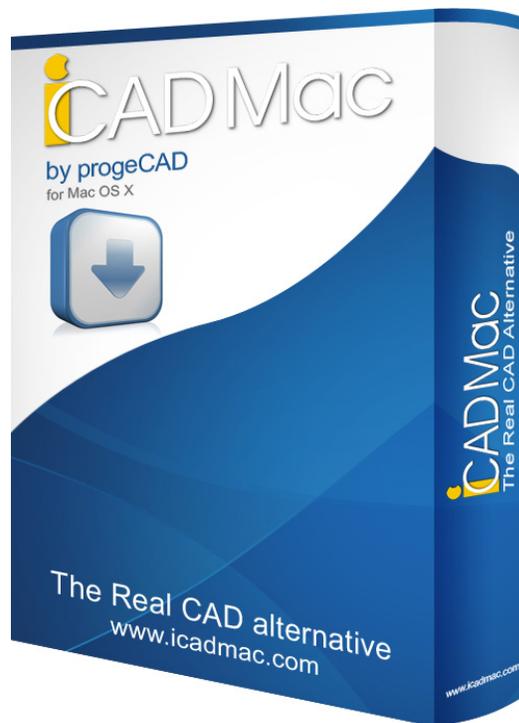


# Inside iCAD Mac



[www.icadmac.com](http://www.icadmac.com)

by Ralph Grabowski

iCAD Mac



[www.icadmac.com](http://www.icadmac.com)

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Part I

# Introduction to iCADMac



## Chapter 1

# Introducing iCADMac for Macintosh

### In This Chapter

- Welcome to iCADMac for Macintosh!
- Information about iCADMac

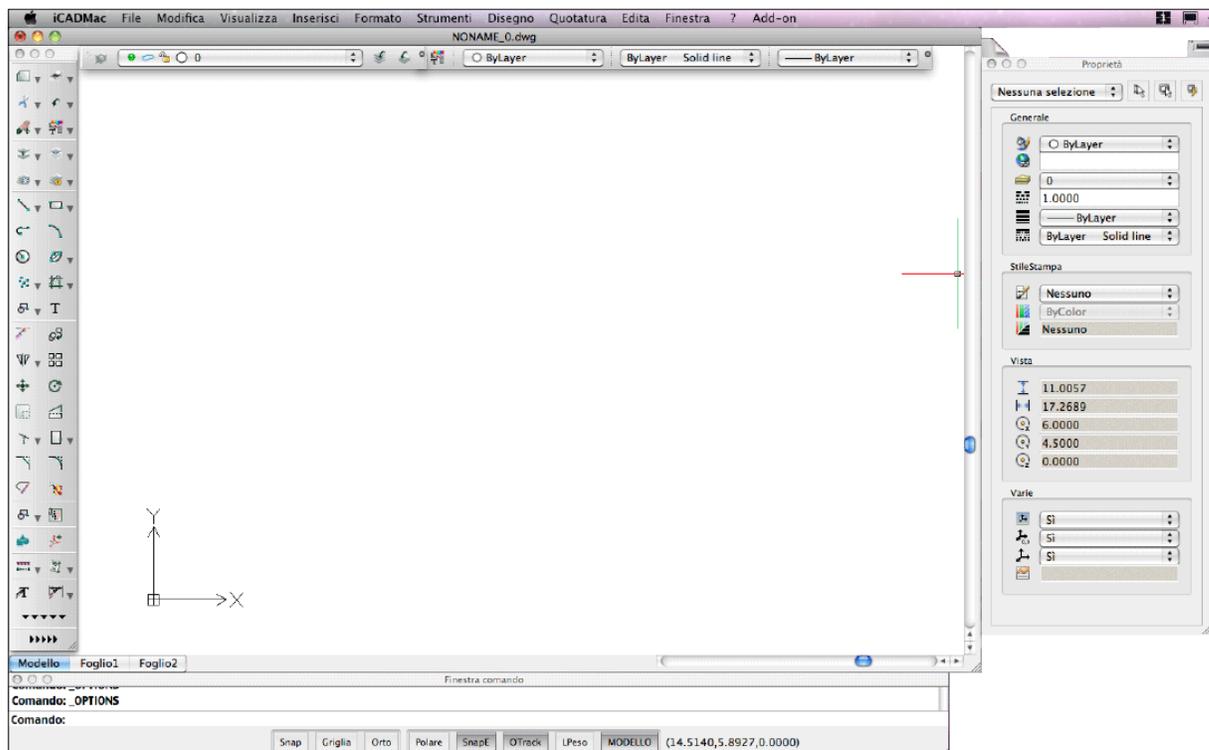
**W**elcome to iCADMac for Macintosh!

In this chapter, you learn about one of the most modern CAD software packages available today. iCADMac is the result of a five-year effort to write a brand new computer-aided design program based on the latest innovations in software.

The iCADMac CAD software is for day-to-day design work and 3D models. It works on computers running OS X.

### About iCADMac

iCADMac is a brand-new CAD program, yet it is compatible with all of your existing DWG and DXF files (up to R2012). It has its own command structure, yet it works with commands, options, and system variables found in programs like AutoCAD and progeCAD. Its user interface will also look immediately familiar to you.



iCADMac has standard 2D drawing commands like Trim and Fillet, as well as advanced functions like associative dimensioning and hatching. In addition, iCADMac displays creates and modifies 3D ACIS drawing objects.

### Benefits of Using iCADMac

ProgeCAD drew its three decades of experience in the CAD business to build iCADMac from scratch, using the most current computer programming technology. iCADMac offers you a number of significant benefits.

#### Impressive Compatibility

iCADMac uses DWG as its primary file format for storing drawings. It incorporates the DWG engine of Open Design Alliance.

It can read and write DWG and DXF files from as far back as R12 and as recently as R2012, and exchanges data with a wide range of graphics and construction software, such as ACIS SAT, PDF, and SVG.

If needed, iCADMac can protect your data investment by recovering DWG and DXF files that may have become damaged.

#### Familiar User Interface

The user interface is built on Qt technology, which means that it looks and works as CAD softwares for Linux and Windows computers. It features dockable toolbars, right-click shortcut menus, command line input, and palettes.

The user interface is fully customizable without programming. (Programmers can, however, define and manage user interface elements through XML resources.) You can create and store multiple

configurations (a.k.a. profiles or workspaces) to adapt the user interface to specific project needs.

Aliases and keyboard shortcuts let you quickly enter frequently-used commands, as well as make iCADMac operate like other CAD programs. The menu structure and toolbar buttons can also be customized.

### **Drafting Tools**

iCADMac contains all the smart drafting tools you expect in today's CAD software, such as numerous entity snap modes, entity tracking, and polar guides. You can create custom coordinate systems to align the construction plane with drawing objects.

You can reposition, align, copy, and resize entities using grips. Filtered selections modify selection sets of entities. The Properties palette directly manipulates the geometry and properties of entities, including layers, colors, line styles, and line weights. The Property Painter copies properties from one entity to others.

Dimensions, hatches, and gradient fills are associative, which means they update automatically. Styles can be created for dimensions, text, tables, and RichLines (multilines) to ensure conformity with industry standards. You can annotate drawings with notes, tables, leaders, geometric tolerances, and text attributes.

### **Collaboration**

Other drawings and images can be attached as references, which lets you share files among a workgroup. Referenced drawings can be clipped to show just portions of them. The References palette lets you control attached drawings and images.

You can add mark up drawings with freehand sketches, and mask portions as needed.

The Options center manages all drafting and system preferences in a single location. Profiles let you store and restore multiple user settings.

### **Organized Layouts**

Drawings can be organized into multiple tiled layouts using tabbed sheets. Each tiled viewport can show the model at a different scale and from a different viewpoint. Drawing borders, title blocks, and equipment lists can be placed on sheets as required.

In sheets, you can switch between model and paper space with just a single click. Each sheet can determine a different style of printing or plotting. Page layouts save and reuse print settings for printers and plotters.

### **Solid Modeling**

iCADMac includes a 3D solid modeler based on ACIS, which lets you work with 3D solid primitives, perform Boolean operations, and slice and intersect solids. You can edit the edges, faces, and bodies of 3D solids.

You can extrude or revolve 2D entities into 3D models, sweep 2D entities along paths, and create

lofts between them. Regions transform areas into 3D solids.

### **Application Programming Interfaces**

iCADMac provides you with a host of programming languages and interfaces for creating your own functions and plug-ins. The Macintosh version supports the following APIs:

- LISP
- LISP reactors (VL- functions)
- C/C++
- Design Runtime Extension (DRX)

### **Platform**

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iCADMac is designed for Macintosh OS X 10.5.8 or higher.

### **Language Support**

iCADMac is initially available in English and Italian. Other languages will follow.

### **Demo Version**

To work through the tutorials in this book, you can download a free, 30-day demo version from [www.iCADMac.com](http://www.iCADMac.com). It requires no registration, and performs all functions, except for saving and printing. After you register yourself, the save and print functions are enabled. (Note that printed output will have a “Demo version” stamp.)

### **After-sales Service**

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ProgeCAD continually updates the software. To ensure you are notified of the latest improvements and have full access to its support teams, consider taking out a service subscription.

- **iCARE** for 12 months of updates and technical support coverage.

For pricing and contract details, contact iCADMac’s Sales Office at [sales@icadmac.com](mailto:sales@icadmac.com)

### **iCADMac for Third-party Developers**

#### **Programming Languages and Interfaces**

iCADMac has a built-in LISP interpreter, as well as full support for DCL (dialog control language), Diesel macros, and most Visual LISP reactors. The program also features autoloading of add-ins, C/fdt header, conversion and headers from ADS (AutoCAD development system), C++, DRX API from Open Design Alliance, Delphi, and toolbar and menu macros.

Because OS X does not support Active X, COM, VSTA, and other APIs proprietary to Microsoft, the Macintosh version of iCADMac also does not support these programming interfaces.



## Chapter 2

# A Quick Tour Through iCADMac for Macintosh

### In This Chapter

- Starting iCADMac
- Becoming familiar with the user interface
- Drawing lines
- Reversing mistakes
- Accessing help

**y**ou begin by learning how to start iCADMac, and then take a tour of its user interface. You get your feet wet by placing a few lines in a new drawing.

### Starting iCADMac

If iCADMac is not yet set up on your computer, install it first. Once installation is complete, you may wish to drag the iCADMac icon from the Application folder to the dock; this makes it easy to find and start the program.

To start iCADMac, click the iCADMac icon found in the Application folder or the dock of your Macintosh computer.



## Key Terms in This Chapter

**Cursors** provide feedback from OS X and iCADMac.

**Pick button** is the left button on the mouse.

**Pickbox** specifies the points being picked (selected).

## Abbreviations

<b>UCS</b>	User Coordinate System
<b>F</b>	Function key
<b>U</b>	Undo (undoes the last command or option)
<b>UI</b>	User interface

## Commands

<b>Command</b>	<b>Shortcut</b>	<b>Menu Selections</b>
Help	? or <i>fn</i> +F1	Help   * Help
UCSIcon	...	View   Display   UCS Icon
Exit	command+Q	File   Exit
Line	L	Draw   Line
Properties	command+1	Tools   Properties
Undo	command+Z	Edit   Undo

\* The vertical bar separates menu selections. For instance, from the Help menu, select the 0 item.

## System Requirements

iCADMac runs on any recent model of MacBook, Mac mini, iMac, and Mac Pro computer. The operating system should be at least Mac OS X 10.5.8, or higher. Here is the recommended system configuration:

Mac with Intel processor

OS X v10.5.8 (Leopard) or v10.6.x (Snow Leopard) or LION (10.7.x)

1GB of RAM (2GB recommended)

2GB of hard disk space available for installation

1024x768 display (1280x800 recommended) with 16-bit video card

Mouse

An Internet connection for activation and installation from download (optional)

DVD-ROM drive for installation from DVD (not required for installation from download)

(Alternatively, click the Application icon on the dock, select **All Programs**, followed by **iCADMac**)



## The iCADMac Window

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When iCADMac is done loading, it displays a window consisting of a graphical drawing area surrounded by information areas on four sides. See the large figure on the next two pages.

Along the top, you see the title bar, menu bar, and several toolbars.

In the middle is the drawing area. Here you see the red-green crosshair cursor with its pickbox, as well as the coordinate system (CS) icon with its x and y axes.

To the left and right are palettes: Tool Matrix on the left, and Properties on the right.

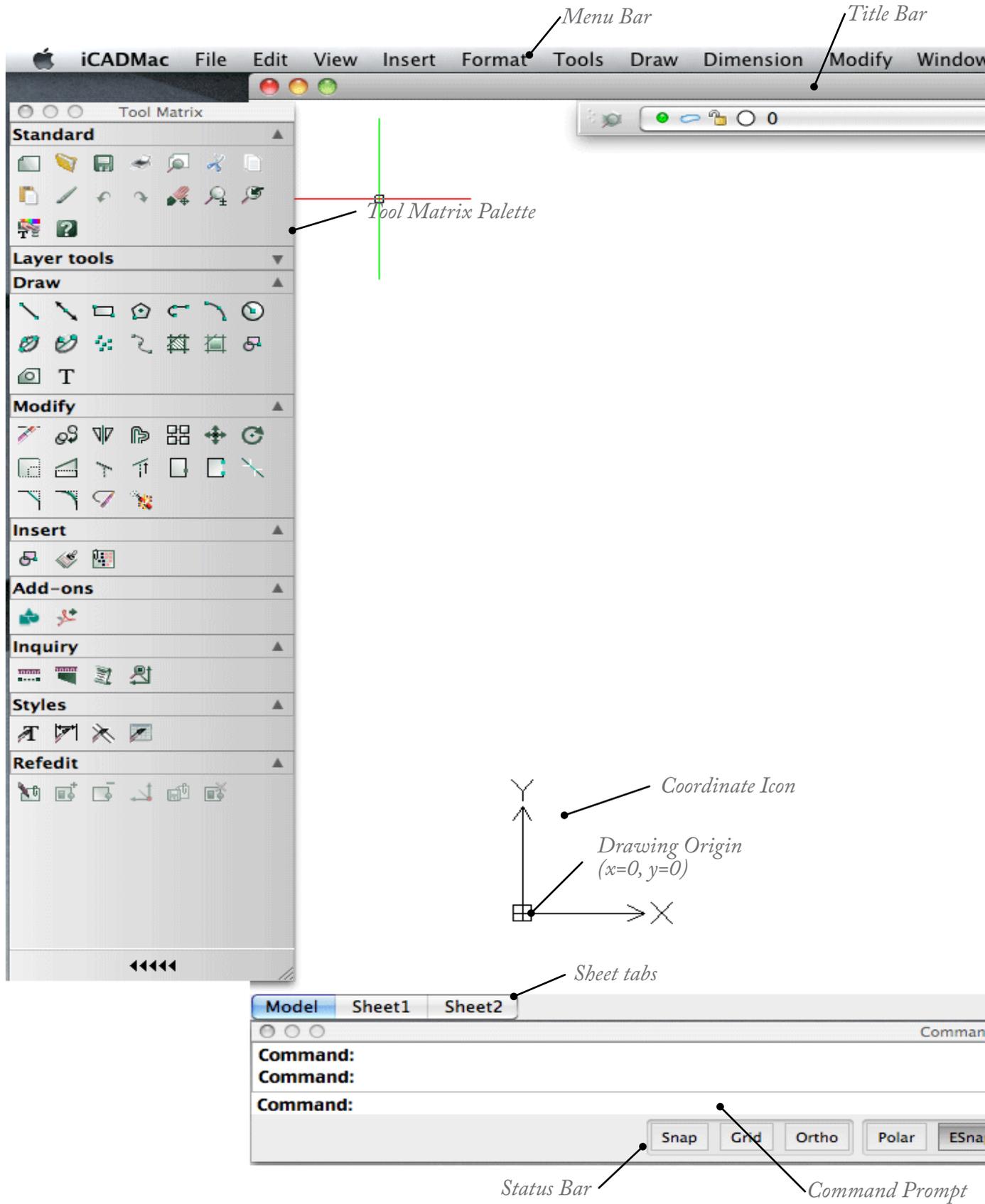
At the bottom are model and sheet tabs, the command prompt window, and the status bar.

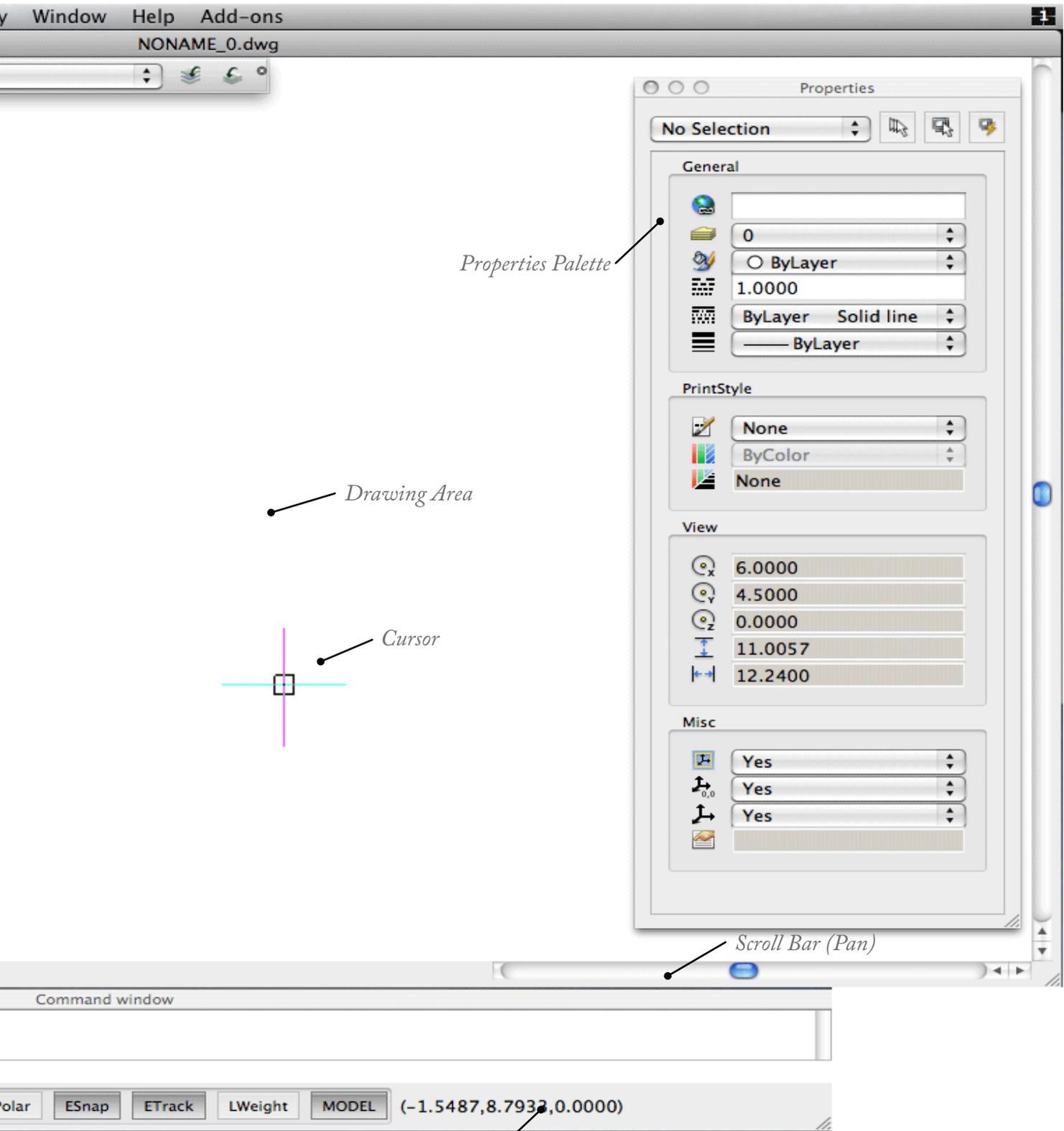
## Basic User Interface Tour

The many aspects of the iCADMac user interface can be daunting to learn all at once, so here we look at just a few items:

- The crosshair and arrow cursors
- The menu bar
- The command line area
- The UCS icon

Chapter 4 provides a detailed look at the iCADMac user interface.





Properties Palette

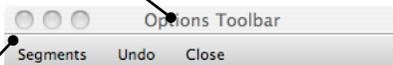
Drawing Area

Cursor

Scroll Bar (Pan)

X, Y Coordinates and Elevation (Z)

Drag to move toolbar



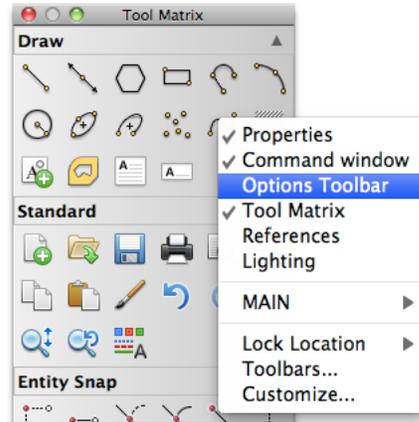
Click to close toolbar

Drag to resize toolbar

## Revealing Hidden UI Elements

iCADMac “hides” some user interface elements, because not all users need them. Here is how to reveal them:

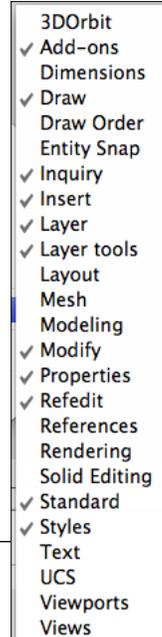
1. Right-click the Tool Matrix palette. Notice the shortcut menu:



2. Select an item from the shortcut menu. A check mark next to an item name indicates it is displayed.

Shortcut Menu	Meaning
---------------	---------

<b>Properties</b>	Toggles the Properties palette (Properties command).		
<b>Command window</b>	Toggles the Command window (CommandWindow command).		
<b>Options Toolbar</b>	Toggles the Options Toolbar.		
<b>Tool Matrix</b>	Toggles the Tool Matrix palette (ToolMatrix command).		
<b>References</b>	Toggles the References palette (References command).		
<b>Lighting</b>	Toggles the Lighting palette (LightList command).		
<b>MAIN</b>	Displays submenu of toolbar names. (See figure at right.)		
<b>Lock Location</b>	Displays submenu for locking floating and/or docked toolbars (LockUI system variable):		
	<table border="1"> <tbody> <tr> <td>Floating Toolbars</td> </tr> <tr> <td><input checked="" type="checkbox"/> Docking Toolbars</td> </tr> </tbody> </table>	Floating Toolbars	<input checked="" type="checkbox"/> Docking Toolbars
Floating Toolbars			
<input checked="" type="checkbox"/> Docking Toolbars			
	<ul style="list-style-type: none"> <li>• <b>Floating Toolbars</b> locks floating toolbars in place.</li> <li>• <b>Docking Toolbars</b> locks docked toolbars in place.</li> </ul>		
<b>Toolbars</b>	Displays the Specify Toolbars dialog box (Toolbars command).		
<b>Customize</b>	Displays the Customize Commands dialog box (Customize command).		



## The Crosshair Cursor and Other Cursors

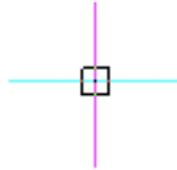
Cursors give you feedback from iCADMac, Windows, and other software. You move the cursor around the iCADMac window by moving your computer's mouse or trackpad. The cursor changes shape, as described here.

While in the iCADMac drawing area, the cursor initially looks like an arrow with a small square at its end. You are probably familiar with this arrow cursor from other Macintosh applications.

The small square is called the *pickbox*. It shows you the area in which you can select (or “pick”) objects in the drawing.



During commands, the cursor changes to a crosshair to show you its precise location in the drawing.



When you move the crosshair cursor out of the drawing area, it changes its shape to that of an arrow. The arrow cursor lets you make menu selections and pick toolbar buttons.

The cursor can change to other shapes, as well. For example, when the cursor becomes a double-ended cursor, you can resize the iCADMac window and palettes.

**TIPS** The size of the crosshair cursor can be changed through the Options command: (a) in the Options dialog box, select the **System Options** tab; (b) open the Graphics Area section; (c) look for **Pointer Size**. The default value is **5**, which means the length of the crosshair is 5% of the screen's size. When set to **100**, the cursor stretches across the entire drawing area.

### Colors of the Command Text

iCADMac uses colors to help you distinguish between different kinds of text in the command bar. The figure below shows some examples:



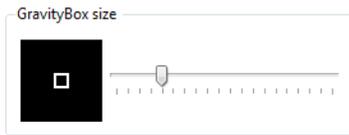
**Black text** specifies prompts from iCADMac.

**Blue text** indicates options. Enter the entire name, or just the option's abbreviation, as indicated by the underlining. For instance, you can enter **arc** or just **a**, and then press **return**.

**Amber text** displays current values for options, either the system default or the value you entered most recently.

**Italicized text** indicates the default option, such as *Enter to continue from last point*.

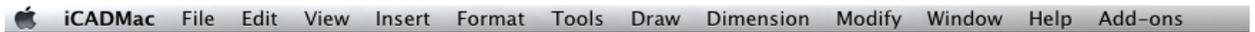
The size of the square pickbox is changed with the same dialog box's **User Preferences** tab: (a) open the **Drafting Options** section; (B) open **Display**; and then (C) the **Gravity Box** section. The size ranges from 0 to 50 pixels; the default size is 3.



Many other user interface options can be changed with the Options dialog box, accessed from the application menu and the **Preferences** item.

## The Menu Bar

The menu bar you see in iCADMac is similar to that used by other Macintosh programs. Some of the words found on the menu bar are common to other programs, such as File, Edit, View, Window, and Help. Other menu items, such as Dimension, Draw, and Modify, are more commonly found in CAD programs.



(As is common with other Macintosh programs, the Preferences option is located in the iCADMac menu, and not the Tools menu, as in the Windows version of this program.)

When you choose items from menus, iCADMac displays a helpful line of text in the status bar.



Here you read a one-sentence description of the command. This is helpful when you are not sure of a command's purpose. In this example, the Line command reports, "Creates straight line segments."

## The Command Window

Along the bottom of the iCADMac window is the Command window. Here you type commands — if you are like me and prefer typing over making menu selections with the mouse. Typing command names and options is a fast way to draw and edit when you are a touch typist, but it is slower if you are not.



The command area is where iCADMac displays additional options and prompts you for additional information it might need to complete a command.

When you see the ' Command: ' prompt by itself, like this:

Command:

it means that iCADMac is ready for you to enter a command.

(If you want to enter a command, but there is text after ' Command: ', press the **esc** key once or twice to cancel the current command.)

Try drawing a few lines now:

1. Enter the Line command, as follows:

Command: line (Press return.)

Type the word **line**, and then press the **return** (or **enter**) key. Pressing **return** tells iCAD-Mac you are finished entering the command name.

- iCADMac changes the prompt from 'Command:' to:

Options: Segments or  
Specify start point: (Move the mouse to move the cursor.)

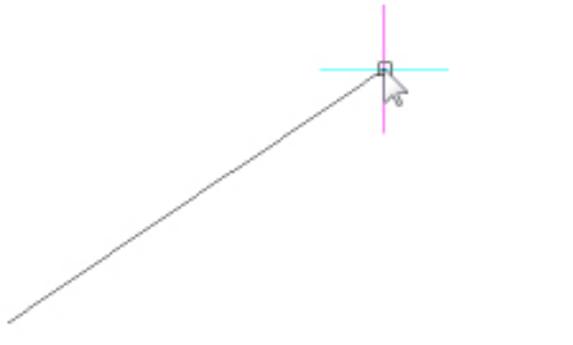
In prompting you 'Specify start point,' iCADMac is asking you to specify a spot in the drawing where the line starts. One way to show the starting point is with the mouse. As you move the mouse, you see the crosshair cursor move about the drawing area of the window.

(The command window shows a second prompt, which reads, 'Options: Segment or'. If you were to enter an 'S' instead of picking a point, iCADMac would switch to segment mode, where it draws a single line segment.)

- Pick a point on the screen by pressing the left button on your mouse, or by tapping the trackpad once. In CAD programs, this action is known universally as "picking," and so the left mouse button is referred to as the "pick button."

iCADMac records the on-screen x, y coordinates of your pick point, making them the start of the line.

- Move the mouse again. Notice that a line stretches from the starting point to the cursor. This line is known as a "ghost line" or as a "rubberband." It previews the position of the line before you pick the next point in the drawing.



- Press the pick button again to position the other end of the line. You have drawn your first line in iCADMac!

In the command window, iCADMac changes the prompt to read:

Options: Segments, Undo, Enter to exit or  
Specify next point or [Undo]: (Pick another point.)

- Continue by drawing more lines by moving the mouse and pressing the pick button. Each time you press the pick button, iCADMac draws another line segment.
- You end the Line command by pressing the **return** key, or pressing the **esc** key, as follows:  
Specify next point: (Press return or esc.)

Pressing the **esc** key cancels *any* iCADMac command; sometimes, complex commands may need you to press the key two or three times.

**TIP** Pressing the **return** key can have three different effects, depending on the prompt active at the time. Here are some examples using the Line command:

- At the 'Specify start point:' prompt, pressing **return** causes iCADMac to continue drawing from the last picked point, when the point was used to make a line or an arc. This is a great way to ensure that a line is drawn perfectly tangent to the end of an arc.
- At the 'Specify next point:' prompt, pressing **return** terminates the command.
- At the 'Command:' prompt, pressing **return** repeats the last command, in this case the Line command.

8. To erase the lines you just drew, type **U** at the ' Command: ' prompt (short for "undo").

Command: u  
Undo LINE

Alternatively, you can execute the U command these ways:

- From the **Edit** menu, select **Undo**
- Press **command+Z**. (Hold down the Command key and then press Z.)
- Select the  undo icon from the toolbar.

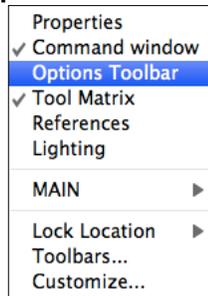
As you can see from this example of undoing, iCADMac provides different ways to perform the same action. You will probably find yourself using a combination of typing at the keyboard, choosing toolbar icons, making menu picks, and entering keyboard shortcuts — whichever you find most convenient.

## Options Toolbar

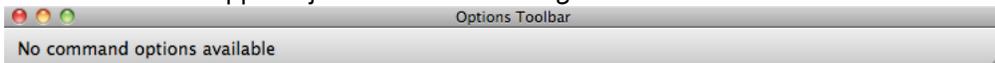
The Options toolbar displays the names of command options. As you start the Line command, for example, it displays option names in synch, as illustrated a bit later.

Normally, however, the toolbar is turned off, so if you want to try using it, open it as follows:

1. Right-click the menu bar.
2. From the shortcut menu, choose **Options Toolbar**.



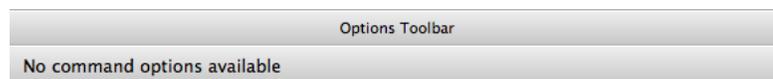
Notice the toolbar that appears just above the drawing area:



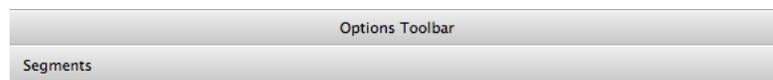
Initially, no command options are listed by the toolbar. Once you enter a command, however, options appear here. To activate one, you can click on its name on the toolbar, or enter its abbreviation in the Command window.

Here is the Line command again, this time showing how its command-line prompts (shown in blue) are coordinated with buttons on the Options toolbar:

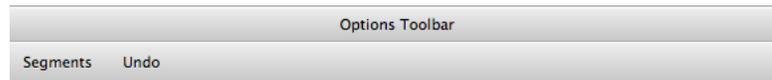
Command: line (Press return.)



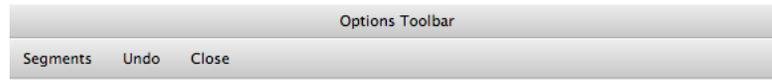
Options: **Segments** or  
Specify start point: (Move the mouse to move the cursor.)



Options: **Segments**, **Undo**, Enter to exit or  
Specify next point or [Undo]: (Pick another point.)



Options: [Segments](#), [Undo](#), [Close](#), Enter to exit or Specify next point» (Enter an option.)

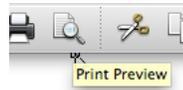


## Buttons and Icons

Palettes like the Tool Matrix and Properties collect related commands and properties as buttons and droplists; icons indicate their purpose. For instance, the Draw section of the Tool Matrix contains buttons for drawing lines, circles, and so on.



Sometimes, I am unsure about the meaning of an icon, and so I pause the cursor over it. After a couple of seconds, iCADMac displays a yellow tooltip showing the name of the command:



Down on the status bar, a line of helpful text appears:

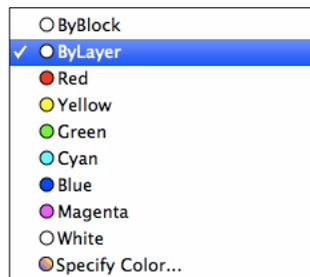


## Droplists

The Palette contains droplists. These let you choose from lists of options. For example, the Line Color droplist changes the colors of entities in your drawing.



Click one the droplist, and a list of colors drops down:



Run the cursor down the list, and then click the option you wish to select. This action is just like using a menu.



## Properties Palette

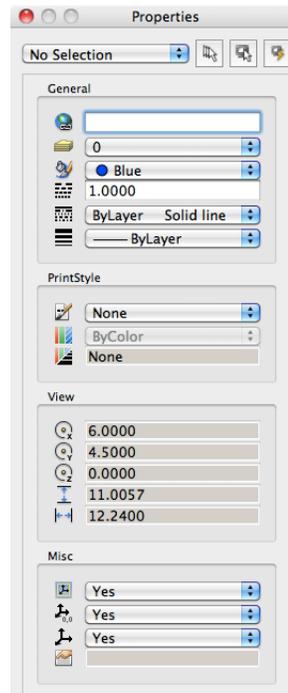
<b>Command:</b>	Properties
<b>Aliases:</b>	pr, props, ch, dmodify, mo
<b>Shortcut:</b>	command+1
<b>Shortcut Menu:</b>	Right-click the menu bar, and then choose <b>Properties Palette</b> from the shortcut menu.
<b>Tool Matrix:</b>	Standard   Properties

Palettes are unlike dialog boxes, in that you don't have to dismiss them; they can always remain open as you draw and edit. (In contrast, you must dismiss a dialog box before you can continue drawing and editing.)

iCADMac has palettes with names such as Properties, References, Tool Matrix, and Lighting. Of these, you will probably find the Properties palette the most useful, because of the two tasks it performs: reporting the properties of selected objects, and changing their properties.

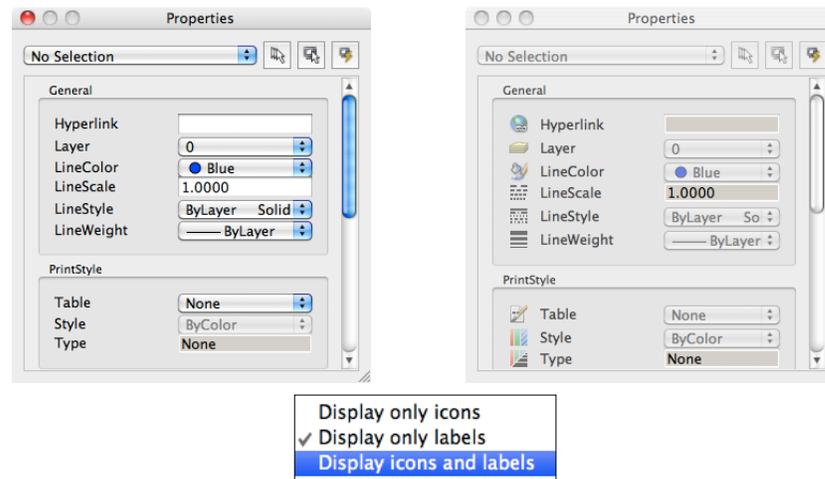
(If the Properties palette is not displayed, there are many ways to turn it on. Choose one of the methods listed above.)

The Properties palette initially looks like this:



The palette shows the default properties, such as the active color, line style, print style, and so on.

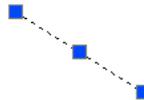
The properties labels can be icons, text, or both. To change the labels: (a) right-click the palette, and then (b) choose an option from the shortcut menu, as illustrated below. Icons make the palette narrower and so it takes up less screen space, but I find the text labels easier to understand.



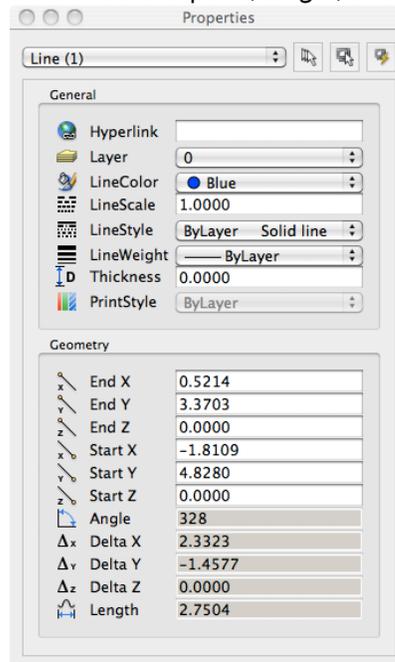
Each label prefixes a *field*. When the background color of a field is white, it means that you can change its value; when in gray, you cannot change it. (The value is set by iCADMac.)

To see how the Properties palette works, let's try it on the lines you drew earlier in this chapter:

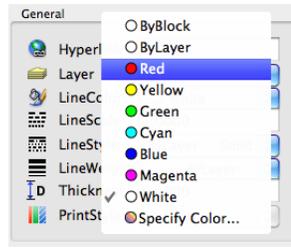
1. Select a line by picking it with the cursor. Notice that two things happen:
  - The line is *highlighted*. It becomes dashed and has three squares attached.



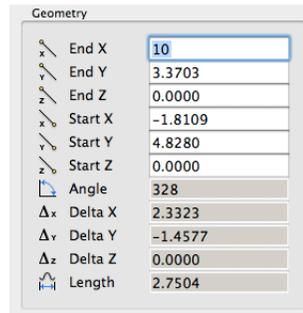
- The content of the Properties palette changes. It displays the properties of the line, as described by the General and Geometry sections. The General section describes the properties of the line, such as color and line weight; the Geometry section describes the x, y, z coordinates of the line's end point, length, and so on.



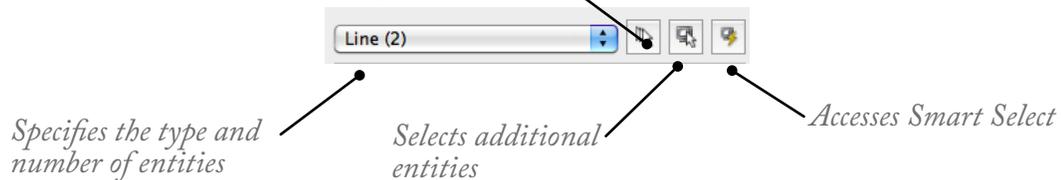
2. To use the Properties palette to change the line's color to red, follow these steps:
  - a. In the **General** section, click the **LineColor** droplist.
  - b. Choose **Red**. Notice that the line turns red.



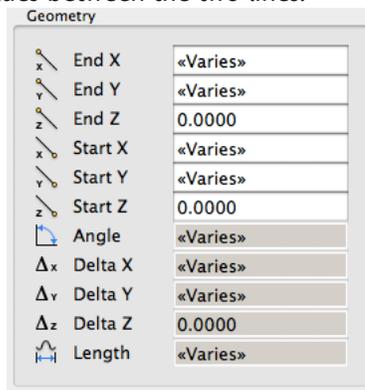
3. Now, let's use the Properties palette to change the line's geometry:
  - a. In the **Geometry** section, click the **End X** text box.
  - b. Erase the text by backspacing.
  - c. Enter a new number, such as **10**, and then press **Tab**. Notice that one end of the line changes its position.



4. Pick a second line. Notice that the Properties palette reports that two lines are selected. *Toggles between adding and replacing entities*



5. Below, in the Geometry section, many properties report, "<Varies>." This means that the properties have different values between the two lines.



6. Press **esc** to unselect the two lines. The one line that you changed retains its red color.

The palette can float, and be positioned anywhere on the screen. At the right end of the top of the palette is a mini toolbar. The buttons perform the following functions:

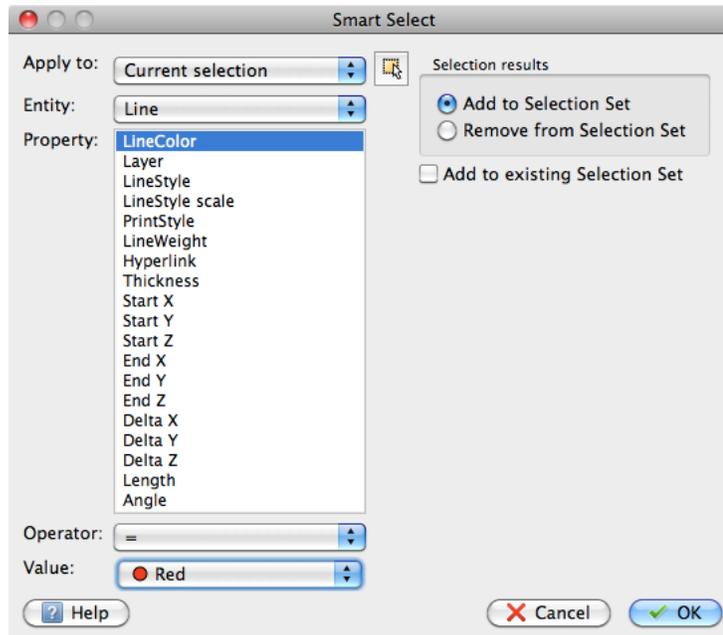
- Adds to Current Selection Set, *or* Replaces Current Selection Set
- Selects Entities
- Smart Selects

The first button performs two actions, either  **Add to Current Selection Set** or  **Replace Current Selection Set**. Click the button to switch between modes:

- In **Add** mode, iCADMac allows you to select additional entities.
- In **Replace** mode, iCADMac replaces the original selection with the newly selected entity.

The  **Select Entities** button chooses other entities in the drawing, replacing the current selection set. (More on selecting entities and selection sets, in the next chapter.)

The  **Smart Select** button displays a dialog box for selecting entities on the basis of their properties. For example, you could use it to select all red lines, or all switch box symbols located on layer “Electrical.”

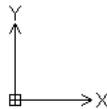


## Turning Off the UCS Icon

**Command:** Cslcon

**Menu bar:** View | Display | UCS Icon | On

The CS icon is located in the lower-left corner of the drawing area. CS is short for “coordinate system.” Its primary purpose is to help you draw in 3D by locating the origin and indicating the rotation of the x, y-plane.

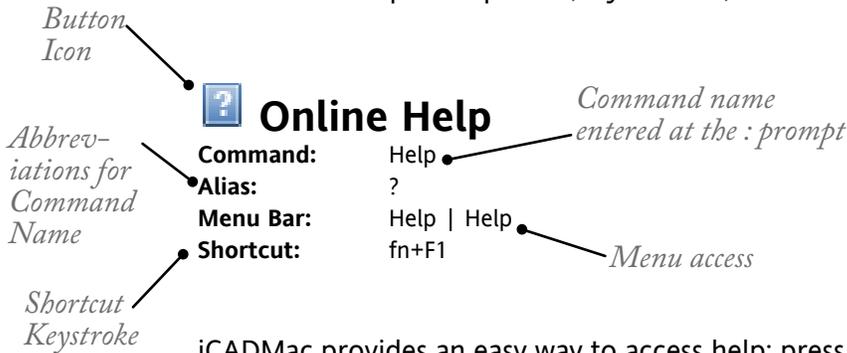


The X and Y arrows point in the direction of the positive x and y axes. Their intersection is usually (but not always) located at the origin of the drawing, where  $x=0$  and  $y=0$ . The square around the origin means you are “looking down” the z axis straight onto the x, y-plane.

The CS icon is not that useful in 2D drafting; in fact, I find it gets in the way, and so I recommend turning it off. From the **View** menu, select **Display | UCS Icon | On**. The vertical bars ( | ) separate menu picks.

## Other UI Elements

You will learn about the other user interface elements in the following chapters of this book. These include the Properties palette, layout tabs, and the status bar.

