact the person who requested the file.

PowerCADD to DWG or DXF

Translating Your PowerCADD Drawings to .dwg or DXF

The translation from a PowerCADD document to a .dwg document is as simple as choosing .dwg as your format in the Save As dialog box. However, when you know initially your document will ultimately need to be in a .dwg format, you might consider using some techniques to ensure success.

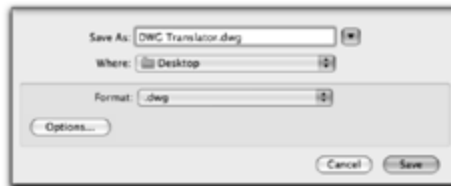
There are many things, we Macintosh users take for granted. Special looks can be achieved in PowerCADD, through custom fill patterns, custom colors, bit image scans, and clipping just to mention a few. For best results, knowing their limitations is the best technique for trouble-free translations. Clipping, the cover-up of lines or objects with other lines or objects, is a natural way for you to create a PowerCADD drawing; however, Windows programs do not 'clip'. If you would like to see how your 'clipped' drawing will look, use the Select All command with Edit All Layers turned On and set the fill pattern to None.

Since the process for writing .dwg or .dxf file names from PowerCADD is the same, we will use the .dwg translation process for illustration and demonstration purposes. Any translation process which differs for .dxf, will be noted.

Saving a .dwg File

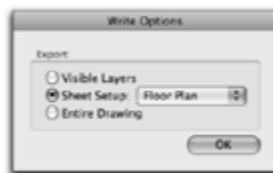
To save your PowerCADD file as a .dwg file:

- ☐ Choose Save As from the File menu.
- ☐ Choose .dwg from the Format pop-up. The .dwg suffix is automatically appended to the file name in the Save As field. This suffix is required for the file to be recognized in Windows.



- ☐ Click Save.
- ☐ If your file contains multiple sheet setups, or various layer status settings the Write Options dialog will appear.
- ☐ Choose the desired settings and press OK to complete the translation.

The **Write Options** dialog allows you to specify what data in your PowerCADD drawing should be translated.



Visible Layers: Click this radio button to write out only those layers in your drawing which have a drawing status of Visible or Dimmed. Objects and layers, which are hidden (off), will not be saved with the translation.

Sheet Setup: Click this radio button to choose a specific sheet setup for translation from the pop-up list. Only objects and layers, which are visible for the specified Sheet Setup, are saved with the translation.

Entire Drawing: Click this radio button to translator your entire drawing. All objects on all layers are saved with the translation. Hidden layers will be off, visible layers will be on, when the file is opened.

Options...

The Options... button in the Save As .dwg dialog opens the Write Settings dialog where you can customize which version of .dwg is used, and map PowerCADD line weights to specific pen color values.



Choosing a version of .dwg

To choose which version of .dwg to save as:

- ☐ Press Option in the Save As dialog.



- ☐ Choose the desired version from the list and click OK.
- ☐ You are returned to the Save As dialog to complete the Save As process.

When checked, the Set Colors for Line Wts box allows you to create a table to translate line weights in your drawing to specific pen color numbers. When not checked all line weights default to a default pen color of black.

- ☐ Press Option in the Save As dialog.
- ☐ Check the Set Colors for Line Wts box.
- ☐ Click Settings. The Line Weight to Color Settings... dialog will appear (pictured and described later in this section).
- ☐ Make the desired settings and press OK.
- ☐ You are returned to the Save As dialog to complete the Save As process.

Some Other Options

Teigha File Converter

There is a free file converter available from Open Design Alliance called . It basically looks for .dwg files in a folder, converts them to your chosed AutoCAD version and places those into another folder. It works very well as a batch converter.

Draftsight

Another option is Draftsight, a free program, will open most dwg files and can then save back to AutoCAD 2010.

Import DXF

Import DXF is now part of WildTools. It is definitely not what you need for a full translation of a complete drawing with dimensions.

It is optimized for dealing with files exported from LIDAR data where you often end up with a file of 250,000 to 400,000 objects that should be combined into a single object, which it does.

It's also optimized for dealing with files exported from the Concepts Pro iPad app where you can do architectural sketches with an Apple Pencil and iPad Pro.

Set Line Weight to Color Settings...

Line Weight to Color Settings... displays a table allowing you to match specific PowerCADD line weight values to specific pen color numbers often used by other applications to create line weights when printing.



- ❑ **Line Weight** column: Displays the PowerCADD line weight to be mapped to a specific color.
- ❑ **Units** column: Displays the line weight units. To change a unit click on the unit name to choose from Points, Millimeters, or Inches.
- ❑ **Color** column: Displays a color swatch of the value specified in the # column. To change the color swatch specify a different numeric value in the # column.
- ❑ **#** column: Displays the color number which will be assigned to a given line weight. Click on a number to change it.
- ❑ **Color Name** column: Displays the standard name assigned to the color specified in the # column. To change a color name specify a different numeric value in the # column.
- ❑ **+ / -** buttons: To add a new line weight to the table click **+**. To remove an existing line weight from the table click on the line weight entry in the table then click **-**.
- ❑ **Extract From Active Drawing**: Click this button to automatically fill the table with line weights from the active drawing. By default all entries are assigned a sequential color number. Click on a value in the # column and enter a new value to change the color.
- ❑ **Save**: Click to save the currently displayed Line Weight to Color table. A standard MacOS Save dialog appears allowing you to specify a location and file name. Saved settings may be used again with other drawings or shared with other PowerCADD users. For example, a unique line weight to color table may be saved for coordination with a specific consultant to match their internal drawing standards.
- ❑ **Load**: Click to load a previously saved Line Weight to Color table. A standard MacOS Open dialog appears allowing you to locate the previously saved settings.
- ❑ **Cancel**: Click to abort any changes in this dialog and return to the Incoming Settings dialog to complete the translation.
- ❑ **OK**: Click to accept any changes in this dialog and return to the Incoming Settings dialog to complete the translation.

Translating .dwg or .dxf files to PowerCADD

The translation from a .dwg or .dxf file is as simple as opening the file and specifying the translation settings to be used to create a new PowerCADD drawing.

While the Translator automatically interprets the content of the file being read, clear communication with the person sending the .dwg or .dxf file can help make the process even easier.

Items to consider for coordination include:

- ☐ What drawing units were used to create the original .dwg or .dxf file? For example: inches, feet, miles, millimeters, centimeters or kilometers could have been used and affect the scale and drawing units in PowerCADD.
- ☐ What was the intended printed scale and sheet size from Windows? This will help you determine what the PowerCADD drawing scale and drawing sheet size should be.
- ☐ Does the file contain external references (X-Ref's)? If so, have the user *bind* the references into a single file or make sure they include all referenced drawings. If they are sending referenced drawings ensure they identify which file is the master so you know which one to open.

Please review the Translator Reference Guide and Frequently Asked Questions section for additional information.

Since the process for reading .dwg or .dxf file names into PowerCADD is the same, we will use the .dwg translation process for illustration and demonstration purposes. Any translation process which differs from .dxf will be noted.

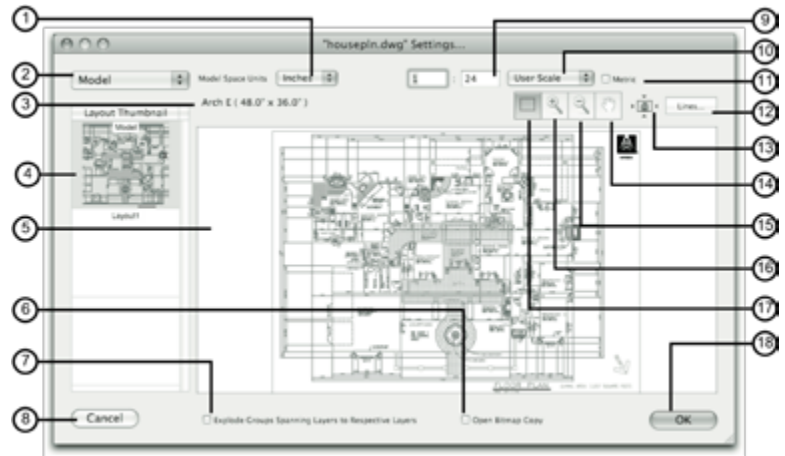


NOTE: To open a .dwg file, the filename must have a .dwg suffix. Files without the correct filename suffix will appear dimmed in the Open dialog.

Incoming .dwg Settings

The Incoming .dwg Settings dialog allows you to assign and preview settings, which directly affect the translation process.

Any time you open a .dwg or .dxf file the following Incoming Settings dialog will appear:



(1) **Model Space Units:** When translating Model Space drawings you may specify the units used to translate the drawing from this pop-up list. When translating Paper Space layouts this pop-up is unavailable.

(2) **Model/Paper Space menu:** Use this menu to select which content should be translated. If there are multiple Paper Space layouts their names are listed in this list.

(3) **Stationery Sheet Size:** Displays the current sheet size of the stationery file selected at the start of the translation process.

(4) **Thumbnail Panel:** A small, thumbnail, view of the .dwg file is displayed here. It will always contain one Model Space view which is displayed first in the panel. Some files will display multiple Paper Space layouts which are displayed in this panel. Click on a thumbnail view to see the contents display in the Preview Pane.

(5) **Preview Pane:** Displays a preview of the thumbnail selected in the Thumbnail Panel. To change the content displayed here click on a different thumbnail. To navigate within the Preview Pane use the Zoom In, Zoom Out, Pan, or Reduce to Fit tools.

(6) **Open Bitmap Copy:** Check this box to open a copy of the drawing in a raster (pixel) format.