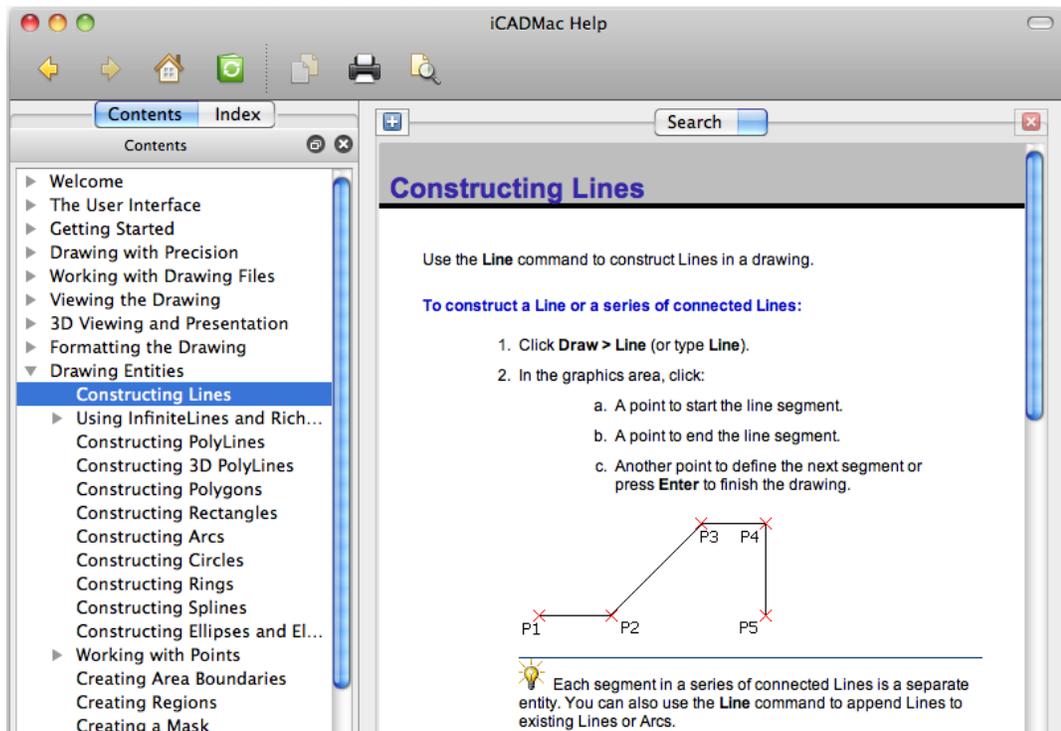


iCADMac provides an easy way to access help: press function key **fn+F1** during a command. (Hold down the Mac's **fn** key, and then press **F1**.) Here's an example using the Line command:

1. Type the **Line** command, then press the F1 function key, as follows:

Command: line  
 Options: Segments, Enter to continue from last point or  
 Specify start point» (Press fn+F1.)

Pressing fn+F1 invokes context-sensitive help. iCADMac opens the Help window to display useful information about the Line command.



Whenever you see underlined text, click it to display the definition of a word or to find out more about an option.

2. You can continue using the Line command with the Help window open:

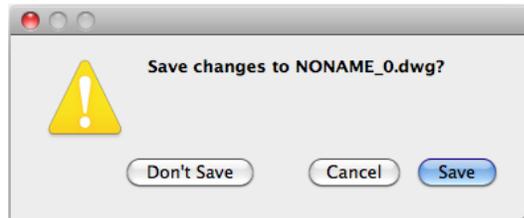
Resuming LINE command  
 Specify start point:

**TIP** To peruse specific subjects, select **Help** from the Help menu, and then select one of the tabs, such as **Contents** or **Search**.

## Exiting iCADMac

**Command:** Exit  
**Alias:** quit  
**Menu Bar:** File | Exit  
**Shortcut:** command+Q

To exit iCADMac, use the **Exit** command. When iCADMac asks if you want to save the drawing, click **No** this time.



(As alternatives, you can use the **command+Q** shortcuts, or else select **Quit** from the **iCADMac Command Edition** menu.)



## Chapter 3

# CAD Concepts

### In This Chapter

- Learning about vectors and how they define entities
- Specifying coordinates and selecting entities
- Inputting data with the mouse/trackpad
- Editing with grips

**T**his chapter introduces you to concepts basic to computer-aided design, such as vectors, coordinate input, and selection methods. If you are experienced with CAD software, feel free to skip to the next chapter.

## Vectors Make Entities

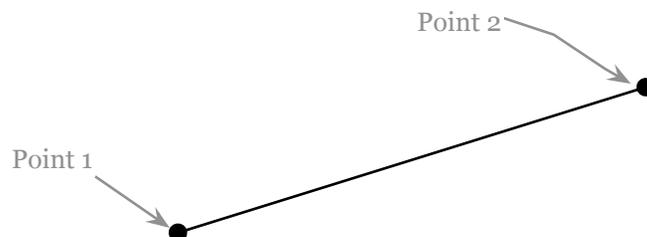
Drawings consist of *entities* — lines, circles, arcs, text, dimensions, and so on — that you create and manipulate by inputting commands and specifying coordinates.

Just like all other CAD software, iCADMac works with *vector* entities. By vectors we mean entities defined by geometric features. For example, lines are defined by their end points, circles by a center point and radius.

It may seem curious, but you never draw the entities themselves; instead, you specify their geometric features, and then iCADMac draws them. For instance, instead of drawing a line, you specify a points at either end of the line, like this:

Command: line  
Options: [Segments](#) or  
Specify start point» (Indicate the starting point of the line, point 1.)  
Options: [Segments](#), [Undo](#), [Enter to exit](#) or  
Specify next point» (Indicate the end of the line, point 2.)

iCADMac then draws in the line segment between the two points you specified, as illustrated below.



## Key Terms in This Chapter

**Axis** — refers to the reference line from which distances and angles are measured; all iCADMac drawings have three axes: x, y, and z.

**Cartesian coordinate** — specifies points in planes (and space) by their distance measured from the origin along the x and y axes (and optionally the z axis).

**Coordinates** — refers to two or more numbers or angles that determine the positions of points in space relative to another reference, such as the origin or another point.

**Cylindrical coordinate** — specifies points in space by their distance from the origin along the x and z axes; the y distance is specified by the angle measured counterclockwise from the positive x axis.

**Direct distance entry** — specifies distances by moving the mouse and then entering a distance.

**Grips** — refers to small squares that permit direct editing of entities.

**Negative distance** — refers to distances measured negatively from the origin.

**Negative angle** — refers to an angle measured clockwise from the positive x axis.

**Origin** — refers to the coordinate center of the drawing, where  $x=0$ ,  $y=0$ , and  $z=0$ .

**Point filter** — allows mixed input of coordinates, some by pointing in the drawing and some by entering values at the keyboard.

**Polar coordinate** — specifies points in space by their distance from the origin along the x axis; the y distance is specified by the angle measured counterclockwise from the positive x axis.

**Positive angle** — refers to an angle measured counterclockwise from the positive x axis.

**Relative coordinate** — refers to points measured relative to another point.

**Spherical coordinate** — specifies points in space by their distance from the origin along the x, y, and z axes; the y distance is specified by the angle measured counterclockwise from the positive x axis.

**Vector** — refers to entities drawn by iCADMac; vectors are defined by endpoints, radii, and additional geometric functions.

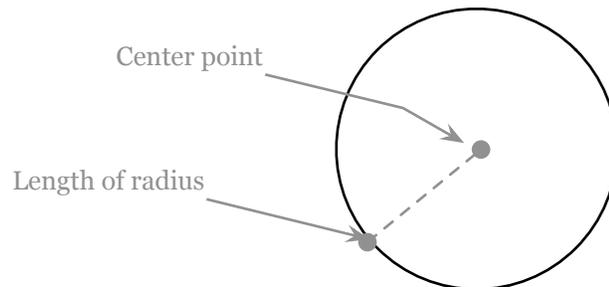
## Abbreviations

@	(pronounced “at”) indicates relative coordinates.
<	(“angle bracket” or “less than”) indicates angles.
-	(“negative sign” or “dash”) indicates negative values.
.	(“point”) indicates point filters.
:	(“colon”) indicates the iCADMac prompt.
<b>x,y</b>	indicates 2D Cartesian coordinates.
<b>x,y,z</b>	indicates 3D Cartesian coordinates.
<b>@x,y</b>	indicates relative 2D Cartesian coordinates.
<b>d&lt;a</b>	indicates polar coordinates (distance, angle).
<b>@r&lt;a</b>	indicates relative polar coordinates.
<b>x,y&lt;a</b>	indicates cylindrical coordinates.
<b>d&lt;a&lt;A</b>	indicates spherical coordinates (distance, angle1, angle2).

The same technique applies to all other entities: iCADMac draws them after you specify their geometric features. Here are a few more examples:

**Circles** are drawn after you indicate the location of the center point and the length of the radius.

Command: circle  
 Options: **3Point**, **2Point**, **Ttr**, **TTT**, **Enter to exit** or  
 Specify center point» (Pick point 1.)  
 Options: **Diameter** or  
 Specify radius» (Pick point 2.)

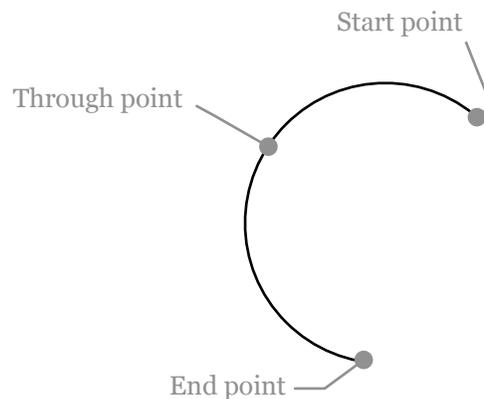


**TIPS** Usually, there is more than one way to draw an entity. For instance, for circles you can specify the radius or diameter, even two or three points on their circumference. You can also draw circles so that they touch other entities tangentially.

During commands, iCADMac highlights the names of options with **blue** text.

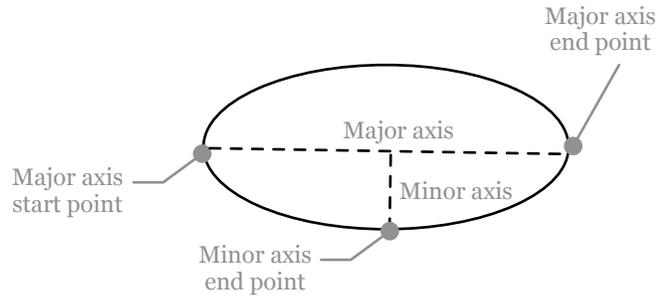
**Arcs** are drawn after you indicate two endpoints and one other point, such as a radius or center point. (Of all CAD entities, arcs have just about the largest number of options.)

Command: arc  
 Options: **Center**, **Append**, **Enter to continue from last point** or  
 Specify start point» (Pick point 1.)  
 Options: **Center**, **End** or  
 Specify through point» (Pick point 2.)  
 Specify end point» (Pick point 3.)



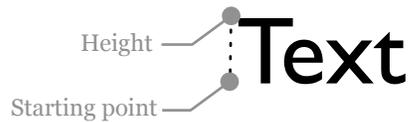
**Ellipses** are defined by two axes, and are drawn after you indicate their location and length.

Command: ellipse  
 Options: **Elliptical arc**, **Center** or  
 Specify axis start point» (Pick point 1.)  
 Specify axis end point» (Pick point 2.)  
 Options: **Rotation** or  
 Specify other axis end point» (Pick point 3.)



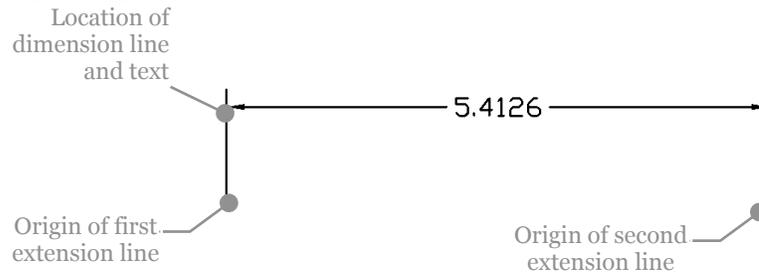
**Text** is created after you indicate its starting point, height, and angle. iCADMac uses the SimpleNote command to place text in drawings, although you can enter “text” as an alias for this command:

Command: text  
 SIMPLENOTE  
 Options: **Justify**, **textStyle** or  
 Specify start position» (Pick point 1.)  
 Default: 2.5  
 Specify height» (Specify the height.)  
 Default: 0  
 Specify text angle» (Specify the angle.)  
 Specify text» (Enter the text.)



**Dimension** are drawn after you indicate the origin of two extension lines and then locate the dimension line. The LinearDimension command draws horizontal, vertical, and rotated dimensions; other commands in iCADMac draw additional types of dimensions.

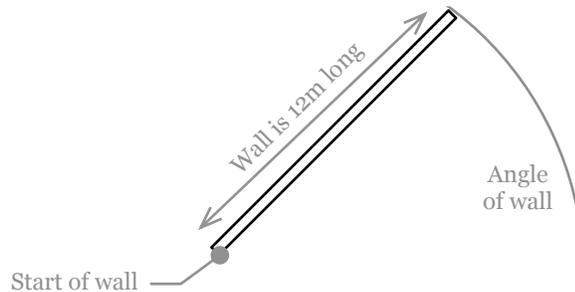
Command: lineardimension  
 Default: Entity  
 Options: **Entity** or  
 Specify first extension line position» (Pick point 1.)  
 Specify second extension line position» (Pick point 2.)  
 Options: **Angle**, **Horizontal**, **Note**, **Rotated**, **Text**, **Vertical** or  
 Specify dimension line position» (Pick point 3.)  
 Dimension Text : 154.31



In addition to the basic entities described above, iCADMac draws many others. You can see the complete list of drawing commands in the Draw, Dimension, and Solids menus.

## Alternative Entity Specification

While you can certainly draw lines by specifying two endpoints, this may not be the most convenient method. For example, in drawing a wall, you often know its length and where it starts, but not where the far end is located.



iCADMac provides a variety of methods for defining the size and location of entities. To draw the 12-foot-long wall, for instance, you can specify its starting point, its length, and its angle to other walls, like this:

Command: line  
 Specify start point» (Pick a point for the start of the wall.)  
 Specify next point» (Move the mouse to show the angle, and then enter the length, such as 12.)

(The *angle* is measured counterclockwise from the positive x axis.)

The same is true for nearly all entities created by iCADMac: often, there are two or more ways to draw each of them. For example, there are more than a dozen different ways to draw an arc.

## Specifying Coordinates

To draw entities, you need to specify the *coordinates* (locations) of geometric features.

iCADMac provides two methods of specifying geometric features:

Keyboard  
 Mouse or trackpad (cursor)

You use the keyboard to enter specific values, such as numbers, exact distances, and angles. You use the mouse is used to show locations in drawing, and enter imprecise values.

### Keyboard Input

You can use the keyboard to enter the names of commands and options, as well as to specify exact x, y coordinates, distances, angles, and amounts. Let's take the Line command as the example again:

Command: line  
 Options: [Segments](#), [Enter to continue from last point](#) or  
 Specify start point»

At the 'Specify start point' prompt you can do any of the following actions at the keyboard:

- To *continue* the line from the last point entered in the drawing, press **return** at the 'Specify start point' prompt:

Specify start point» (Press return.)

iCADMac then starts the line at the end of the last line or arc that was drawn earlier.

- To specify the Segments option, press **S** and then **return**.  
Specify start point» s (and then press return.)  
iCADMac then presents the prompts for drawing single line segments.
- To specify x ,y coordinates, enter values separated by a comma, such as **1,2**, and then press **return**.  
Specify start point» 1,2 (and then press return.)  
iCADMac then starts drawing the line at the point in the drawing defined by x=1 and y=2.

## Coordinate Syntax

---

Although entering x, y coordinates is a very common practice in CAD programs, they are not the only type and syntax of coordinates.

- **Comma** ( , ) separates distances, such as 2,3 for 2D coordinates and 4,5,6 for 3D.
- **Angle bracket** ( < ) signifies angles, such as <45.
- **Negative** distances and angles are prefixed by a dash ( - ). Negative angles are measured clockwise from the x axis.  
Specify start point» -1,-2<-45
- **Relative** coordinates use the ‘at’ symbol ( @ ). Most coordinates are measured relative to the *origin* of the drawing (0,0), and so they are called “absolute coordinates.” By placing the @ symbol in front of a coordinate, you are telling iCADMac to measure distances and angles relative to the *last point* picked in the drawing.  
Specify start point» @1<45  
End point: @7,5

It may seem counterintuitive, but all CAD systems measure positive angles *counterclockwise* from the positive x axis. This direction can be reversed with the Units command, as can the location from which angles are measured.

**TIP** You can enter coordinates with imperial or metric units:

- Imperial units use ‘ for feet and “ for inches, such as **1’2”,5”**.
- Metric units use no symbols, just the numbers.

## Coordinate Systems

---

iCADMac works with many kinds of measurement systems. Here is an overview of all of them.

### 2D Cartesian Coordinates

**x, y** specifies two-dimensional coordinates used for 2D drafting. Note that a comma ( , ) is required to separate the x and y values:

Specify start point» 1,2

The z coordinate is fixed by the Elevation system variable and usually is set to 0.

### 3D Cartesian Coordinates

**x,y,z** specifies three-dimensional coordinates, also known as *Cartesian* coordinates.

Specify start point» 1,2,3

## Polar Coordinates

**d<a** specify *polar* coordinates, which describe the distance and angle from the origin (0,0). Think of being at the North Pole, and then finding any point on the earth by a distance (from the North Pole) and an angle around the pole.

Specify start point» 23<45

The angle bracket ( < ) prefix indicates the angle (45 degrees).

## Cylindrical Coordinates

**x,y<a** specifies cylindrical coordinates, which combines an x, y coordinate with an angle.

Think of being inside a pop can: the **x** measures the radius of the can, the **y** measures the height of the can, and the **a** measures the angle around the can.

Specify start point» 1,2<45

## Spherical Coordinates

**d<a<A** specifies spherical coordinates, the 3D version of polar coordinates. Here you specify a distance and two angles.

Think of being in the center of the Earth: the **d** measures the distance from the center, the **a** measures the angle that sweeps around the equator (in the x, y plane), while the **A** measures the angle along the International Date Line (in the y, z plane).

Specify start point» 1<45<90

**TIP** You can specify angles in degrees, radians, grads, or surveyor units. To distinguish between them, you employ the following symbols:

- Degrees use no symbols, such as 181.2.
- Radians are indicated by **r**, such as 2.17r
- Grads by **g**, such as 400g
- Surveyor units use **N** and **E**, such as 24N45E

## Point Filters

You don't even have to enter the entire coordinate set. iCADMac allows you to enter partial coordinates through *point filters*. The word "point" refers to the points placed in the drawing, while "filter" means that you are providing the coordinates partially.

Typically, you enter the points in three steps:

1. Enter a point filter name, such as .x, which alerts iCADMac of your plan.
2. iCADMac prompts you 'of,' at which you enter the value of the x coordinate.
3. And then iCADMac prompts you for the missing portion, such as '(need YZ).'

When you enter a point filter, iCADMac holds the following conversation with you:

Specify start point» .xy  
of (Pick a point.)  
(need Z) 4

The table below illustrates all possible combinations of point filter:

Point Filter	First specify...	...and then specify
.x	x coordinate	y, z coordinates
.y	y coordinate	x, z coordinates
.z	z coordinate	x, y coordinates
.xy	x, y coordinates	z coordinate
.xz	x, z coordinates	y coordinate
.yz	y, z coordinates	x coordinate

## Mouse Input

As an alternative to the keyboard, you can use the cursor to show iCADMac locations in drawings; the mouse (or trackpad) controls the cursor.

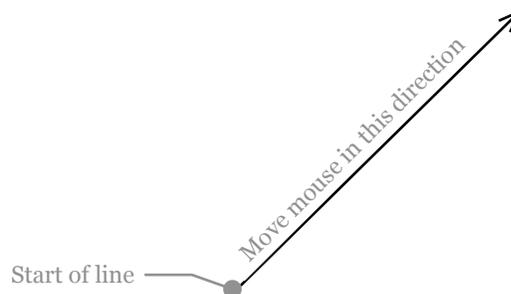
### Direct Distance Entry

iCADMac has a “hidden” form of coordinate entry known as *direct distance entry*. In this method, you don’t enter x, y coordinates; instead, you move the cursor in a specific direction, and then type in the distance.

This is an interactive form of relative polar coordinates (@d<a). Here’s how to use direct distance entry with the Line command:

Specify start point» (Move mouse/cursor in any direction, and then enter a distance:) 5

To indicate the angle, you move the cursor in that direction.

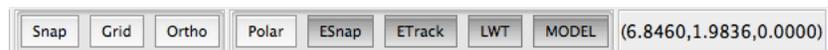


The problem with using the mouse is that it is not particularly accurate by itself. As you draw, you could keep an eye on the status bar to see the size of the entity being drawn. (See the status bar shown below.) The drawback is that watching the status bar is too slow and imprecise for production use. Similarly, for example, you would find it difficult to draw a line at 22.5 degrees precisely using just the mouse.

For this reason, iCADMac provides several aids that let you draw accurately with the mouse and cursor, such as orthographic mode, snap distances, and polar mode. To understand these modes, we first look at the status bar, which controls them.

### Status Bar

The *status bar* reports the status of several drawing aids, as well as the distance and angle from the last pick point.



### Coordinates

The coordinate field is at the extreme right end of the status bar. When you are just moving the

cursor around (and not in a drawing or editing command), the status bar reports the current location of the cursor as x,y coordinates, as well as the “z” coordinate as the current elevation (usually 0 units). In the figure below, the cursor is at x = 10.5 and y= 4.0.



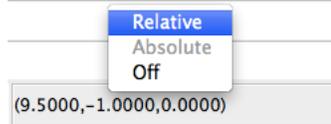
(10.5000,4.0000,0.0000)

During drawing and editing commands, you can choose to have the status bar continue to report x,y coordinates or change display to show the distance and angle from the last pick point. This change can be made only during a command, as follows:

1. Start a drawing command, such as Line.

Command: line  
Specify start point» (Move cursor.)

2. Right-click the coordinate field. Notice the short cut menu.



3. Choose **Relative**.

(The Off option turns off the real time updating of the coordinate display during cursor movement; it updates only when you pick a point.)

4. Move the cursor around to see the relative coordinates displayed by the status bar. In the figure below, the cursor has moved 2.5 units at an angle of 180.0 degrees.



(2.5000< 180.0000 0.0000)

5. Press **esc** to end the Line command. Notice that the coordinate display reverts to absolute, until you next start a command.

## Snap Mode

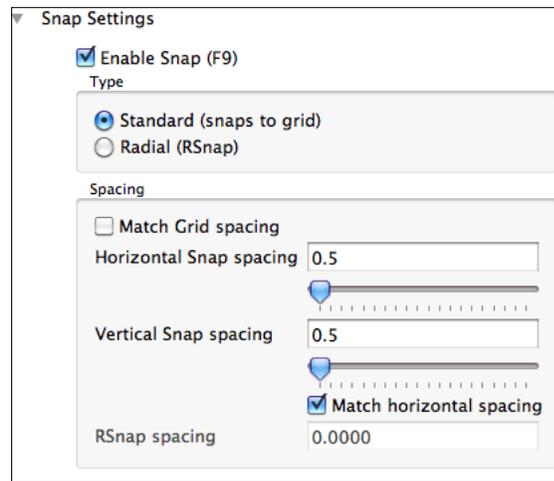
The **snap** distance specifies the cursor resolution. For instance, when you set the snap distance to 1mm, then mouse movements in the drawing area are accurate to the nearest one millimeter. (Snap does not affect the cursor outside of the drawing area.)

 To turn on Snap mode, click the **Snap** button; notice that it turns dark gray to indicate the mode is turned on.

To change the snap distance: (a) right-click the **Snap** button, and then (b) choose **Settings**, as displayed below.



iCADMac displays the Snap section of the Options dialog box. The meanings of these settings are described in detail in a later chapter.

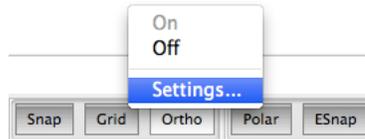


The snap distance is invisible, however, so many users also turn on the grid to match the snap distance.

### Grid Mode

The **grid** is an array of evenly-spaced dots that provides an idea of distances in the drawing. For instance, when the grid spacing is 1000mm, then you get an idea of how far a meter stretches, no matter how close or far away the zoom level is.

Just as with Snap mode, you turn on grid mode by clicking the **Grid** button; grid options are set by right-clicking the button, and then choosing **Settings** from the shortcut menu.



### Ortho Mode

Ortho mode forces the cursor to travel in horizontal or vertical directions only — in 90-degree increments. This is a useful setting, because many drawings contain lines at right angles.

Turn on ortho mode by clicking the **Ortho** button. This mode has no options.

### Polar Mode

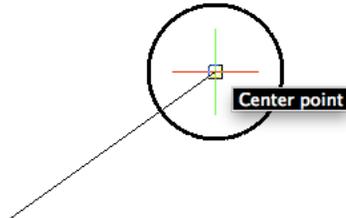
**Polar** mode lets you draw accurately at fixed angles, such as in 15-degree or 45-degree increments. (Ortho mode is limited to 90-degree increments.) Together with snap mode, polar mode solves the problem of drawing accurately using only the mouse.

You turn on polar mode by clicking the **Polar** button; options are set by right-clicking the button, and then choosing **Settings** from the shortcut menu.



### ESnap Mode

**Entity snap** modes cause the cursor to snap to geometric features of entities, such as the end point of a line, the mid point of an arc, or the tangent point to a circle. This is crucial when you want to draw from one geometric feature to another, for example from a line to the precise center of a circle, as illustrated below.



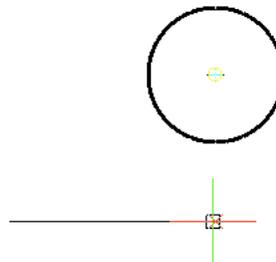
Entity snaps are called “esnaps” for short. To help you further, iCADMac displays *tooltips* to identify geometric features, such as the “Center point” tooltip in the figure above. In all, iCADMac has 13 entity snaps; tutorials in later chapters make use of most of them.

You turn on entity snap mode by clicking the **ESnap** button; entity snap options are set by right-clicking the button, and then choosing **Settings** from the shortcut menu.

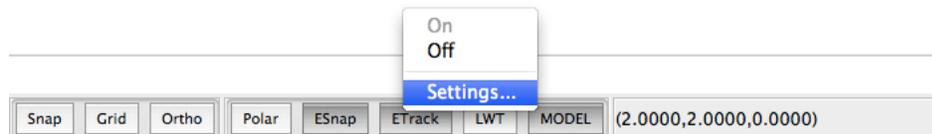
### ETrack Mode

**Entity tracking** tells you when geometric features line up with the cursor, such as the ends of lines or centers of circles. This mode works in conjunction with esnaps.

In the figure below, iCADMac found the center point of the circle while I was drawing another line below it. The alignment is noted by the icon appearing at the circle’s center point.



You turn on entity tracking mode by clicking the **ETrack** button; options are set by right-clicking the button, and then choosing **Settings** from the shortcut menu.



## Selecting Entities

When it comes to editing drawings, you need a way to select the entities to be edited. *Selection* tells iCADMac which group of entities you need to modify. In most cases, you use the mouse to select entities; in a few cases, you might use the keyboard instead.

There are two ways to select entities:

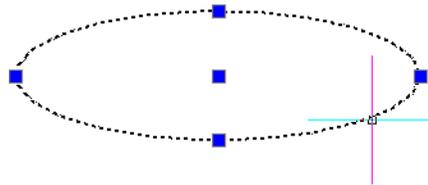
- *Before* starting an editing command (called “grips editing”).
- *After* starting the editing command (called “entity selection”).

You will probably find use for both methods but for different kinds of editing operations. Let’s look at both.

### Grips Editing

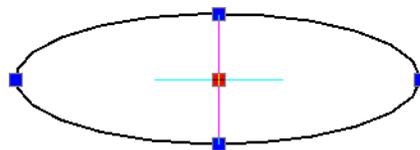
Grips editing starts when no command is active. You select one or more entities by picking them with the cursor.

Notice that the entities change their look to dashes, such as for the ellipse illustrated below. In addition, one or more blue squares appear, called “grips.” iCADMac responds in this way to let you know which entity you picked.



When grips are colored blue, they are known as “cold grips.” Their sole purpose is to show you where grips are located on the selected entity.

Move the cursor over a grip, and then click on it. Notice that the grip turns red. A red grip is known as a “hot grip,” because its purpose is to edit the selected entity. All editing takes place relative to the hot grip, which becomes the de facto base point.



You can perform the following editing operations with hot (red) grips:

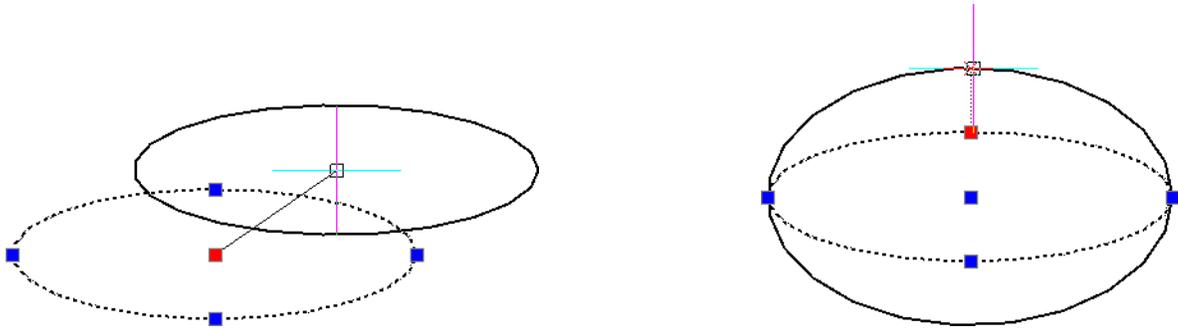
- Move the entity
- Make one or more copies
- Stretch the entity by making *part* of the entity longer or shorter
- Scale the entity by making *all* of it larger or smaller
- Mirror (or flip) the entity
- Rotate the entity by an angle
- Delete (or erase) the entity

The editing action also depends on *which* cold grip you selected to make hot, as described below.

### Moving and Stretching with Grips

Drag the hot grip. Notice that the action edits the entity in different ways, depending on which

grip is hot.



Left: The center grip moves the ellipse.

Right: The edge grip stretches the ellipse.

The exact type of editing depends on the location of the grip. In general:

**Centrally**-located grips tend to move the entity.

**Peripherally**-located grips tend to stretch the entity.

You will find additional editing options are displayed in the command window:

```
STRETCH
Options: Base point, Copy, Undo, eXit or
Stretch point» (Pick a point, or enter an option.)
```

**Base point** relocates the base point (the “start point”).

**Copy** copies the entity, instead of moving it.

**Undo** undoes the last action.

**eXit** exits grips editing. You can also press **esc** to exit.

Recall that you need to type only the capitalized letter of an option, such as ‘c’ to copy the entity or ‘u’ to undo the last action. In addition, you can press the **spacebar** to review additional editing options — move, mirror, rotate, and scale:

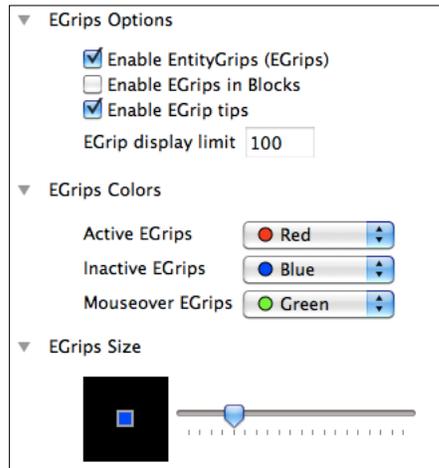
```
MOVE
Options: Base point, Copy, Undo, eXit or
Stretch point» (Press spacebar.)
MIRROR
Options: Base point, Copy, Undo, eXit or
Stretch point» (Press spacebar.)
ROTATE
Options: Base point, Copy, Undo, eXit or
Stretch point» (Press spacebar.)
SCALE
Options: Base point, Copy, Undo, eXit or
Stretch point» (Press spacebar or esc.)
```

Press the spacebar one more time to return back to Stretch, and then repeat the cycle.

### Erasing with Grips

To erase selected entities, press **delete** on the keyboard.

**TIP** To change the size and color of grips, enter the **DraftingOptions** command, and then expand the **Entity Selection** node (illustrated below). Sizes are changed with the **EGrips Size** node, colors with the **EGrips Colors** one.



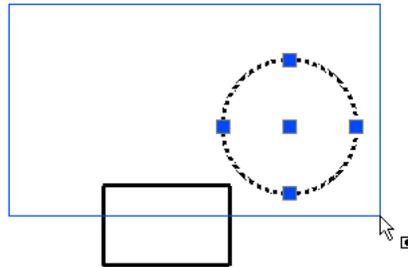
## Selecting Multiple Entities

You select a single entity by picking it with the cursor. Ensure part of the entity is within the cursor's *pickbox*. To select additional entities, keep picking them.

### Selecting Multiple Entities by Rectangle

After a while, picking entities one-by-one becomes tedious, and so iCADMac provides ways to pick two or more entities at once. One method is the selection *rectangle*. It selects all entities within an area specified by a rectangle, which you create like this:

1. Pick in a blank point in the drawing away from any entities.
2. Form a rectangle by moving the cursor diagonally, enclosing the entities you wish to select.



iCADMac reacts differently, depending on whether you move the cursor to the right or the left:

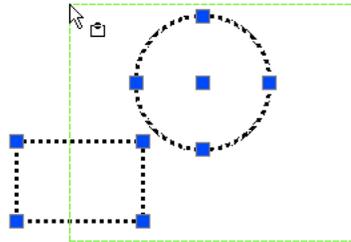
**Right** — moving the cursor right forms a *windowed* selection. iCADMac selects all entities that fall entirely within the selection rectangle. It ignores entities that cross the rectangle or fall outside the rectangle. Notice the cues displayed by iCADMac:

- The selection rectangle is **blue**.
- An  icon near the cursor reminds you of the selection mode (“window”). The icon shows a dot *entirely inside* the tiny rectangle.

Those entities selected by the rectangular window are highlighted by iCADMac. In the figure above, only the circle is selected, because only it is *entirely* within the selection rectangle.

**Left** — moving the cursor left forms a *crossing selection*. iCADMac selects *all* entities within the selection rectangle, as well as those crossing the rectangle. Notice the cues displayed by iCADMac:

- The selection rectangle is green and dashed.
- The  icon near the cursor reminds you of the selection mode (“crossing”). The icon shows the dot resting on the edge of the rectangle.



Again, entities selected by the crossing rectangle are highlighted by iCADMac, but this time both the circle and ellipse are selected.

## Selecting Entities by Location

But, wait! There is more! Selection areas need not be rectangles; they can also be polygons and fences. These selection modes become useful when editing commands ask you to specify entities. For example, the Copy command prompts you, as follows:

Command: copy  
Specify entities» (Enter a selection mode.)

At the ‘Specify entities’ prompt, you can pick entities with the mouse, or enter one of the following options. I’ve highlighted each option’s abbreviation with underlined boldface text.

<b><u>S</u>election Options</b>	<b><u>M</u>eaning</b>
<b><u>A</u>LL</b>	Selects all non-frozen entities in the drawing.
<b><u>A</u>Uto</b>	Selects an entity when picked directly; otherwise, picking a blank spot starts Window or Crossing mode.
<b><u>B</u>OX</b>	Starts Windows and Crossing mode, depending on how the cursor is moved: Right to left = Crossing mode. Left to right = Window mode,
<b><u>C</u>rossing</b>	Selects entities contained within or crossing the boundary of a rectangular selection area.
<b><u>C</u>Polygon</b>	Selects entities contained within or crossing the boundary of a polygonal selection area.
<b><u>F</u>ence</b>	Selects entities crossing a series of line segments.
<b><u>L</u>ast</b>	Selects the entity most recently added to the drawing.
<b><u>P</u>revious</b>	Selects entities included in the previous selection set.
<b><u>W</u>indow</b>	Selects entities contained entirely within a rectangular selection window.
<b><u>W</u>Polygon</b>	Selects entities contained entirely within a polygon selection window.

<b><u>S</u>election Modes</b>	<b><u>M</u>eaning</b>
<b><u>A</u>dd</b>	Adds one or more entities to the selection set; used after the Remove option.
<b><u>E</u>Group</b>	Selects entities of a named group (created by the EntityGroup command).
<b><u>R</u>emove</b>	Removes one or more entities from the selection set.

<b>M</b> ultiple	Selects multiple entities without highlighting them.
<b>S</b> ingle	Selects one entity, and then carries on with the command; the 'Specify entities' prompt does not return.
<b>U</b> ndo	Undoes the last selection.

You can create selection sets with the Select command. Then, during an editing command, use the Previous option to employ the stored set, like this:

Command: select  
Specify entities» (Choose one or more entities.)

Command: copy  
Specify entities» p

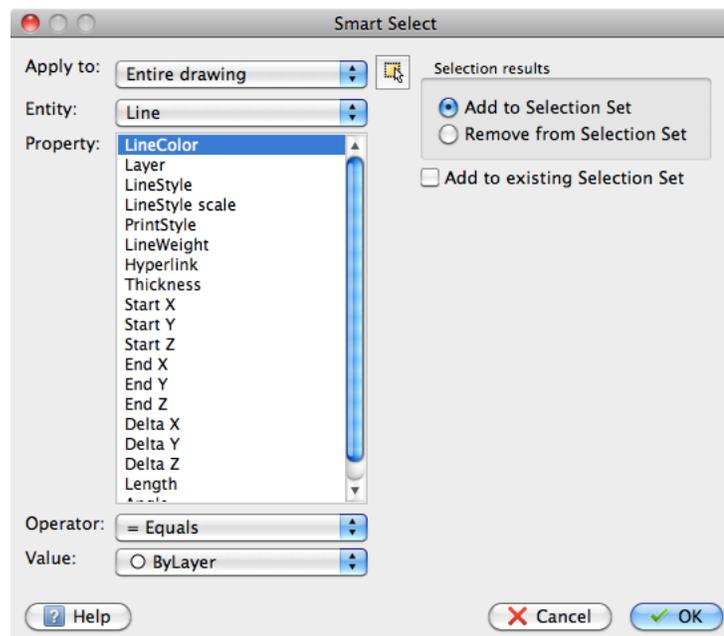
**TIP** The Select command does not normally lists its options; to see them, enter a question mark at the 'Specify entities' prompt, like this:

Command: copy  
Specify entities» ?  
«Invalid selection» Specify a point or [Window](#), [Last](#), [Crossing](#), [BOX](#), [ALL](#), [Fence](#), [WPolygon](#), [CPolygon](#), [EGroup](#), [Add](#), [Remove](#), [Multiple](#), [Previous](#), [Undo](#), [AUto](#), [SIngle](#)

## Selecting Entities by Properties

The selection options listed in the table above are meant for selecting entities based on their location in the drawing. A different method is to select them by their properties, such as all entities colored red or with a specific hyperlink.

The SmartSelect command displays a dialog box that lets you choose entities based on any of their properties. The most common properties include color, Layer name, line style, line style scale, print style, lineweight, or hyperlink.



(This dialog box is also available through the Properties palette by clicking the  Smart Select icon.)

**TIP** You can mix keyboard and mouse entries. For instance, the prompts below show a line drawn using point filters, esnap modes, mouse/cursor picks, and relative coordinates:

```

Command: line
Specify start point» .xy
of: mid
of (Pick an entity with the mouse/cursor.)
(need z) 3
Specify next point» @1<-45

```

## AutoCAD Command Compatibility

iCADMac command names are compatible with those in AutoCAD and progeCAD, for the most part. Some command names are identical, such as Line, Copy, and Stretch.

Others are different, such as PEdit (“EditPolyline”) and DimLinear (“LinearDimension”). However, iCADMac ensures compatibility through the use of *aliases*. This means that you can enter PEdit to edit polylines, DimLinear to start linear dimensioning, and so on.

In addition, iCADMac recognizes the aliases used by AutoCAD, such as L for the Line command and PE for PEdit (or EditPolyline).

Appendix C, “Command Aliases,” lists commands and corresponding aliases, and boldfaces those that are identical with AutoCAD.

## Macintosh Multi-touch Compatibility

iCADMac for Macintosh understands some of the multi-touch gestures available through the MacBook’s trackpad and the MagicMouse. The following actions work with the trackpad; mouse actions typically requires one less finger.

Trackpad Gesture	Action in iCADMac
Single-finger click	Pick
Two-finger click	Right-click (usually displays a context-sensitive shortcut menu)
Two-finger drag up	Zoom in at the cursor
Two-finger drag down	Zoom out at the cursor
Four-finger drag	Switch between drawings (displays Expose)



Part II

# 2D Architectural Drafting



## Chapter 4

# Starting New Drawings

### In This Chapter

- Preparing drawings for first-time use
- Understanding how layers organize drawings
- Saving drawings
- Learning the importance of automatic backups

**t**his chapter teaches you how to prepare iCADMac for new drawings. At the end of the chapter, you learn how to save your work to disk and how to exit iCADMac.

### Before You Begin

To learn how to use iCADMac, you work with a drawing based on an office floor plan. The example used for the 2D drafting portion of this book is illustrated on the following pages.

#### Preparing iCADMac for Drawing

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Before creating any new drawing, you must prepare it by carrying out the following tasks:

- Name the drawing
- Select a unit of measurement
- Set the snap and grid spacings
- Indicate the bounds of the drawing
- Set up layers by naming them

## Key Terms in this Chapter

**File name** refers to names used for drawing files and other documents.

**Grid** provides a visual guide consisting of an array of evenly-space dots.

**Layer** organizes drawings by segregating entities logically.

**Bounds** specify the nominal limits of drawings and constrain the range of grid marks.

**Snap** constrains cursor movement to discrete distances.

**Unit** specifies forms of linear and angular measurements, such as metric, architectural, and engineering.

**Zoom** enlarges and reduces the visual size of drawings.

## Commands

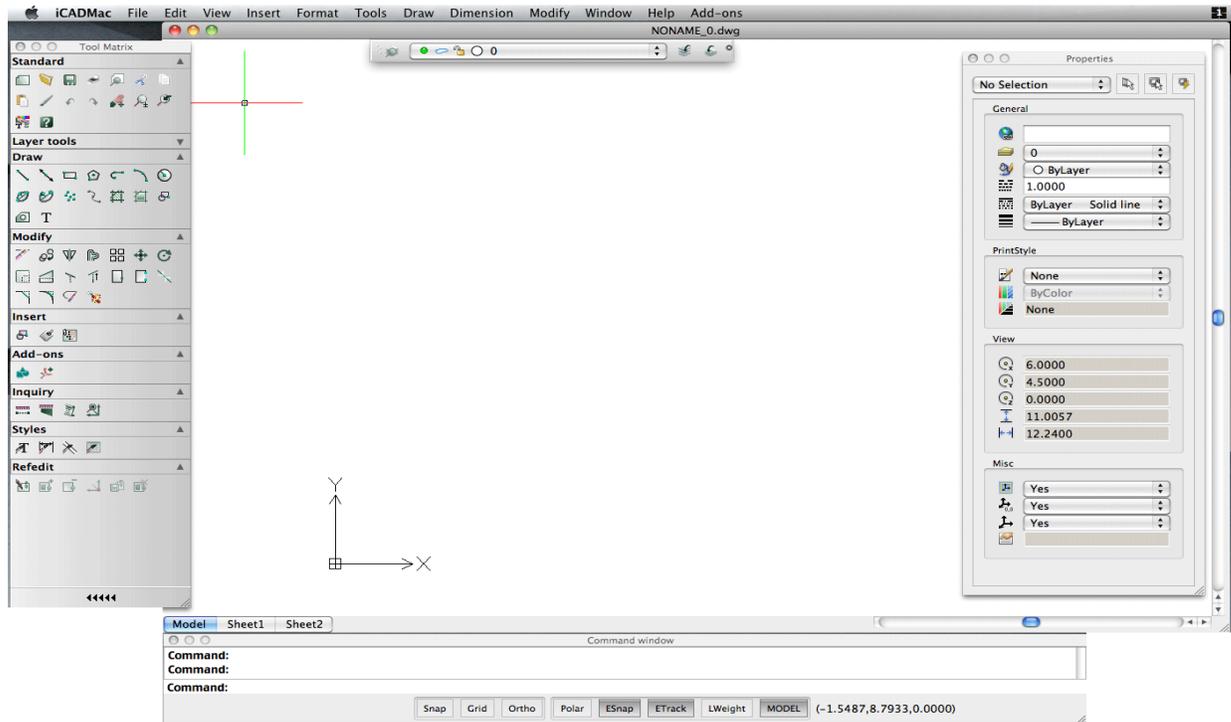
<b>Command</b>	<b>Shortcuts</b>	<b>Menu Selection</b>
DrawingBounds	bounds, limits	Format   Drawing Limits
Grid	<b>fn+F7</b>	...
Layer	la	Format   Layer
New	<b>command+N</b>	File   New
Options	op	iCADMac   Preferences
Save	<b>command+S</b>	File   Save
Snap	<b>fn+F9</b>	...
UnitSystem	un, units	Format   Units
Zoom	z	View   Zoom

## Starting iCADMac With a New Drawing



Start iCADMac by one of the methods described in the previous chapter: click the icon on the dock, or else use the Application folder to access **iCADMac**.