(7) Explode Groups Spanning Layers to Respective Layers: When checked a multi-layer block is ungrouped and all objects are inserted onto their appropriate layer in the translated drawing. When not checked, blocks are not ungrouped. A block is translated as a group onto a PowerCADD layer corresponding to the block insertion layer in AutoCAD.

(8) Cancel Button: Click to abort a translation operation.

(9) Scale Fields: When translating Model Space drawings you may directly enter scale values in these fields or choose a scale from the User Scale popup menu to display its values here. When translating Paper Space layouts these fields will display the translated drawing scale and cannot be edited.

(10) User Scale Menu: When translating Model Space content, click here to display a list of standard drawing scales to choose from.

(11) Metric Checkbox: Check this box to change the display options in the User Scale Menu from Imperial (feet and inches) to Metric units.

(12) Lines Button: Click to access additional color to line weight translation features as described later in this section.

(13) Reduce To Fit: Click to change the zoom percentage so all the content of a selected thumbnail fits to the extents of the Preview Pane.

(14) Pan Tool: Click to select this tool. With the cursor in the Preview Pane, click and drag to change which portion of the preview is shown.

(15) Zoom Out Tool: Click to select this tool. With the cursor in the Preview Pane, click to zoom-out of the previewed thumbnail.

(16) Zoom In Tool: Click to select this tool. With the cursor in the Preview Pane, click to zoom-in on a portion of the previewed thumbnail.

(17) Select Area Tool: This tool is available only when the Model Space thumbnail is selected. Click to select this tool. With the cursor in the Preview Pane, click and drag to define a selection rectangle around the model space content to be translated.

(18) OK Button: Click to proceed with a translation based on settings in the Settings dialog.

Model Space and Paper Space

The translator is capable of loading the file from either Model Space or a Paper Space layout.

Model Space and **Paper Space** layouts are the two primary spaces in which objects reside.

Paper Space is used for creating a finished layout for printing or plotting, as opposed to drafting or design work.

Model Space is used for creating and editing the drawing.

Objects created in paper space are visible only in paper space.

Objects in model space can only be edited in model space.

All drawings will contain Model Space information.

Most drawings will contain both Model Space and at least one Paper Space layout.

When translating a drawing you have a choice of opening either the Model Space or Paper Space views.

Opening Model Space Content



The process of opening Model Space content is the same whether the .dwg file contains only Model Space content, or both Model and Paper Space content.

NOTE: You must use the Select Area tool in the Settings dialog to define a selection rectangle (marqu e) which includes the data you want translated in Model Space. If no selection is made the entire model space content is translated.



HINT: To follow along with the Model Space instructions use the housepln.dwg file included on your **master installation CD**.

To translate Model Space content:

- ☐ Choose Open from the File menu, locate the .dwg file, and click Open.
- ☐ The progress indicator will appear. After the drawing has processed the New Document dialog opens asking you to choose a Stationery drawing to be used to process the file.
- ☐ Choose the desired Stationery drawing and click Choose. The Settings dialog will open.



- ☐ If the drawing has only model space data then Model Space is automatically selected from the Layout Thumbnail Panel. If your drawing contains both Model Space content and Paper Space layouts, click on the Model thumbnail. A preview of the drawing appears in the Preview Pane.
- ☐ Choose the Select Area tool. Click and drag in the Preview Pane to define the area of the drawing to be translated. Drag a marquee that includes all the model space content to translate the entire drawing. If you can't see all the content use the Zoom Out tool.
- ☐ The scale fields activate in the upper left corner of the Settings dialog. Type a scale or select from any predefined industry standard drawing



scale from the popup list. If you enter a scale that would cause the selection to not fit on the drawing page, PowerCADD will prompt you with the minimum scale required.

- ❑ Click OK and the drawing appears in your custom stationery pad at the specified scale.



NOTE: A new layer titled “ES IMPORT INFO” is added to the drawing. It is the top most layer and hidden by default. This layer contains detailed information about the settings used to translate the file.



NOTE: Remember drawing scale and sheet size are related when translating Model Space content. If the drawing scale you choose will not fit on your chosen sheet size, Cancel the translation operation and start again, choosing a larger sheet size from your list of Stationery drawings.



NOTE: All layers of the file are translated. The layer status (On or Off, locked) in PowerCADD matches the layer status based on the last time the file was saved.



TIP: To translate only a portion of the Model Space content, use the Select Area tool to drag a marquee over just a portion of the drawing area displayed in the Preview Pane.

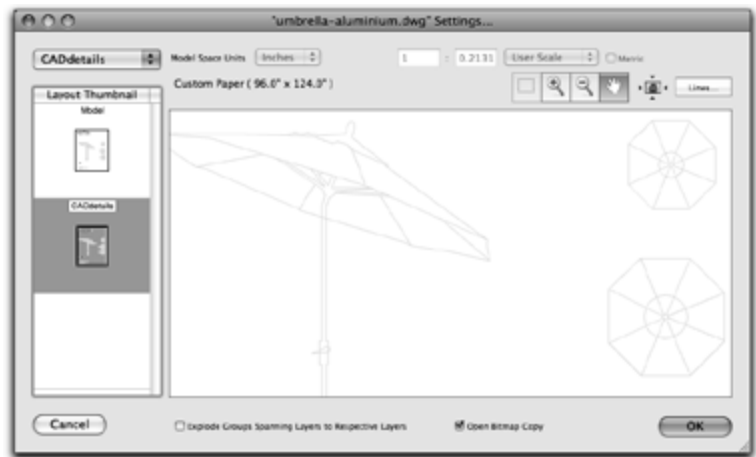
Opening Paper Space Content

A Paper Space layout translation allows you to create, with a single click, a PowerCADD drawing which matches the Paper Space layout content.

Paper Space Layouts are typically used to display the Model Space content at a printable scale, similar to how you start drawing in PowerCADD (to scale). In most cases the layouts also contain a title block and additional drawing notes.

To translate content from a Paper Space layout:

- ☐ Choose Open from the File menu, locate the .dwg file, and click Open.
- ☐ A progress indicator will appear. After the drawing has processed, the New Document dialog opens asking you to choose a Stationery drawing to be used to process the file.
- ☐ Choose the desired Stationery drawing and click Choose. The Settings dialog will open.



- ☐ Click on a thumbnail of the Paper Space layout. The Preview Pane updates to display the entire layout.
- ☐ Optional: Use the zoom tools to zoom in, out, or pan to examine the drawing more closely prior to translating.
- ☐ The paper size and drawing scale has been set by the sender and cannot be changed.
- ☐ Click OK and the drawing appears in your custom stationery pad at the scale specified by the originator.

If the custom stationery drawing you specified was larger than the actual Paper Space layout, simply perform a page setup to adjust the page size accordingly.

If the user specified a scale that was too large to fit on your stationery you may need to use Scale from the Edit menu to make it small enough to

fit the page, or translate the file again using Model Space and specify an appropriate scale.

Examine the information on the “ES IMPORT INFO” layer from the first translation for helpful information on determining an appropriate scale.



NOTE: The Custom Print Scale assigned to the View Port, as defined by the originator when creating the layout, is used to determine the drawing sheet size and scale of the translated drawing.

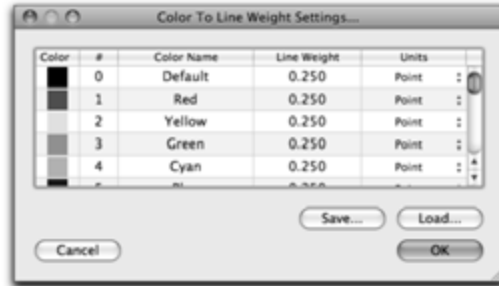


NOTE: Only layers visible in the View Port are included when the drawing is translated into PowerCADD. Hidden layers in the View Port are not translated. If all hidden and visible layers are required translate using Model Space instead.

The Lines button in the Settings dialog gives you access create a table for mapping incoming Pen Colors to PowerCADD Line Weights.

To change the Line Weight translation settings:

- ☐ Click Lines in the Settings dialog. The Color To Line Weight Settings dialog will open.



Color To Line Weight Settings... displays a table allowing you to match specific pen color numbers to PowerCADD line weights.

Color column: Displays a color swatch of the value specified in the # column. This value cannot be edited.

column: Displays the color number to which a line weight will be assigned. This value cannot be edited.

Color Name column: Displays the standard name assigned to the color specified in the # column. This value cannot be edited.

Line Weight column: Displays the PowerCADD line weight to be assigned to a specific color. Click in the field and enter a number to change the line weight.

Units column: Displays the line weight units. To change a unit click on the unit name to choose from Points, Millimeters, or Inches.

Save: Click to save the currently displayed Color to Line Weight table. A standard MacOS save dialog appears allowing you to specify a location and file name. Saved settings may be used again with other drawings or shared with other PowerCADD users. For example, a unique line weight to color table may be saved for coordination with a specific consultant to match their internal drawing standards to yours.

Load: Click to load a previously saved Line Weight to Color table. A standard MacOS open dialog appears allowing you to locate the previously saved settings.

Cancel: Click to abort any changes in this dialog and return to the Incoming Settings dialog to complete the translation.

OK: Click to accept any changes in this dialog and return to the Incoming Settings dialog to complete the translation.

- ☐ Make the desired changes to the Line Weight and Units columns for each color number and click OK.
- ☐ You are returned to the Settings dialog to complete the translation.
- ☐ The translated drawing will contain specified line weights for each color specified. The objects color is also retained.

Xrefs (Referenced Drawings)

To correctly translate .dwg files containing Xrefs, all Xref files must be in the same folder as the Master Drawing before translating.