Notice that iCADMac starts with a new, blank drawing. (In your copy of iCADMac, the drawing area may be white or black, depending on settings. You learn how to change this later in this chapter.)



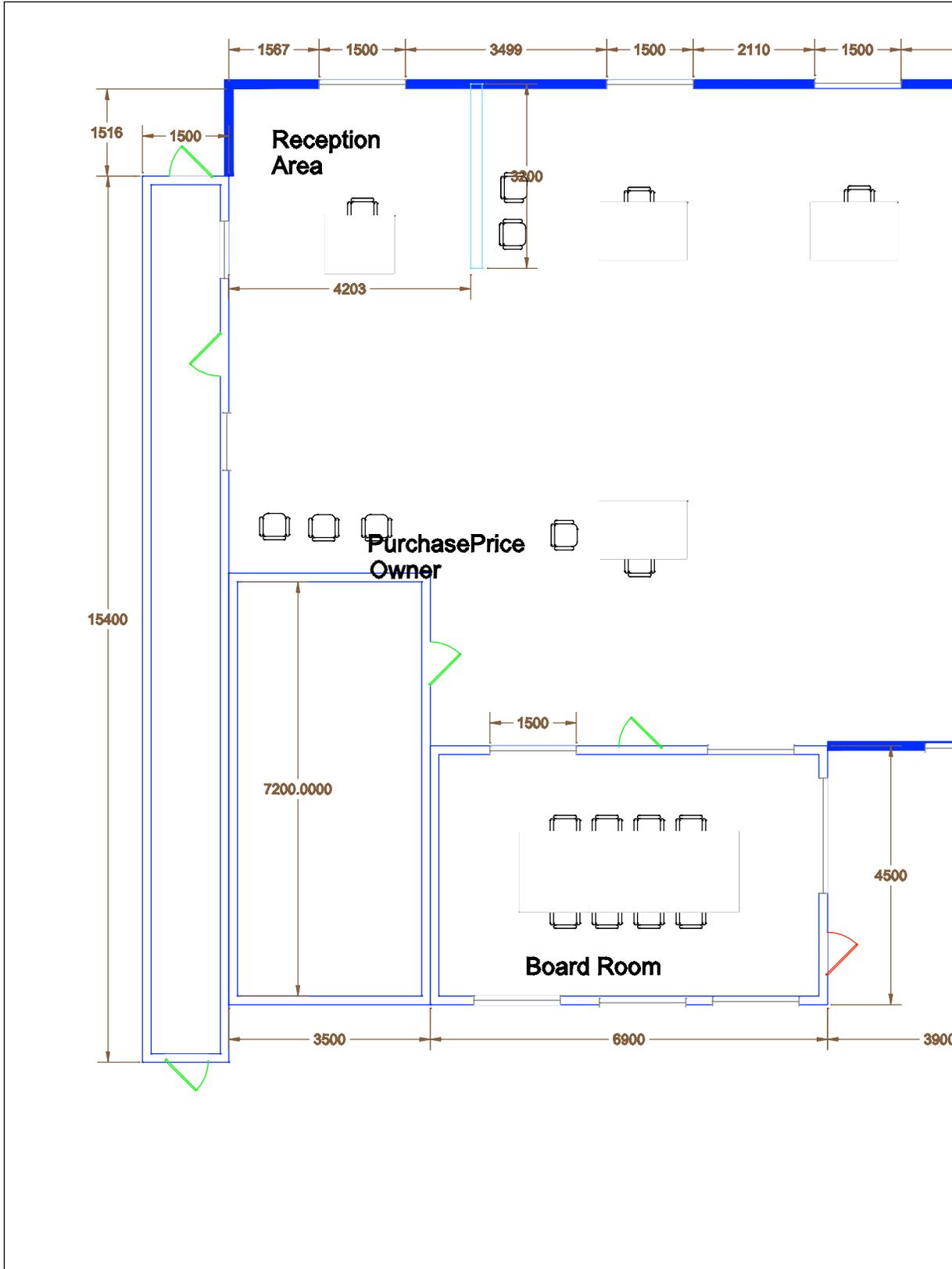
## Saving Drawings

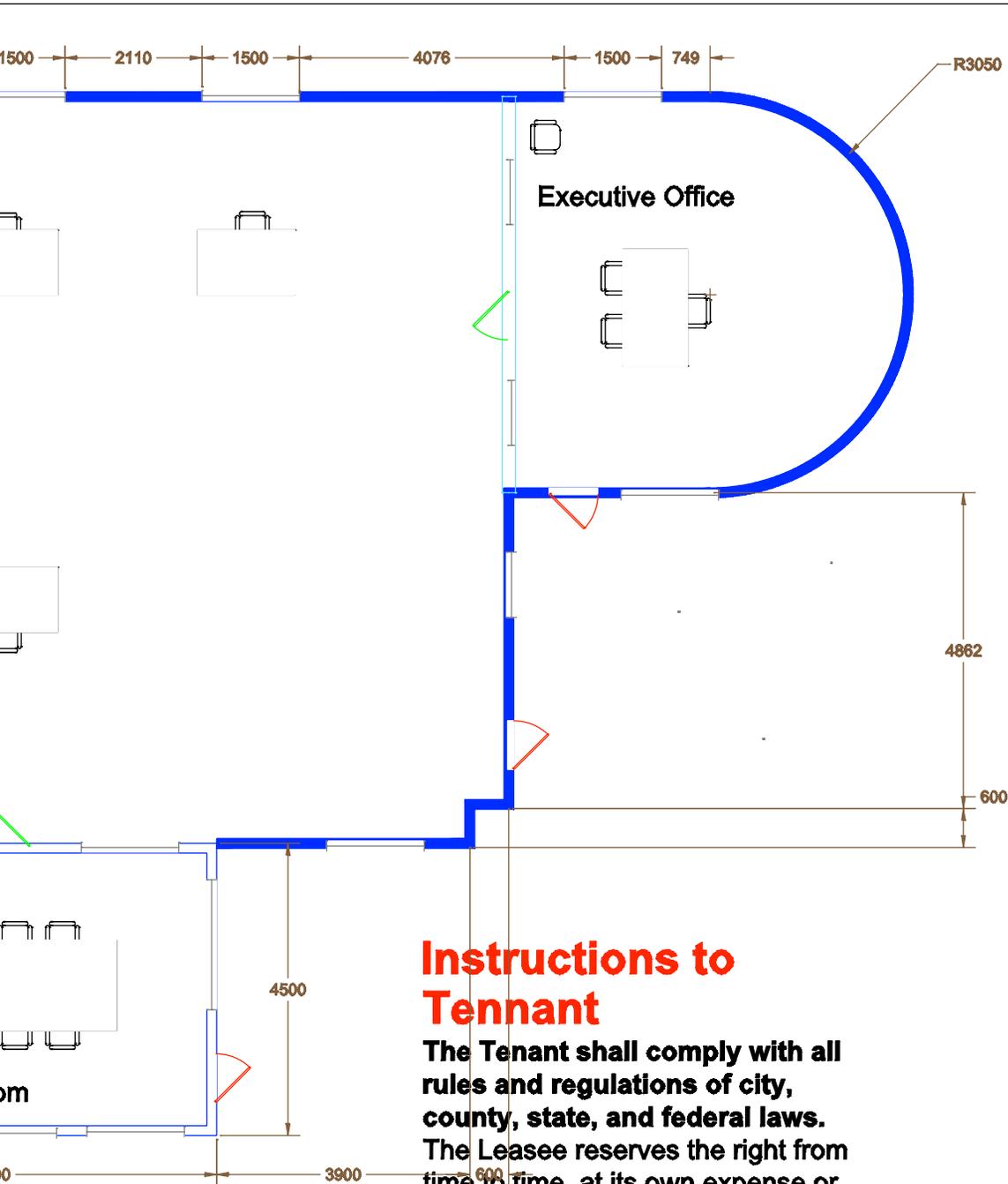
**Command:** Save  
**Alias:** qsave  
**Tool Matrix:** Standard | Save  
**Menu Bar:** File | Save  
**Shortcut:** command+S

From the text displayed by the title bar, you can see that the drawing has the generic name of “NONAME\_0.dwg.” By saving the drawing now, you can change its name to something meaningful.

Drawings are saved with the Save command, as follows:

1. In the Tool Matrix’s Standard section, click the icon that looks like a diskette  (tooltip = Save).
2. Because this drawing has the generic name “NONAME\_0.dwg,” iCADMac automatically displays the Save File dialog box. (When drawings have already been given a name, then this dialog box does not appear; iCADMac just saves the drawing.)

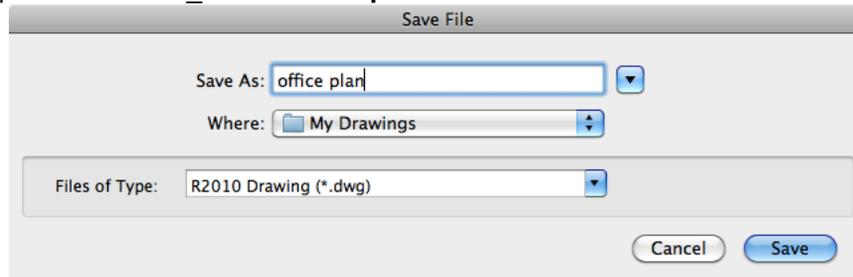




## Instructions to Tenant

**The Tenant shall comply with all rules and regulations of city, county, state, and federal laws. The Leasee reserves the right from time to time, at its own expense or the expense of its agents or contractor, to make renovations, repairs and alterations in and about the leased premises, and may enter upon the premises at a reasonable hour for these purposes.**

3. Decide on the name of the drawing file, as well as the folder in which you want to store it. For this tutorial, use the following settings:
  - Store the drawing in the *My Drawings* folder, the default used by iCADMac.
  - Replace “NONAME\_0” with **office plan**.



4. Click the **Save** button.

iCADMac saves the drawing with the name *office plan.dwg* in the *My Drawings* folder. On the title bar, the name changes to “office plan.dwg.”

From now on, the Save command saves the drawing under this name; it will not display the Save File dialog box. Should you wish to save this drawing under a different name, use the SaveAs command.

## Setting the Units

**Command:** UnitSystem  
**Aliases:** un, units  
**Menu bar:** Format | Unit System

iCADMac can display measurement units in a variety of styles, including metric, fractional, and exponential. (The latter expresses very large numbers). You can choose the type of length and angle measurements, and as well as their precision.

After you select a measurement style, iCADMac displays all measurements in that style. But don't worry: you may switch measurement styles at any time with the UnitSystem command.

By default, iCADMac starts new drawings with the following measurement styles:

### About the SaveAs Command

The Save File dialog box lets you save drawings in formats read by several CAD programs. To see the complete list, click on the Save As Type droplist (shown at right).

**Drawing DWG** is the format used natively by AutoCAD, progeCAD, and quite a few other CAD programs. iCADMac reads and writes DWG files from between Release 12 and 2010.

*Warning!* It is crucial that you match the version number. This is because older releases of CAD software cannot read newer releases of the DWG format. (Newer versions of CAD software, however, can always read older DWG files.) When saving drawings to older DWG releases, iCADMac might erase or alter some entities, because they are not supported by early formats of DWG.

**Template DWT** is a format meant to be used for template drawings. (See chapter 11.)

**Drawing DXF** (drawing interchange format) is read by many CAD, CAM, analysis, and graphical programs.

**Design Web DWF** is a simplified format suitable for displaying drawings on Web sites.

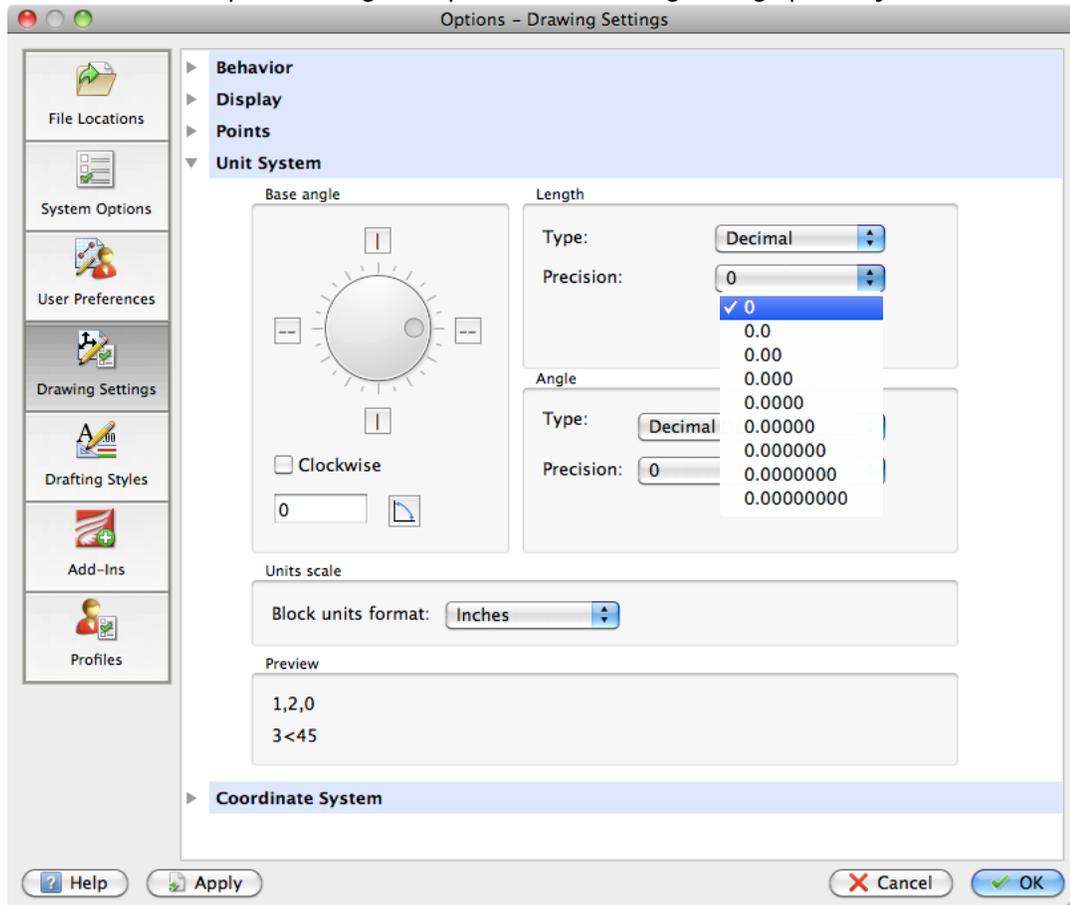
- **Lengths** are in decimal units (another term for “metric”), with four decimal places of precision.
- **Angles** are in decimal degrees, with zero decimal places of precision measured counterclockwise.

The defaults are almost good enough for the office layout drawing, which was measured in millimeters. Only the precision of lengths should be changed, since four places is four too many when working with floor plans.

Follow these steps to change the settings for units:

1. Start the UnitSystem command by one of the methods listed at the start of this section:
  - At the command prompt, enter **unitsystem**.
  - Or, type in one of its aliases, **un** or **units**.
  - Or, from the **Format** menu, choose **Unit System**.

Notice that the Options dialog box opens to the Drawing Settings | Unit System section.



2. The dialog box has many options, but we are interested in just the *precision* of lengths, because the measurements of the floor plan were made to the nearest millimeter. Precision should be changed to 0, as follows:
  - a. In the Length section, locate the Precision drop list.
  - b. Click the down arrow, and then select **0**.
3. Click the droplist next to **Block Units Format**, change from Inches to **Millimeters**. (This setting becomes important in a later chapter when inserting blocks.)
4. Click **OK** to dismiss the dialog box.

You are free to enter distances more accurate than 1 millimeter; iCADMac remembers distances to 14 decimal places of accuracy. It is only when iCADMac displays coordinates that it rounds them up or down to the nearest millimeter.

### Limiting the Drawing Boundary

**Command:** DrawingBounds  
**Aliases:** bounds, limits  
**Menu Bar:** Format | Drawing Boundary

There is no limit to the size of drawings you can create with iCADMac. You could draw the entire solar system full size, if you want, and it would fit a drawing. Office plans are a great deal smaller.

It can be useful to set a boundary to the drawing, because this constrains the extent of grid marks and the Zoom Bound command. (More on these two later.) Here is how to figure out the outer boundary of the drawing:

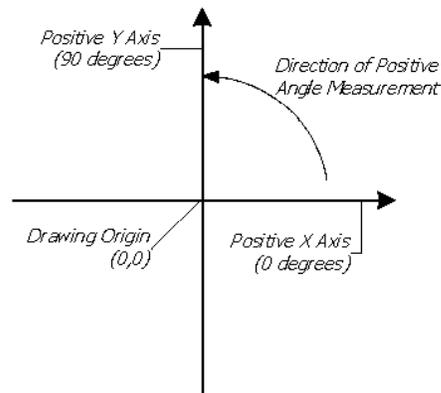
1. Examine the size of the office plan in the sketch. The building is 22.5m wide and 16.4m deep. Leaving a bit of “breathing room” around the plan, the drawing will need about 25m of width and 20m of depth.
2. Start the DrawingBounds command, and then enter the values at the prompts:
  - Command: drawingbounds
  - Options: OFF, ON or
  - Specify lower left corner» 0,0
  - Default: (12,9)
  - Specify upper right corner» 25000,20000

You will not notice any change to the drawing until you turn on the grid, as described next.

#### Units of Angle Measurement

Normally, iCADMac measures angles this way:

- 0 degrees starts from the positive x axis (East).
- Angles are measured using decimal degrees, such as 45.6789 degrees.
- Positive angles are measured counterclockwise. as illustrated below.



The UnitSystem command can change the default settings so that 0 degrees starts at any angle. Angles are measured in degrees-minutes-seconds, grads, radians, or surveyor's units; and either/or clockwise.

**TIP** There are 400 grads and  $2\pi$  radians in a 360-degree circle.

## Setting the Snap and Grid

The primary advantage to drafting with CAD is that it permits you to create very accurate drawings. iCADMac has several features that help you draw with perfect accuracy.

One such feature is called “snap mode.” Snap can be thought of as *setting the drawing resolution*. For instance, a snap of 10mm means that as you move the mouse, the cursor moves by 10mm, precisely.

Another feature is called the “grid,” which helps you visualize distances by displaying an array of dots. For instance, a grid spacing of 1000mm means that a dot appears every 1m, vertically and horizontally. Grid dots are not plotted.

You set the two of them through the Options dialog box; in later chapters of this book, you learn about additional aids to accuracy.

### Snap Spacing

**Command:** Snap

**Alias:** sn

**Status Bar:** right-click Snap

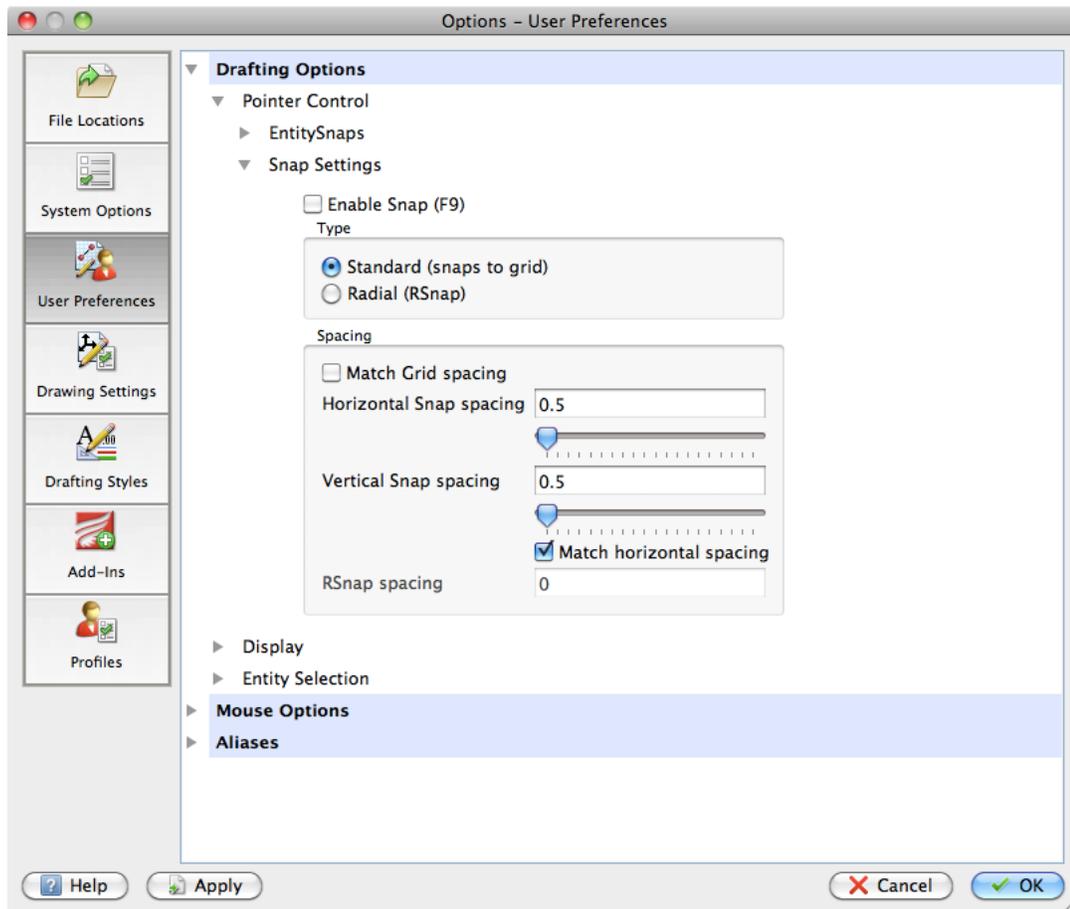
Follow these steps to set the snap spacing:

1. On the status bar, right-click the **Snap** button.



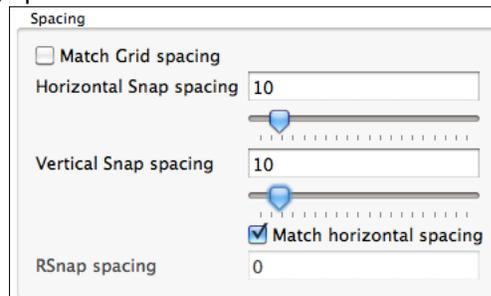
(Why not use the Snap command? This is a case where *not* using the command is the easier approach. This is because the Snap and Grid commands list their options in the command bar, which can be more difficult for new users to handle than a dialog box.)

2. In the shortcut menu, choose **Settings**. Notice that the Options dialog box opens at the Snap Settings section.



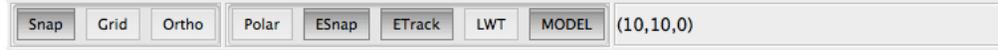
(If the Snap Settings section is not visible, follow these steps (a) choose the **User Preferences** tab, (b) open the **Drafting Options** node, (c) open **Pointer Control**, and then (d) open **Snap Settings**. The advantage to right-clicking the status bar's Snap button is that it takes you directly to this part of the dialog box.)

3. To move the cursor in ten-millimeter increments, make the following changes to the dialog box:
  - a. Click the white square (called a "check box") next to **Enable Snap (F9)**.  
 Enable Snap (F9) (The "F9" text is a reminder that you can turn snap on and off at any time by pressing function key fn+F9.)
  - b. Change **Horizontal Snap Spacing** to **10**.  
 Notice that the Vertical Snap Spacing matches the change (10), because the Match Horizontal Spacing option is turned on.



- c. Leave other snap settings, and then click **OK**.
4. The drawing looks no different, but reacts differently. Move the cursor about. Notice that it jumps, rather than moves smoothly. The cursor is moving in increments of 10mm.

Look at the coordinate display and notice it changes by 10s.



**TIP** When entering text and numbers in a dialog box, pressing the Tab key is a quicker way to get to the next field. Fields are buttons, text entry boxes, list boxes, and other dialog box elements that you can change. To return to previous fields, press **Shift+Tab** (hold down the **Shift** key, and then press **Tab**).

## Grid Spacing

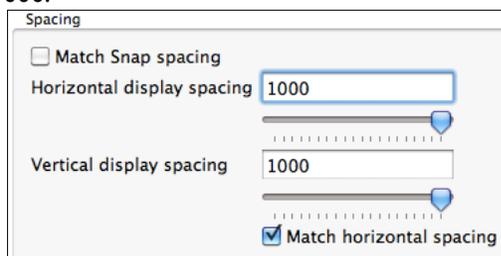
**Command:** Grid

**Status bar:** *right-click* Grid

The grid is meant as guide; the default spacing of 0.5mm gets in the way because it spaces the dots far too closely. In fact, iCADMac won't even display grid dots in this case, because there would be too many on the screen. By changing the spacing to 1000mm, you get just enough to help you get a feel for 1-metre distances.

To use the grid as a visual guide, you need to turn it on. Follow these steps to set the grid spacing to 1000 and turn it on:

1. On the status bar, right-click the **Grid** button.
2. In the shortcut menu, choose **Settings**. Notice that the Options dialog box opens at the Grid Settings section (User Preferences | Drafting Options | Display | Grid Settings).
3. To display grid dots in 1000-millimeter increments, make the following changes to the dialog box:
  - a. Click the check box next to **Enable Grid (F7)**.  Enable Grid (F7) ("F7" reminds you that the grid can be turned on and off at any time by pressing function key fn+F7.)
  - b. Turn on **Match Horizontal Spacing**.
  - c. Change **Horizontal Snap Spacing** to **1000**. The Vertical Snap Spacing options matches the change to 1000.



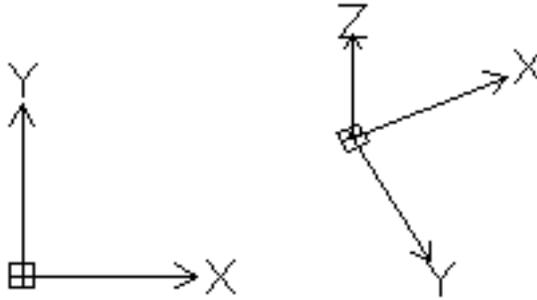
- d. Leave the other grid settings, and then click **OK**. A grid of dots should appear in the drawing area (a.k.a. graphics area).

## Turning Off the CCS Icon

**Command:** Cslcon

**Menu bar:** View | Display | CCS Icon

The double-arrowheaded icon in the lower left corner is known as the “CCS icon,” short for *custom coordinate system*. Its purpose is to orient you in three-dimensional space.



Left: CS icon in 2D drawings.

Right: CS icon in 3D drawings.

Because the floor plan is only drawn in two dimensions, you can turn off the CS icon, as follows.

1. Start the **csicon** command.
  - Command: csicon
  - Default: ON
  - Options: All, No origin, OFF, ON or ORigin
  - Specify option» off
2. Enter the **Off** option.

Notice that the icon disappears. You get to use the CS icon in later chapters that deal with 3D drawing.



## Seeing the Entire Drawing

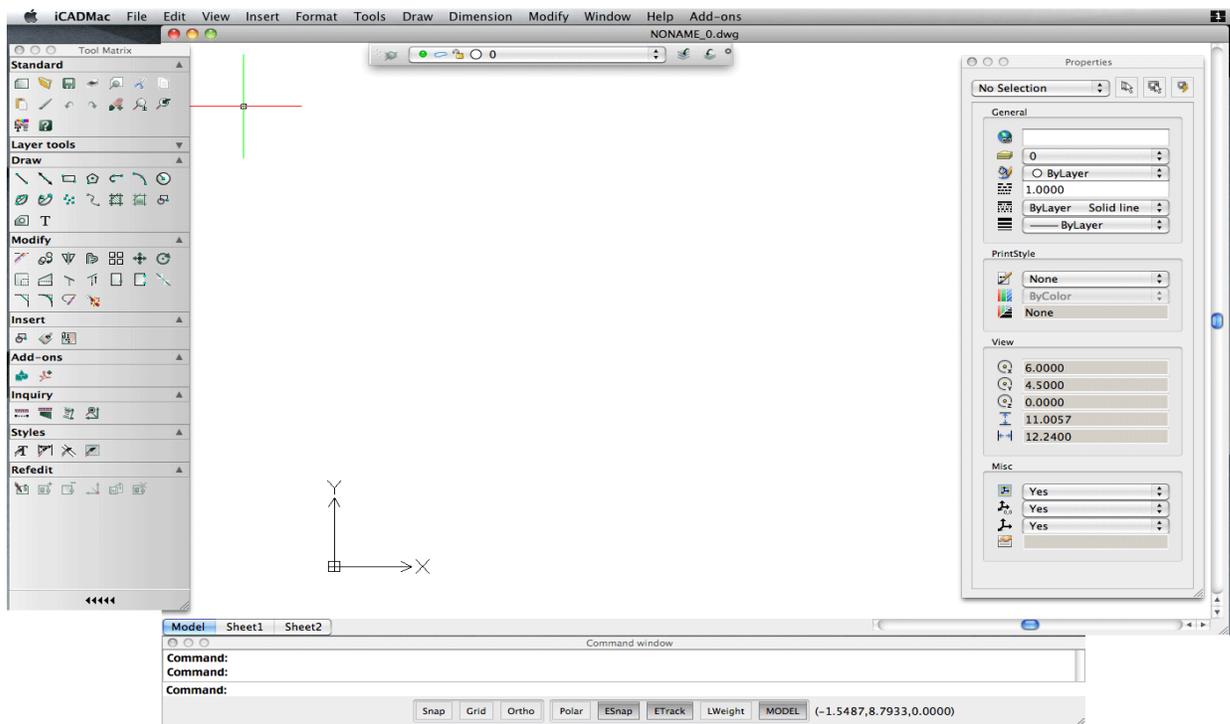
**Command:** Zoom Bounds  
**Alias:** z b  
**Menu bar:** View | Zoom | All

Use the Zoom command with the Bounds option to see the entire grid, giving you a visual indication of the extents of the drawing limits.

: zoom  
 Default: Dynamic  
 Options: Bounds, Center, Dynamic, Fit, Previous, SElected, specify a scale factor (nX or nXP) or  
 Specify first corner» b

Don't be intimidated by the many options of this command; generally, you use just a couple of them.

**TIP** For many options of iCADMac' commands, you need only type the first letter of the option. Above, you typed “b” as the abbreviation for the “Bounds” option. When two options begin with the same letter, you need to type the first two characters of the option.



When you began, iCADMac displayed an area of about 9mm by 12mm; now it displays an area of 25m by 20m. Thus, the **Zoom** command let you see the “big picture,” as well as details.

## Adding Layers

If you have worked with *overlay drafting*, then you are familiar with the concept of *layers*. In overlay drafting, you draw the base plan on a clear sheet of Mylar (plastic drawing media), the electrical plan on another sheet, the structural on a third, and so on. Since the Mylar is transparent, you can overlay the three drawings to create a single blueprint.

Layers in iCADMac operate similarly. You draw parts of drawings on different layers. Then, you can turn layers off and on to display the drawing in different ways. For example, the electrical contractor would be interested only in the layers showing the base plan and the electrical schematics.

A further advantage to layers is that global properties can be applied to entities on a per-layer basis. For instance, all entities on a layer called “grass” could be colored green (or maybe golden brown, depending on the time of year!). Changing the color of the layer to brown instantly changes the color of all entities assigned to that layer.

You can assign line weights, line styles, and print styles to layers, as well as determine whether they should be printed, are *frozen* (hidden from sight), or *locked* (made uneditable).

It is possible to create hundreds and even thousands of layers in drawings for complex projects, but it is more common to work with just a few dozen layers. In this book, you work with a mere half-dozen layers to segregate items like the text, different types of walls, and so on.

## About Layers in iCADMac

iCADMac provides quite a bit of control over layers and the properties of layers. Every layer has a set of properties identified by the header bar in the Layers Manager dialog box:

Status ▾	Name	Show	Frozen	Lock	LineColor	LineStyle	LineWeight	PrintStyle	Print	Description
→	0	●	☐	🔒	● White	Conti... line	— Default	Color_7	🖨	

**Status** reports the status of each layer; the → arrow indicates the active layer.

**Name** identifies layers, and can be up to 255 characters long. You can use numbers, letters, and the following punctuation: dollar signs (\$), hyphens (-), underlines (\_), and spaces. Two layers cannot have the same name in the same drawing. There is no limit to the number of layer names in drawings.

**Show** displays entities assigned to the layer. When show is *off*, you cannot see or edit entities nor are they plotted.

**Frozen** prevents entities on layers from being seen, edited, or plotted. In addition, frozen entities are not included when iCADMac determines drawing rebuilds; this is the only manner in which Freeze differs from Show being turned off; It is better to freeze layers than to turn off their Show status.

When layers are not frozen (thawed), entities can be seen, edited, and plotted. To *thaw* means to turn off the frozen status.)

**Lock** prevents entities from being edited. To *unlock* layers means to make the entities available for editing.

**LineColor** specifies the color of entities on the layer. The default color is white, which is displayed as white or black depending on the background color. You can use the LineColor command to override the color for selected entities.

**LineStyle** specifies the line style for entities assigned to the layer. The default is Continuous Solid Line. You can use the LineStyle command to override line styles for selected entities.

**LineWeight** specifies the line width for entities assigned to the layer. The default is 0.00mm; the maximum is 2.11mm (about 0.08", or 6 points wide). You can use the LineWeight command to override line weights for selected entities.

**PrintStyle** determines the named plot style with which to print entities assigned to the layer. Print styles define colors, widths, and other properties to be used during printing. You can use the PrintStyle command to override print styles for selected entities.

**Print** prints layers; when print is off, entities are not printed. You use the Print command to print drawings.

**Description** includes explanatory text for each layer. This field is optional, and if you wish to add a description, then double-click the field twice.

#### Extra Layer Properties for Sheet Mode

When drawings are in sheet mode (layouts or paper space), the layers dialog box displays two more columns:

Active ViewPort	New ViewPort
☐	☐

**Active ViewPort** freezes/thaws the layer in the active (current) viewport of layouts independently.

**New ViewPort** freezes/thaws the layer the moment new viewports are created in layouts.

## Naming Layers

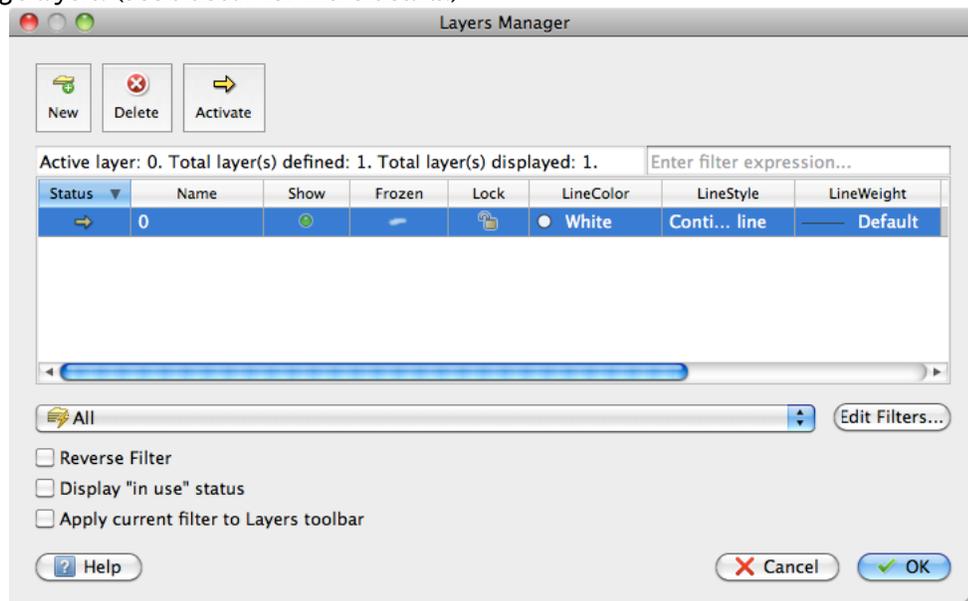
**Command:** Layer  
**Alias:** la  
**Menu Bar:** Format | Layer  
**Tool Matrix:** Layers | Layers Manager

Layers are created by naming them. Through the Layers Manager dialog box, iCADMac lets you name layers with up to 255 characters.

But before beginning to name them, think about the kinds of layers you need to create. Looking at the sketch of the floor plan, you can see a number of areas that could benefit from being segregated by layers. There are the external and internal walls, the furnishings, doors, and windows. Each of these would benefit by being segregated to its own layer.

To create new layers in the drawing, follow these steps:

1. In the Tool Matrix's Layers section, click the **Layer Manager**  button. iCADMac displays the Layers Manager dialog box, which lets you control almost every aspect of the drawing's layers. (See sidebar for more details.)



(For now, we'll ignore the filters section at the bottom of the dialog box.)

Notice that the drawing already has one layer, 0. Every new iCADMac drawing has that one layer. It is fixed and cannot be removed.

2. Create a layer by clicking the  **New** button. Notice that iCADMac creates a new layer called "Layer1."

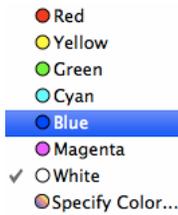


3. Change the name from "Layer-1" to **Walls-External**.

Status ▾	Name	Show	Froz
➔	0	●	⚡
🗑️	Walls-External	●	⚡

4. Assigning colors to layers makes it easier to determine which lines belong to which layer. To change the color of the Walls-External layer to dark blue, follow these steps:

- Click the white circle  under the **LineColor** column (across from the layer name Walls-External).
- A droplist appears. Select **Blue**.



Notice that the color of the circle changes to blue. All entities drawn on layer Walls-External will now be colored blue.

Status ▾	Name	Show	Frozen	Lock	LineColor	Line
➔	0	●	⚡	🔒	○ White	Conti...
🗑️	Walls-External	●	⚡	🔒	● Blue	Conti...

5. Create another layer, naming it “Walls-Internal” and assigning it a light blue color. Light blue is not listed on the LineColor droplist, so here is how to select a color other than the seven shown in the droplist:
- In the LineColor droplist, choose **Specify Color**.

### Controlling Layer Listings

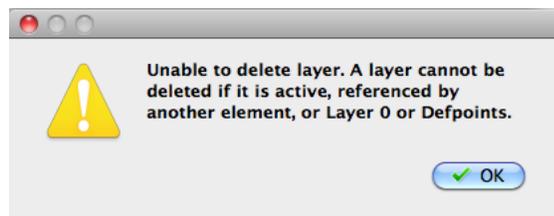
To control drawings with many layers, use the Layers Manager dialog box to sort layers and display selected groups of layers. To sort layers alphabetically, click header such as Name or Freeze. The layers are sorted alphabetically by name or by freeze/thaw status. (Notice the arrowhead.) Click the headers a second time to sort in reverse order (Z to A).

#### Erasing Layers

Status	Name ▾	Show	Frozen	Lock	Line
➔	0	●	⚡	🔒	● Wh

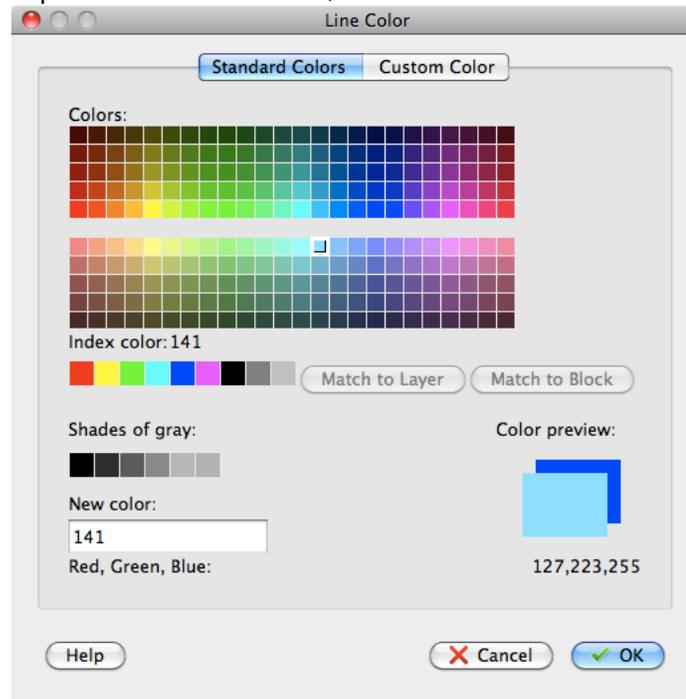
To erase one or more layers from the drawing, select them and then press the **delete** key. iCADMac changes the layers status icon to , indicating the layer will be removed once you click **OK**. To reverse the accidental deletion of layer(s), use the **U** command.

You can only erase layers that are empty, not active, not part of externally referenced drawings, and not layer 0. If you try, iCADMac displays a warning dialog box.



- Red
- Yellow
- Green
- Cyan
- Blue
- Magenta
- White
- Specify Color...

- b. Notice the Line Color dialog box. Click on a light blue square, such as 141. (The Index Color item reports the color's number.)



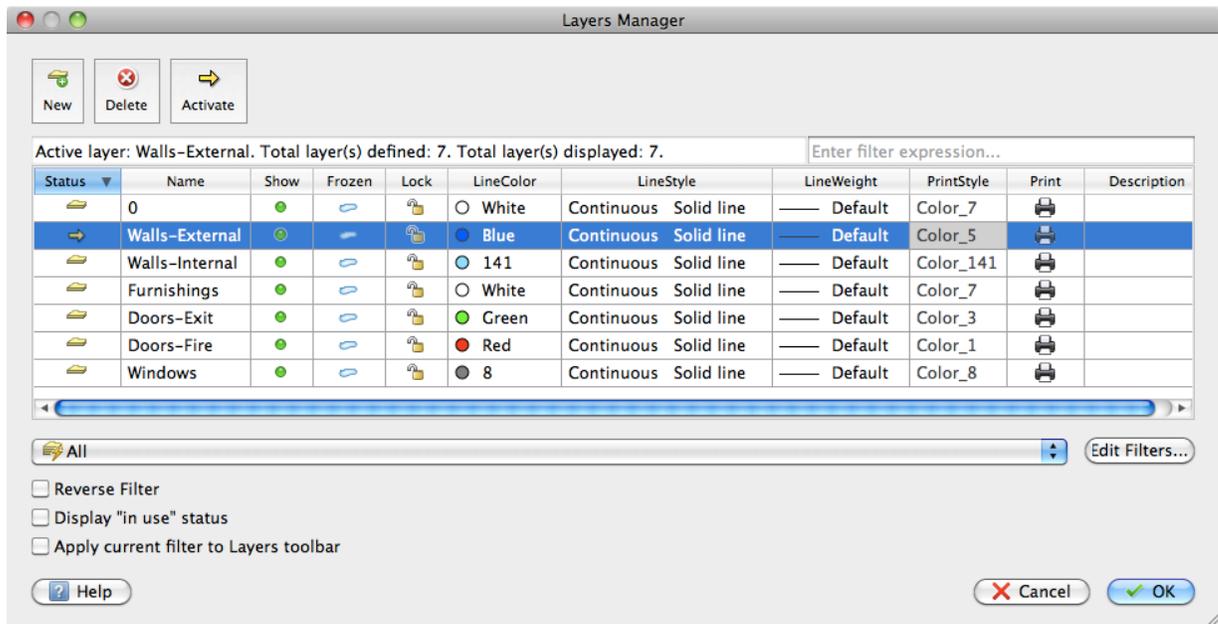
- c. Click **OK**. Notice that the color for layer Walls-Internal changes to light blue.
6. Continue adding layers and assigning colors according to the table below:

<b>Layer Name</b>	<b>LineColor</b>
Furnishings	White
Doors-Exit	Green (color 4)
Doors-Fire	Red (color 1)
Windows	Gray (color 8)

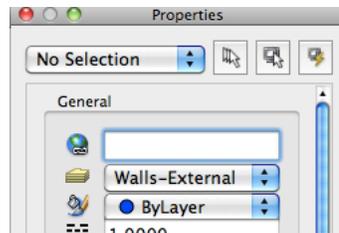
If you make a mistake in spelling a layer name, just click the name twice, and then correct it.

7. When you finish, pick the Walls-External layer, and then click **Activate**. From now on, all drawing takes place on the Walls-External layer — at least until you activate another layer.