Xrefs, or references drawings, are commonly used to combine multiple drawings into one Master, or Root, Drawing. The Translator is capable of binding those Xref files for you during the translation process. The Master Drawing is the file you want to translate as it links all the other drawings together. If you don't know which drawing is the Master Drawing contact the original drawing author.

To translate a .dwg file that uses Xref's:

- ☐ Place the Master Drawing and all Xref files in the same folder.
- ☐ Choose Open from the File menu and open the Master Drawing.
- ☐ Proceed with the file translation process as described earlier.



NOTE: There is no warning dialog for missing reference drawings. If your drawing looks incomplete, perform a visual audit of the file and contact the original file author for all related reference files.

Translator Reference Guide

The Reference Guide is designed to provide information to you and your clients who send or receive drawings. Plan ahead by alerting your clients of potential problems. You will save time in the long run.

The .dxf file format allows users of different CAD software to exchange drawings through a common format. It also allows those drawings to be sent to CNC equipment and analytical programs. The Drawing Interchange Format (DXF) is supported by most CAD programs. The .dwg file format also allows for the exchange of drawings, but is not supported by all CAD programs. Generally, .dwg will be a preferred method of file exchange when dealing in a cross-platform environment with most design professionals.

As with any channel of communication, a less destructive interference means more accurate and timely data. While the transfer of data between PowerCADD and other CAD programs is virtually seamless, communication problems can and do occur. Planning is beneficial to success in all aspects of drawing, especially when exchanging data between clients. Communication between drawing file users is mandatory.

Many translation problems can be prevented by substituting or avoiding certain types of objects. It helps to know when drawing is destined to be translated so substitutions can be made during a drawing's initial construction. If that is not possible then editing the file before translation, using the wide range of features in PowerCADD, is possible and may be necessary to ensure the best possible translation result.

The information presented is applicable to most any .dxf or .dwg file created with many programs. If AutoCAD will not open the file, there is a good chance the Translator will not either.

Whether you use the Translator External to read files on a daily basis or once a month, you should read this entire guide at least once. It will help you respond more quickly to problems and ask the right questions.

While the most common use for this guide will be a trouble-shooting reference, you could also use it as a translation checklist.

Before you send a file to a client, *check off* each item in the appropriate section.

If you receive a drawing that is difficult to translate, check off the possible problems. If all else fails, call your client and use the guide to ask questions. Communicate!

The guide has two main sections:

- ☐ PowerCADD to dwg & dxf
- ☐ dwg & dxf to PowerCADD

Each main section is separated into smaller divisions that address specific commands, objects, and ideas. Each division is listed in the table of contents for quick referencing.

PowerCADD to .dwg and .dxf

The following information is applicable to sending files created in PowerCADD to other CAD programs running on a Windows-based personal computer. With the exception of DOS file naming conventions, and a few minor changes, opening a .dwg or .dxf file in other applications on an Apple or Windows operating system are similar to those described here. Please reference the documentation for the other applications or operating systems for additional information.

File Names

DOS is very specific about file names. Spaces are not allowed. When naming your file for a DOS-based system, use up to eight characters or numbers. The Translator External will add a period (.) and the 3-digit file extension, .dxf or .dwg, automatically when saving. For example:

FILENAME.dwg

While the Windows Operating System does support longer file names, it still requires a 3-digit file extension. As a general guide, avoid the use of special characters in file names, as those may not translate correctly between all operating systems.

Scale

PowerCADD handles scale differently than Windows programs.

For example, some programs will allow you to draw a large commercial building site at full scale with a paper size measured in acres rather than inches. The user then creates a paper space layout with a scale assigned for printing.

PowerCADD scales a drawing for use on standard drafting paper sizes, just like manual drafting.

A drawing that leaves PowerCADD at 1:48 scale (1/4"=1'0) will arrive in Windows programs at full scale 1" = 1".

Text styles and fonts set up in PowerCADD will become the standard text style in other programs. All PowerCADD text, whether caption or multiple lines will become single line text when translated into .dwg.

If you are sending text content that will not be edited (or should not be edited) and want to retain your exact look and feel, you might consider PowerCADD Convert to Outlines menu item. Convert to Outlines can convert your text content to Bézier curve outlines. While the content cannot be edited as text, it will retain the precise look and feel. A practical use for Convert to Outlines might be for a company logo in a drawing title block.

Caption text in PowerCADD (single line text) will be single lines of text when translated via .dwg or .dxf. Paragraph text in PowerCADD will map to a single text block when translated. In general the following font mapping is used when reading and writing .dwg or .dxf files (PowerCADD Font/Windows Font):

Geneva/Arial
Courier/Courier New
Times/Times New Roman
Helvetica/Helvetica
Arial/Arial
Times New Roman/Times New Roman
Georgia/Georgia
Arial Black/Arial Black
Impact/Impact
Verdana/Verdana12
Tahoma/Tahoma
Palatino/Palatino.

Font styles of regular bold, italic, and underline are supported for the above-noted fonts.