When done, the Layers Manger dialog box should look like this:



8. Click **OK** to exit the Layers Manager dialog box. Notice that the Properties palette changes the name of the layer from 0 to External-Walls, and that the color changes from white to blue.



## Automatic Backups

**Menu Bar:** iCADMac | Preferences

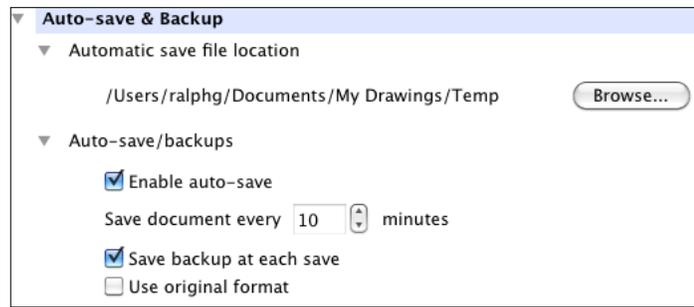
To make software run as quickly as possible, all programs keep their data in the computer's memory (RAM). iCADMac is no different. The drawback to this approach is that when OS X crashes or when the power is cut to the computer, you lose your work. It is thus an excellent idea to save drawings every 10 or 15 minutes.

To do so, get into the habit of pressing **command+S** regularly. (This is the shortcut keystroke for the Save command.)

iCADMac also quietly saves your drawings automatically every ten minutes to a folder named `/var/folders/-z`. Automatically-saved drawings are given the extension of `.ds$`. (The \$ sign is traditionally used in computing to signify temporary files.) The backup file of this tutorial's drawing is `office plan_19172.ds$`.

To enable automatic backups, follow these steps:

1. From the **iCADMac** menu, select **Preferences**. iCADMac displays the Options dialog box.
2. Select the **System Options** tab.
3. Open the **Auto-save & Backup** node, and then the **Auto-save/backups** node.



4. If necessary, click the check box next to **Enable Auto-save** to turn on the option.
5. The **Save Document Every** box specifies how much time elapses before iCADMac saves the drawing. Don't set this number too low, or the computer spends so much time saving to disk that you cannot do other work.
6. If haven't already done so, you might want to turn on **Save Backup at Each Save**. This means that iCADMac makes backup (\*.bak) copies of drawing files with each save. The older version is renamed *office plan.bak*.
7. To save backups in a convenient folder location, click Browse, and then choose a location such as *My Drawings/Temp*.
8. Click **OK**. Although iCADMac will automatically save your drawings every ten minutes, it is still a good idea for you to save your work with **command+S** after finishing significant amounts of editing.

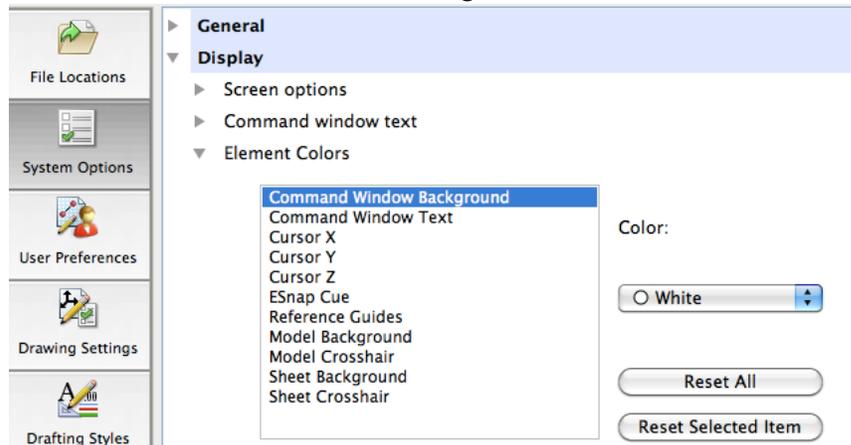
#### What Color is White?

There can be some confusion over the color “white” — or is it black? iCADMac switches between white and black depending on the background color of the drawing area. When the background is black, iCADMac displays white lines; when white, iCADMac displays black lines. In both cases, however, iCADMac calls the color “white.” So, white can be black — at least in the world of CAD!

#### Changing the Background Color

To change the background color of the drawing area, follow these steps:

1. From the **iCADMac Command Edition** menu, select **Properties**.
2. In the Options dialog box, click the **System Options** tab.
3. Open the **Display** node, and then the **Element Colors** node.
4. Select the window element (**Model Background**), and then choose a color.

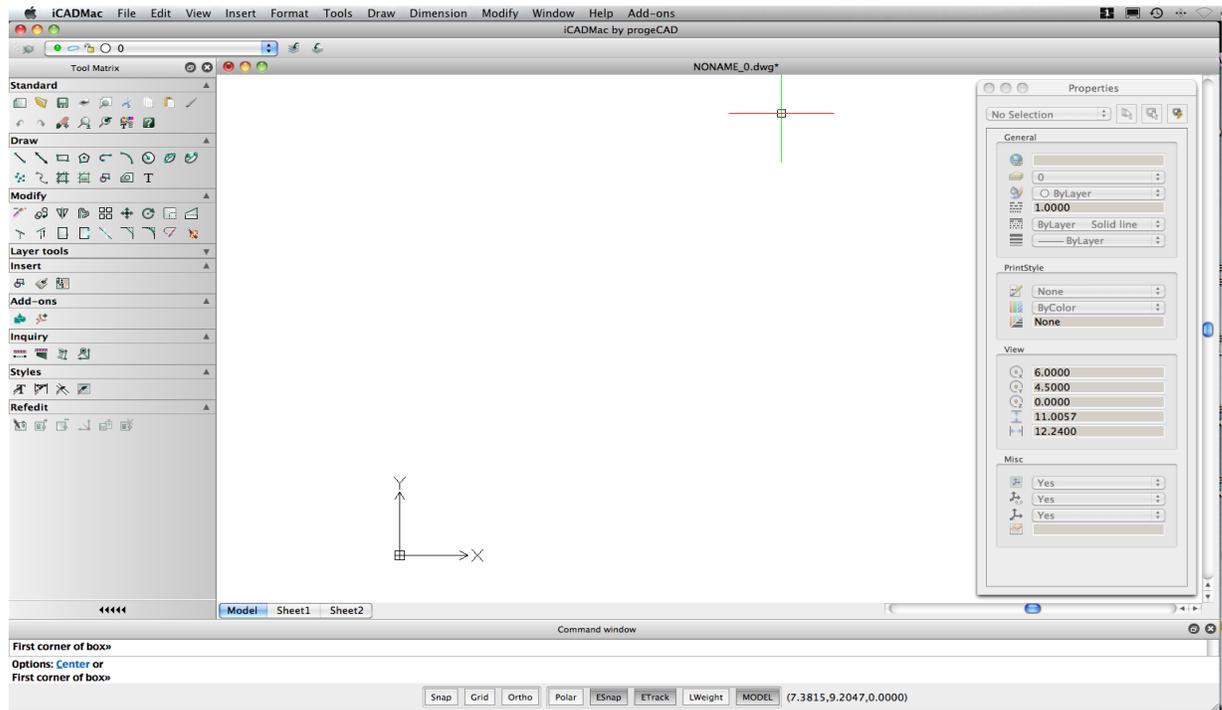


**TIP** Automatic saves and backups are not made to the original files. During automatic saves, iCADMac saves drawings with the extension of .ds\$ and backed up files with .bak.

9. If you need to stop at this point, use **iCADMac | Quit**. iCADMac asks to save open drawings, and then closes down.

## Summary

Let's review the drawing to this point. Although you haven't drawn anything yet, the drawing file contains a fair amount of information.



On the Properties palette, you see that the color of the current layer is blue and its name is Walls-External. On the status line, you see that the coordinates are displayed in millimeters, and that drafting modes snap and grid are turned on.

In the next chapter, you begin drawing the external walls of the floorplan.

## Chapter 5

# Drawing Walls

### In This Chapter

- Drawing walls with lines, rectangles, polylines, and richlines
- Creating richline styles, and modifying polyline widths
- Offsetting and trimming entities
- Understanding absolute and relative distances

**I**n the last chapter, you learned how to start iCADMac, set up new drawings, and save drawings to the computer's hard drive. In this chapter, you learn how to draw lines accurately, make simple changes to drawings, and produce copies of drawings on your printer.



### Bringing Back the Office Drawing

**Command:** Open

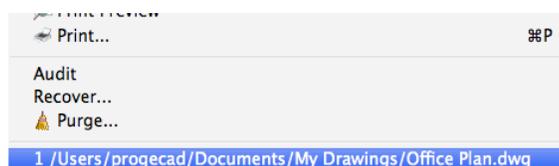
**Menu Bar:** File | Open

**Tool Matrix:** Standard | Open

**Shortcut:** command+O

If you exited iCADMac at the end of the last chapter, you need to restart iCADMac, and load the office plan drawing. Here's how:

1. Start iCADMac by selecting the **iCADMac** icon from the dock. (Alternatively, click the iCADMac icon on the Applications folder.)
2. iCADMac remembers the names of the last ten drawing files you saved. Here is how to access them. Click the **File** item on the menu bar. Notice the list of recently-opened drawings that appears at the bottom of the menu.



3. Select **office plan.dwg**. The drawing should look exactly the same as when you last saw it, that is to say, blank, except for the grid.

## Key Terms in This Chapter

**Absolute coordinate** — refers to measurements made relative to the drawing's origin.

**Entity snap** — snaps cursor to geometric features, such as the ends of lines.

**Origin** — locates 0,0, usually at the lower-left corner of drawings.

**Polyline** — draws connected lines and/or arcs treated as a single entity.

**Read-only** — refers to drawings that cannot be saved to their original files.

**Relative coordinate** — refers to measurements made relative to the last point.

**RichLine** — draws up to 16 parallel lines with color fill and optional endcaps.

## Commands

Command	Shortcuts	Menu Selection	Status Bar
Cancel	esc	... <sup>1</sup>	...
EditPolyline	pedit, edpl, pe, edpline	Modify   Entity   Polyline	...
Delete	delete	Edit   Erase	...
Line	l	Draw   Line	...
LineWeight	lw, lweight	Format   Line Weight	LWT
Open	command+O	File   Open	...
Ortho	<b>fn+F8</b>	...	Ortho
EntitySnap	es, esnap, os, osnap	...	ESnap   Settings
Offset	o, qoffset	Modify   Offset	...
Polyline	pl, pline	Draw   Polyline	...
Rectangle	rec, rect, rectang	Draw   Rectangle	...
Redo	command+Y	Edit   Redo	...
RichLine	rl, ml, mline	Draw   RichLine	...
RichLineStyle	rls, rlstyle, mlstyle	Format   RichLine Style	...
Trim	tr	Modify   Trim	...
Save	command+S	File   Save	...
U	command+Z	Edit   Undo	...
Zoom	z	View   Zoom	...

<sup>1</sup> No menu selection.

## Drawing Walls

Let's get some lines on the screen! The office floor plan consists largely of straight lines, some of which form rectangles. For the remainder of this chapter, you draw straight lines and rectangles using a variety of methods.

**Line** command draws independent line segments.

**Rectangle** command draws rectangles.

**Polyline** command draws connected line and arc segments with optional width.

**RichLine** draws connected parallel lines.

Using each one gives you experience with the different methods of drawing lines in iCADMac. You gain exposure to the pros and cons of each, and then you can decide which is best for particular situations.

### Drawing Lines with Absolute Coordinates

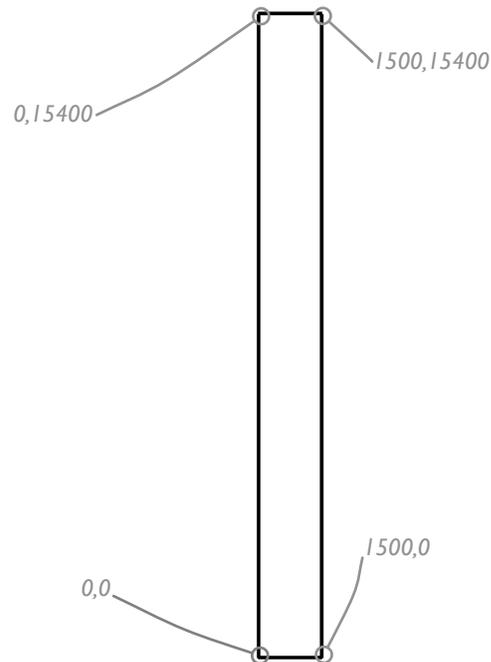
**Command:** Line

**Alias:** l

**Menu Bar:** Draw | Line

**Tool Matrix:** Draw | Line

You start the drawing with the common hallway using the Line command. The lines begin at the lower-left corner known as the “origin,” where  $x = 0$  and  $y = 0$ , and then are drawn using absolute coordinates. The  $x, y$  coordinates are shown in the following illustration.



Absolute coordinates are just that: the actual  $x, y$  coordinates for each point in the drawing. (Later, you draw with the other kind of coordinates, relative.)

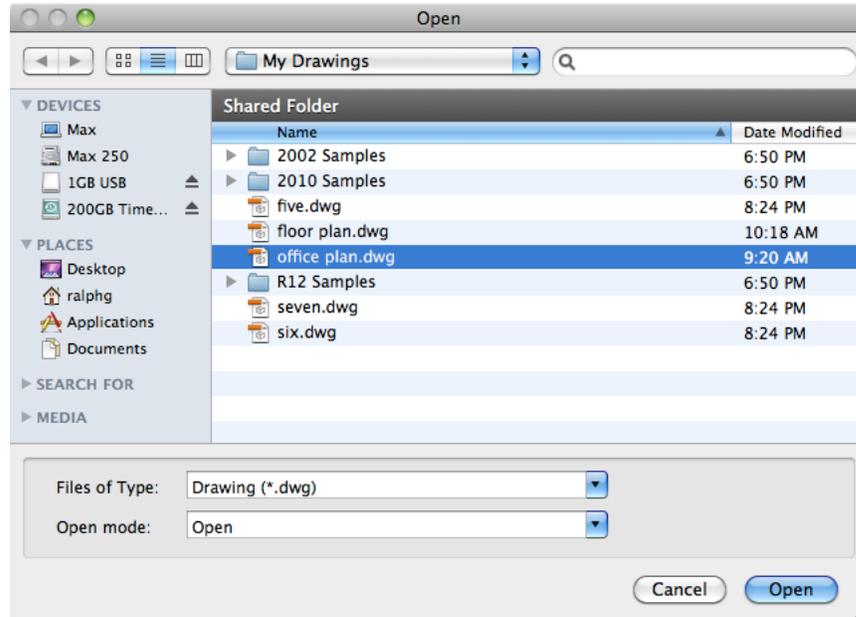
1. Start the Line command using one of the methods listed above. (Enter **line** or its alias **l** at the ‘Command:’ prompt; or from the **Draw** menu, select **Line**; or click the **Line** button on the **Draw** tool matrix.)
2. As confirmation, iCADMac prints “\_LINE” at the command line. Respond to the ‘Specify first point:’ prompt by typing the coordinates of the origin:

## Opening Drawings with the Open Dialog Box

By default, iCADMac saves drawings in the `c:\users\\documents\my drawings` folder. Naturally, you can use any folder you wish. But if you cannot find your drawings, more than likely they can be found in the `\my drawings` folder.

When drawing names are not listed in the File menu (described above), then you need to use the Open dialog box.

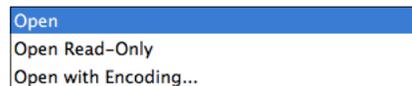
1. Enter the **Open** command, or click the  Open button on the Tool Matrix. Notice the dialog box.



2. Navigate to the folder in which drawing files are stored. *My Drawings* is the default folder.
3. (Optional.) Choose a file format from the **Files of Type** droplist: drawing DWG, Web DWF, Interchange DXF or template DWT.
4. Select a file name.
5. Click **Open**. The drawing appears in iCADMac.

### Open Options

The Open Mode droplist contains two other ways to open drawings:



- **Open Read-only** opens the drawing in *read-only* mode, which means you cannot save (*write*) changes to the original file; instead, you have to save the drawing by a different name.
- **Open with Encoding** opens drawings with alternate character sets. “Coding” refers to the language codepage number the software uses to determine character encoding, such as Baltic or Arabic. This option displays a dialog box that allows you to choose the appropriate codepage number.

**TIPS** You can open the same drawing in iCADMac more than once. The second and subsequent drawings are opened read-only.

Command: `_LINE`  
 Options: `Segments` or  
 Specify start point» 0,0

- To draw the lower 1500-long line, you need to tell iCADMac that the other end of the line is located at the x, y coordinates of 1500,0:  
 Options: `Segments`, `Undo`, `Enter to exit` or  
 Specify next point» 1500,0

**TIPS** Notice the words `Segments`, `Undo`, and `Enter`. These are options of the Line command. **Segments** draws single line segments; **Undo** undraws the last line segment. **Enter** (return) or **esc** exit the command.

Some letters are underlined, such as `Undo`. This indicates the abbreviation. You can enter **undo** or **u**. **return** is in boldface to indicate a keystroke: you press the **return** key; you do not type r-e-t-u-r-n.

- The next line is 15400mm north. Its endpoint is located at coordinates 1500,15400:  
 Options: `Segments`, `Undo`, `Enter to exit` or  
 Specify next point» 1500,15400

Remember that these numbers are large because they represent millimeters.

- Draw the topmost segment:  
 Options: `Segments`, `Undo`, `Close`, `Enter to exit` or  
 Specify next point» 0,15400

**TIP** If you make a mistake entering coordinates, simply type **u** to undo the last segment, and then re-enter the coordinates.

- To finish the lot boundary, use this shortcut: type **c** (short for “Close”), instead of typing the final coordinates (0,0).  
 Options: `Segments`, `Undo`, `Close`, `Enter to exit` or  
 Specify next point» c

iCADMac automatically draws a line from the current endpoint to the beginning of the first line.

- Save your work by pressing **command+S**.

## Drawing Lines with Relative Coordinates

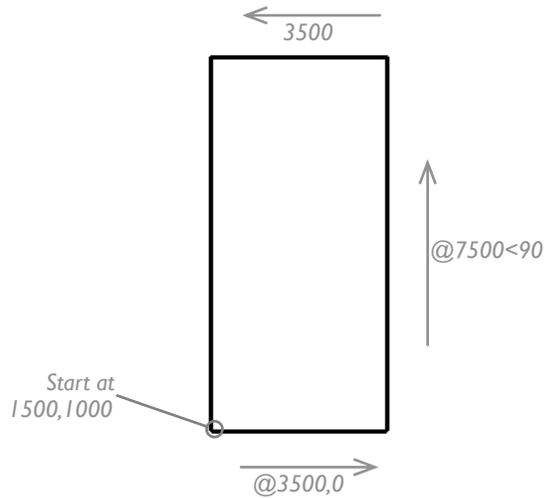
You drew the hallway with *absolute coordinates*, coordinates based on measurements from origin (at 0,0). iCADMac can also draw lengths and angles, known as *relative coordinates*. In this case, iCADMac draws a line from the current point relative to the last point.

It is easier to see relative coordinates in action, rather than to read about them. Draw the next rectangle using the Line command and relative coordinates. (The next rectangle is the 3500x7500mm storage room, located 1000mm up from the x axis.)

- Turn on ortho mode by clicking the  **Ortho** button on the status bar. (It becomes dark gray when turned on.) iCADMac acknowledges this at the command prompt:

Command: <Ortho On>

Ortho is short for “orthographic.” This mode forces iCADMac to draw all lines precisely to the horizontal or vertical — no angled lines, something that is useful for drawing rectangles.



2. Start the **Line** command.
3. The first point is at 1500,1000:
  - Command: line
  - Options: **Segments**, **Enter to continue from last point** or
  - Specify start point» 1500,1000

1500,1000 is the absolute coordinate for the lower left corner of the rectangle.
4. To draw the bottom segment of the rectangle, define its length using relative coordinates, as follows:
  - Options: **Segments**, **Undo**, **Enter to exit** or
  - Specify next point» @3500,0

iCADMac draws the line to the right by 3500 units from the last point (1500,1000), and 0 units up.
5. Draw the next line, specifying the relative polar coordinates:
  - Options: **Segments**, **Undo**, **Enter to exit** or
  - Specify next point» @7500<90

**TIP** When you tell iCADMac to draw a line with the above relative polar coordinates, you enter a special notation that has the following meaning:

Notation	Meaning
@	Use relative coordinates.
7500	Distance is 7500mm from the current point.
<	Draw the line at an angle...
90	...of 90 degrees

Lines are drawn relative to the current point; the angle, however, is measured in absolute degrees using the East-is-0-degrees convention: 90 degrees is drawn straight up. Using relative polar coordinates makes sense when you have many angled lines to draw.

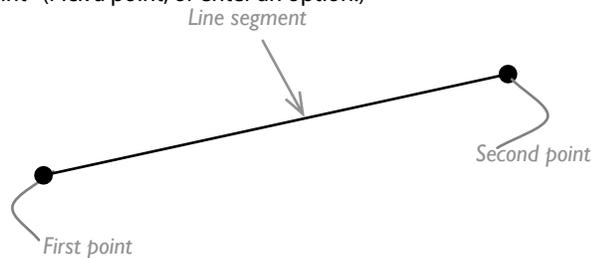
6. You enter 3500 to draw the top of the rectangle. This time, though, use a different method to indicate the length. Follow these steps:
- Move the mouse to the left. Notice a ghost line that appears.



- Enter **3500**, and then press **Enter**.  
Options: Segments, Undo, Close, Enter to exit or  
Specify next point» 3500

#### Summary of Line Command Options

: line  
Options: Segments, Undo, Enter to exit or  
Specify next point» (Pick a point, or enter an option.)



**Specify start point** — specifies the coordinates of the starting point of the line segment. You can specify the point by any of the following methods:

- Pick a point in the drawing using the cursor.
- Enter coordinates at the keyboard.
- Obtain a point from existing geometry using entity snaps.
- Press **return** to continue the line from the last-drawn line or arc.

**Segments** — draws single line segments, prompting for start and end points of each segment.

**Enter to exit** — exits the command when you press the **Enter** key.

Options: Segments, Undo, Close, Enter to exit or  
Specify next point» (Pick a point, or enter an option.)

**Undo** — undraws the last line segment; repeat this option to undraw previous segments.

**Close** — closes the polygon by drawing a segment that connects the last and first points automatically.

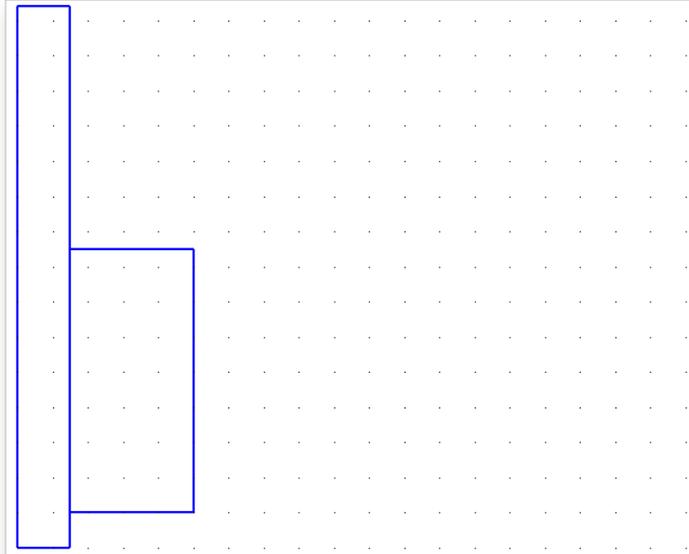
**Specify next point** — specifies the endpoints of line segments.

- Use the Close option to close the rectangle:

Options: [Segments](#), [Undo](#), [Close](#), [Enter to exit](#) or  
Specify next point» c

“Closing the rectangle” means that iCADMac draws the fourth segment for you.

- Save your work by pressing **command+S**. The drawing now contains two rectangles, representing the hallway and the storage room.



Unlike drafting on paper, you are drawing the floor plan full size (even if it looks small on your computer screen). This is one of the most powerful aspects of CAD: everything is drawn full size. There is no need to use a scale ruler or to divide distances by a scale factor as in manual drafting.

### **ESnap** Using Entity Snaps

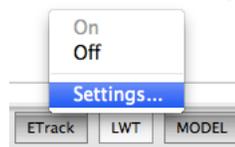
**Command:** EntitySnap  
**Aliases:** es, esnap, os, osnap  
**Status Bar:** ESnap | Settings  
**Tool Matrix:** Entity Snap  
**Shortcut:** fn+F3

Next draw the conference room. Notice that its lower right corner starts at the storage room’s lower left corner. You can take advantage of that geometry to start the rectangle precisely there using entity snaps.

*Entity snaps* (“esnaps” for short) snap the cursor to specific geometric features, such as to the intersections of lines, the centers of circles, and the midpoints of lines. Here you use Intersection esnap.

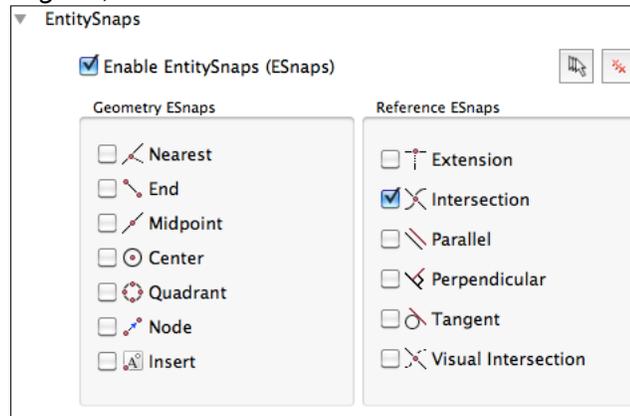
To ensure the Intersection esnap is turned on, follow these steps:

- Right-click the **ESnap** button, and then choose **Settings** from the shortcut menu.



- In the Options dialog box, Click **Enable EntitySnaps (ESnaps)** to turn on entity snapping.

- In the Options dialog box, ensure that **Intersection** is checked.



- Click **OK** to exit the dialog box.

With Intersection esnap turned on, draw the rectangle.

## Drawing Rectangles

**Command:** Rectangle

**Aliases:** rec, rect, rectang

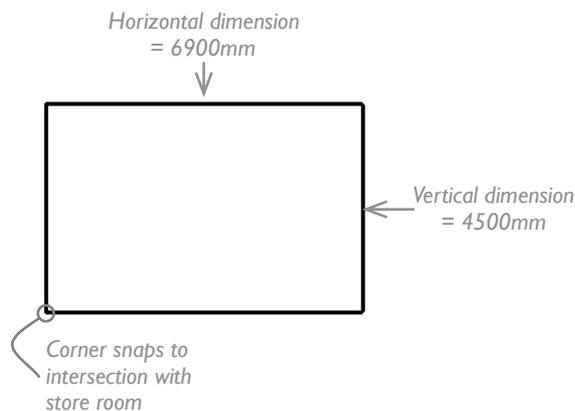
**Menu Bar:** Draw | Rectangle

**Tool Matrix:** Draw | Rectangle

You have drawn two rectangles with the Line command, one with absolute coordinates, the other with relative ones. iCADMac has a command that draws rectangles directly, the Rectangle command.

There is a difference, however, in *what* the two commands draw: Line draws line segments, whereas Rectangle draws polylines. *Polylines* are connected lines. Move one segment and the entire polyline moves.

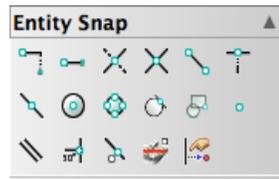
You will use the Rectangle command to draw the conference room.



- Start the **Rectangle** command using one of the methods listed above.
- At the prompt, move the cursor to the intersection indicated by the figure below:  
: rectangle  
Options: **Chamfer, Elevation, Fillet, Thickness, line Width or**  
Specify start corner» (Click at the intersection.)

## All Entity Snaps

The Entity Snap icons on the Tool Matrix illustrate all of the entity snaps available in iCADMac:



On the top row, left to right, these are:

**Snap from** — snaps at a distance from geometry.

**Inference point** — snaps to temporary points.

**Virtual intersection** — snaps to the apparent intersection (used in 3D drawings).

**Intersection** — snaps to the point where two entities cross.

**Endpoint** — snaps to the endpoints of lines, polylines, arcs, and other open entities, including end points of planes and solids.

**Extension** — snaps to the imaginary endpoint, if an open entity were to be extended sufficiently.

On the middle row, these are:

**Midpoint** — snaps to the midpoints of lines, polylines, and arcs.

**Center point** — snaps to the center points of circles, ellipses, rings, polyarcs, and arcs.

**Quadrants** — snaps to the 90-degree points (quadrants) of arcs and circles.

**Tangent** — snaps to the point of tangency on circles and arcs.

**Insertion point** — snaps to the insertion point of text, attributes, blocks, and shapes.

**Node** — snaps to points.

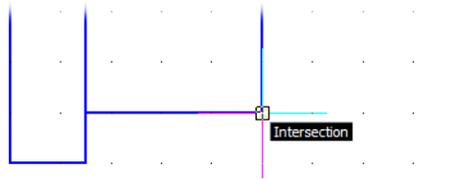
On the bottom row, these are:

**Parallel** — snaps parallel to the geometry.

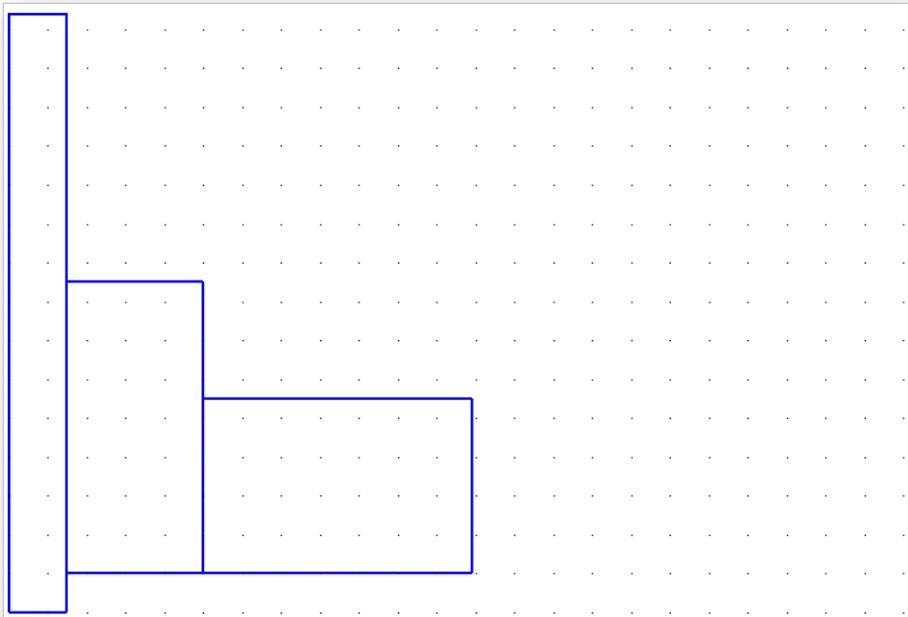
**Perpendicular** — snaps perpendicularly to the nearest geometry.

**Nearest** — snaps to the nearest geometry.

**Off** — turns entity snapping off.



3. Enter **d** to specify the dimensions of the rectangle:  
Options: **A**rea, **D**imensions, **R**otation or  
Specify opposite corner» **d**
4. The dimensions of the rectangle are 6900mm horizontally and 4500 vertically:  
Default: 10  
Specify horizontal dimension» 6900  
Default: 10  
Specify vertical dimension» 4500  
Notice that iCADMac draws the rectangle.



## Drawing Polylines

**Command:** Polyline  
**Aliases:** pl, pline  
**Menu Bar:** Draw | Polyline  
**Tool Matrix:** Draw | Polyline

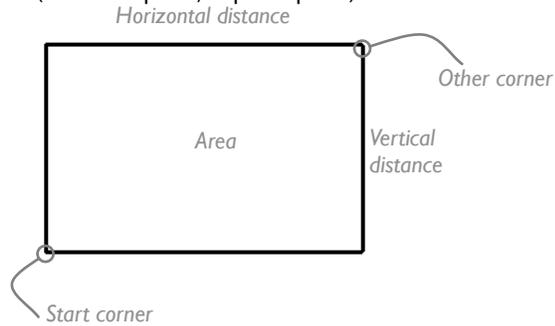
When you used the Rectangle command earlier in this chapter, you drew a rectangle made of polylines — lines connected together into a single object. As the prefix “poly” suggests, polylines are made up of many features — lines and arcs, or splines of varying widths — all connected as a single entity.

## Summary of Rectangle Command Options

: rectangle

Options: **Chamfer**, **Elevation**, **Fillet**, **Thickness**, **line Width** or

Specify start corner» (Enter an option, or pick a point)



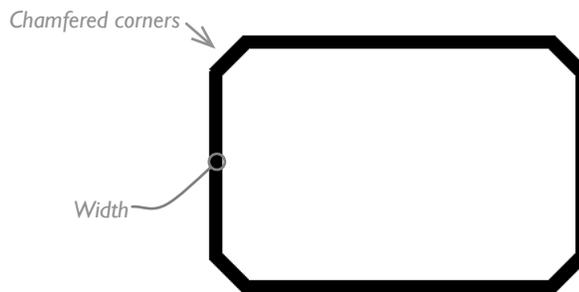
**Specify start corner** specifies one corner of the rectangle:

- Pick a point in the drawing using the cursor.
- Enter coordinates at the keyboard.
- Obtain a point from existing geometry using entity snaps.

**Chamfer** specifies the two distances for bevelling the corners of the rectangle; the same size chamfer is applied to all four corners.

Specify first chamfer length» (Enter a distance.)

Specify second chamfer length» (Enter a distance.)

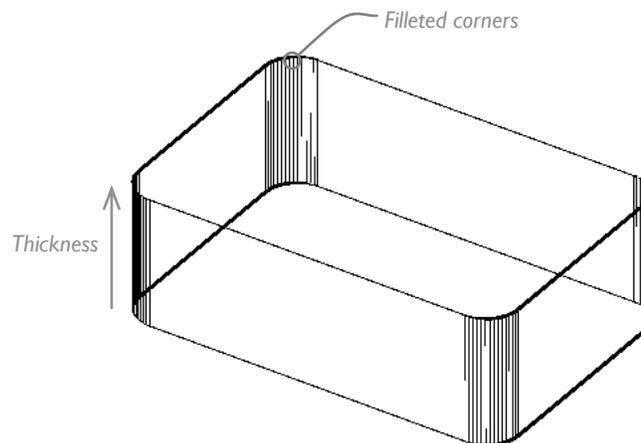


**line Width** specifies the width of the rectangle's lines.

Specify line width» (Enter a width.)

**Fillet** specifies the radius of rounded corners; the same fillet is applied to all four corners.

Specify fillet radius» (Enter a radius.)



*Rectangle Command: Options continued...*

**Thickness** specifies the thickness of the rectangle (height in the z direction).  
Specify thickness» (Enter a thickness.)

**Elevation** specifies the height of the rectangle above or below the x,y-plane; positive numbers locate the rectangle above the plane; negative ones below.  
Specify elevation» (Enter a height.)

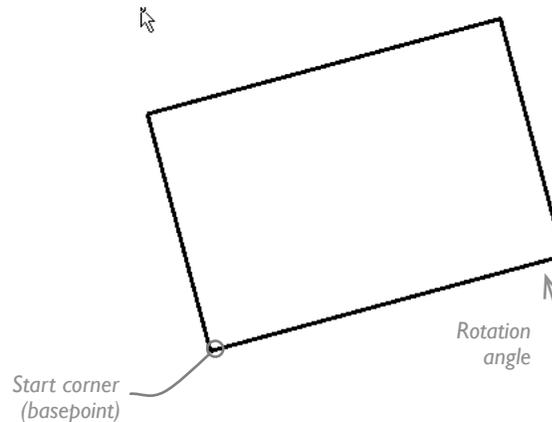
The following options appear when you are prompted for the opposite corner:

Options: **Area**, **Dimensions**, **Rotation** or  
Specify opposite corner» (Enter an option.)

**Area** specifies the size of the rectangle based on its area and the length of one side.  
Specify total area» (Specify the area of the rectangle.)  
Options: **Horizontal**, or **Vertical**  
Specify known dimension» (Enter H or V.)  
Specify horizontal dimension» (Enter a length.)

**Dimensions** specifies the length and width of the rectangle, measured from the start corner.  
Specify horizontal dimension» (Enter a length.)  
Specify vertical dimension» (Enter a length.)

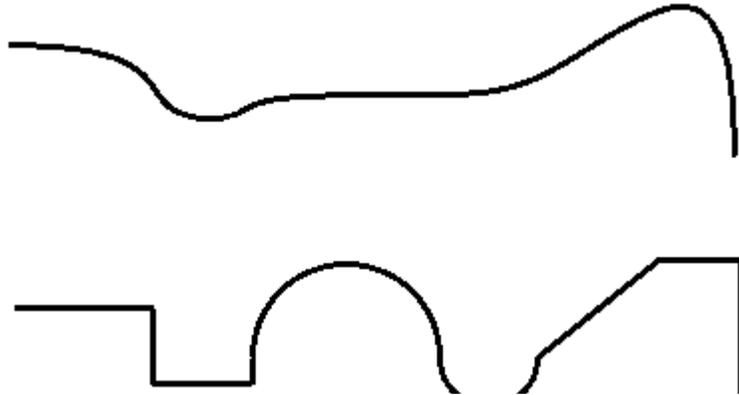
**Rotation** specifies the angle of the rectangle, measured counterclockwise from the positive x axis. The rectangle rotates about the start corner.  
Options: Pick points or  
Specify rotation» (Specify an angle.)  
Options: **Area**, **Dimensions**, **Rotation** or  
Specify opposite corner» (Pick the other corner, or enter an option.)



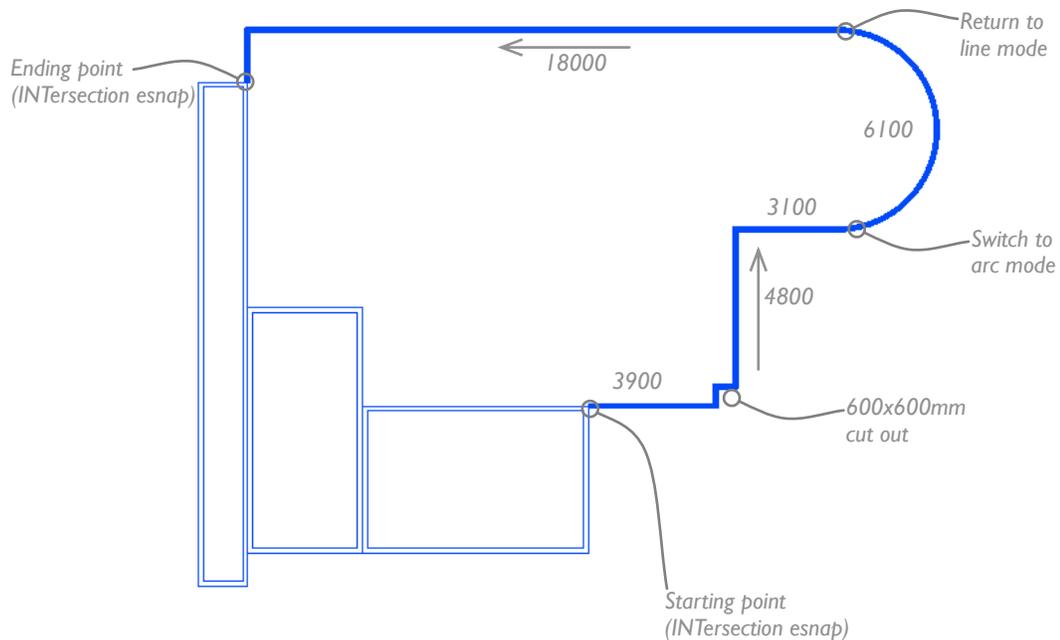
**TIP** To change the thickness and elevation, use the **Modify** command. To change the line width, use the **EditPolyline** command. The size of chamfers and radius of fillets cannot be changed, except by erasing them and then reapplying the **Chamfer** and **Fillet** commands, respectively.

In addition to Rectangle, iCADMac has a command specifically for drawing any shape of polyline. (It also has the special Polygon and Ring commands for creating regular polygons and filled circles from polylines.)

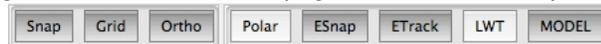
The Polyline command is good for drawing complex entities, such as the ones illustrated below: a splined polyline, and below it a complex polyline made of lines and arcs.



You use the Polyline command to draw the remaining external walls, shown in black by the figure below:



1. Ensure the drawing aids are turned on: snap, grid, ortho, and esnap.



2. Start the **Polyline** command using one of the methods listed above.  
:polyline
3. Start at the conference room rectangle's upper right corner. The Intersection snap ensures a precise pick.  
Options: [Enter to continue from last point](#) or  
Specify start point» (Pick the upper right corner of the conference room.)
4. Move the cursor to the right, and then enter 3900. This draws the first polyline segment 3900mm long.  
Options: [Arc](#), [Halfwidth](#), [Length](#), [Undo](#), [Width](#), [Enter to exit](#) or

Specify next vertex» 3900

5. Continue drawing the polyline using the dimensions shown in the figure.

Options: [Arc](#), [Close](#), [Halfwidth](#), [Length](#), [Undo](#), [Width](#), [Enter to exit](#) or  
Specify next vertex» 600

Options: [Arc](#), [Close](#), [Halfwidth](#), [Length](#), [Undo](#), [Width](#), [Enter to exit](#) or  
Specify next vertex» 600

Options: [Arc](#), [Close](#), [Halfwidth](#), [Length](#), [Undo](#), [Width](#), [Enter to exit](#) or  
Specify next vertex» 4800

Options: [Arc](#), [Close](#), [Halfwidth](#), [Length](#), [Undo](#), [Width](#), [Enter to exit](#) or  
Specify next vertex» 3100

6. To draw the arc around the executive office, switch to arc drawing mode. To do so, enter **a** at the prompt, as follows:

Options: [Arc](#), [Close](#), [Halfwidth](#), [Length](#), [Undo](#), [Width](#), [Enter to exit](#) or  
Specify next vertex» a

7. There are many ways to define an arc. Here, you will define the end point of the arc:

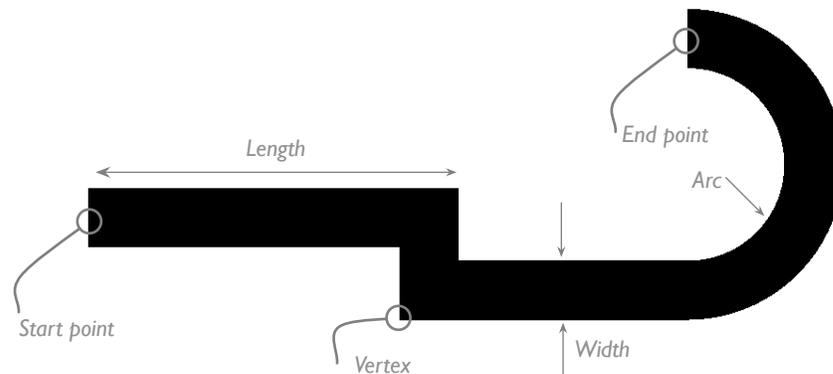
Options: [Angle](#), [CEnter](#), [Close](#), [Direction](#), [Halfwidth](#), [Line](#), [Radius](#), [Through point](#), [Undo](#), [Width](#), [Enter to exit](#) or  
Specify arc end point» 6100

8. With the arc in place, you can switch back to line drawing mode. Enter **l** at the prompt:

Options: [Angle](#), [CEnter](#), [Close](#), [Direction](#), [Halfwidth](#), [Line](#), [Radius](#), [Through point](#), [Undo](#), [Width](#), [Enter to exit](#) or  
Specify arc end point» l

#### Summary of Polyline Command Options

Polylines consist of the a number of elements. illustrated below:



The Polyline command has the following options for drawing polylines:

**Specify start/next vertex** — specifies the location of the polyline's next *vertex* (endpoint of a polyline segment).

**Arc** — draws polyline arcs using prompts similar to the Arc command; you can switch between line and arc mode.

**Close** — joins the last endpoint with the starting point.

**Halfwidth** — specifies the width of polylines by the distance from their center line to the outside edge.

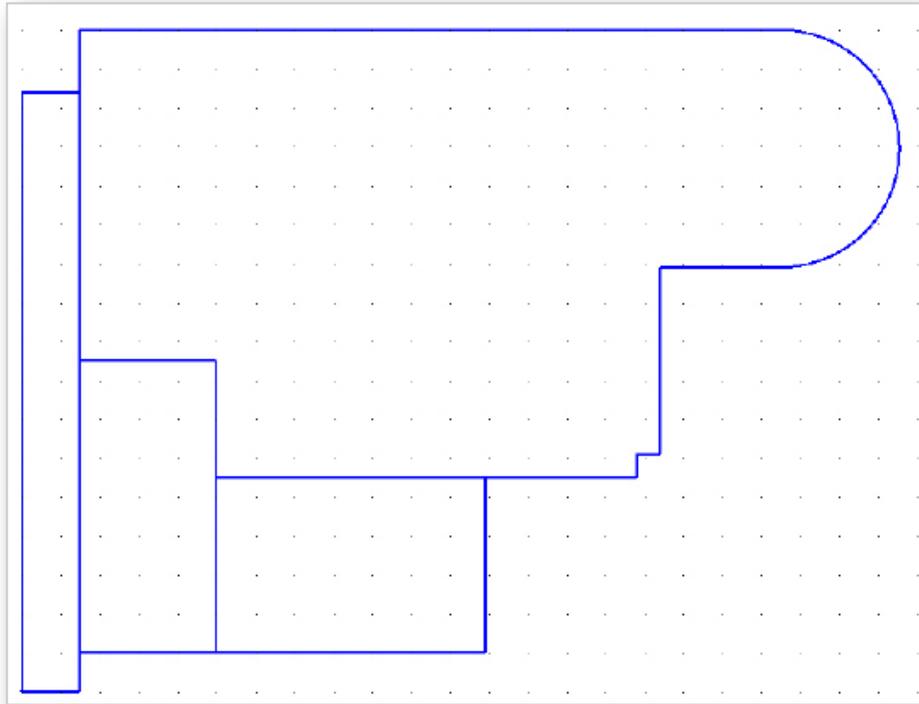
**Length** — specifies the length of the segment in the manner similar to direct distance entry.