Fill Patterns

Macintosh fill patterns do not translate. There is no such thing as a fill pattern in AutoCAD. Convert all fill patterns to hatch patterns.

Line Types and Pen Fills

Macintosh pen patterns do not translate. Convert all pen patterns to solid or dashed lines before translating. By default, all objects with pen patterns are converted to solid during the translation process. Dash patterns on a line with pen patterns are retained.

Pen Widths

Pen widths do not translate as WYSIWYG line weights; they will become the default pen width.

Symbols

Symbols created in PowerCADD will translate as a block, which is simply a group of objects. The block name will assume the PowerCADD symbol's name. If your symbols used additional Symbol Tag fields, that data is ignored when saved as .dxf or .dwg.

Groups

Grouped objects created in PowerCADD will translate as an unnamed block.

Attributes

Object attributes such as opacity, shadows, fill or pen pattern, fill or pen color do not translate.

Sending DXF or .dwg files created in PowerCADD for use in other applications

Here are a few tips on sending files:

- ❑ If you are sending files on a CD-ROM to be used in the Windows operating system, format the CD to be used on that system. Some third-party CD Burning software requires you choose that option separately. If you are burning the CD using the MacOS built-in burning software the CD is automatically burned in a format supported by both the MacOS and Windows.
- ❑ Always compress files before sending them by email as this protects the file from possible data corruption during the email encoding and decoding process.

Use third-party applications such as ZipIt, DropZIP, or the MacOS Create Archive command. PC users will generally not be able to read files compressed in formats other than zipped (*.zip).

If you are sending files to another Macintosh user, you may use either the MacOS Create Archive command to create a zipped archive (*.zip) or a third party utility such as Stuffit to create a Stuffit Archive (*.sit or *.sitx).

If you are using a Stuffit Archive the recipient must have Stuffit Expander installed which is not common on all systems.



For ease of coordination, it is typically easiest to use a *.zip archive created with the MacOS Create Archive menu item as it can be read natively by PC and MacOS users alike.

- ❑ Be sure your files have a valid file extension at the end of the name such as: FLOORPLN.dwg or FLOORPLN.dwg.

If the file was zipped, it would have a .ZIP extension. The file should have ONLY ONE file extension and only one 'dot' (.) in the file name.

For example, FILENAME.dwg.ZIP is INCORRECT and should be changed to FILENAME.ZIP.

- ❑ Ask the person receiving the .dwg or .dxf file what version of software they are using to find out what version their application can read. When saving your PowerCADD file as .dwg or .dxf, click Options in the dialog and choose the appropriate version from the list.

Dxf and dwg do not support bit images.

If you have important bit-imaged shapes in the PowerCADD file, trace or redraw them with lines and B-Spline or Bézier curves.

This will create a polygon, which will translate as polylines.

If it is critical your bit image be part of the file, you can send them as separate attachments along with the .dxf or .dwg file.

The receiver will then need to use the Insert command to bring your bit images into the drawing.

*Opening the .dxf File
in AutoCAD*

DXF is imported using the DXFIN command, which can be full or partial.

A full DXFIN will load all definitions, layer information, block references, etc. along with the entities.

In order to get a full DXFIN, the user must load into a new (generic or empty prototype) drawing.

A partial DXFIN is one that is loaded into an existing drawing. Incoming data like block definitions and layer information will be ignored and overridden by the current drawing's definitions. Only entities will import.

*AutoCAD Error: "Not a new
drawing, only Entities section
will be input."*

When a full DXFIN is opened into an existing drawing or a modified default drawing, an error message will state: "Not a new drawing, only Entities section will be input."

General

Make sure you send a clean drawing. Delete objects that are off the drawing page, using PowerCADD's Cleanup Drawing menu item. Convert any smashed polygons, arcs, circles, and curves back to one-piece polygons with Attach. Delete stray marks that may appear outside of the main drawing's border.

Tips when requesting Windows files for use in PowerCADD

Following are recommendations for coordination when requesting files for use in PowerCADD. Take the time to discuss and test the translation process prior to a deadline to establish the specific process for a given project.