**Undo** — undoes the last polyline drawing operation.

**Width** — specifies the width of polyline segments; with different starting and ending widths, iCADMac draws tapered polylines.

9. You are almost done! Draw the final two polyline segments:  
Options: *Arc*, *Close*, *Halfwidth*, *Length*, *Undo*, *Width*, *Enter to exit* or *Specify next vertex»* 18000  
  
Options: *Arc*, *Close*, *Halfwidth*, *Length*, *Undo*, *Width*, *Enter to exit* or *Specify next vertex»* (Move the cursor down to meet the corner of the hallway, and then use Intersection entity snap to complete the segment.)
10. Press **return** to exit the Polyline command:  
Options: *Arc*, *Close*, *Halfwidth*, *Length*, *Undo*, *Width*, *Enter to exit* or *Specify next vertex»* (Press return.)
11. Press **command+S** to save your work.

The drawing should now look like the figure illustrated below.



**TIP** You can cancel commands at any time by pressing **esc**. Sometimes, however, you may need to press **esc** two or even three times in commands that have large numbers of suboptions, such as the **EditPolyline** command.

## Drawing Internal Walls

The interior walls are drawn on a layer different from the external ones. In the following section, you change layers, and then use yet another kind of line command to draw the interior walls.

### Changing Layers Quickly

---

By placing entities on different layers, you ensure that they automatically take on the properties of the layers. Properties include things like color, line style, and visibility. When you drew lines on

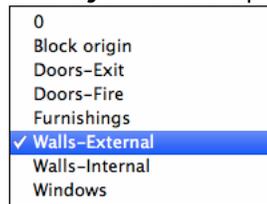
the Walls-Exterior layer, they were colored blue, automatically. When you make the Wall-Interior layer active, lines will be drawn light blue.

In the previous chapter, you used the Layer Manager dialog box to create new layers, and then to set Walls-Exterior as the active (or working) layer. You could use this dialog box to change the active layer, but I find the Layers droplist in the Properties palette more convenient. (If the Properties palette is not visible, press **command+1**.)

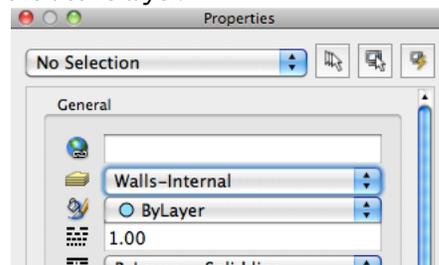


To make the Walls-Interior layer active, follow these steps:

1. In the Properties palette, click on the **Layer** name drop list. Notice the list of layer names.



2. Move the cursor down to **Walls-Interior**, and then click again. The layer drop list now displays Walls-Internal as the active layer.



**TIP** If you prefer to use the keyboard, then you might find the CLayer system variable to be a quicker way to change layers. Enter **CLayer**, and then the name of the layer to be made active, like this:

```
Command: clayer
Enter new value for CLAYER : walls-internal
```

## Drawing Walls with RichLines

So far you have drawn walls with lines, rectangles, and polylines. There is one more way to draw straight lines in iCADMac, the richline.

Richlines are made of one or more parallel lines. The space between the lines can be filled with color, and the ends can be capped. You can define many kinds of rich lines, and apply them with *styles*.

There are two primary drawbacks to multilines: they can only be straight lines, no arcs; they do not

clean up their intersections automatically, where pairs of lines overlap.



### Placing RichLines

**Command:** RichLine

**Aliases:** rl, ml, mline

**Menu Bar:** Draw | Multiline

There are two interior walls you draw with multilines. One defines the reception area; the other walls off the executive office. Let's do the executive office first.

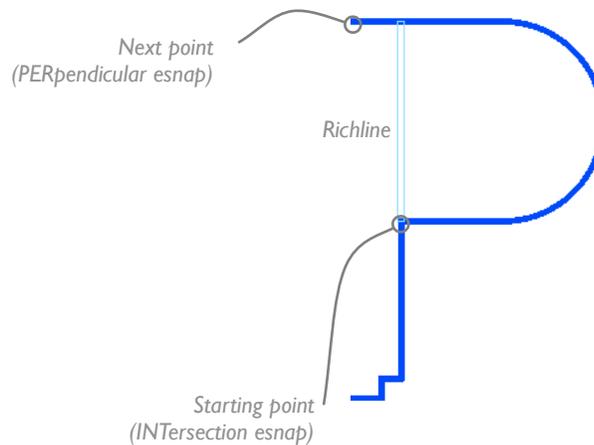
1. Start the **RichLine** command with one of the methods listed above.
2. At the prompt, pick the intersection indicated by the figure.

Command: richline

Active settings: Justification = Top, Scale = 1, Style = STANDARD

Options: **Justification**, **Scale**, **Style** or

Specify start point» (Pick point at intersection.)



3. Move the cursor up to the upper wall.
4. Enter “per” and then press **return**.

Specify next point» per

*Per* is the abbreviation for “perpendicular,” another one of iCADMac’ entity snaps. Entering it at the ‘Specify next point:’ prompt makes the esnap temporary; it is in effect for the next pick point only.

(This is different from using the Options dialog box to set Intersection esnap, which remains in effect until you turn it off.)

5. iCADMac asks “to,” meaning where the lines should be drawn to:  
to (Pick on the wall.)
6. Press **return** to exit the command.

Options: **Undo** or

Specify next point» (Press return.)

### Zooming In and Out

The multiline probably looks to you like any other line (or polyline) in the drawing — other than being light blue in color. In fact, it consists of two closely spaced lines, just 0.5mm apart.

You can see them by zooming into the drawing:

1. Position the cursor over the richline.
2. Roll the mouse's scroll wheel forward. Under the cursor, the drawing should become larger.



3. Keep rolling the wheel until you see two distinct lines, as illustrated below.

#### Summary of RichLine Command Options

Command: richline  
 Active settings: Justification = Top, Scale = 1, Style = STANDARD  
 Options: **Justification**, **Scale**, **Style** or  
 Specify start point» (Pick a point.)

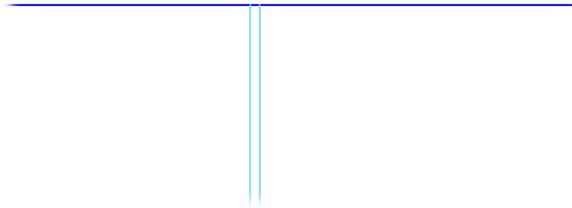
**Justification** — aligns the richlines to the right, left, or center of pick points. The options are named Zero (the default), Top, and Bottom, as illustrated below.



**Scale** enlarges or reduces the width of the richline by a scale factor. (The width of the richline is defined by its style.) Enter a number at the 'Enter RichLine scale' prompt:

- Smaller than 1 makes the richline narrower.
- 1 makes the richline the same width as defined by its style.
- Larger than 1 makes the richline wider.
- Negative numbers flip the richline about its centerline.

**Style** selects a predefined style for the richline. Styles are created and edited with the RichLineStyle command.



- To return to the original view, enter the Zoom Bounds command as an alias:

Command: z b

(To zoom out, you could roll the scroll wheel backwards, but it is faster to enter “z b.”)



### Defining MultiLine Styles

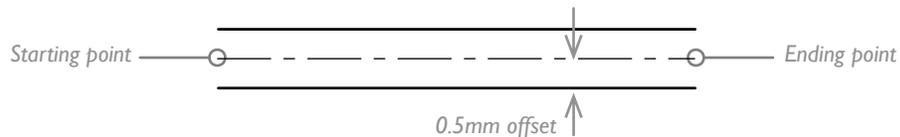
**Command:** RichLineStyle

**Aliases:** rls, rlstyle, rlinestyle, mlstyle

**Menu Bar:** Format | MultiLine Styles

Interior walls are often 100mm thick, so the current richline of 0.5mm is too thin. You change the width with the RichLineStyle command.

The default style consists of two parallel lines, each 0.5 mm from an imaginary center line. (The center line path is defined by your pick points in the drawing.)



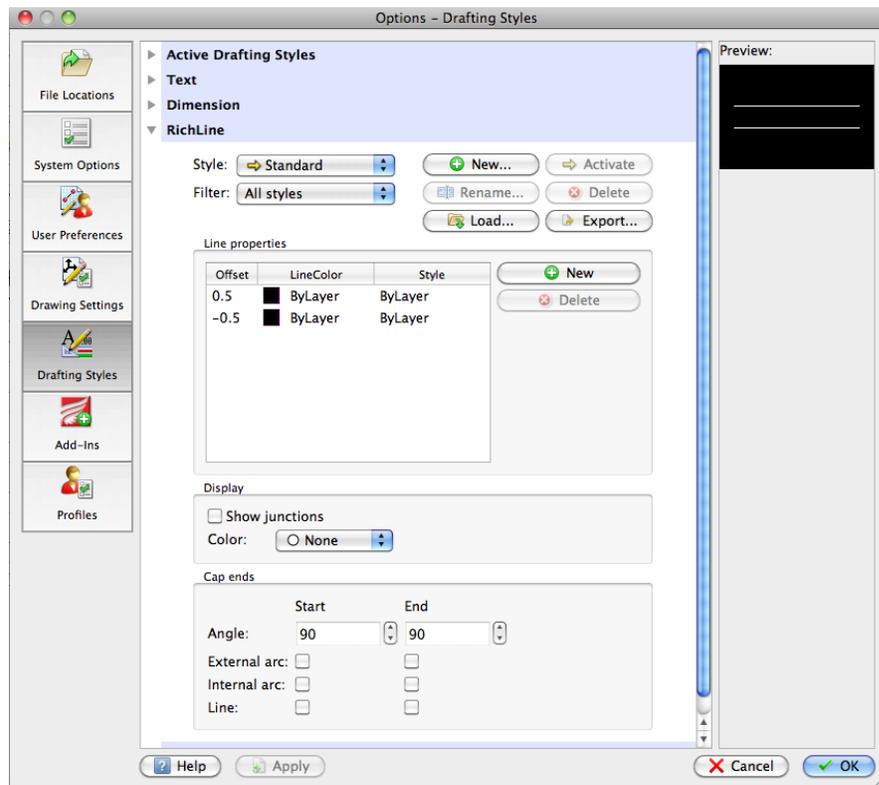
But richlines can consist of as many as 16 parallel lines, each located a specified distance from the center. In addition, each can have its own color and linestyle. Richlines can also have the following optional properties:

**Display Color** — a solid fill color that floods the entire richline.

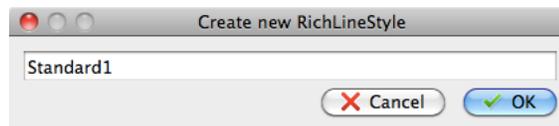
**Junctions** — a diagonal line at every intersection.

**Cap ends** — a line or arc at either end; the cap can be at any angle.

- Start the **RichLineStyle** command using one of the methods listed above. Notice that the Options dialog box opens to the Drafting Styles | RichLine section, featuring a preview window to the right.



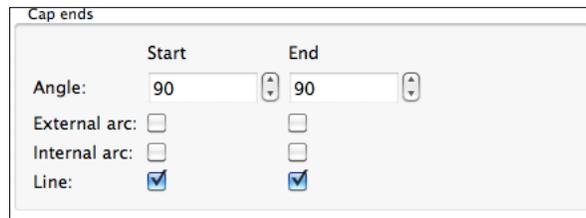
- To create a new style, click **New...** **New**. Notice the Create New RichLineStyle dialog box.



- Change the name from STANDARD1 to "Walls-Interior," and then click **OK**.
- With the new style named, you can now define it. It consists of two lines spaced 100mm apart — or 500mm from the centerline.
  - In the Line Properties section, under Offset, click 0.5, and then change it to **100**.

Offset	LineColor	Style
100	ByLayer	ByLayer
-100	ByLayer	ByLayer

- Change the second line from -0.5 to **-100**.
- Add end caps to the richline. In the Cap Ends section, check both boxes next to **Line**.



- That's all you need to do for this richline style. Click **OK** to exit the dialog box.



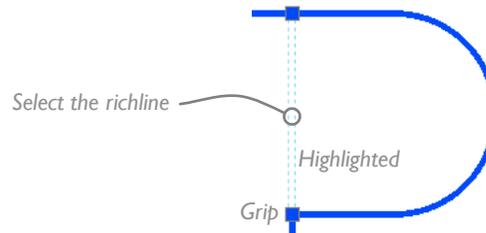
## Erasing Entities

**Command:** Delete  
**Aliases:** e, erase, del  
**Menu Bar:** Edit | Erase  
 Modify | Erase  
**Tool Matrix:** Modify | Delete  
**Shortcut:** fn+delete

A flaw in the design of richlines means that you cannot simply apply this new style to the existing lines, as you might to other entities that accept styles, such as text and dimensions. This means that you need to (a) erase the existing richline, and then (b) redraw it with the new style.

You erase entities as follows:

- Select the richline. Notice that it is highlighted, as illustrated below.



**Highlighting** indicates the selected entities.

**Grips** allow you to edit entities directly, as described in a later chapter.

- Press the **fn+delete** key. The entity disappears from the drawing.



## Oops! Unerasing Entities

**Command:** U  
**Menu Bar:** Edit | Undo  
**Tool Matrix:** Standard | U  
**Shortcut:** command+Z

When you erase objects by accident, you can return them to the drawing with the U command, as follows:

Command: u

**TIP** To undo an undo, use the **Redo** command, or press **command+shift+Z**.

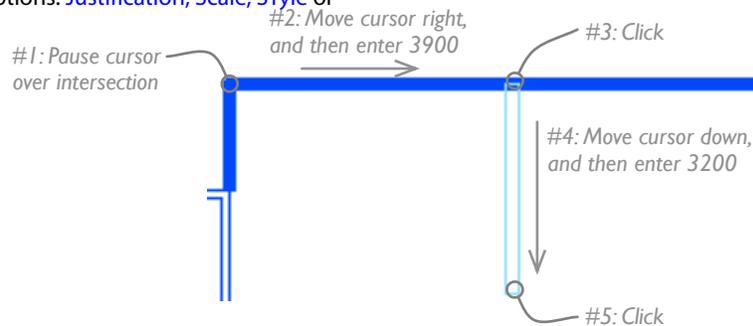
## Replacing RichLines

With the unwanted richline removed, you now redraw the wall defining the executive office with the newly-defined richline style.

1. Start the **RichLine** command.  
Command: richline
2. Notice that the style is set to “Standard.” Enter **st** to change the style, as follows:  
Active settings: Justification = Top, Scale = 1, Style = STANDARD  
Options: **Justification, Scale, STyle** or  
Specify start point» st
3. Enter “walls-interior” for the new style name:  
Options: **? to list** or  
Specify name» walls-interior
4. Draw the richline as before, from the intersection to the perpendicular of the upper wall.  
Notice that the lines are 100mm apart.  
Specify start point» (Pick point at intersection.)  
Specify next point» per  
to (Pick on the wall.)  
Options: **Undo** or  
Specify next point» (Press return.)
5. Add the wall that defines the reception area. (If style is not set to “walls-interior,” you may need to enter the **STyle** option again.

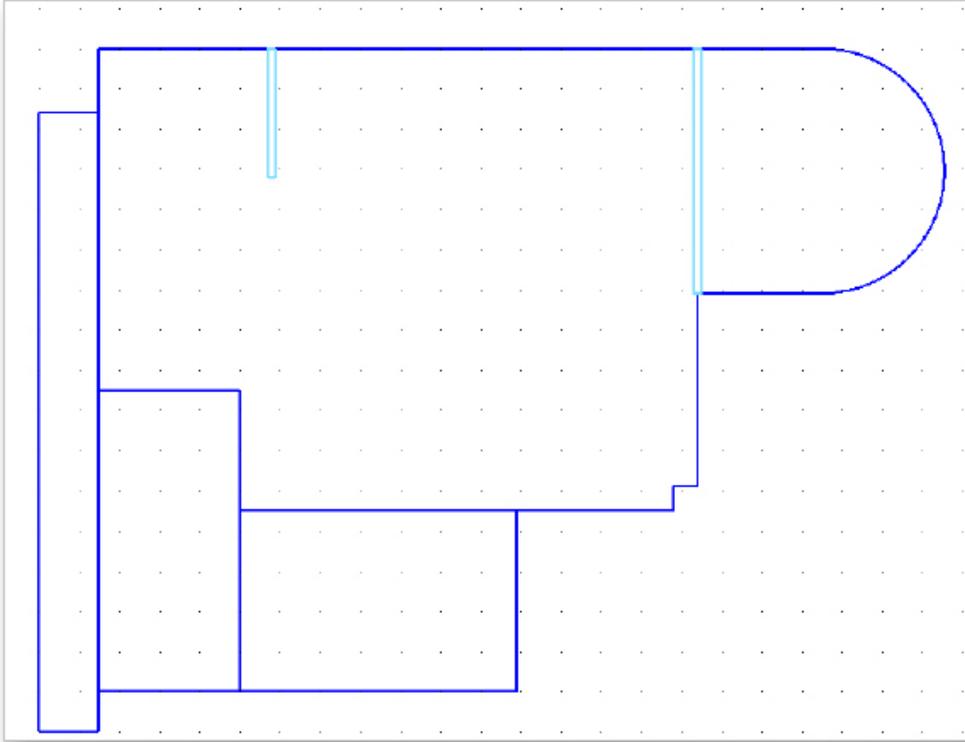
To repeat the previous command, just press **return**:

Command: (Press return.)  
Active settings: Justification = Top, Scale = 1, Style = WALLS-INTERIOR  
Options: **Justification, Scale, STyle** or



6. Use direct distance entry to locate the 3900mm-offset, as follows:  
Specify start point» (Position cursor at intersection #1; do not click!)  
(Move cursor right.) 3900  
(And then click to start the richline.)
  - a. Position the cursor over intersection #1.

- b. Move the cursor to the right, and then enter the offset distance, **3900** (#2 in the figure above).
  - c. Click to position the starting point of the richline (at point #3).
  - d. Move the cursor down, and then enter the length of the wall, **3200** (#4).
  - e. Click to position the endpoint of the richline (#5).
7. Press **return** to end the command. The result should look like the figure below.
  8. Press **command+S** to save your work.



## Thickening Walls

You have drawn wall lines four ways now, with the Line, Rectangle, Polyline, and RichLine commands. Of these, only the rich lines have the correct wall thickness. In this section, you learn how to thicken walls made of lines and polylines.

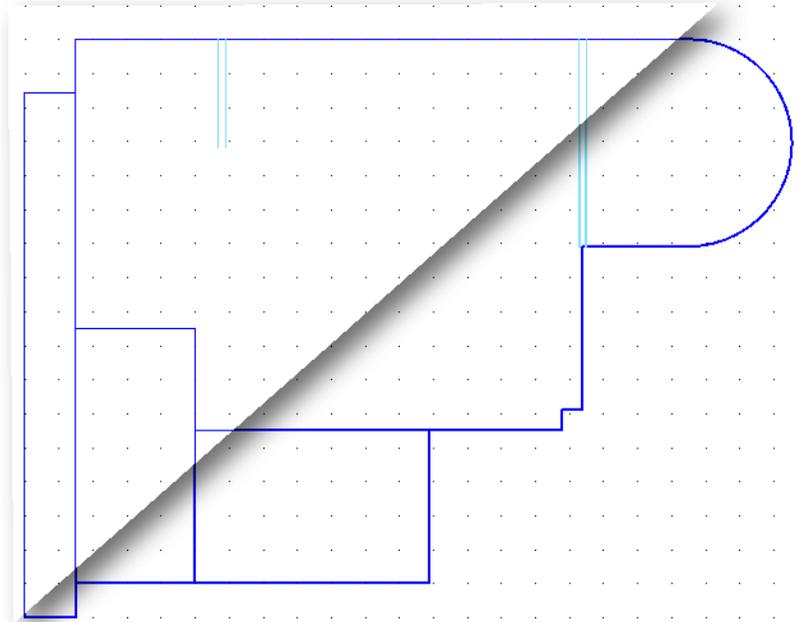
### Changing the Line Weight

**Command:** LineWeight  
**Aliases:** lw, lweight  
**Menu Bar:** Format | Line Weight  
**Status bar:** LWT  
**Properties:** LineWeight

Line weights change the visual width of lines and all other entities. They don't actually thicken entities. I find it easier to view drawings, however, when line weights are turned on, so I always keep them on.

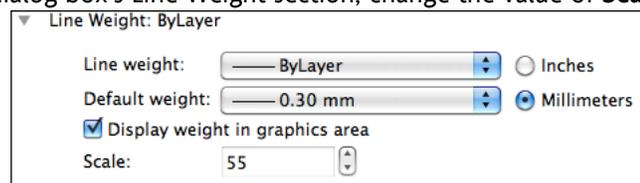
To turn on line weights, click the **LWT** button on the status bar. Entities should thicken, as illustrated

by the composite image:



Lines can look too thin or too thick. You can change the relative thickness, as follows:

1. Right-click **LWT** on the status bar, and then choose **Settings** from the shortcut menu.
2. In the Options dialog box's Line Weight section, change the value of **Scale**.

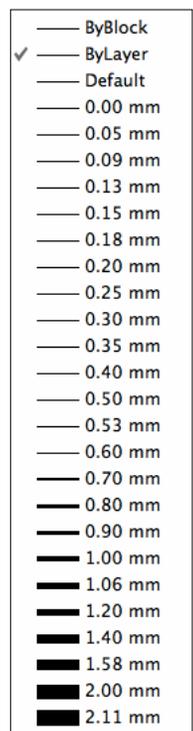


3. Click **Apply** to see if the weights are now appropriately scaled.
4. Click **OK** to exit the dialog box.

**TIPS** You can assign line weights individually to layers and entities. For layers, use the LineWeight column in the Layer Manager dialog box (Layer command).

For entities, use the LineWeight droplist in the Properties palette, as follows:

1. Select the entities.
2. From the LineWeight droplist, choose a line weight value.





## Widening Polylines

**Command:** EditPolyline

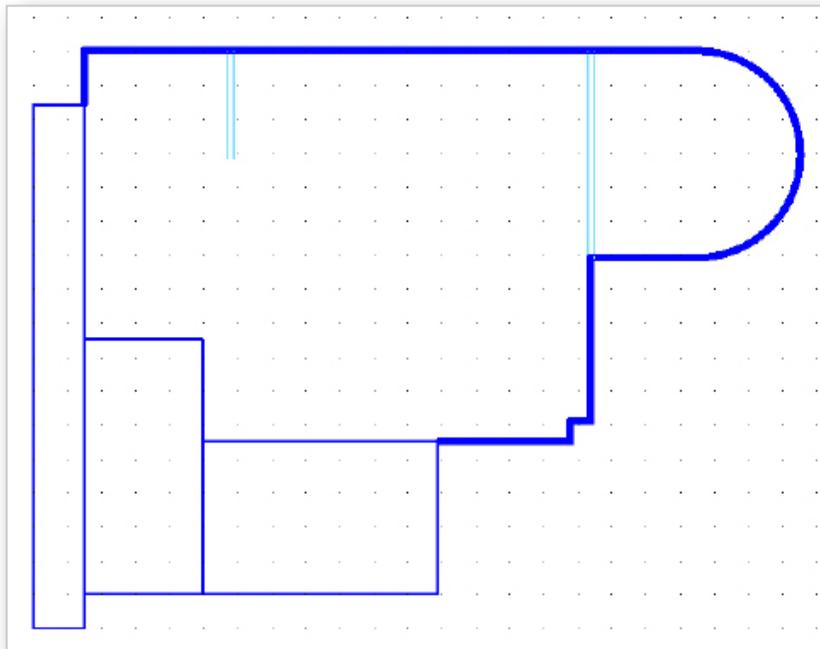
**Aliases:** pedit, edpl, edpline, pe, polyedit

**Menu Bar:** Modify | Entity | Polyline

**Tool Matrix:** Modify | EditPolyline

Unlike line weights, the widths of polylines are real. You can assign the width while drawing them with the Polyline command, or else change the width afterwards with the EditPolyline command. Follow these steps:

1. Start the **EditPolyline** command using one of the methods listed above.  
Command: editpolyline
2. Choose the long polyline.  
Options: **Multiple** or  
Specify polyline» (Pick the polyline.)
3. Enter **w** to choose the Width option.  
Options: **Close**, **Decurve**, **Edit vertex**, **Fit**, **Join**, **Linegen**, **Spline**, **Undo**, **Width**, or **eXit**  
Specify option» w
4. Specify a new width. Outside walls are typically 150mm, so enter **150**:  
Specify new width» 150
5. Press **return** to end the command. Notice that the polyline instantly thickens.  
Specify option» (Press return.)
6. Press **command+S** to save the drawing.



## Summary of EditPolyline Command Options

The EditPolyline command has the most extensive set of options, sub-options and sub-sub-options of all commands in iCADMac. Here is an overview of the most important ones:

:editpolyline  
Options: **Multiple** or  
Specify polyline» (Pick a polyline.)

**Multiple** — works with more than one polyline at a time.

**Specify polyline** — selects the polyline to be edited; if you select a line or an arc, iCADMac offers to turn it into a polyline.

Options: **Close**, **Decurve**, **Edit vertex**, **Fit**, **Join**, **Linegen**, **Spline**, **Undo**, **Width**, or **eXit**  
Specify option»

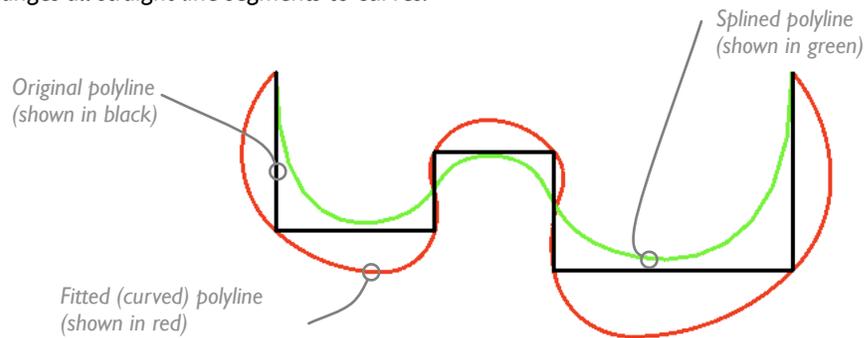
**Close** — closes the polyline: a segment is drawn between the last and first endpoints.

**Decurve** — removes curve information from fitted and splined polylines.

**Edit vertex** — edits the vertices; adds, changes, and removes vertices; and changes the widths in individual segments. The edit vertex options are the following:

Options: **Break**, **Insert**, **Move**, **Next**, **Previous**, **REgen**, **Straighten**, **Tangent**, **Width**, or **eXit**

**Fit** — changes all straight line segments to curves.



**Join** — joins other polylines and lines to the current polyline; other entities must be touching.

**Linegen** — (linestyle generation) determines how linestyles are treated at vertices, continuous or dashed.

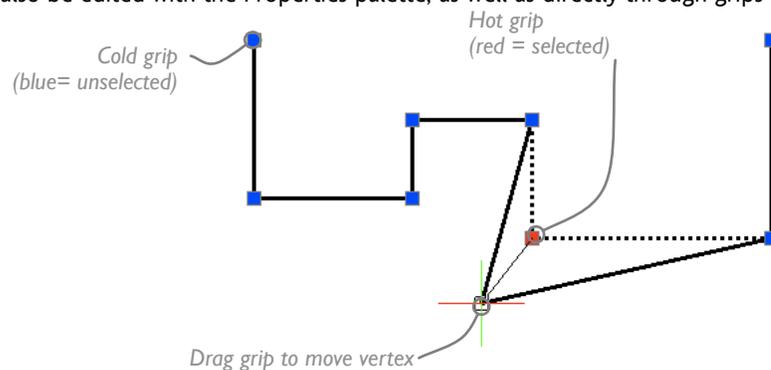
**Spline** — changes the polyline into the equivalent of a spline curve (figure above).

**Undo** — undoes the last editing changes.

**Width** — changes the width of the entire polyline; constant width only, no tapers.

**eXit** — exits the command.

Polylines can also be edited with the Properties palette, as well as directly through grips editing.



## Offsetting Lines

**Command:** Offset

**Aliases:** o, qoffset

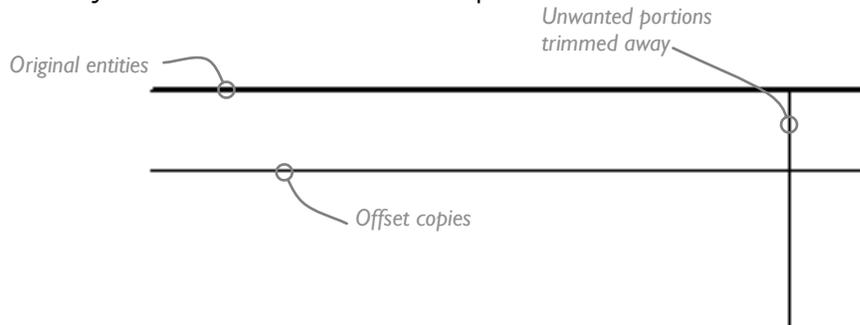
**Menu Bar:** Modify | Offset

**Tool Matrix:** Modify | Offset

While the EditPolyline command nicely thickens polylines, it makes them solid filled; they are not hollow, like the rich lines. You can create an effect similar to rich lines by using the Offset and Trim commands. Here's what the two commands do:

**Offset** creates parallel entities.

**Trim** cuts away the excess where entities overlap.



In this tutorial, you apply the Offset command to the polyline drawn with the Rectangle command and to the rectangle drawn with the Line command — because the Offset command works slightly differently with polylines and lines.

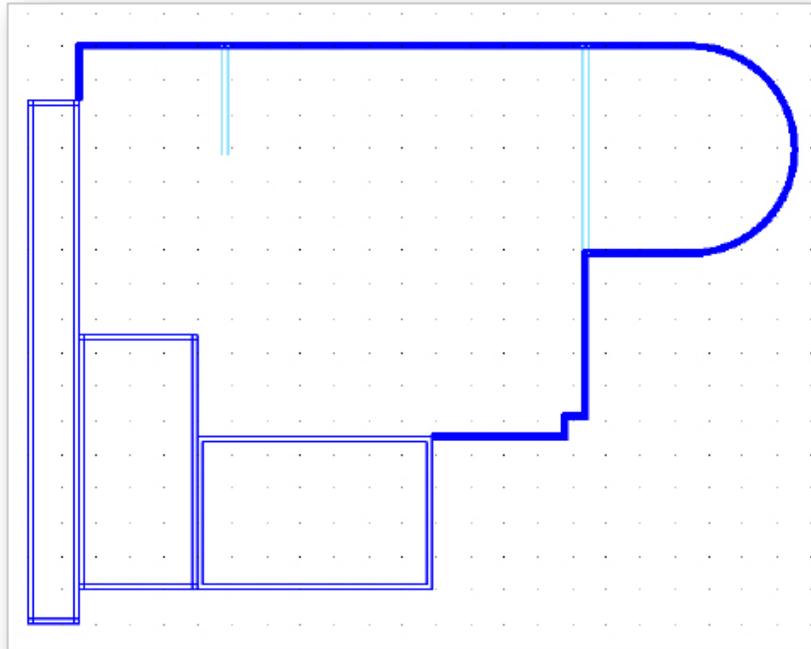
1. Start the **Offset** command using one of the methods listed above.  
Command: offset
2. Specify the offset distance, 150mm.  
Active settings: Delete source=No Layer=Source OFFSETGAPTYPE=0  
Default: 1  
Options: [Delete](#), [Distances](#), [destination Layer](#), [Through point](#) or  
Specify distance» 150
3. Select the rectangle representing the conference room, which was drawn with the Polyline command.  
Default: Exit  
Options: [Exit](#), [Undo](#) or  
Specify source entity» (Pick the polyline.)
4. Indicate on which side you wish the parallel copy made; for this tutorial, click inside the conference room.  
Default: Exit  
Options: [Both sides](#), [Exit](#), [Multiple](#), [Undo](#) or  
Specify side for destination» (Pick inside the polyline.)

Notice that iCADMac draws parallel polylines instantly.

5. Now carry on to make offsets of the storage room and hallway, which were drawn with the Line command. This will be more work, because you have to select all eight lines making up the two rooms, one at a time.  
Specify source entity» (Select a line.)

Specify side for destination» (Pick inside the rectangle.)

... and repeat seven more times until the drawing looks like this:



### Trimming Lines

**Command:** Trim

**Alias:** tr

**Menu Bar:** Modify | Trim

**Tool Matrix:** Modify | Trim

The drawback to applying offsets to lines is overlapping lines. I have enlarged part of the drawing to illustrate the problem more clearly:

#### Summary of Offset Command Options

Command: offset

Active settings: Delete source=No Layer=Source OFFSETGAPTYPE=0

Default: 1

Options: **Delete**, **Distances**, **destination Layer**, **Through point** or

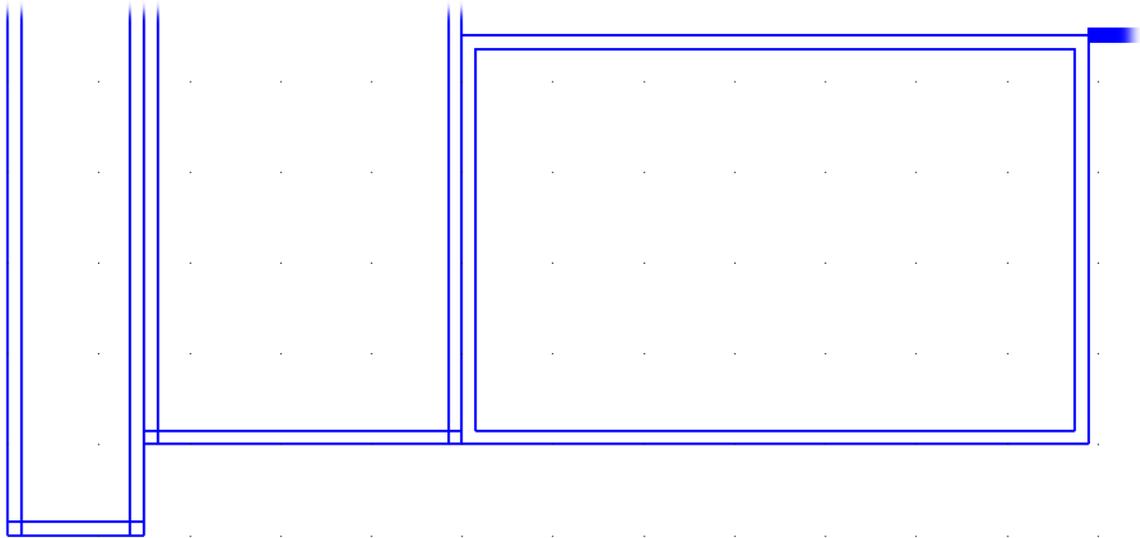
Specify distance» (Enter an option.)

**Delete** — deletes the original after making the offset copy.

**Distances** — specifies multiple offsets.

**destination Layer** — determines if offset copies are placed on the active (current) layer or remain on the source's layer.

**Through point** — places the offset copies at the cursor's pick point.

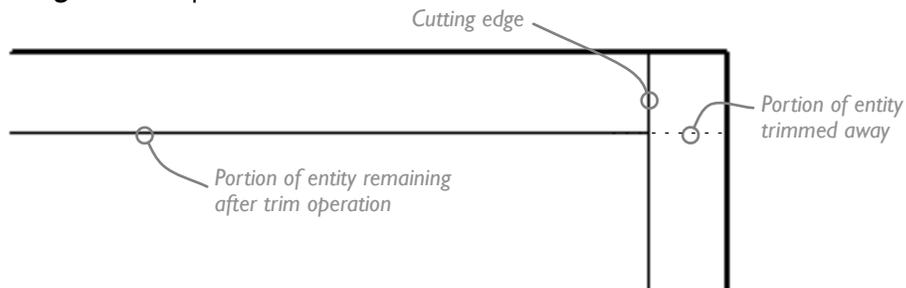


The rectangle to the right has clean intersections, because it was drawn with the polyline; those to the left need cleaning up, because they were drawn with lines.

The solution to the mess is to trim the overlaps, cleaning up the drawing with the Trim command. There are two types of entities you select during the command, edges that cut and ones that are trimmed:

**Cutting edges** are entities that determine where other entities are trimmed.

**Trimmed edges** are the portions of entities to be removed.



Entities can be both cutting edges and trimmed.

1. Start the Trim command.

Command: trim

2. Press **return** to select all entities in the drawing as cutting edges. This is a safe move, because cutting edges are not modified by this command.

Active settings: Projection=CCS, Edge=None

Specify cutting edges ...

Options: [Enter to specify all entities](#) or

Specify entities» (Press return.)

Note that richline entities are not supported by this command.

3. One by one, pick the portions to be trimmed. *Important!* Pick the part of the entity you want removed. If you make an error, use the Undo option.

Options: [Crossing](#), [croSsline](#), [Project](#), [Edge](#), [eRase](#), [Undo](#) or

Specify segments to remove» (Select portion to remove.)

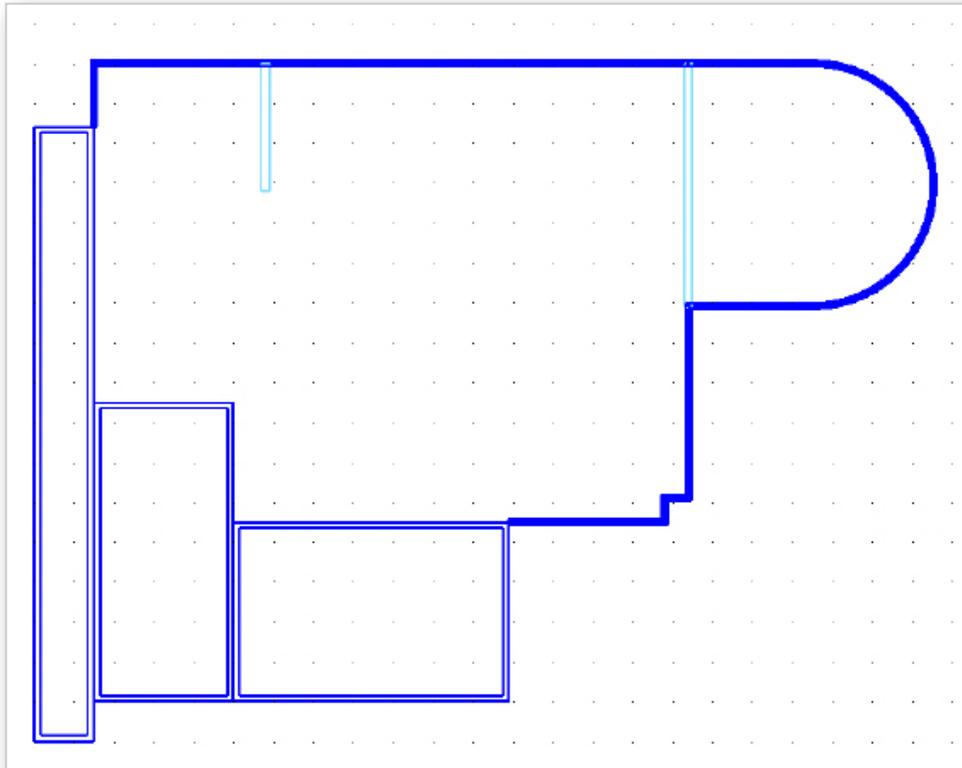
**TIP** Because it can be difficult to pick small entities accurately, employ these tricks:

- Zoom into the area where you make the trims. (Roll the scroll wheel.)
- Turn off **Snap** on the status bar to unrestricted cursor movement.

4. Continue with the other lines that need trimming. When done, press **return** to exit the command.

Specify segments to remove» (Press return.)

5. Press **command+S** to save the drawing.



## Summary

Congratulations! You've drafted your very first drawing using iCADMac for Mac.

In this chapter, you learned how to represent walls by lines, polylines, and richlines — thereby learning the pros and cons of each type of entity. You also learned how to make them look more like walls by giving them styles, weight, width, and parallel lines.

In the next chapter, you print the drawing on paper.





## Chapter 6

# Printing Drawings

### In This Chapter

- Printing drawings
- Previewing drawings before printing them
- Reviewing detailed print options
- Creating page layouts and print styles

**The most common** result of a CAD drawing is that it is ultimately printed. CAD drawings can be printed on paper or printed electronically. The Print command sends the drawing to printers, plotters (oversize printers), and files. It can print a drawing “quickly” or with a great deal of control. In this chapter, you learn how to quick print, preview, print in detail, and export drawings by a variety of methods.



### Printing the Office Drawing

**Command:** Print

**Alias:** plot

**Menu Bar:** File | Print

**Tool Matrix:** Standard | Print

**Shortcut:** command+P

iCADMac provides a range of ways to specify how drawings should be printed, from the very general to the very specific. This chapter discusses all of them:

**Quick print** prints the drawing using the default settings. This approach is suited to draft printouts, because you don't fuss over settings.

**Presets** lets you choose among all settings available in the Print dialog box. This approach controls how *any* drawing is plotted.

**Page layouts** memorize print settings for individual drawings, and even for each model and layout sheet. iCADMac can import from page layouts stored in other DWG, DXF, and DWF files.

**Plot styles** specify the plot settings for every layer and entity in the drawing. This approach provides the greatest control of all.

The tutorials in this chapter require that your computer be attached to a printer. If necessary, open the *office layout.dwg* file you completed in the last chapter.

## Key Terms in This Chapter

**Default printer** — specifies the printer used by iCADMac when no other printer is selected.

**Dither** — simulates more colors through combinations of colors.

**Grayscale** — prints colors as shades of gray.

**Page layout** — refers to preset print settings.

**Plot** — refers to an older term for printing drawings on paper.

**Plotter** — refers to an old style of CAD printer that used pens to draw lines.

**Preview** — shows before printing how the drawing will fit the paper.

**Print** — prints drawings on paper.

**Print range** — determines which part of the drawing is printed.

**Print scale** — specifies the size of the drawing relative to the paper.

**Print style** — defines how entities are printed.

**Print to file** — sends the drawing to a file instead of a printer.

**Quick print** — prints drawings scaled to fit the paper using the default printer.

**Screening** — prints lines with less ink to make them look faded.

**Virtual pen** — simulates pens in non-pen printers.

## Abbreviation

**PDF** — portable document format, used by Adobe Acrobat software.

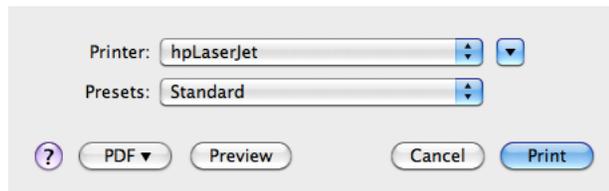
## Commands

<b>Command</b>	<b>Shortcuts</b>	<b>Menu Selection</b>
PageLayout	pagesetup	File   Page Layout Manager
Preview	pre	File   Print Preview
Print	command+P, plot	File   Print
PrintStyle	plotstyle	Format   Print Style

## Draft Prints

In this first printing tutorial, you produce a *draft* (or quick) printout of the office layout drawing.