- ❑ Request a printed hard copy or Adobe Acrobat PDF file of the drawing sheets you will be translating. This will assist you in setting up the correct drawing scale, units and sheet size when reading the file into PowerCADD.
- ❑ Given the choice between .dwg or .dxf files, request .dwg. Using .dwg is easier for both you and the Windows user as it requires fewer steps for them in preparing the files and retains more drawing information resulting in a better translation. If .dxf is the only file type available, make sure the user exports the file using the DXFOUT command as an ASCII (not binary) file with an accuracy of 6 decimal places (the default). PowerCADD and other Macintosh applications cannot read Windows binary data.
- ❑ Have the user create a Paper Space Layout containing the data you need in PowerCADD. This includes the sheet size, printed drawing scale (your PowerCADD drawing scale) and visible layers. Typically most Windows users already create paper space layouts for their various drawing sheets, but if you don't see one in the printed hard copy as noted in item 1 above then ask for what you need. The process of creating the layout is a common task and one that will take only a few moments for them.
- ❑ Make sure they package together all the X-refs (referenced) drawings along with the main (master) file. It is common practice to use one master drawing with multiple referenced drawings, but if information you received does not have all the referenced content the translation will be incomplete.
- ❑ If it is important to maintain precise font formatting, use one of the standard supported fonts as outlined earlier in this section.
- ❑ If the drawing is being issued via email, make sure they zip (compress) the file before attaching it to the email message. They should NOT create a self-extracting archive (*.EXE) as you will not be able to extract that content without using the Windows Operating System. If the files are being issued on CD or DVD discs compressing them is not required.
- ❑ If the drawing is to be issued on a CD or DVD disc, no special steps are necessary on the part of the Windows user. Your Macintosh can read a Windows formatted CD or DVD.

Tips for opening Windows files in PowerCADD

Keep these tips in mind when receiving the .dxf or .dwg files.

- ❑ Copy the file(s) from the CD-ROM disc to your hard disk. Do not try to translate the file directly from the disc.
- ❑ If the files are compressed (with a “.Zip” extension in the name) decompress them with ZipIt, Stuffit Deluxe or another compression program that reads ZIP files. In MacOS, double-click the zip file to automatically extract its contents.
- ❑ Open the file from the File menu in PowerCADD or drag and drop the file onto the PowerCADD application icon in your Dock or in the PowerCADD application folder. If you have not specified PowerCADD as the default application to open .dwg or .dxf files in MacOS, double-clicking the file in the Finder may open the file in another application. Reference the MacOS help files to learn how to change the application that opens a document.
- ❑ If you can't open the file in PowerCADD, try opening the file with a word-processing application to make sure it is a valid .dwg file. The first entry in a valid .dwg file will have something similar to “AC1018” as the first entry. This refers to the version number used to create the file so the actual numbers shown in your example may vary from what you see. However, it will always begin with ‘AC####’. If you don't see that, then check with the original file author to determine just what type of file was sent.
- ❑ If the file was received via email, make sure it was compressed first. It is not uncommon for uncompressed files to become damaged when sent as email attachments. If the sender is using the Windows operating system have them use WinZIP to create a zip archive (filename.zip). If the sender is using a Macintosh operating system have them create a zip (filename.zip) or Stuffit archive (filename.sit).
- ❑ Make sure you have all the X-ref (referenced) drawings and any special fonts you might need. Please reference “Tips for requesting Windows files for use in PowerCADD” discussed earlier in this section.

.dwg or .dxf to PowerCADD

The following information is applicable to opening .dxf or .dwg files in PowerCADD.

Limits and Extents

It is possible to receive a drawing whose Limits are much larger than its Extents. Limits can also be smaller than Extents. This tends to be confusing and for good reason.

Unusually Large Limits

The textbook method of setting Limits is to set them slightly larger than the objects drawn to accommodate a border and title block. If the Limits are huge compared to the actual object boundary, it is possible the drawing you received is a small part of a much larger drawing.

External References (Xrefs)

By creating Xrefs, users can work on many different parts of the same drawing simultaneously. Each time the Xref's source drawing is updated, it appears in each of the group member's Xrefs. This allows members of the design team to make corrections on their part of the drawing during the initial design phase. In principle, this is similar to using References in your PowerCADD drawings.



NOTE: Some attribute information may come through in the translation, if it was actually visible on the drawing itself. It will appear as text, usually near the object it represents, but it will not be linked to the object as an attribute. Make sure you request all Xrefs from the original file author to ensure a complete translation.

Scale Basics

Look at the Architectural scale $1/4" = 1'0"$. It says each $1/4$ inch of the drawing represents one foot of the actual size of the building at the site. You can also say that one inch of the drawing represents four feet of the real size of the building because $4 \times 1/4" = 1"$.

Now, four feet is equal to 48 inches, so you can say one inch of the drawing represents 48 inches of the real size of the building, or 1:48 scale. These are the two most common ways of representing one of the many scales used in drafting, $1/4" = 1'0"$ and 1:48.

PowerCADD Drawing Scale

Drawing scale is a difficult concept for many CAD users, especially those who have not done much manual drafting. PowerCADD *drafts* on the Macintosh very similarly to the way you draw on a drafting board, in a reduced scale, so it will fit on a standard sheet of paper.

Both manual drafting and PowerCADD require *paper and rulers*, and both will produce a scaled-down drawing when printed.

For example, if you are creating a house plan in PowerCADD in $1/4" = 1'0"$, you would first get out a sheet of paper (Page Setup) and then your architect's ruler or *scale* (Drawing Setup).

The next item needed is your pencil (Line Tools).

When completed, a builder will take your scaled drawing to the building site and produce a real, life-sized model.