1. Start the **Print** command using one of the methods listed above. Notice the simple dialog box.



2. Choose the printer from the **Printer** droplist; you can select only from those that are set up on your Macintosh.

(The other options perform the following functions:

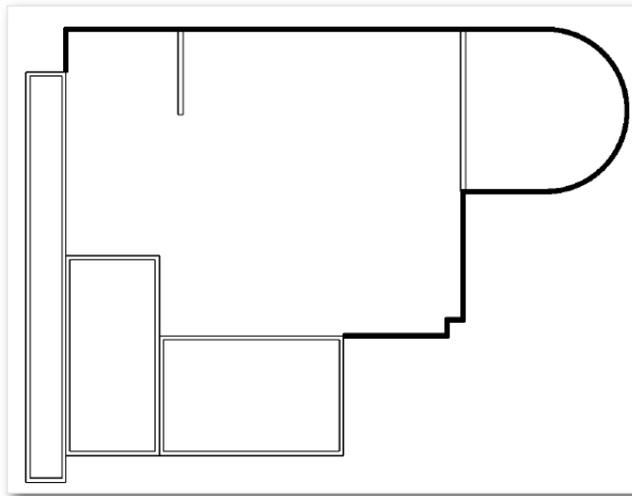
**Presets** droplist selects previously saved printing settings, discussed below.

**PDF** droplist outputs the drawing as a PDF file to a specific location, described later in this chapter.

**Preview** button displays the drawing in the OS X Preview application, described later.

**Cancel** button returns you to the drawing without printing.)

3. Click **Print**. Within a few moments, the drawing should emerge from the printer.



If the printout does not appear as you expect, then you can adjust printing options manually, as described next. Once you establish the settings to your satisfaction, you can save them, and then recall them for future printing jobs through the Presets droplist.

## Presets — Printing the Office Drawing Manually

By clicking the  button, you can set the printout to meet your needs. For instance, you can change the printer's settings, specify a scale factor, and select a drawing view to print.

In this tutorial, you change the following options:

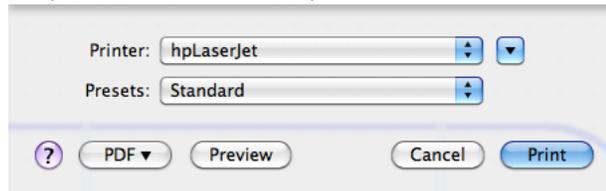
**Printer/Plotter Name** selects the printer from the Name droplist.

**Print Scale** ensures the entire drawing will be printed, and that it isn't too large or too small.

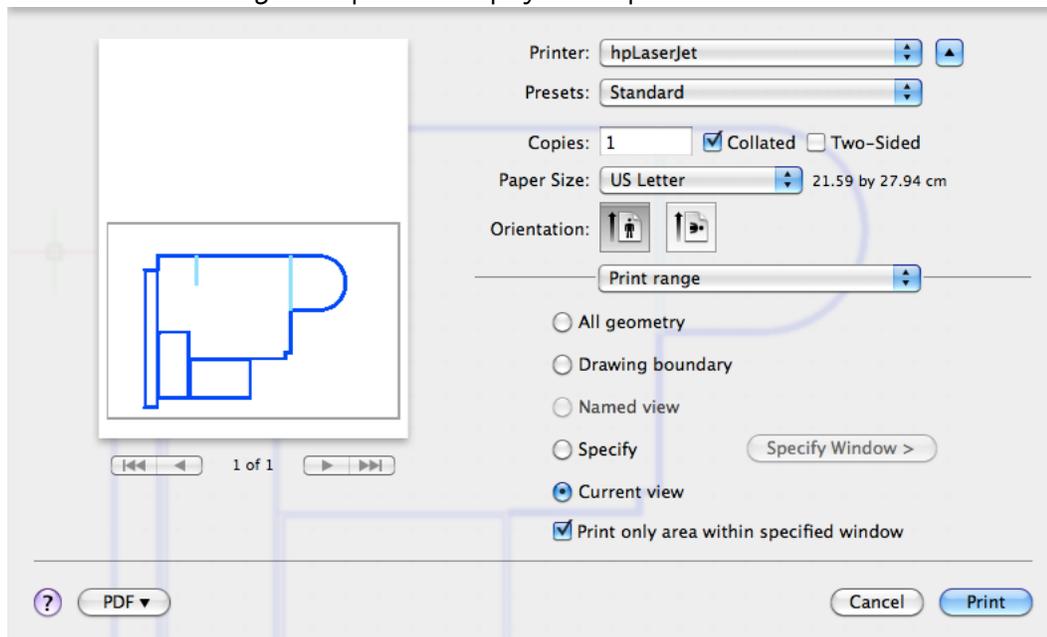
**Print Range** specifies the area of the drawing to print.

Follow these steps:

1. Start the **Print** command.
2. Click the  button (across from "Printer.")



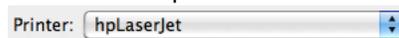
Notice that the dialog box expands to display more options.



### Printer Options

To change the printer or its settings, follow these steps:

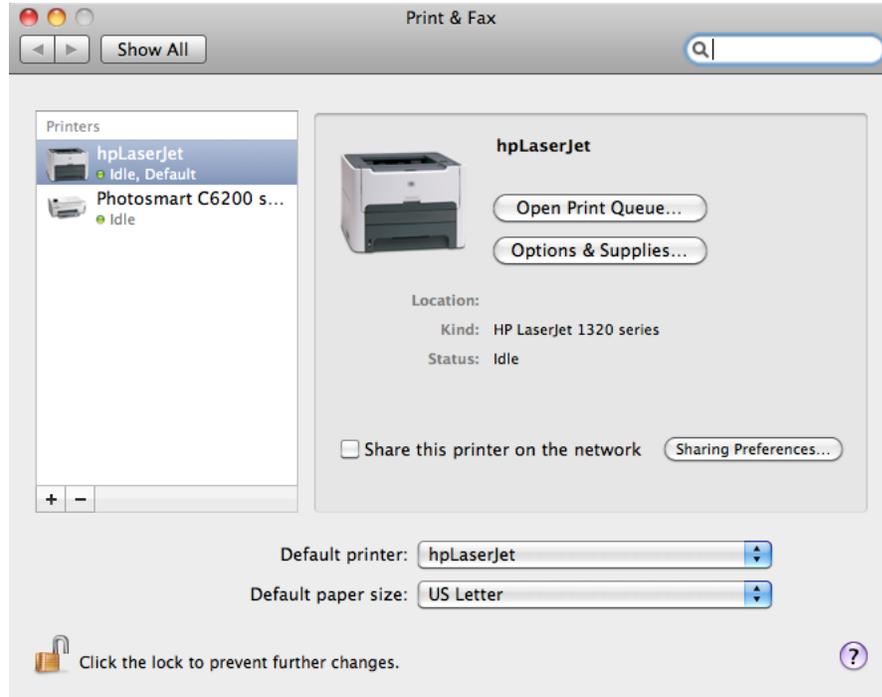
1. In the Printer section, click the **Name** droplist.



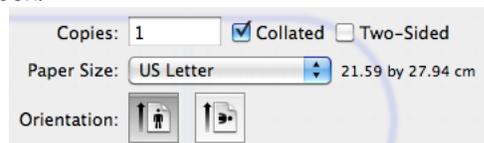
Notice the list of printer and file names.



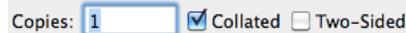
- To change the printer's settings, click **Print & Fax Properties**. Choose a printer, and then click the **Options and Supplies** button. The properties available depend on the printer model.



- Make changes as necessary, and then click the  close button. You are returned to the iCADMac' Print dialog box.



- Copies** specifies the number of copies to print; maximum is 9,999 copies.



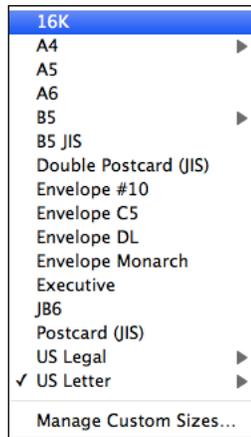
The **Collated** option applies only when printing sets of drawings. Instead of printing 10 copies of page 1 followed by 10 of page 2, the Collated option prints pages 1 and 2, followed by 1 and 2 again.

The **Two-Sided** option applies only to duplex printers, ones that can print on both sides of the paper.

- The **Paper Size** droplist lets you choose different sizes of paper on which to print the drawing.



Shown below are the standard sizes supported by HP's ubiquitous LaserJet laser printer.



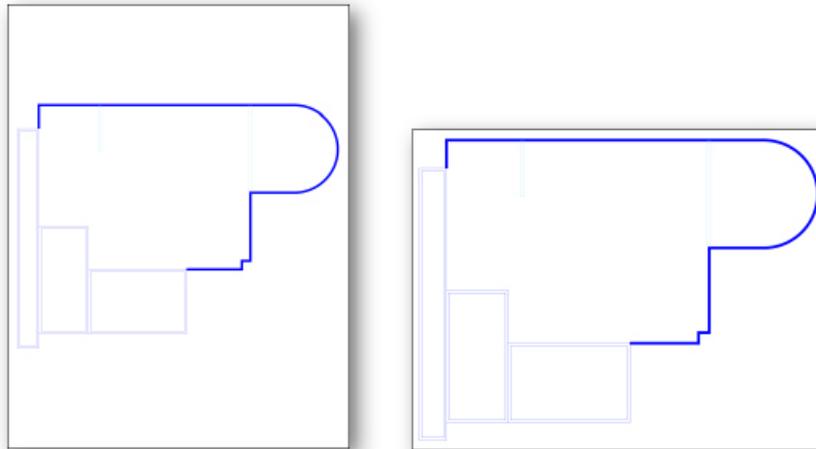
6. The **Orientation** section offers these options:



From left to right, these are:

**Portrait** orients the paper vertically; commonly used with A and A4 size paper. See figure below.

**Landscape** rotates the paper horizontally by 90 degrees; commonly used with all other sizes.



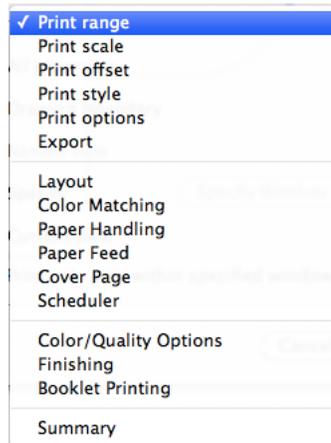
Left: Paper in portrait mode...

Right: ... and in landscape orientation.

7. Additional options are “hidden” by the unnamed droplist that currently shows “Print range.”



Click the droplist to see all groups of printing options, which we will cover in this chapter.



## Print Range Options

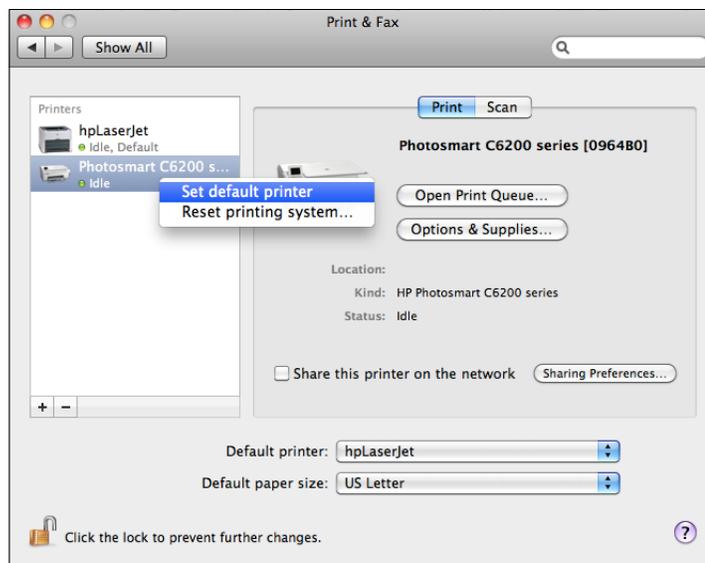
You can print drawings the way they look on the screen at the time you invoke the Print command, or you can print only a portion. These options affect the size of the printed drawing, when scale is set to Fit.

### Selecting the Default Printer in OS X

iCADMac' Quick Print mode uses the printer designated as the *Default Printer* in OS X. The default printer is the one that is used by iCADMac and other applications for printing when no other printer is specified.

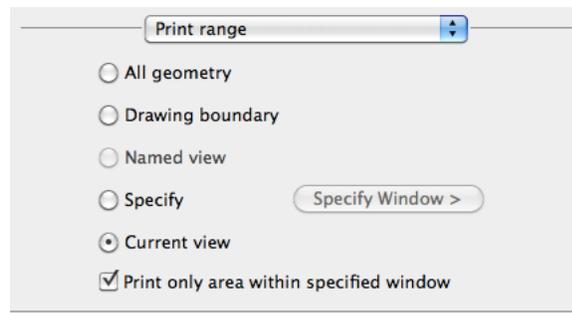
You change the default printer through the OS X Settings application. To do so, follow these steps:

1. On the OS X dock, click the **System Preferences** icon. (Alternatively, click the Apple logo on the menu bar, and then choose System Preferences.)
2. Choose **Print and Fax**. Notice that the default printer is tagged with the "Default" label.
3. In the Printers list box, right-click the printer you wish to make the default.
4. From the shortcut menu, choose **Set Default Printer**.



5. Close the dialog box.

When you next use Quick Print mode, iCADMac will list this newly-selected printer as the default.

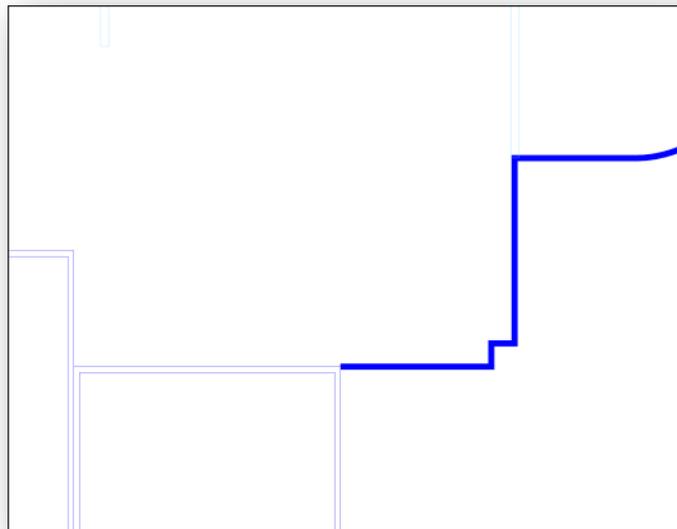


**All Geometry** prints all visible entities, no matter the zoom level; technically, iCADMac creates an invisible bounding rectangle that encompasses all non-frozen entities.

**Drawing Boundary** prints the area of the drawing defined by the DrawingBounds command. (When you select this option, the Print Only Area Within Specified Window option becomes available.)

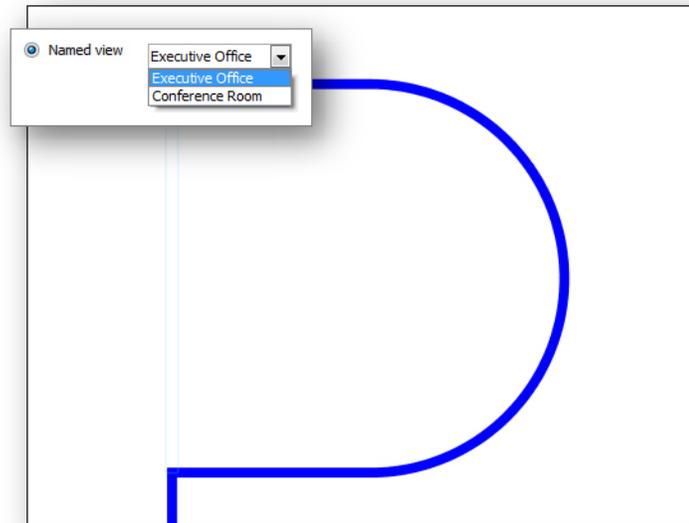
**Named View** prints the area of the drawing specified by a named view; this option is available only when the drawing contains views named by the Views command.

**Specify** prints the area of the drawing defined by a rectangular window. This option is good for printing details or portions of drawings, as illustrated below.



**Current View** prints the area of the drawing currently visible; this option is affected by the current zoom level and pan position.

**Print Only Area Within Specified Window** cuts off entities located outside of the area defined by the Drawing Boundary, Named View, Specify window, and Current View options.



### How to Create and Print a Named View

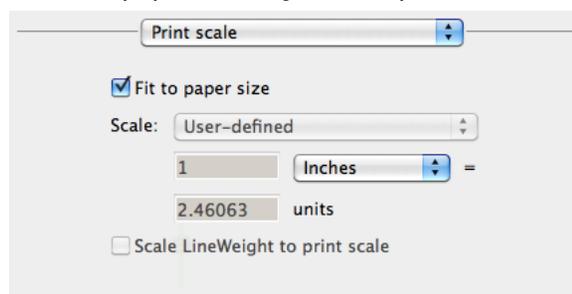
1. Before starting the Print command, create one or more named views with the View command.
2. Enter **Print**, and then choose the **Named View** option. Notice the droplist that appears.
3. Select a named view from the droplist.
4. Click **Print Preview** to see that the area to be plotted matches that of the named view, as illustrated above.

### How to Print a Specified Window

1. Click **Specify Window**.
2. Define the printable area by picking two opposite points, as iCADMac prompts you:  
Specify first corner: (Pick a point, or enter x,y coordinates.)  
Specify second corner: (Pick another point, or enter x,y coordinates.)

### Print Scale Options

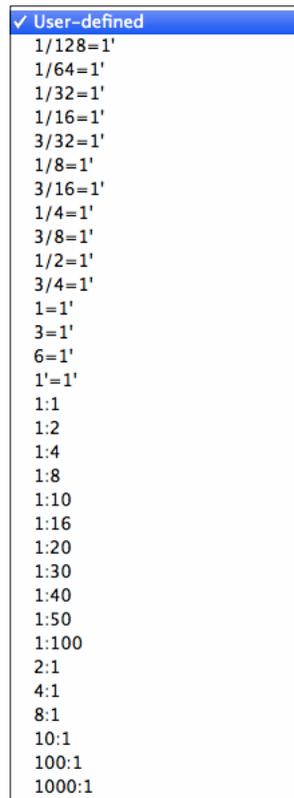
Drawings can be printed to fit the paper, or they can be printed to scale.



**Fit to Paper Size** ensures the drawing fits the paper; the drawing area is determined by the Print Range options described above. Turn off this option to specify the scale factor at which the drawing is printed.

**Scale** specifies the printed drawing's scale factor. (To access the scale factors, you must turn off the Fit to Paper Size option.) Choose a predefined scale factor from the droplist, or else choose User-defined to specify your own scale factor.

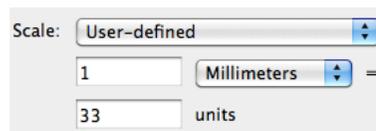
The droplist contains common scale factors for Imperial units (such as 1/4"=1') and metric units, such as 1:20.



**Scale LineWeight to Print Scale** scales lineweights to match the print scale. This ensures that the width of lineweights does not overwhelm drawings printed at small scale factors.

### How to Create a Custom Plot Scale

To specify a custom scale factor, such as 1:33mm, follow these steps:



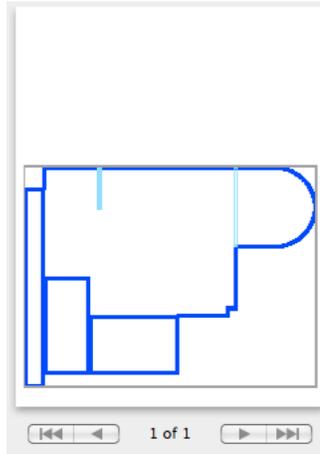
1. Select **User-defined** from the Scale droplist.
2. Enter **1** as the number of millimeters. This is the distance measured on the paper.
3. Select **Millimeters** from the droplist.
4. Enter **33** as the number of units in the drawing. Remember to check the preview area to see how the drawing now fits the paper.

### Print Offset

The Print Offset section determines where the drawing is placed on the paper:

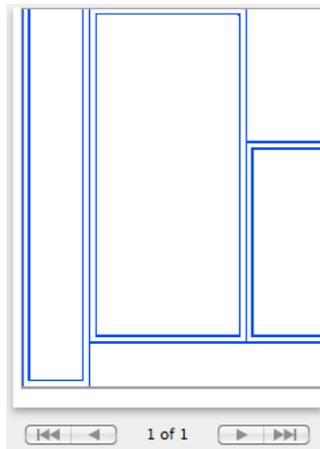
### The iCADMac Early Warning System

The preview window of the Print dialog box shows you how the drawing fits the sheet.

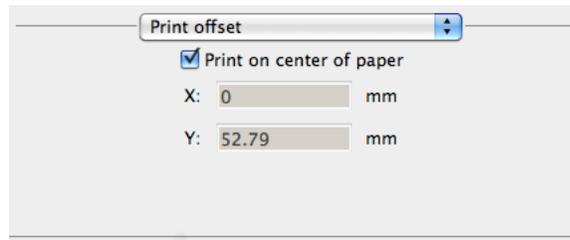


**Gray rectangle** — extents of the drawing.

As you change parameters, the drawing preview changes its size:

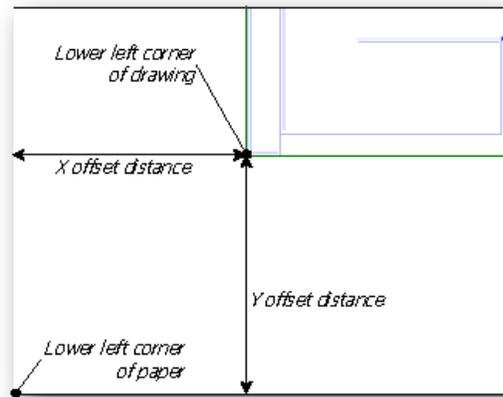


This preview is available only when you click the  button to expand the Print dialog box. When the dialog box is in its unexpanded state, you would click the Preview button to see a preview of the plot in OS X's Preview application, as described later.



**Print on Center of Paper** centers the drawing on the paper. This is handy for making sure the drawing is nicely centered.

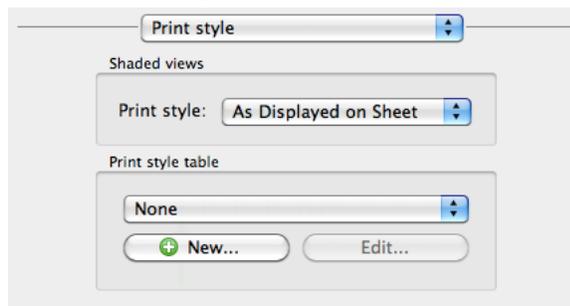
**X** and **Y** specify the distance from the lower left corner of the paper to the lower left corner of the drawing, as illustrated below. Use these distances to move the drawing out of the way of title blocks and other pre-printed areas of the paper.



**TIP** You can use layout mode to position the drawing visually relative to the paper. See chapter 13.

## Print Style

**Print Style** specifies how 3D entities are printed.



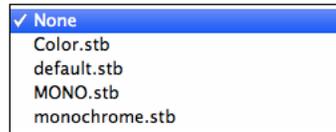
**Shaded Views** controls the look of the entire 3D drawing. Choose a style from the droplist:



- **As Displayed on Sheet** prints the way it looks.

- **Hidden** removes hidden lines.
- **Rendered** prints the model as rendered.
- **Wireframe** prints with no rendering and no hidden lines removed.

**Print Style Table** controls how individual entities and layers are plotted, assuming a plot style table has been attached. Choose a print style (.ctb) file name from the droplist. (CTB is short for “color-based table.”)

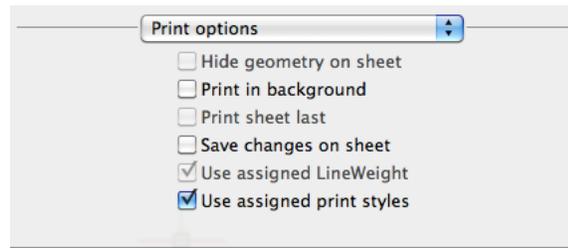


**New** creates new print styles; see the section on print styles later in this chapter.

**Edit** modifies the selected print style.

## Print Options

The following options come into effect when they are turned on:



**Hide Geometry on Sheet** does not print hidden lines of 3D entities shown through viewports in sheets (layouts). This option is unavailable (grayed out) when you plot from the Model tab.

**Print in Background** prints the drawing in the background, allowing you to return to the drawing editor almost immediately. Only turn off this option when printing occurs too slowly.

**Print Sheet Last** prints paper space entities before printing model space entities shown in viewports. This option is unavailable (grayed out) when plotting from Model tab.

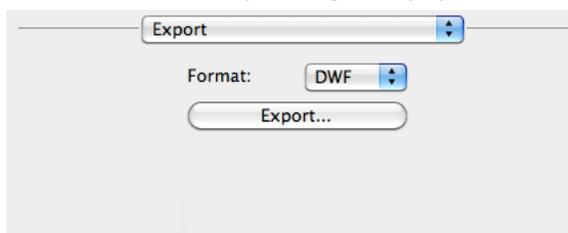
**Save Changes on Sheet** saves the Print dialog box’s settings with the active sheet.

**Use Assigned LineWeight** prints entities with the LineWeights assigned to them in the drawing. This option is unavailable (grayed out) when Use Assigned Print Styles option is turned on, because print styles assign linewidths.

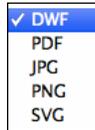
**Use Assigned Print Styles** prints entities with the PrintStyles assigned to them. This option is unavailable (grayed out) when print styles were not assigned to the drawing.

## Export

You can save the drawing as a file, instead of printing it to paper. Choose **Export** from the droplist.



iCADMac lets you save the drawing in any of the following file formats:



**DWF** — exports the drawing in DWF format (drawing Web format), often used to display drawings in Web pages.

**PDF** — exports the drawing in PDF format (portable document format), often used for reports.

**JPG** — exports the drawing in raster JPEG format, the same format used by digital cameras. While this format generates very small files relative to the other formats, the high levels of compression create artifacts that interfere with line drawings created by CAD software. Selecting this option is similar to using the ExportJPG command.

**PNG** — exports the drawing in raster PNG format, commonly used to display images in Web pages. This format is preferred over JPEG, since it creates no artifacts. It is similar to using the ExportPNG command.

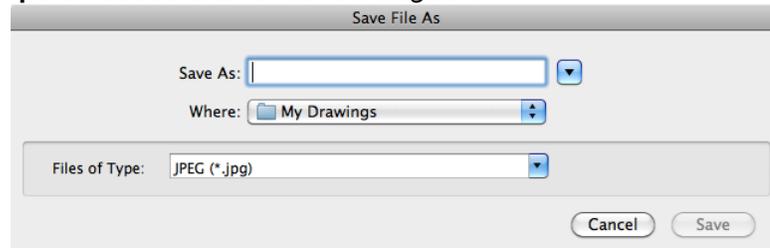
**SVG** — exports the drawing in vector SVG format (scalable vector graphics), sometimes used to display drawings in Web pages; equivalent to using the ExportSVG command.

If you are unsure which format to select, choose DWF for vector output and PNG for raster. If you are unsure whether to use Print or one of the Export commands, consider these benefits: the Print command gives you more control, while the Export commands are faster.

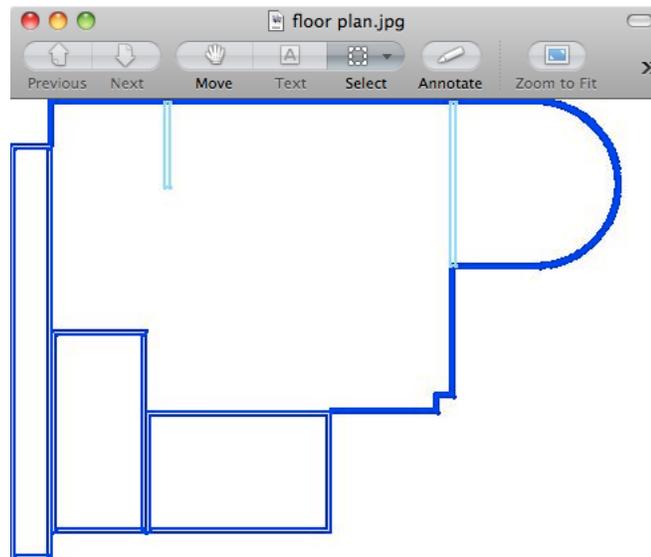
### How to Export Drawings as Raster and Vector Files

Here is how to use **Export** to print drawings to a file:

1. In the Print dialog box, choose the **Export** option.
2. Select a raster or vector file format.
3. Click **Export**. Notice the Save File As dialog box.



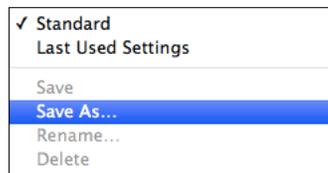
4. Enter a file name into the **Save As** field.
5. If necessary, change the destination folder. By default, iCADMac saves files to the `\my drawings` folder.
6. Click **Save**.
7. To view the exported image, double-click the file in the destination folder. For example, double-click `floor plan.jpg` in `\my drawings`. The file may open in the Preview app or another application assigned to handle JPEG files.



### Saving Settings as Presets

After you change one or more settings, you may want to preserve them for future prints. I tend to print drawings (a) with all geometry, (b) scaled to fit the paper, and (c) centered.

To save customized settings, click the **Standard** drop list. Above the horizontal line are the names of saved settings. Initially, there are two: Standard (the default settings) and Last Used Settings.



Below the horizontal line are the controls:

**Save** — saves the current settings to the current name, overwriting previous settings.

**Save As** — saves the settings by name.

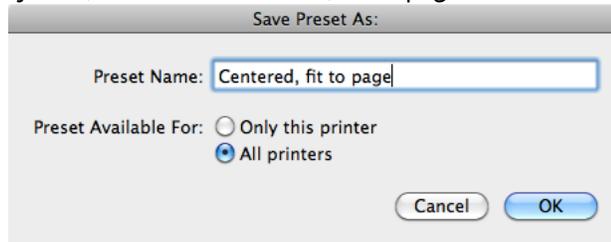
**Rename** — changes the name of existing settings.

**Delete** — removes the current named setting from the list. Select the name to be removed, and then choose **Delete**.

## How to Save Print Settings

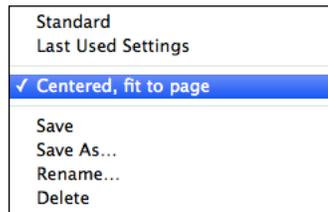
To save print settings for reuse, following these steps:

1. From the Preset Name droplist, choose the **Save As** item. Notice the dialog box.
2. Enter a name that describes the settings. iCADMac prevents you from using a name that already exists. In my case, I entered “Centered, fit to page.”



3. Decide whether the settings apply to the current printer, or to all. In my case, I always want my prints to be centered and fitted, so I would choose **All Printers**.
4. Click **OK**.

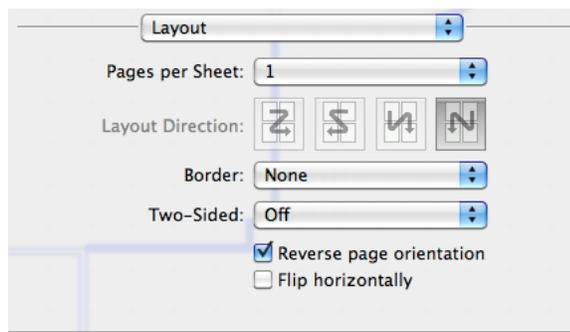
Notice that the name is added to the list of preset names.



From now on, you need only select the preset name, instead of going through the five sections.

## OS X Options

A second set of options is common to the Macintosh, not specific to iCADMac. Many of these are not usually of interest to CAD users, but some of them may be useful from time to time. Here is a summary of the options:



**Layout** lets you print the drawing over two or more pages. The **Flip Horizontally** option mirrors the print, which can be useful when you need to print the drawing on the underside of transparent media.

**Color Matching** ensures the colors you see on the screen are reproduced by the printer. This is of no consequence to monochrome laser printers, and matters little for color prints of drawings.

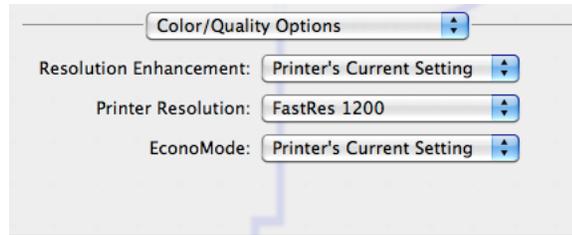
**Paper Handling** prints multi-page output in forward, reverse, odd-only, or even-only order.

**Paper Feed** specifies which paper tray to use, useful for printers that have two or more trays hold-

ing different sizes of paper.

**Cover Page** prints a cover page, useful for separating multiple print jobs, especially for networked printers serving numerous computers.

**Scheduler** determines when the drawing is printed, now or later; useful for holding off large print jobs till overnight.

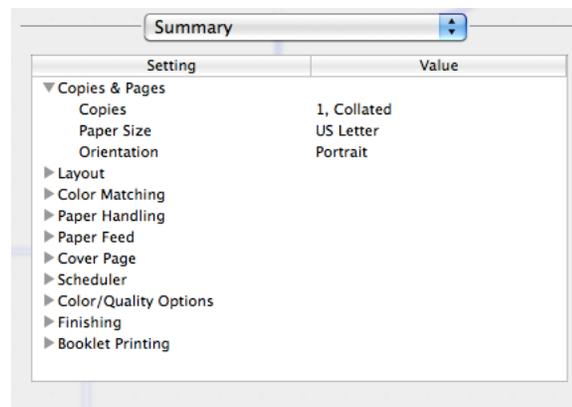


**Color/Quantity Options** vary greatly according to the printer; the settings illustrated above are for the HP LaserJet. A higher resolution produces a higher quality print, but takes longer to output.

**Finishing** specifies the type of medium, such as paper or transparency.

**Booklet Printing** is the opposite of Layout; here, multiple drawings are printed on both sides of single sheets of paper.

**Summary** lists all options.



## Previewing Before Printing

**Command:** Preview

**Alias:** pre

**Menu Bar:** File | Print Preview

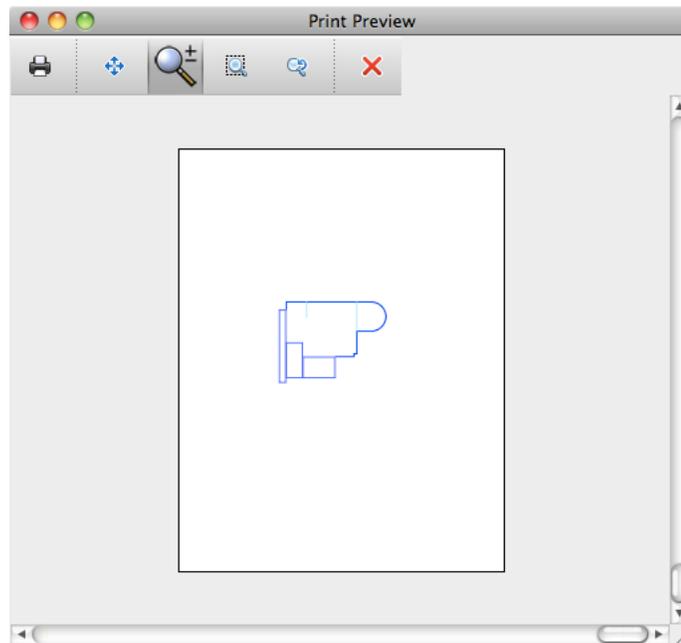
**Tool Matrix:** Standard | Preview

The Preview command lets you see the drawing before it is printed. It displays the Print Preview window inside iCADMac.

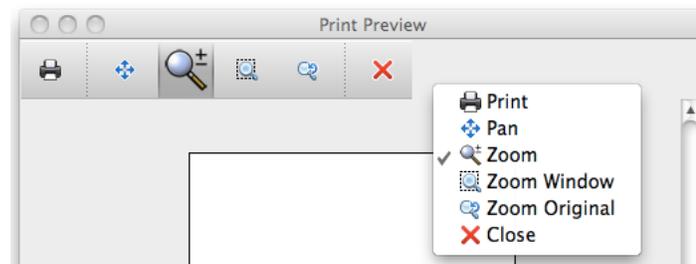
(A second Preview command is accessible in the Print dialog box; it displays drawings using Mac OS X's Preview app.)

The preview allows you to correct errors before you waste a sheet of paper and a bunch of ink, errors such as incorrect size or offset.

Enter the **Preview** command to see the drawing in a new window:



You control the preview image through the toolbar or the right-click menu.



The options have the following meanings:

**Print** closes the preview window and then prints the drawing.

 **Pan** moves the preview image; this works only after you have zoomed in to the preview image. During panning, the hand cursor appears, as shown at left. (The Pan and Zoom options do not affect the printed output; that is, they do not move, offset, enlarge, reduce, or scale the drawing printed on the paper.)

 **Zoom** enlarges and reduces the preview image as you roll the mouse's scroll wheel. Roll forward to zoom in; roll back to zoom out.

**Zoom Window** enlarges the preview image defined by a rectangle. Pick two points to define the rectangle.

**Zoom Original** returns the preview image to fit the preview window, following zooming and panning operations or resizing the window.

**Cancel** closes the preview window.

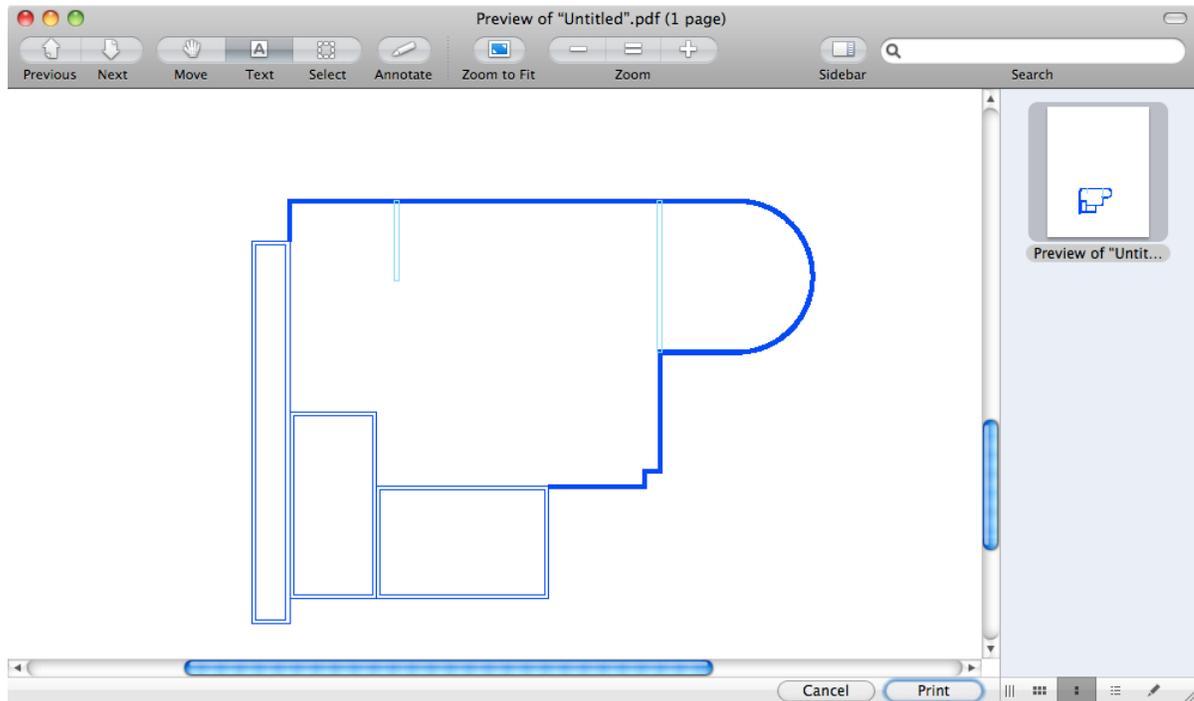
**TIP** Upon entering the Preview command, you might receive the following error message:

```
Command: preview
No printer has been set.
No previous print settings available.
```

To get around this problem, enter the ? option, and then choose one of the names listed:

Default: \_?/?  
Options: ? to list or  
Specify printer name» ?  
Available print devices.  
hpLaserJet  
Photosmart C6200 series [0964B0]  
DWF  
PDF  
JPG  
PNG  
SVG  
Default: \_?/?  
Options: ? to list or  
Specify printer name» hplaserjet

Alternatively, enter the **Print** command and then click the **Preview** button. This approach, however, displays the drawing in the OS X Preview app, as illustrated below:



The remainder of this chapter deals with advanced printing topics.

## Using Page Layouts with Drawings

**Command:** PageLayout  
**Alias:** pagesetup  
**Menu Bar:** File | Page Layout Manager

The presets you saved in the tutorial above are stored with iCADMac. They can apply to *any* drawing you want to print. iCADMac also allows you to store presets with *each* drawing, known as “page layouts.” This is useful when a project consists of many drawings, some of which must be printed on smaller paper with a laser printer, others on large paper with a color inkjet printer.

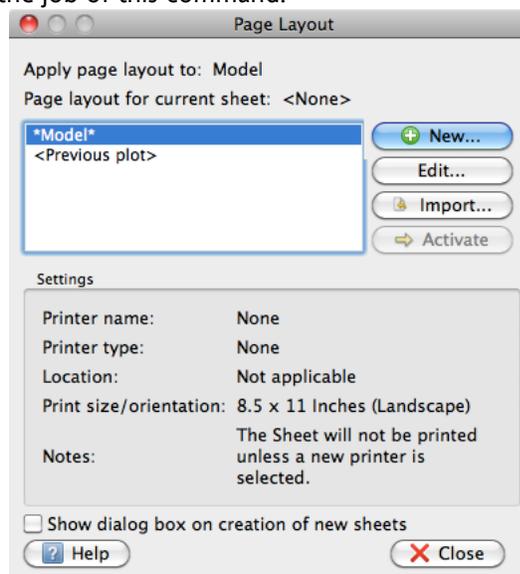
The **Use Page Layout** mode lets you use print options stored with the drawings.

You cannot use this mode unless at least one page layout was defined before starting the Print command. Page layouts are defined with the PageLayout command; sometimes, they are called “page setups.” Because page layouts are named, you can use them for more than one drawing.

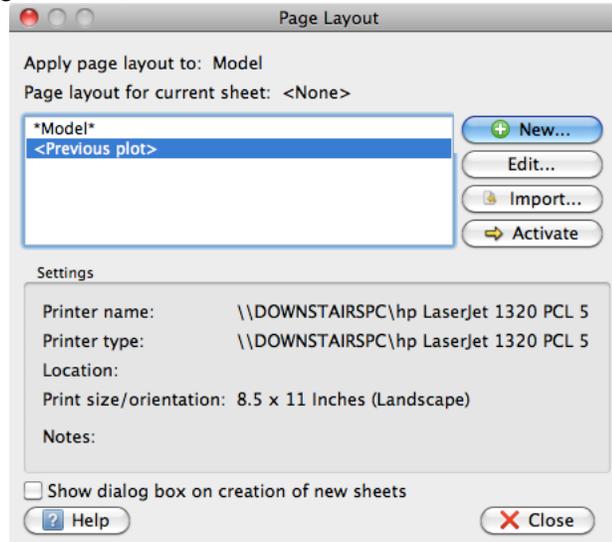
### How to Create Page Layouts

To assign printing setup to a specific drawing, follow these steps:

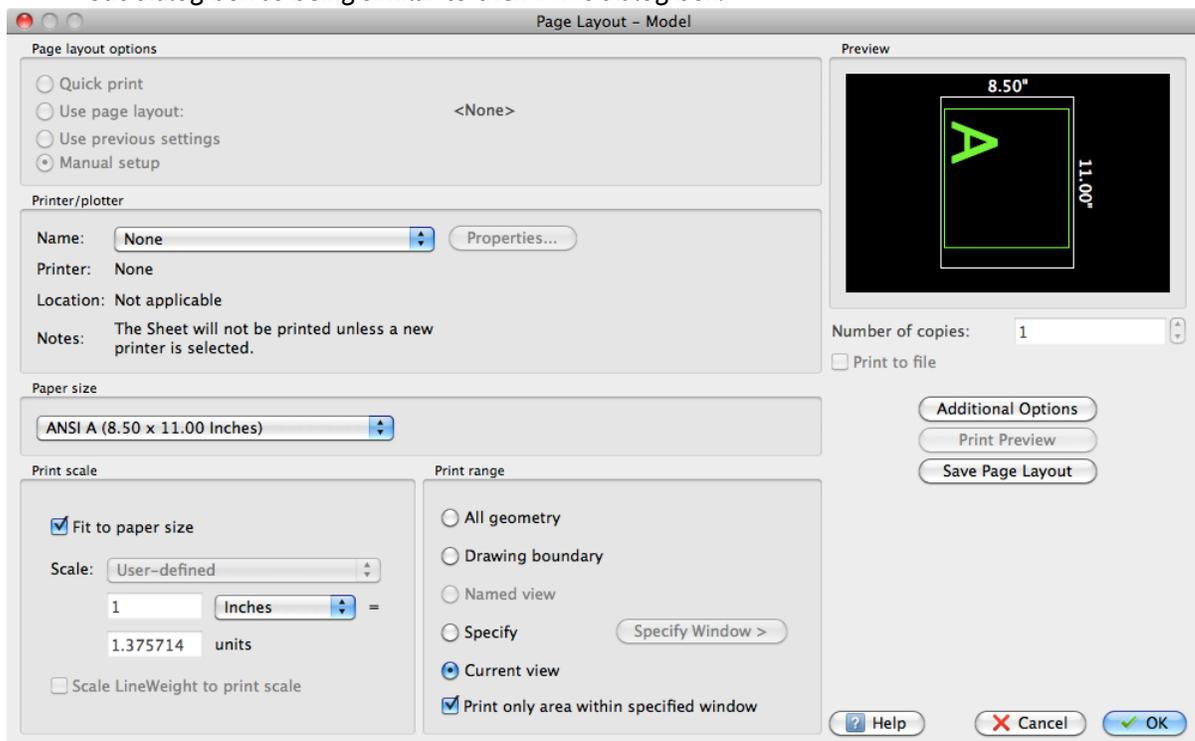
1. Enter the **PageLayout** command. Notice that the Page Layout dialog box reports that the sheet (drawing) cannot be printed with \*Model\*, for no printer is assigned to it. Assigning printers to layouts is the job of this command.



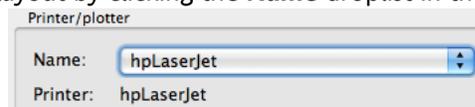
- Choose **<Previous Plot>**, and notice that the settings you applied in earlier tutorials appear in the Settings window, such as those shown below.



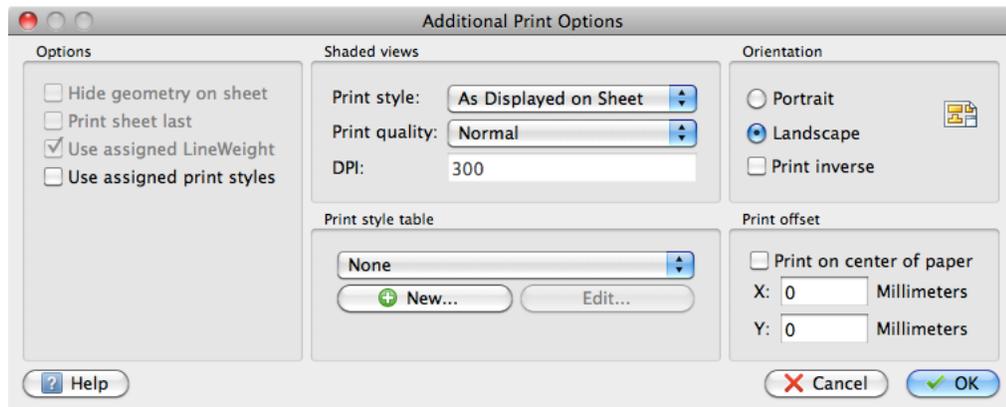
- Go back to \*Model\*, and then click **Edit** to personalize the printer settings for the Model tab of the office layout drawing. Windows users of iCADMac will recognize the Page Layout dialog box as being similar to their Print dialog box.



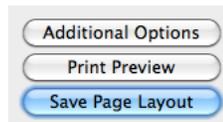
- Assign a printer to the layout by clicking the **Name** droplist in the Printer/plotter section.



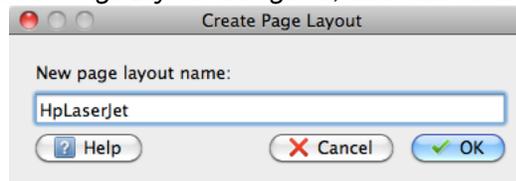
- Change any other settings in the dialog box. They correspond to settings described earlier in this chapter. Additional settings are available by clicking the **Additional Options** button. (See figure below.)



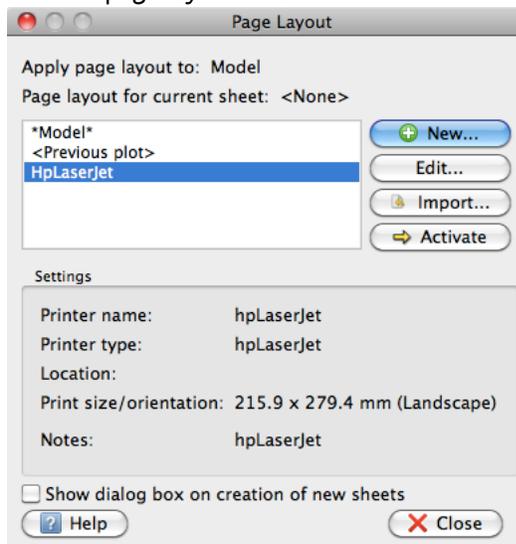
6. When done, click **Save Page Layout**.



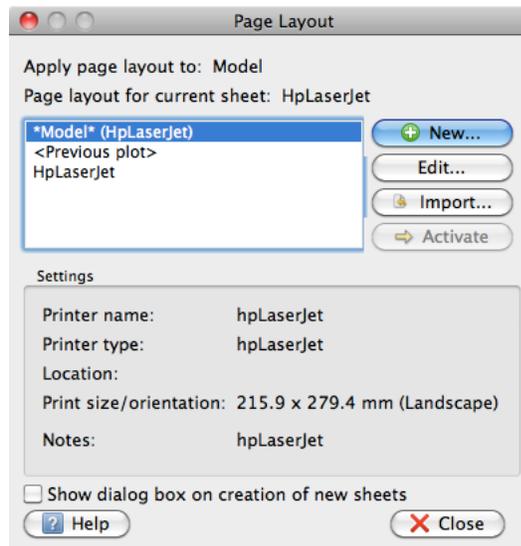
7. Enter a name in the Create Page Layout dialog box, and then click **OK**.



8. Click **OK** to exit the Page Layout dialog box. Notice that the original Page Layout dialog box reappears, listing the new page layout's name.



9. Ensure HpLaserJet is selected, and then click **Activate**. This applies the page layout to the Model layout, as illustrated below.



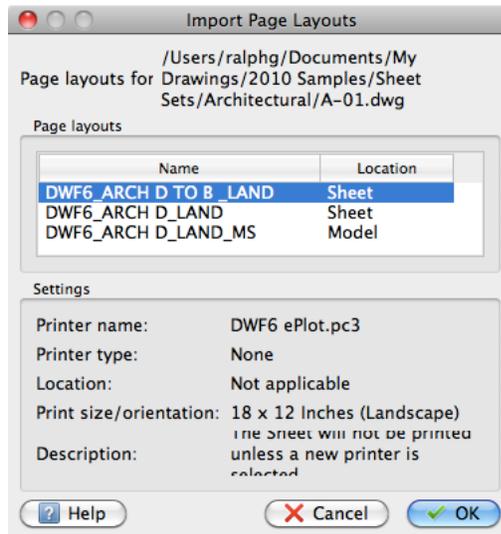
10. Click **Close** to dismiss the dialog box.

Now when you use the Print and Preview commands on this drawing, the HpLaserJet settings will be used. The Preview command no longer embarrasses you by announcing, “No printer has been set.”

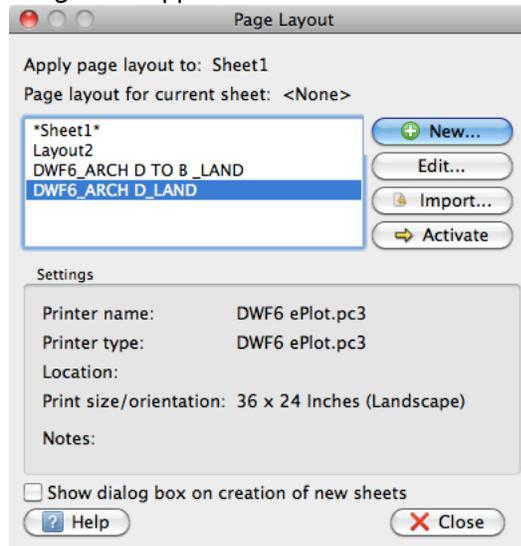
### How to Borrow Page Layouts

You can borrow a page layout from another drawing when the current drawing does not have one, or when you want to import it from other drawings. Follow these steps:

1. Enter the **PageLayout** command, and then click **Import**. Notice the Open dialog box.
2. Go to the folder containing drawing files with page layouts. Here you have to guess, because there is no way through this dialog box to tell which drawings contain layouts.
3. If necessary, select a file type. iCADMac can import page layouts from the following types of files:
  - DWG** drawing files from iCADMac, AutoCAD, and so on
  - DXF** interchange files
  - DWF** drawing Web files
4. Select the file, and then click **Open**. Notice the Import Page Layouts dialog box. If you are lucky, it will contain the names of previously defined page layouts, as shown below. (If not, then the list will be empty.)



- Choose one or all of the page layouts listed in the dialog box, and then click **OK**. Notice that the Page Layout dialog box reappears.



## Assigning Print Styles

**Command:** PrintStyle  
**Aliases:** plotstyle, pstyle  
**Menu Bar:** Format | Print Style

The PrintStyles command defines how individual entities and layers are printed. Generally, you can ignore print styles and this section of the chapter — unless you require extreme control over the printing process.

There are many, many print style options, and they let you specify the manner in which every layer and/or entity is to be printed. You can control the following properties:

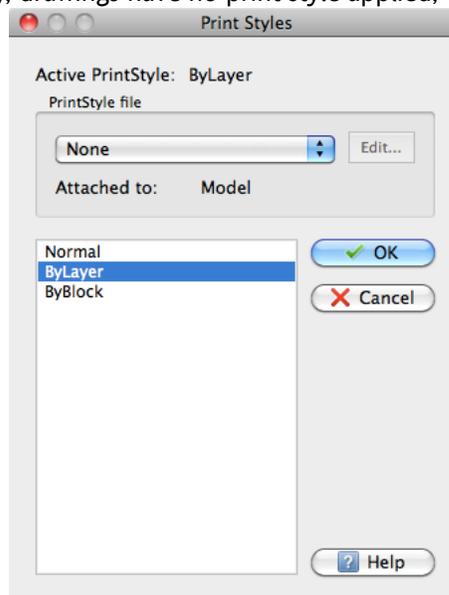
Line color	Line weight	Line style
Dither	Screening	Pen number
Force end segments	Line cap style	Corner style
Grayscale	Fill style	Virtual pen number
	Scaling	

I'll discuss these properties in detail later; first, let's see how to create and apply print styles.

### Creating Print Styles

If you want ultimate control over the look of printed drawings, then this is how you create print styles:

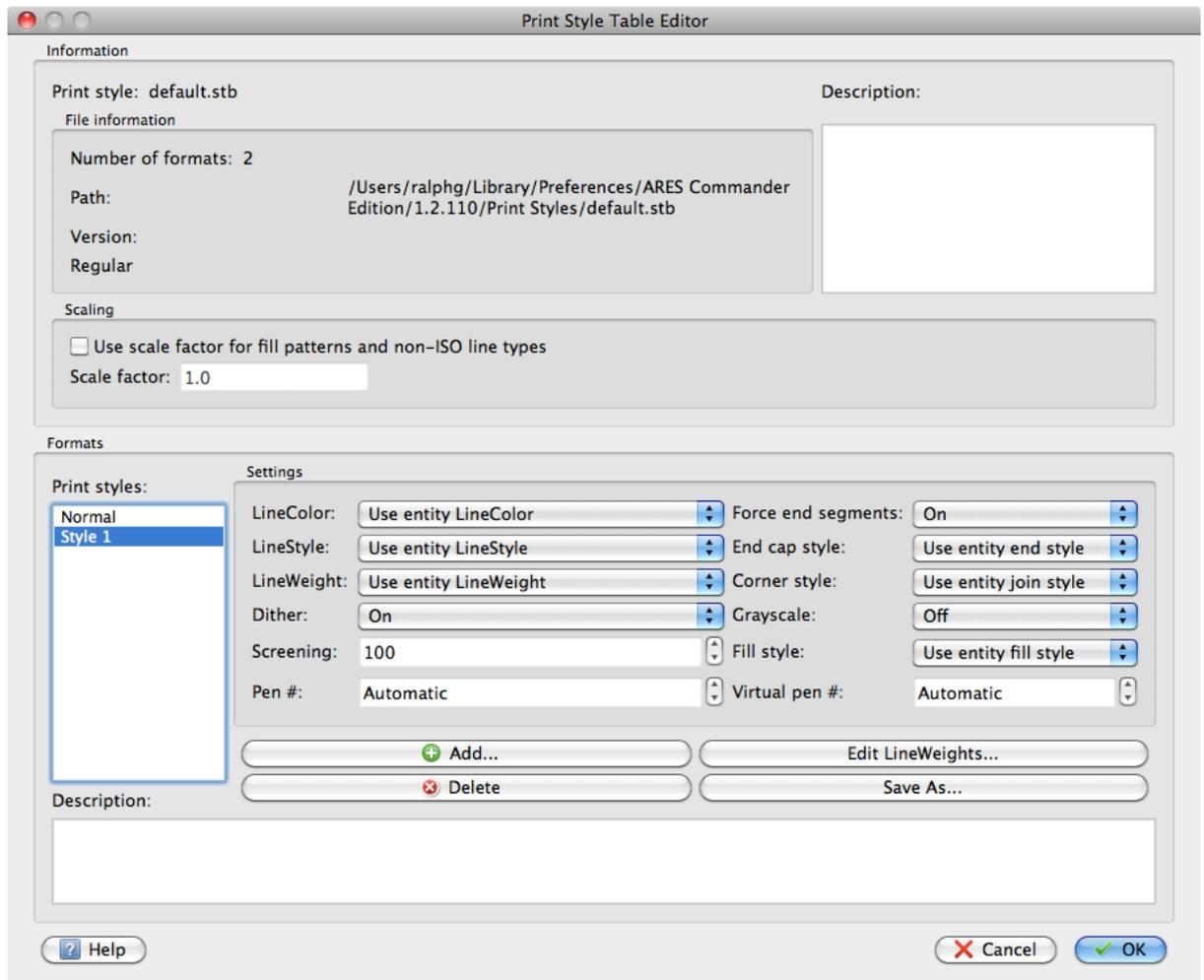
1. Start the **PrintStyle** command using one of the methods listed above. Notice the Print Styles dialog box. Initially, drawings have no print style applied, "None."



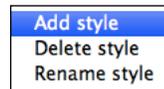
2. From the **PrintStyle File** drop list, choose a print style name. For this tutorial, choose *default.stb*, and then click **Edit** to review the available print style properties.

(Whereas page layouts are stored in drawings, print styles are stored in external *.stb* files. That's because page layouts are specific to drawings, while print styles can be used with any drawing. STB is short for "Style TaBle.")

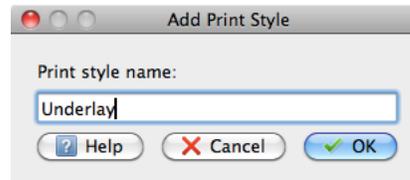
Notice the Print Style Table Editor dialog box.



3. To create the new print style, follow these steps:
  - a. In the Print Styles list, right-click, and then choose **Add Style** from the shortcut menu.



- b. In the Add Print Style dialog box, enter a name for the new print style. For this tutorial, enter "Underlay," and then click **OK**.



Notice that the new Underlay print style is added to the list of Print Styles.

4. You can now change the settings. For this tutorial, change Screening to 30%.

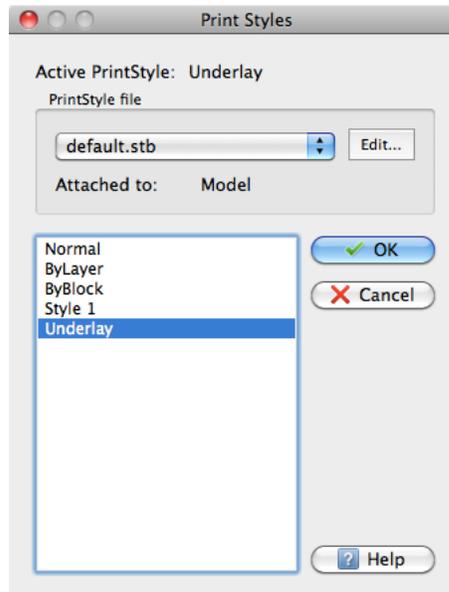


5. Click **OK** to exit all of the dialog boxes. You have created the new print style named Underlay.

## Applying Print Styles

In this tutorial, you learn how to apply print styles to drawings — well, not drawings. Print styles are applied to layers and entities, not entire drawings. To apply styles, follow these steps:

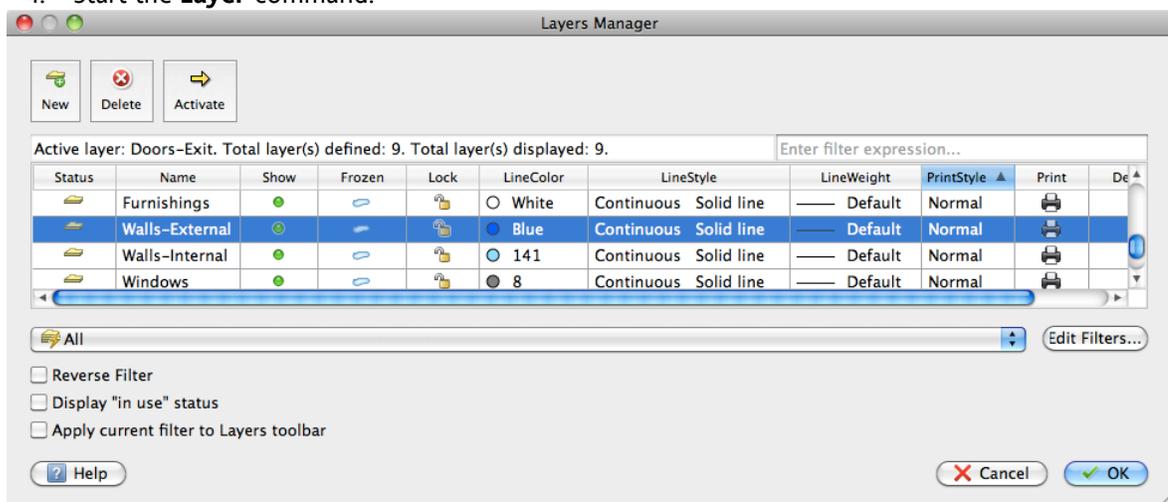
1. Enter the **PrintStyle** command. The newly-created Underlay style should be listed in the dialog box.



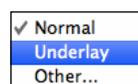
2. Attach the Underlay print style. (If necessary, choose **Default.stb** from the PrintStyle File droplist.) In the list of print style names, choose **Underlay**. Notice that the dialog box reports, “Attached to: Model.”

This action attaches the Underlay print style to the model tab of the *office plan.dwg*. The drawing does not look any different, because print styles are not applied to drawings. In the next step, you apply the print style to entities through the Layer command.

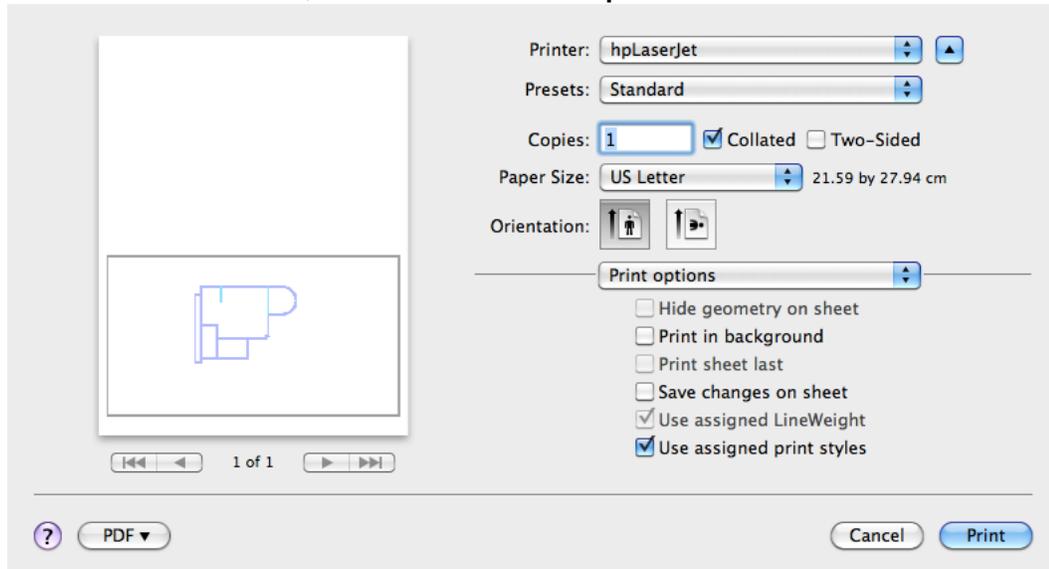
3. Click **OK** to dismiss the dialog box.
4. Start the **Layer** command.



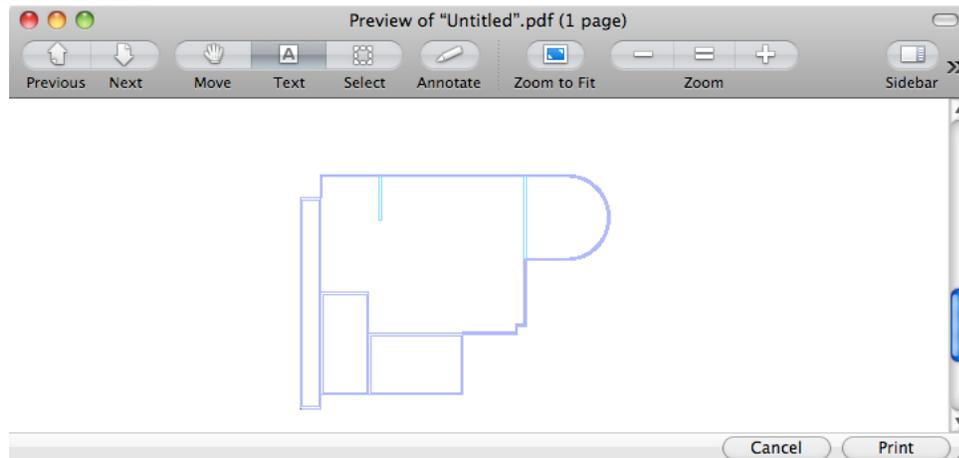
5. In the Layers Manager dialog box, choose the **Walls-External** layer.
6. In the PrintStyle column, click “Normal,” and notice that it turns into a droplist of plot style names.



7. From the PrintStyle droplist, select **Underlay**.  
(The **Other** item opens the Print Style dialog box.)
8. Click **OK**. Again, the drawing looks no different. That's because print styles apply to prints, not to drawings.
9. Start the **Print** command, and then choose **Print Options**.



10. Ensure that the **Use Assigned Print Styles** option is turned on.  
(Notice that the Use Assigned LineWeight option gets turned off automatically; that's because print styles override lineweights.)
11. Click  to collapse the dialog box, and then click **Preview**. Notice that the external walls look faint.



- TIP** To apply print styles to individual entities, use the Properties palette, as follows
1. Press **command+1** to open the Properties palette.
  2. Select the entities to which you wish to apply a print style.
  3. In the Properties palette, choose a print style from the PrintStyle section's Style droplist.



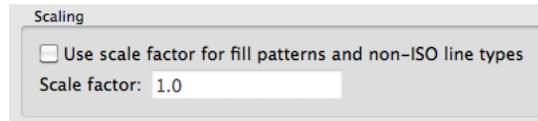
Use Print Preview to ensure the style has been applied correctly.

## All About Print Style Properties

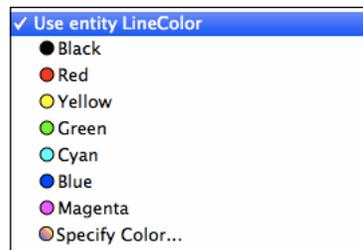
---

Print styles control a baker's dozen properties, as detailed here.

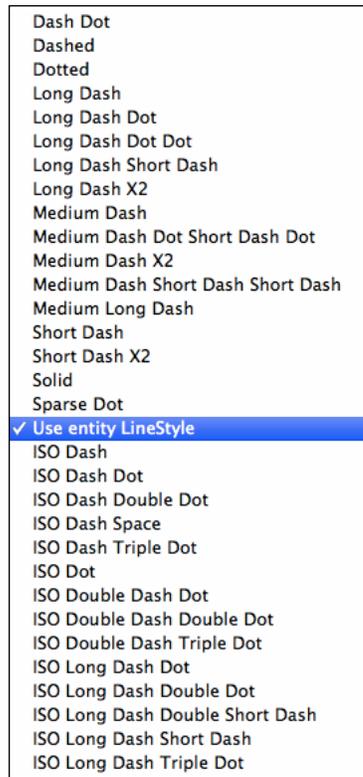
**Scaling** specifies the size of fill (hatch) patterns and line types. (ISO patterns are scaled in the Hatch dialog box.) Enter a scale factor smaller than 1 to plot them smaller; larger than 1 to plot them larger.



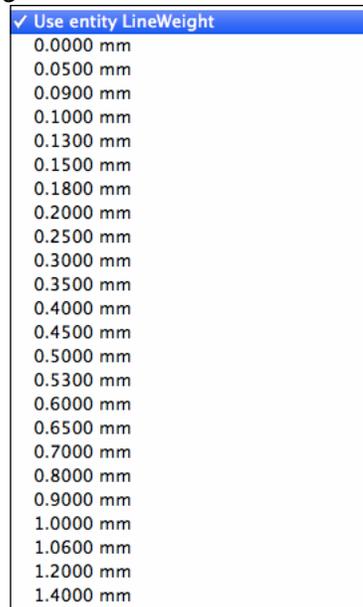
**Line Color** prints the entity with the color assigned in the drawing (with the Color or Layer commands), or overrules the assigned color with another one. Usually, this option should be left at "Use Entity Color," unless you have a reason to overrule the drawing color.



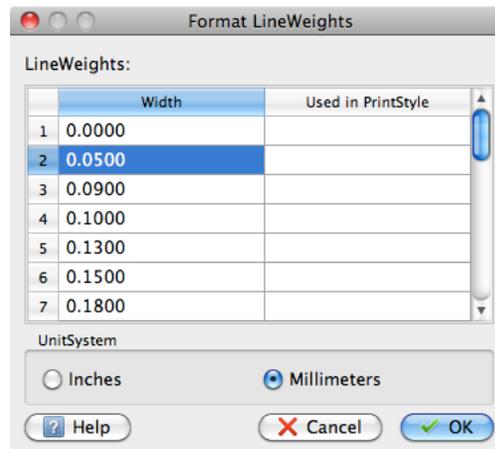
**Line Style** prints the entity with the linestyle (linetype) assigned in the drawing (with the LineStyle or Layer commands), or overrules the assigned style with another one. Usually, this option should be left at "Use Entity Linetype" unless you have a reason to overrule the drawing linestyle.



**Line Weight** prints the entity with the weight assigned in the drawing (with the Line-Weight or Layer commands), or overrules the assigned lineweight with another one. Usually, this option should be left at “Use Entity LineWeight,” unless you have a reason to overrule the drawing lineweight.



**TIP** Whereas the widths of lineweights are fixed in the iCADMac drawing editor, you can edit their values in the Print Style Table Editor dialog box, illustrated below. Click **Edit LineWeights**, and then enter new values.



**Dither** toggles the use of dithering, which is useful when the printer outputs only a few colors. Turning on dithering helps the printer simulate more colors and shades of gray. Usually, this option should be left **off**, unless you know the printer needs it.



**Screening** adjusts the amount of color. For example, 50% screening prints half white and half color. This is useful for making colors less intense. Usually, this option should be left at **100%**.



**Pen Number** allows you to specify the pen number the printer should use. This is a hold-over from the 1980s, when pen plotters were popular. These plotters had between 1 and 8 pens, which could have different colors or different thicknesses. You can specify that certain layers or entities use a specific pen in the plotter.

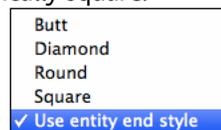


The range is 1 to 32. Usually, however, you keep this option set to “Automatic,” which means the plotter uses the pen number that corresponds to the color number. For instance, red is color #1, and so the plotter uses pen #1 — even if it is not red.

**Force End Segments** determines if the linestyle scale is adjusted to complete the linestyle pattern along the length of the entity. Usually, this option should be turned **on**.



**End Cap Style** determines how the ends of lines (and other open entities) should be printed. You can choose from Butt, Diamond, Round, Square, or just leave it alone with “Use Entity End Style,” which is typically square.



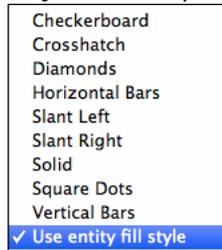
**Corner Style** determines how the intersections of lines (and other entities) should be printed — not just corners. You can choose from Bevel, Diamond, Miter, Round, or just leave it alone with “Use Entity End Style,” which is typically overlapping.



**Grayscale** toggles the use of shades of gray, instead of color. Use this option if your monochrome printer prints all colors as black, or if you want a color printer to print the drawing colors in gray. Usually, this option should be left **off**.



**Fill Style** specifies the pattern with which filled areas should be printed. Usually, this option should be left as “Use Entity Fill Style,” which is a solid fill. However, you do have a choice of patterns, which is required by some disciplines.



**Virtual Pen Number** is just like the Pen Number option, but is meant for printers that support the concept of pen numbers. For a laser printer, for example, virtual pen #1 might correspond to solid lines, and pen #2 to crosshatched lines. Usually, you keep this option set to “Automatic,” unless you know that the printer supports virtual pens.



## Summary

Now you know all about printing and plotting, including the very sophisticated ways in which the output can be controlled through print styles.

In the next chapter, you learn about an important technique that will make your drafting more efficient: using blocks (symbols)

## Chapter 7

# Adding Symbols

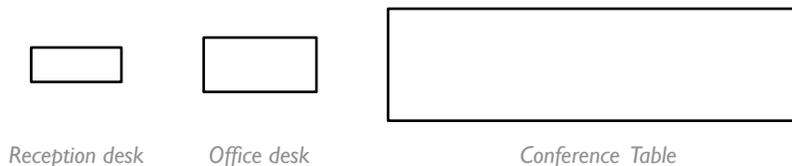
### In This Chapter

- Drawing symbols
- Converting symbols to blocks
- Inserting blocks
- Downloading blocks from Web sites
- Extracting blocks from drawings
- iCADLib Blocks Manager

**With the walls** in place, it is time to populate the office with furnishings. Desks, chairs, windows, and doors are usually drawn as symbols, which can be inserted as reusable blocks. In this chapter, you draw a variety of symbols common to office layouts, and then add them to the *office layout.dwg* drawing.

### Drawing a Desk Symbol

The desk is perhaps the most basic of symbols, and thus the easiest to draw. It consists of rectangles placed at various sizes that represent desks that are small, large, or as enormous as a conference table — as illustrated below.



If you were to draw one symbol for every possible size of desk, you would end up with many, many rectangles — indeed a large number that becomes unmanageable. Instead, you draw one rectangle, and then use a trick to make it suitable for any size of desk.

The desk is drawn with the Rectangle command, with an opaque filling added by the Mask command. (The mask becomes useful later when you add chair symbols to the desks.)

**TIP** iCADMac uses the term “block” in place of symbol, following the practice of other CAD programs, like AutoCAD and progeCAD. In this book, I use the term “symbol” for a groups of objects before it is turned into a block with the MakeBlock command.

## Key Terms in This Chapter

**Block** — refers to a collection of entities that is treated as one.

**Block Definition** — defines a block; invisible in the drawing.

**Draw Order** — defines which overlapping entities are visually “on top.”

**Explode** — refers to the reduction of complex entities to basic equivalents.

**Exploded Block** — refers to constituent entities, instead of the whole block.

**Fillet** — rounds corners.

**Insert** — places blocks in drawings.

**Mask** — refers to colored areas (usually white or black) meant to hide objects underneath them.

**Mirror** — mirrors copies of entities.

**Pattern** — copies entities in evenly-spaced horizontal, vertical, or circular patterns.

**Symbol** — refers to a graphical representation of a real world object.

**Uniform Scale** — refers to blocks that have the same x, y, and z scale factor.

**Unit Block** — refers to a block that fits inside a 1x1-unit square.

## Commands

<b>Command</b>	<b>Shortcuts</b>	<b>Menu Selection</b>
Arc	a	Draw   Arc
DisplayOrder	dr, draworder	Tools   Draw Order
Explode	x	Modify   Explode
ExportDrawing	w, wblock	File   Export   Export DXF/DWG
Fillet	f	Modify   Fillet
InsertBlock	i, insert	Insert   Block
MakeBlock	b, block	Draw   Block   Make Block
Mask	wipeout	Draw   Mask
Mirror	mi	Modify   Mirror
Pattern	ar, array	Modify   Pattern

## Drawing a Unit-size Desk

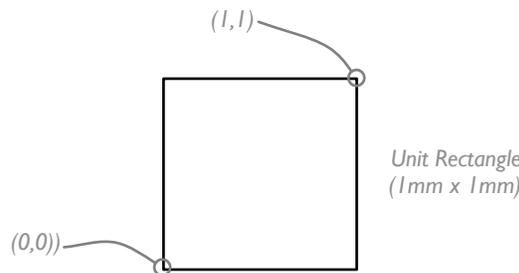
**Command:** Mask  
**Aliases:** wipeout  
**Menu Bar:** Draw | Mask

The trick to placing different sized symbols is the “unit block.” This is a symbol drawn to fit a square sized 1mm by 1mm. Later, when you insert the block in the drawing, you specify the actual size of the desk, such as 600mm by 900mm. In this way, a single block can represent *any* size of desk, cabinet, or other rectangular shape.

To draw the unit block, follow these steps.

1. Start a new drawing with the **New** command, and then choose *standardiso.dwt* from the Specify Template dialog box.
2. Use the **SaveAs** command to save this drawing with the *office desk.dwg* file name.
3. Enter the **Rectangle** command, and then draw a rectangle sized 1mm by 1mm, as follows:

Command: rectangle  
 Options: Chamfer, Elevation, Fillet, Thickness, line Width or  
 Specify start corner» 0,0  
 Options: Area, Dimensions, Rotation or  
 Specify opposite corner» 1,1



4. To see the rectangle, zoom the drawing so that it fits the window. Feel free to use *aliases* to reduce the number of keystrokes. Enter “z” for Zoom and “f” for Fit, as follows:

Command: z  
 ZOOM Default: Dynamic  
 Options: Bounds, Center, Dynamic, Fit, Previous, SElected, specify a scale factor (nX or nXP) or  
 Specify first corner» f

**TIP** If you prefer entering the options of the Zoom command as found in AutoCAD or progeCAD, iCADMac accepts them as well. For example, you can enter **z e** to zoom the drawing to its extents.

5. With the **Mask** command, fill the rectangle with a solid white color:

Command: mask

- a. With this command, you can create new mask outlines with the ‘Specify start point’ option, or use existing entities to define the boundary of the mask.

Since the Rectangle command makes rectangles of polylines, enter **p** for the Polyline option.

Default: Polyline  
 Options: Frames, Polyline or

Specify start point» p

- b. Select the rectangle:  
Specify entity» (Select the rectangle.)
  - c. iCADMac can keep or erase the entity; for this tutorial, keep the polyline that represents the outline of desks. Enter y:  
Default: No  
Confirm: Keep polyline?  
Specify Yes, or No» y
6. Press **command+S** to save your work.

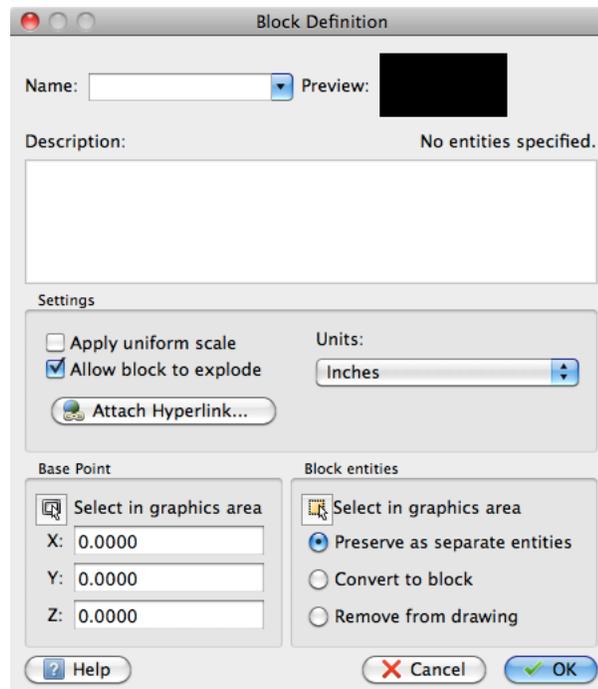
The desk symbol looks no different, because the mask is white. It will come in handy when you place chairs at the desks later in this chapter.

## Converting Symbols Into Blocks

**Command:** MakeBlock  
**Aliases:** b, block, mblock, partdef, bmake  
**Menu Bar:** Draw | Block | Make Block  
**Tool Matrix:** Draw | MakeBlock

The desk symbol consists of two entities, the rectangle and the mask. It is useful to combine them into a single entities, *blocks*. For example, office desks and cabinets, which you can use repeatedly. In iCADMac, you convert symbols into blocks with the MakeBlock command.

1. Start the **MakeBlock** command using one of the methods listed above. Notice the Block Definition dialog box.



The dialog box has many options, but as a minimum you need to provide three pieces of information for iCADMac to convert a group of entities into a block:

- i. A *name* by which the block can be identified.
- ii. The *entities* to be converted.

iii. The *location* where the block is later placed in drawings.

Let's enter the three pieces of information now.

2. In the **Name** field, enter "unit desk." This name will identify the block later when it comes time to insert the block into the office plan drawing.

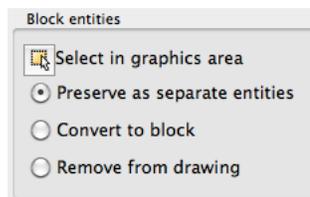
**TIP** When the drawing contains two or more blocks, you can click the **Name** droplist arrow to see their names. This ensures that you do not accidentally use an existing block name.

(If you do reuse an existing block name, the block associated with the name is redefined with the new entities you select. Every instance of the block is redefined by the new one. Thus, this can be a powerful but dangerous feature.)

3. To choose the entities to be converted into a block, follow these steps:
  - a. Click the  **Select in Graphics Area** button found in the Block Entities section.
  - b. Notice that the dialog box disappears, and that iCADMac prompts you in the command window. Select the two entities, and then press **Enter**.
    - Specify entities» (Choose the rectangle and mask.)
    - Specify entities» (Press return to return to the dialog box.)
  - c. Up in the Preview area, notice that iCADMac displays a thumbnail image and reports the number of specified entities.



4. There are three options in the Block Entities section:
  - **Preserve as Separate Entities**
  - **Convert to Block**
  - **Remove from Drawing**



**Preserve as Separate Entities** leaves the entities in the drawing as entities; the block definition is stored in the drawing, but is not visible until you use the InsertBlock command.

**Convert to Block** replaces the entities with a block insertion. This convenient option combines the following commands: Erase to remove the entities, followed by InsertBlock to insert the block.

**Remove from Drawing** removes the entities from the drawing, leaving only the block definition (invisible until inserted with the InsertBlock command). This option tends to dismay new users, who think their new block has disappeared!

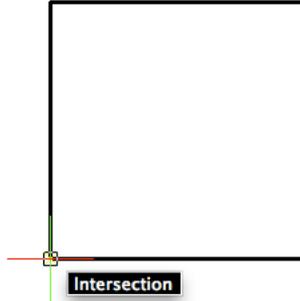
For this tutorial, keep the default — **Preserve as Separate Entities** — because you will use this drawing file as the source for the block. (Later, you will see how to insert blocks from other drawing files.)

5. Choose the *insertion point* for the block. A block consists of many entities. Some point of the block designate where it is inserted in the drawing.

For this table block, specify the lower left corner, as follows:

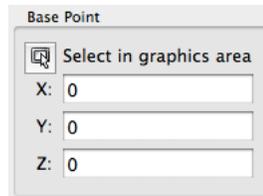
- Click the  **Select in Graphics Area** button, found in the Base Point section.
- Notice that the dialog box disappears, and that iCADMac prompts you in the command window.

Pick the lower left corner with the help of the INTERsection entity snap, and the press **return**.



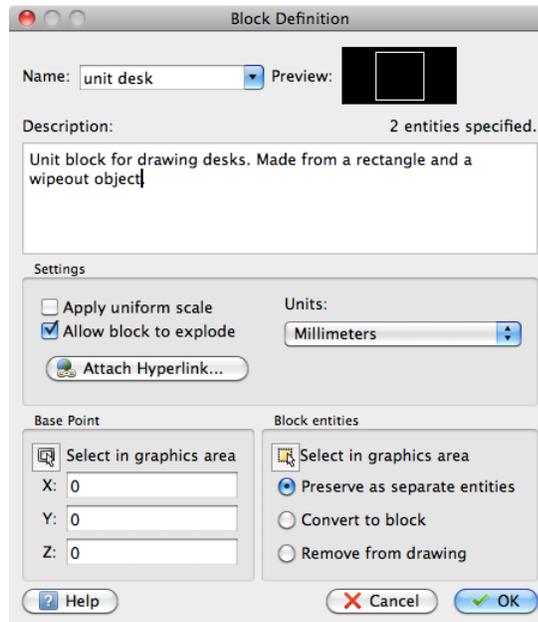
Specify insertion base point» int  
of (Pick the lower left corner of the rectangle.)  
(Press return.)

- When the dialog box returns, notice that the **X**, **Y**, and **Z** coordinates are filled in. (Because you drew the rectangle starting at 0,0, the values reported by X, Y, Z are likely to be 0,0,0.)



- Ensure the following options are left at their default values:
  - Apply Uniform Scale** — off.
  - Allow Block to Explode** — on.
  - Attach Hyperlink** — ignore.
  - Units** — Millimeters.
- The other items in this dialog box are optional. For instance, you can describe the purpose of the block in the **Description** field.

Now the content of the dialog box should be like the one illustrated below:



8. Click **OK** to close the dialog box and define the block.

The drawing looks no different. In the next tutorial, you learn how to insert this newly-created block.

## Inserting Blocks into Drawings

**Command:** InsertBlock

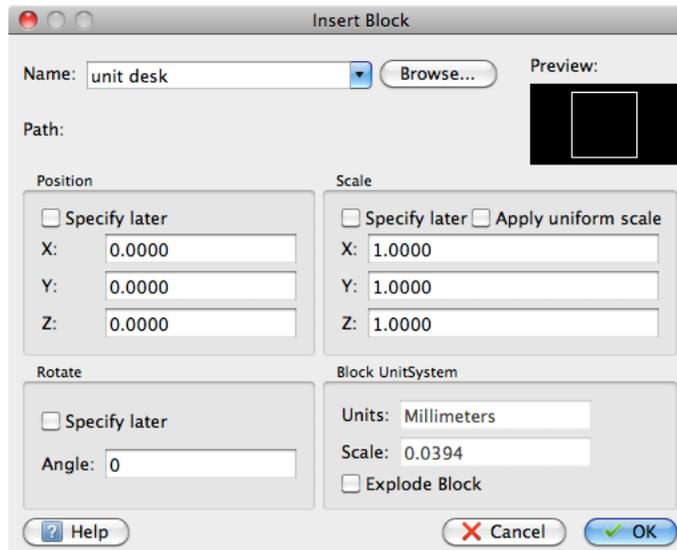
**Aliases:** i, insert

**Menu Bar:** Insert | Block

**Tool Matrix:** Insert | Insert Block

Blocks are placed in drawings with the InsertBlock command.

1. Start the **InsertBlock** command using one of the methods listed above. Notice the Insert Block dialog box.



This dialog box has many options, just like the Block Definition dialog box; as a minimum, there are only two bits of information that you must provide for iCADMac to insert a block:

- i. The name of the block to be inserted.
- ii. The position in the drawing at which to place the block.