Each $1/4"$ on the drawing will equal $1'0"$ at the building site. The builder will read your scaled drawing in real life numbers, or real world coordinates.

Windows Drawing Scale

Windows handles drawing scale opposite of manual drafting.

For example, when you set a sheet of paper (set Drawing Limits), you are choosing a *sheet of paper* in full scale, or the actual size of the building site.

Drawing is like going to the building site with a long tape measure and really big pencil and laying out the house plan in the dirt, one foot equals one foot.

Drawings are stored on disk this way in World Coordinates. Only when it is time to plot, on standard-sized drafting paper, do you apply a reduced scale to the drawing.

It is the lack of scaling that can be confusing when trying to set the scale of an incoming file.

Many of the other CAD programs also draw in full scale, but may use different terminology to describe it.

Scale Information Received

When opening drawings in Model Space, the Translator External treats the incoming drawing coordinates as FULL SCALE INCHES. For example, if the user typed in 10'6" while drawing a line, that line would appear as 126.0 in the file.



NOTE: There are some rare conditions where the scale of a translated drawing may appear to be off by a factor of twelve, which is easy to correct. Choose Drawing Setup from the Layout menu. Multiply the scale by 12 and type that into the scale text field. For example, if you specified 1:40 scale in the translation dialog, you should now specify 40 x 12 or 480. Alternately, you may perform the math in the dialog by typing the scale as 40*12.

Paper Space

Paper space allows you to plot different views of the same drawing, at different scales all at once. The plotted end result looks like several different drawings on one sheet. On disk, it appears as one drawing file. The Translator will show you any paper space layouts in the translation preview window. Select the desired paper space layout to translate the file.

Rel. 9 or Earlier

The Translator External may not read .dxf or .dwg files correctly if created with versions before Rel. 9.

Text/Fonts

Because the Macintosh handles text and fonts differently from Windows, any non-proportional text will appear as normal text in PowerCADD. Text aligned or fitted to the starting and ending points will be left justified only from the starting point.

Blocks

Blocks will translate as PowerCADD groups. The block names will be translated and can be read and printed with the PowerCADD's Symbol Report. Unnamed blocks will appear as grouped objects. Write Blocks, Wblocks as they are called, will be treated only as blocks.

Identifying Solid Hatches

Once the drawing is in PowerCADD, look for arrowheads used in section details, North indicators, and dimensions or leaders. If you spot any object that appears to have a solid fill, select it and try to ungroup it a couple of times. You may see the handles of several hundred lines appear.

Converting Solid Hatches to PowerCADD Fills

To convert solid hatches to fills:

- ☐ Open the drawing in PowerCADD.
- ☐ Delete all of the lines that created the filled area.
- ☐ Use Attach to create a polygon from the hatch's border.
- ☐ Use a PowerCADD fill pattern to refill the object.

3D Objects

True 3D objects, meshes, and solid models are not supported in PowerCADD. If the file being translated contained 3D data, the last saved view will be translated as a 2D view in PowerCADD.

Conclusion

This reference makes no attempt to give all of the comparisons, nor can it address every single situation when translating .dxf and .dwg files. It does provide information about these programs you can utilize when communicating via .dwg and .dxf file exchanges. We hope the guide will help you avoid problems. Knowing something about how your clients' files originate will save time, effort, and aggravation. Plan and communicate.

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Raker, D. & Rice, H (1991) Inside AutoCAD. Gresham. Oregon: New Riders

Frequently Asked Questions

Will using .dwg instead of .dxf retain my fill patterns? No, there is no equivalent to bit image fills.

Why do my fonts change when sending a .dwg file? Text is mapping to a single character type, but the bounds of the text should be correct. Caption text in PowerCADD (single line text) will be single lines of text when translated via .dwg or .dxf. Paragraph text in PowerCADD will map to a single text block when translated. In general the following font mapping is used when reading and writing .dwg or .dxf files (PowerCADD Font/Windows Font):

Geneva/Arial
Courier/Courier New
Times/Times New Roman
Helvetica/Helvetica
Arial/Arial
Times New Roman/Times New Roman
Georgia/Georgia
Arial Black/Arial Black
Impact/Impact
Verdana/Verdana12
Tahoma/Tahoma
Palatino/Palatino

Font styles of regular bold, italic, and underline are supported for the above noted fonts.

Is there any way to create a drawing without the use of clipping? Absolutely. Using the various trim, clipping and break tools or smart tools such as the parallel line, door and window tools that automatically cut and heal lines are options to consider when creating your PowerCADD drawing.

What I can do to improve the performance of translating? Ungrouping objects will significantly improve the performance, sometimes as much as 6X.

My curved arrowheads seem straight in Windows, what happened? There is no object for curved arrowheads in AutoCAD, and they are turned into polylines, which are straight.

Why do my layer names change after the conversion to .dwg? The case of layer names is changed to all UPPERCASE

Will Layer Scales be preserved? No. The drawing will have the scale selected in drawing setup applied.

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