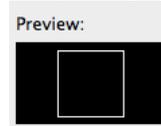
Since you are working with a unit block, you must also specify one more item:

- iii. The scale (size) of the block.

2. To specify the name of the block, choose “unit desk” from the **Name** droplist.



(When a drawing contains just one block, its name is automatically shown. When a drawing contains many blocks, the Preview image helps you distinguish them visually.)



**TIP** To insert any other drawing as a block, click **Browse**, and then choose a DWG or DXF file name in the Open dialog box. The drawing is treated just like a block. (The name of folder in which the file resides would be listed next to **Path**.)

3. To specify the position of the block, turn on the **Specify Later** option. This means that after you click OK, iCADMac will prompt you to for the block’s position.
4. To place this 1mm x 1mm unit block as a 600mm x 900mm desk, change the values in the **Scale** section:

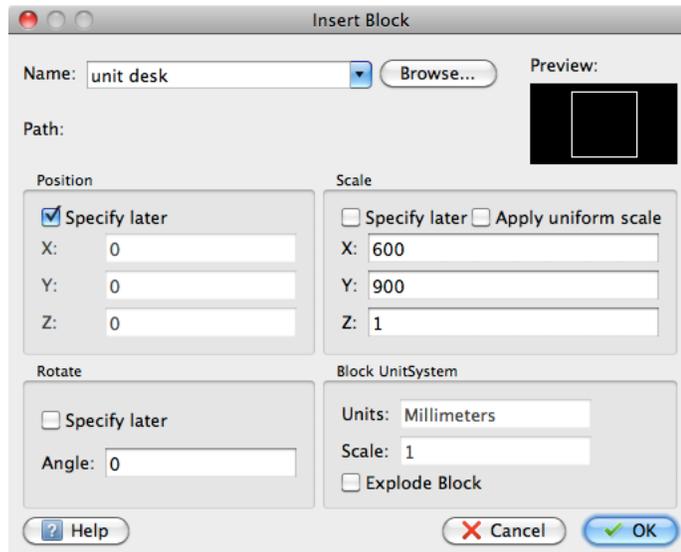
**X** — 600

**Y** — 900

**TIPS** To make the unit block smaller, enter values less than 1. For instance, scale factors of 0.5 make the block half as large.

You can enter negative numbers for the scales, such as X = -600. This mirrors the block about the x axis. (You cannot enter 0 as a scale factor.)

5. Leave all other options as are they are:
  - Apply Uniform Scale** — leave off. (When on, this option forces the Y and Z scale factors to be the same as X.)
  - Rotate Specify Later** — leave off. (When on, this option prompts you for the block’s rotation angle in the command window after exiting the dialog box.)
  - Angle** — leave at 0. (This is the value by which the block is rotated.)
  - Explode Block** — leave off. (When on, this option causes the constituent entities to be inserted, instead of the block itself; this is useful if you want to edit the block.)
6. At this point, the options in the dialog box should look like the ones illustrated below.

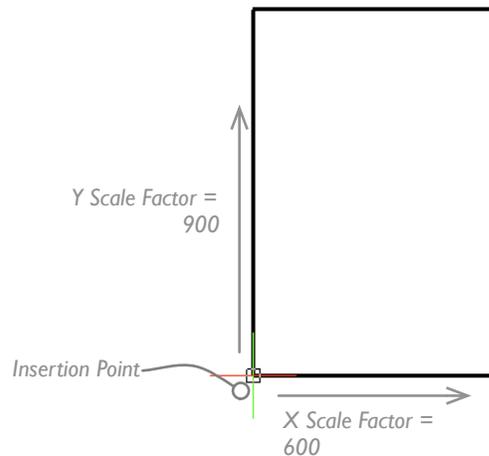


Click **OK**.

7. Notice that iCADMac prompts you in the command window:

Options: Angle, reference Point, uniform Scale, X, Y, Z or  
Specify destination» (Pick a point in the drawing.)

As you move the mouse, notice that the block moves around the drawing. The cursor is located at the block's insertion point, which you defined earlier with the MakeBlock command.



When you pick a point in the drawing, iCADMac places the block.

**TIP** You are prompted for a destination point, because you ticked that option in the Insert Block dialog box. No other prompts are displayed, such as scale factor or rotation angle, because you left those options turned off.

You can, however, override the defaults set in the dialog box by entering at the command prompt the names of the **Angle**, **reference Point**, **uniform Scale**, **X**, **Y**, and/or **Z** options.

8. Press **command+S** to save the drawing file.

With the block placed, the InsertBlock command ends. In the next tutorial, you draw an office chair block. Later in this chapter, you add the desk and chair blocks to the office layout.



## Chair Block

**Command:** Explode

**Alias:** x

**Menu Bar:** Modify | Explode

**Tool Matrix:** Modify | Explode

Unlike table symbols, chair symbols must be drawn full-size, because of their details. They don't work well as unit blocks: a chair inserted with x and y scale factors of 100 and 300 looks distorted. In this tutorial, you draw an office chair full-size, using some new commands.

1. Start a new drawing with the *standardiso.dwt* template file.

**TIPS** There is no need for the UCS icon, so turn it off with the **CsIcon Off** command.

The drawing is easier to see when lineweights are turned on, so click the **LWT** button on the status bar. The other status bar buttons can be left off, including Snap, Grid, Ortho, and ESnap.

### Drawing the Chair's Seat

2. Draw the seat of the chair as a rectangle between 0,0 and 330,330.

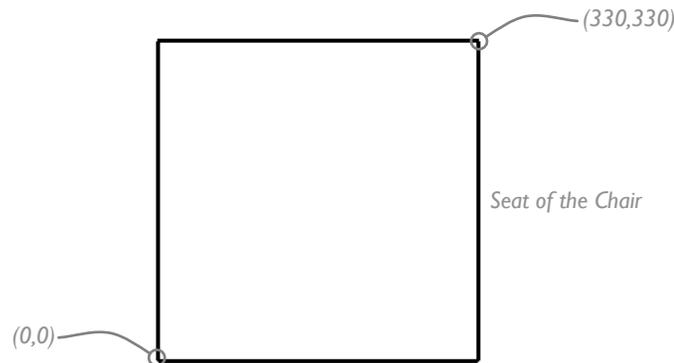
Command: rectangle

Options: Chamfer, Elevation, Fillet, Thickness, line Width or

Specify start corner» 0,0

Options: Area, Dimensions, Rotation or

Specify opposite corner» 330,330



3. Use the Zoom command twice to see the rectangle.
  - a. First, use the Fit option to have the rectangle fill the screen:
 

Command: z  
ZOOM Default: Dynamic  
Options: Bounds, Center, Dynamic, Fit, Previous, SElected, specify a scale factor (nX or nXP) or Specify first corner» f
  - b. Second, make the drawing 70% smaller to add some space around the rectangle:
 

Command: (Press spacebar to repeat the Zoom command.)  
ZOOM Default: Dynamic  
Options: Bounds, Center, Dynamic, Fit, Previous, SElected, specify a scale factor (nX or nXP) or Specify first corner» 0.7

4. You need to explode the rectangle to change its polyline into lines, so that they can be correctly offset later in this tutorial.
  - a. Start the **Explode** command using one of the methods listed above:
 

Command: explode
  - b. At the 'Specify entities' prompt, you select the rectangle. You can move the mouse over to pick it, but I find it is faster to enter 'l' to tell iCADMac to select the last-drawn entity.
 

Specify entities» l  
1 found, 1 total  
Specify entities» (Press return to end the command.)

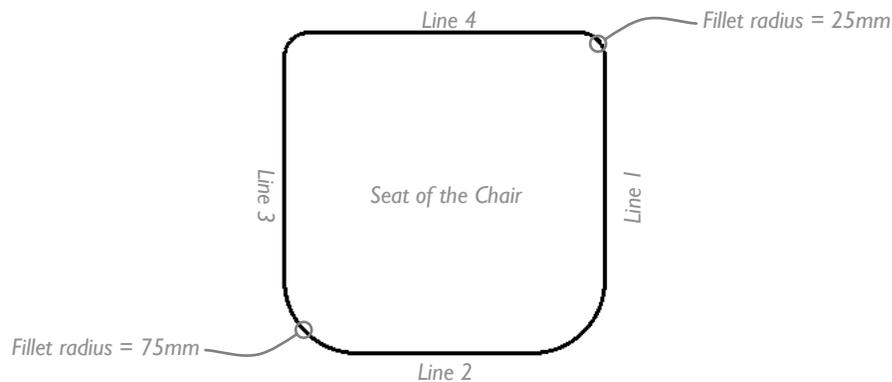
The rectangle looks no different after being exploded.



### Rounding Corners

**Command:** Fillet  
**Alias:** f  
**Menu Bar:** Modify | Fillet  
**Tool Matrix:** Modify | Fillet

Use the Fillet command to round the corners of the seat. In this tutorial, you apply two different radii to the corners of the seat, 75mm and 25mm.

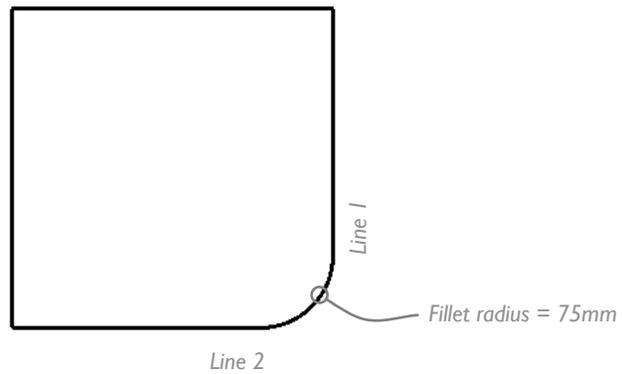


1. Enter the **Fillet** command. Notice that the default radius is 10mm:
 

Command: fillet  
Mode = TRIM, Radius = 10  
Options: Multiple, Polyline, Radius, Trim mode, Undo or
2. Enter 'r' to change the radius to 75 (mm):
 

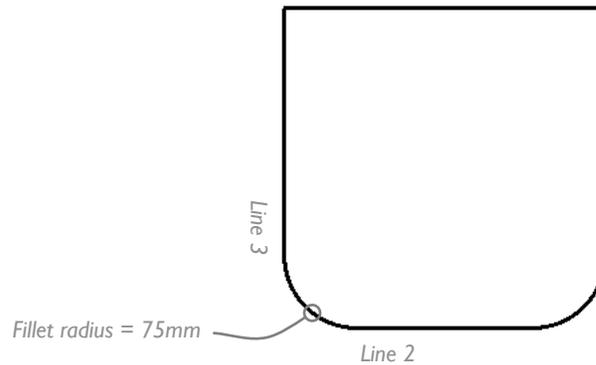
Specify first entity» r  
Default: 10  
Specify radius» 75
3. Select the two lines indicated by the figure:
 

Options: Multiple, Polyline, Radius, Trim mode, Undo or  
Specify first entity» (Pick line 1.)  
Options: Shift + select to apply corner or  
Specify second entity» (Pick line 2.)



4. The Fillet command ends automatically, and so you need to restart the command to place the next fillet. The easiest way to do this is to press the spacebar:

Command: (Press the spacebar.)  
 FILLET Mode = TRIM, Radius = 75  
 Options: Multiple, Polyline, Radius, Trim mode, Undo or  
 Specify first entity» (Pick line 2.)  
 Options: Shift + select to apply corner or  
 Specify second entity» (Pick line 3.)



5. The seat has two more fillets, but of a different radius. First, though, use the Multiple option to make the command repeat itself:

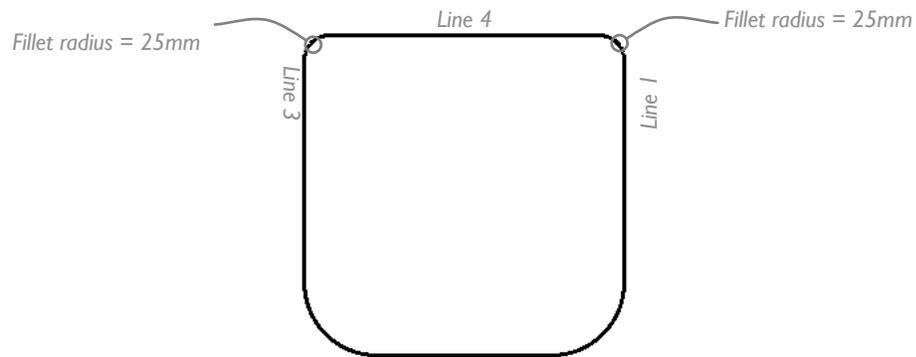
Command: (Press the spacebar.)  
 FILLET Mode = TRIM, Radius = 75  
 Options: Multiple, Polyline, Radius, Trim mode, Undo or  
 Specify first entity»

6. Use the Radius option to change the radius to 25mm:

Command: (Press the spacebar.)  
 Specify first entity» r  
 Specify radius» 25

7. Fillet the remaining corners:

Specify first entity» (Pick line 3.)  
 Specify second entity» (Pick line 4.)  
 Specify first entity» (Pick line 1.)  
 Specify second entity» (Pick line 4.)



8. Save the drawing as *office chair.dwg* with the SaveAs command.  
Command: saveas



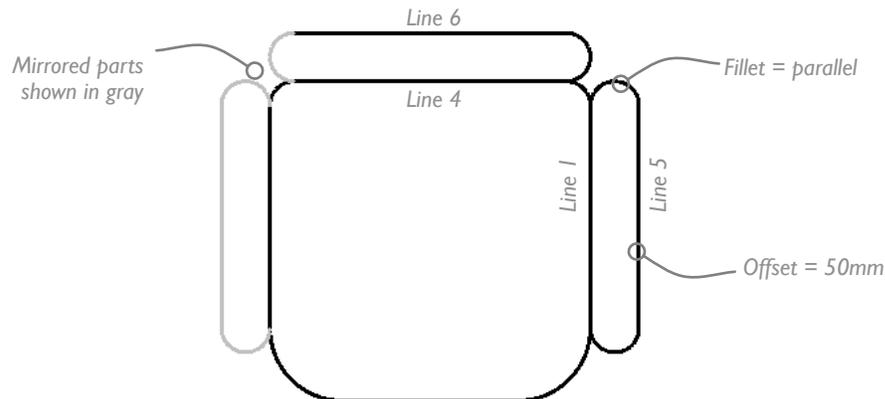
### Adding Arms and Backs

**Command:** Mirror  
**Alias:** mi  
**Menu Bar:** Modify | Mirror  
**Tool Matrix:** Modify | Mirror

The two arms and the back of the chair are drawn by offsetting existing lines, filleting parallel lines, and then mirroring some of the entities.

As you saw in an earlier chapter, the Offset command makes parallel copies. You use it here to draw the straight portions of one arm and the back.

1. The back of the chair and the arms are each 50mm wide. You can draw parts of them, and then add the rest by copying them with the Mirror command.



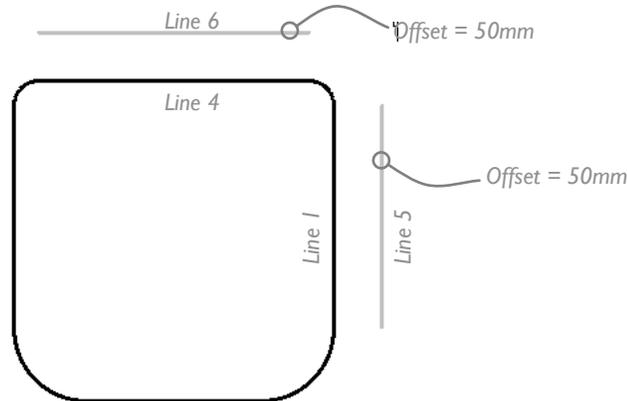
- a. To begin drawing the arms and back, start the **Offset** command to create offset parts of the chair seat:  
 Command: offset  
 Active settings: Delete source=No Layer=Source OFFSETGAPTYPE=0
- b. The default offset distance is 1, so change it to 50:  
 Default: 1  
 Options: Delete, Distances, destination Layer, Through point or Specify distance» 50
- c. Select line 1:  
 Default: Exit

Options: Exit, Undo or  
Specify source entity» (Pick line 1.)

- d. And then pick a point to the right:

Default: Exit  
Options: Both sides, Exit, Multiple, Undo or  
Specify side for destination» (Pick a point to the right.)

Notice that iCADMac makes the offset copy 50mm to the right.



- e. The Offset command repeats itself automatically, unlike the Fillet command. Show iCADMac where to place the second offset line:

Default: Exit  
Options: Exit, Undo or  
Specify source entity» (Pick line 4.)

Default: Exit  
Options: Both sides, Exit, Multiple, Undo or  
Specify side for destination» (Pick a point to the top.)

- f. Press **return** to exit the command:

Default: Exit  
Options: Exit, Undo or  
Specify source entity» (Press return.)

2. The curves of the arm and back are made with the Fillet command. In this case, you do not need to specify the radius, because iCADMac is filleting parallel lines; the distance between them determines the radius, automatically.

Command: fillet  
Mode = TRIM, Radius = 25  
Options: Multiple, Polyline, Radius, Trim mode, Undo or  
Specify first entity» (Pick line 1.)  
Options: Shift + select to apply corner or  
Specify second entity» (Pick line 5.)

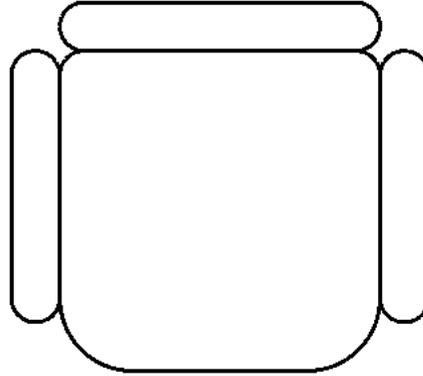
3. Repeat the Fillet command twice more to place two more fillets:

Command: (Press return to repeat the command.)  
FILLET Mode = NOTRIM, Radius = 25  
Options: Multiple, Polyline, Radius, Trim mode, Undo or  
Specify first entity» (Pick line 1.)  
Options: Shift + select to apply corner or  
Specify second entity» (Pick line 5.)

Command: (Press return to repeat the command.)  
Mode = NOTRIM, Radius = 25  
Options: Multiple, Polyline, Radius, Trim mode, Undo or  
Specify first entity» (Pick line 4.)  
Options: Shift + select to apply corner or  
Specify second entity» (Pick line 6.)



Confirm: Delete source entities?  
Specify Yes, or No» n



5. Press **command+S** to save the drawing file.

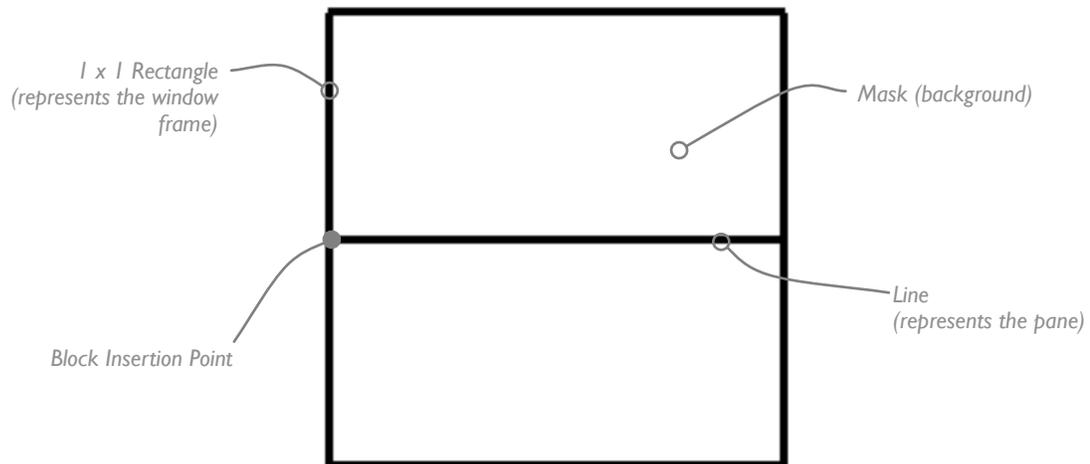
### Making Window and Door Blocks

Use the commands you have learned so far to create the window and door symbols for the walls.

Windows typically are drawn as simple blocks, especially for the fixed windows that are commonly used for offices. One line represents the pane of glass, and two more represent the sides. The block should be drawn as a unit block using the Line and Mask commands. (Use a mask to hide the wall lines.)

Follow these guidelines:

- i. To ensure the unit block is 1mm tall and wide, first draw a rectangle sized 1x1.
- ii. Convert the rectangle to a mask with the Mask command's Polyline option, and then send the mask to the background with the DisplayOrder command.
- iii. Use the Line command to draw the pane line.
- iv. Save the drawing as *window-block.dwg*.



**TIP** Always draw blocks on layer 0. This ensures blocks are inserted on the active layer of the drawing. (If you were to draw the block on any layer other than 0, such as “Blocks,” then iCADMac will always insert it on layer Blocks.)

When inserting the window block, use the x and y scale factors to size the window correctly. Remember that this office has walls of two thicknesses: 150mm for outer walls and 100mm for inner walls: This means that some windows are scaled 150x wide, while others are scaled 100x wide.

**X** — scale factor sizes the window for the specified width, such as 1200mm.

**Y** — scale factor makes the window deep enough for the wall (150 or 100mm)

**Rotate** — rotates the block by 90 degrees for walls running vertically.



Top: X scale factor = 150, Y scale factor = 1000.

Bottom: X scale factor = 100, Y scale factor = 1200.

## Drawing Door Symbols

**Command:** Arc

**Alias:** a

**Menu Bar:** Draw | Arc

**Tool Matrix:** Draw | Arc

**Command:** DisplayOrder

**Aliases:** dr, draworder

**Menu Bar:** Tools | Display Order

**Tool Matrix:** Modify | Display Order

Door blocks are drawn from lines, and employ arcs to indicate the door swing. The use of an arc, however, prevents us from drawing doors as unit blocks, because circular arcs end up looking elliptical when inserted into drawings with unequal x and y scale factors.

The workaround, unfortunately, is to draw a door block for every size and type of door. Typical sizes include a 750mm width for regular doors, and 650mm width for narrow doors. Among door types, there are bi-fold doors that are drawn like Ws.

Doors swing in and out; this can be shown by flipping the block using a negative scale factor. Below, I illustrate some door styles.

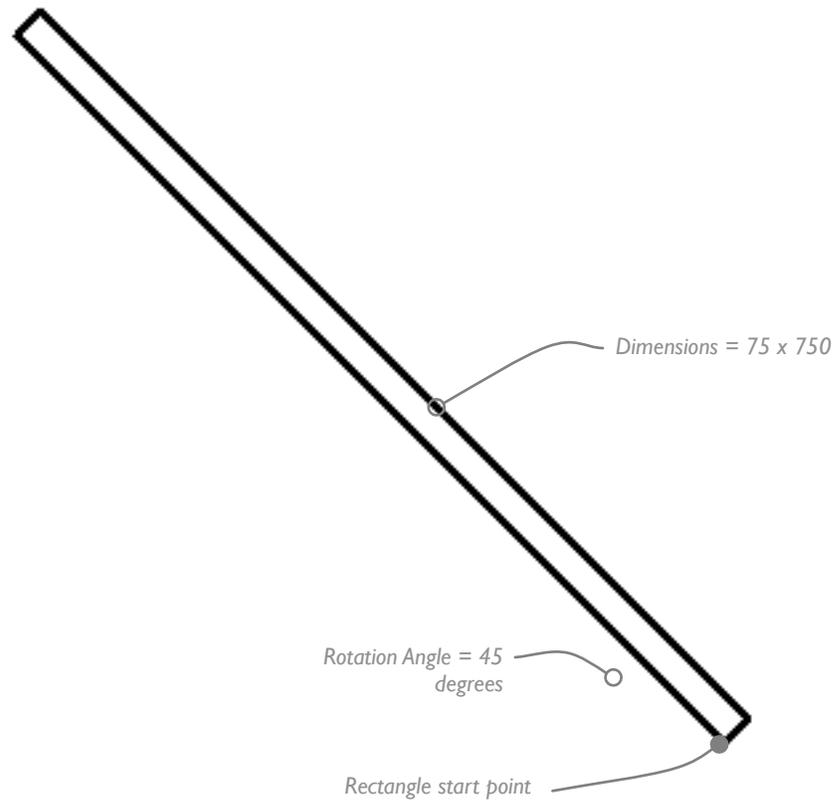
Follow these guidelines for drawing door blocks:

1. To start drawing a door symbol, begin with **Rectangle** command:

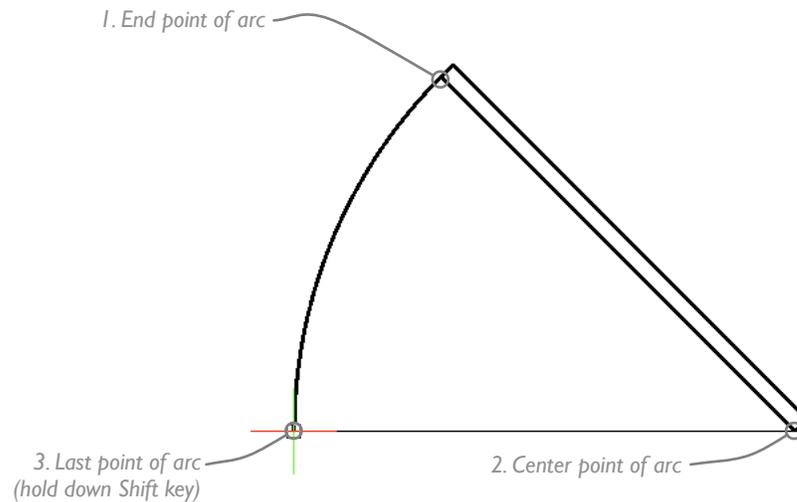
Command: rectangle

Options: Chamfer, Elevation, Fillet, Thickness, line Width or

Specify start corner» (Pick a point in the drawing.)



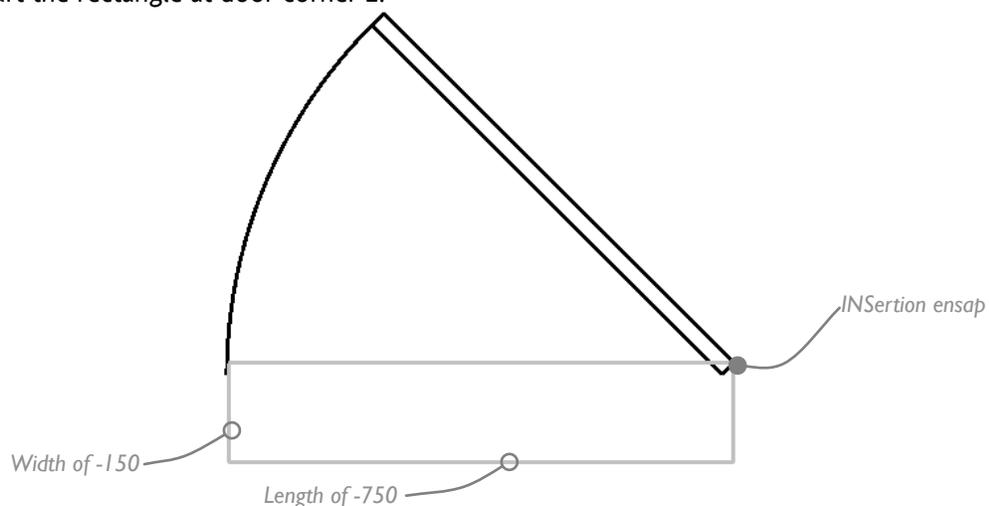
- a. To show the door in a partially open position, use the **Rotation** option to rotate the rectangle by 45 degrees:
    - Options: Area, Dimensions, Rotation or
    - Specify opposite corner» r
    - Default: 0
    - Options: Pick points or
    - Specify rotation» 45
  - b. Draw the door with the command's **Dimensions** option, specifying a thickness of 25mm and a width of 750mm:
    - Options: Area, Dimensions, Rotation or
    - Specify opposite corner» d
    - Default: 10
    - Specify horizontal dimension» 25
    - Default: 10
    - Specify vertical dimension» 750
2. Draw the door swing with the **Arc** command:
- Command: arc
- a. To specify the arc's start point accurately, use **ENDpoint** entity snap:
    - Options: Center, Append, Enter to continue from last point or
    - Specify start point» end
    - of (Pick corner #1 of the rectangle.)



- b. For the next point of the arc, use the command's **Center** option, and then use **END-point** entity snap to position it:
  - Options: Center, End or
  - Specify through point» c
  - Specify center point» end
  - of (Pick corner #2 of the rectangle.)
- c. For the third point of the arc, turn on **Ortho** mode to position the point exactly across from the arc's center point:
  - Options: Angle, chord Length or
  - Specify end point» (On the status bar, click Ortho. In the drawing, click at point #3 to position the end of the arc.)
3. Place a second rectangle in the wall position. It will use masking to blank out the section of wall where the sill is located.

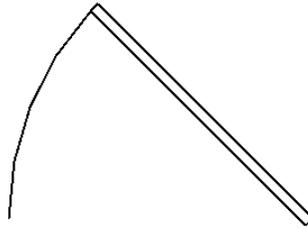
For 150mm-thick walls, dimension the rectangle as Horizontal = -750 and Vertical = -150; use negative dimensions to draw the rectangle to the left and downwards.

Remember to reset the Rotation parameter to 0 degrees. Use **INSertion** entity snap to start the rectangle at door corner 2:



4. Convert the rectangle to a mask with the **Mask** command's **Polyline** option.
5. The mask is on top of the door, obscuring it partially. Use the **DisplayOrder** command to move the mask visually below the door with the **Back** option:
  - Command: displayorder

Specify entities» (Select the mask.)  
 Specify entities» (Press return to end entity selection.)  
 Default: in Back  
 Options: Above entities, in Back, in Front, or Under entities  
 Specify option» b



5. Save the drawing as *door-100x750.dwg*, which identifies it as a door that fits 100x750mm openings.
6. Using the same procedure, create additional door blocks for 150mm-thick walls and of varying widths.

## Placing Furnishings

**Command:** Pattern  
**Aliases:** pat, ar, array  
**Menu:** Modify | Pattern  
**Tool Matrix:** Modify | Pattern

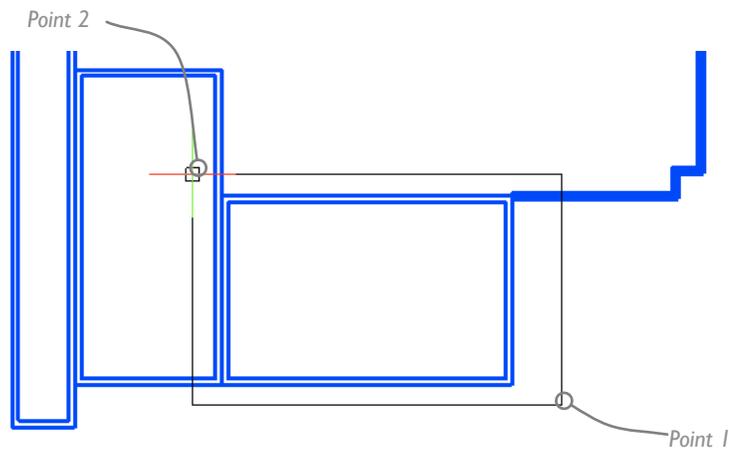
You have created two furniture symbols, one of a table that can be used for any size of table, and one of a chair. Let's place them into the office drawing. (Later, you add the doors and windows.)

In this tutorial, you populate the conference room with a conference table and several chairs.

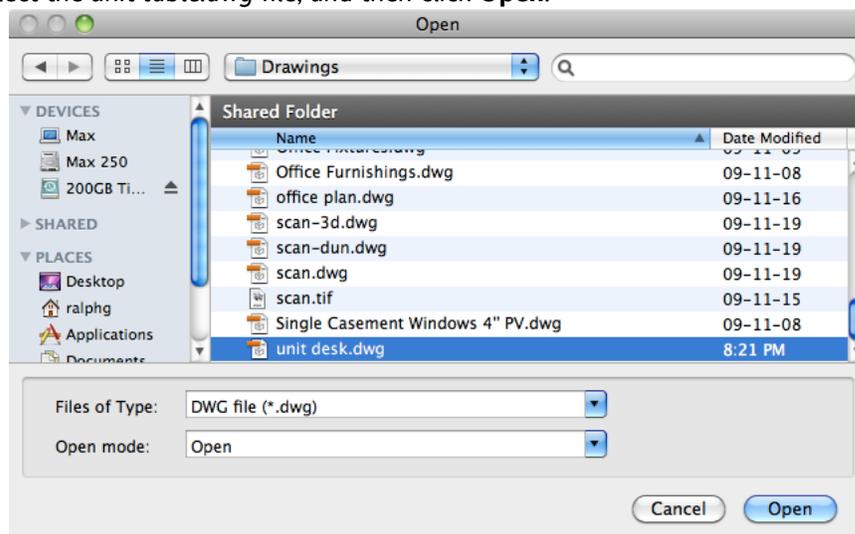
1. Open the *office plan.dwg* drawing file in iCADMac.
2. Change the active layer to **Furnishings** with the Properties palette.
3. Use the **Zoom Window** command to get a closer look at the conference room, as follows:
  - Command: z
  - ZOOM Default: Dynamic
  - Options: Bounds, Center, Dynamic, Fit, Previous, SElected, specify a scale factor (nX or nXP) or
  - Specify first corner» w

The Window option requires you to pick two points, forming a rectangle. The rectangle specifies the zoomed-in view:

Specify first corner» (Pick point 1.)  
 Specify opposite corner» (Pick point 2.)

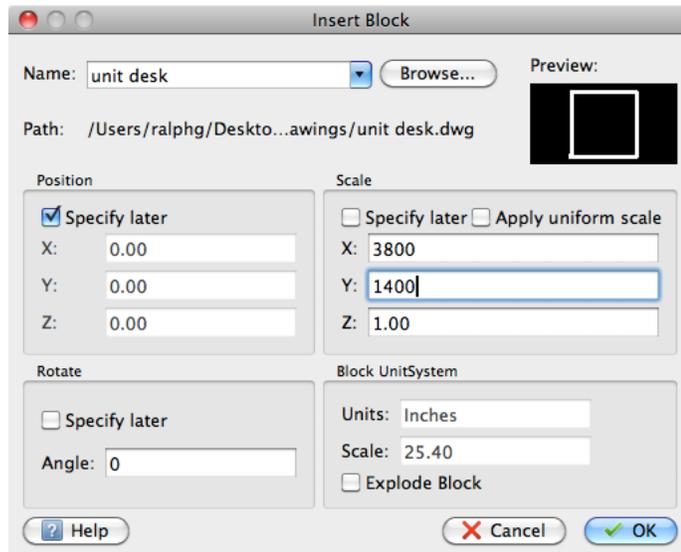


4. Recall that blocks are placed with the InsertBlock command. Place a conference table sized 3800mm x 1400mm by following these steps:
  - a. Start the **InsertBlock** command.
  - b. You need to access the “unit table” block stored in the *unit table.dwg* drawing file. To do so, click **Browse**.
  - c. Select the *unit table.dwg* file, and then click **Open**.

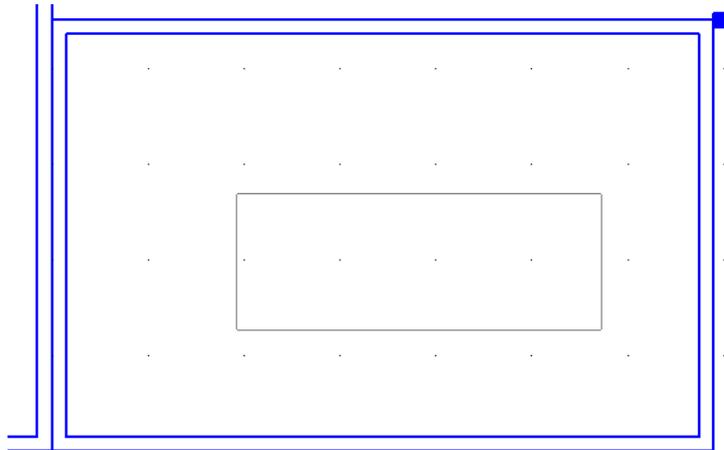


- d. Back in the Insert dialog box, set the following options:

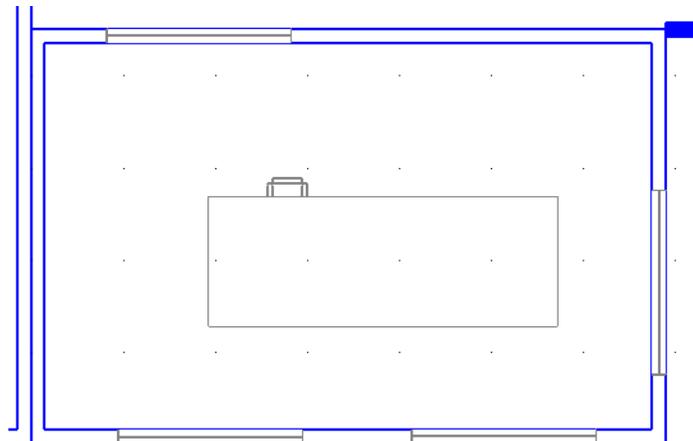
<b>Position</b>	Specify later
<b>Scale</b>	X = 3800
	Y = 1400
<b>Rotate</b>	0



- e. Click **OK**.
- f. iCADMac prompts you in the Command window:  
Specify destination» (Pick a point inside the conference room.)



5. Place the other blocks, such as chairs around the table, and doors and windows in the walls.



6. You can use the Insert command repeatedly to place more chairs. A more efficient method is to use the Pattern command, which makes multiple copies in a line, in an area, or around a circle. Follow these steps to add three chairs:
  - a. Start the **Pattern** command using one of the methods listed above. Notice the Pattern dialog box.
  - b. Choose the following options:

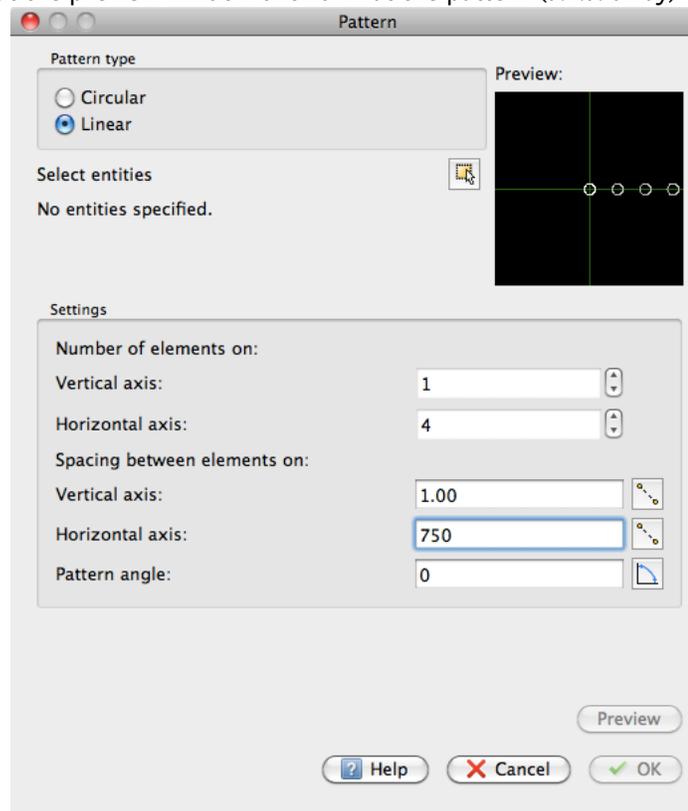
<b>Pattern Type</b>	Linear
<b>Number of Elements On:</b>	
<b>Horizontal Axis</b>	1
<b>Vertical Axis</b>	4

**TIP** When you want three copies, you enter 4 for the number of elements. That's because iCAD-Mac counts the first element as the one already in the drawing — just like entering 1 for elements on the horizontal axis.

**Spacing Between Elements On:**

<b>Horizontal Axis</b>	1	<i>(doesn't matter)</i>
<b>Vertical</b>	750	<i>(millimeters)</i>
<b>Pattern Angle</b>	0	

Notice that the preview window shows what the pattern (a.k.a. array) will look like.



- c. Click the **Select Entities** button. At the prompt, select the chair block:
  - Specify entities» (Select the chair block.)
  - 1 found, 1 total
  - Specify entities» (Press return to return to the dialog box.)
- d. Click **OK** to add the chairs.

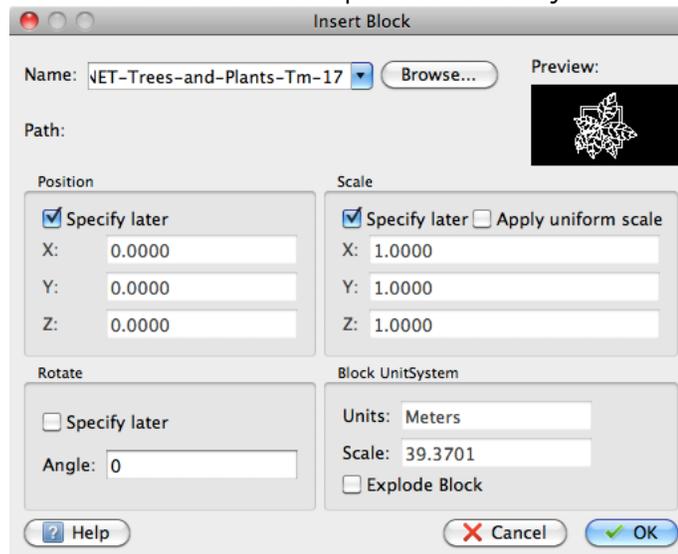


7. If necessary, use the **DisplayOrder** command to place the chair under the table.
8. Save your work.

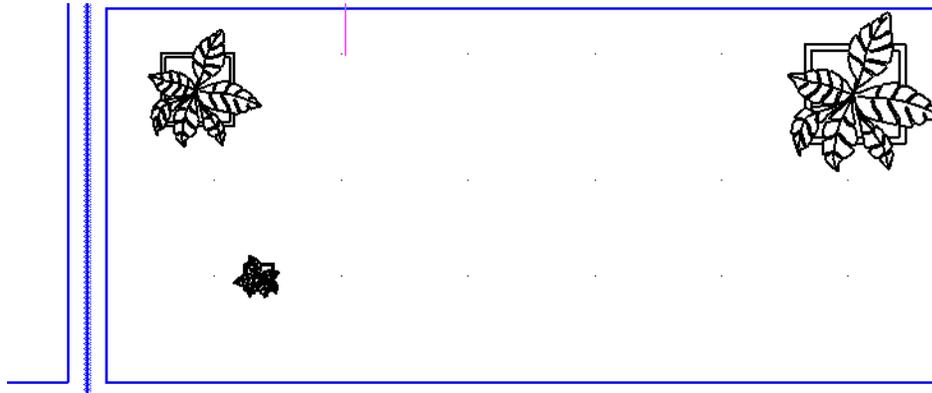
In this tutorial, you insert the downloaded block into the office plan drawing.

1. Switch to the office plan drawing.
2. Enter the **InsertBlock** command, and then click **Browse** to choose the downloaded file.
3. Set **Position** to “Specify Later.” This lets you position the plant anywhere in the office.

Set **Scale** to “Specify Later” and “Apply Uniform Scale.” These let you size the plant to suit, and to ensure the block maintains its aspect ratio correctly.



4. Click **OK**. Notice the prompts in the command window:
  - a. At the ‘Specify destination’ prompt, pick a point in the drawing to place the plant.  
Options: Angle, reference Point, uniform Scale or  
Specify destination» (Pick a point in the drawing.)
  - b. At the ‘Specify scale’ prompt, drag the cursor. Notice that the plant becomes larger.  
When its size suits you, click. This positions the plant block.  
Default: 1  
Specify scale» (Drag the cursor, and then click.)
5. Repeat the command (by pressing the **spacebar**) to place several more plants, each in a new location and at a different size (scale factor).



6. Save the drawing.

**TIP** You might come across two similar looking terms, block definition and block reference. Here's the difference between them:

- When a symbol is defined as a block by the Block command, it becomes a “block definition”; block definitions are invisible.
- When the block definition is inserted into the drawing with the InsertBlock command, the block you see is known as the “block reference.”

Here are some tips for placing blocks in the floor plan drawing:

- Doors should swing out from the inside. This is sometimes a fire code requirement, so that it is easier for people to flee a damaged building.
- Use the **InsertBlock** command's **Rotate** option to place blocks at 90-degree increments, such as the doors illustrated below.
- There should be windows in all of the walls of the office, especially in the executive office and the conference room.
- There should be no windows in the storage room, for reasons of security.
- The window block is a unit block; when inserting it, use these scale factors and rotation angles:

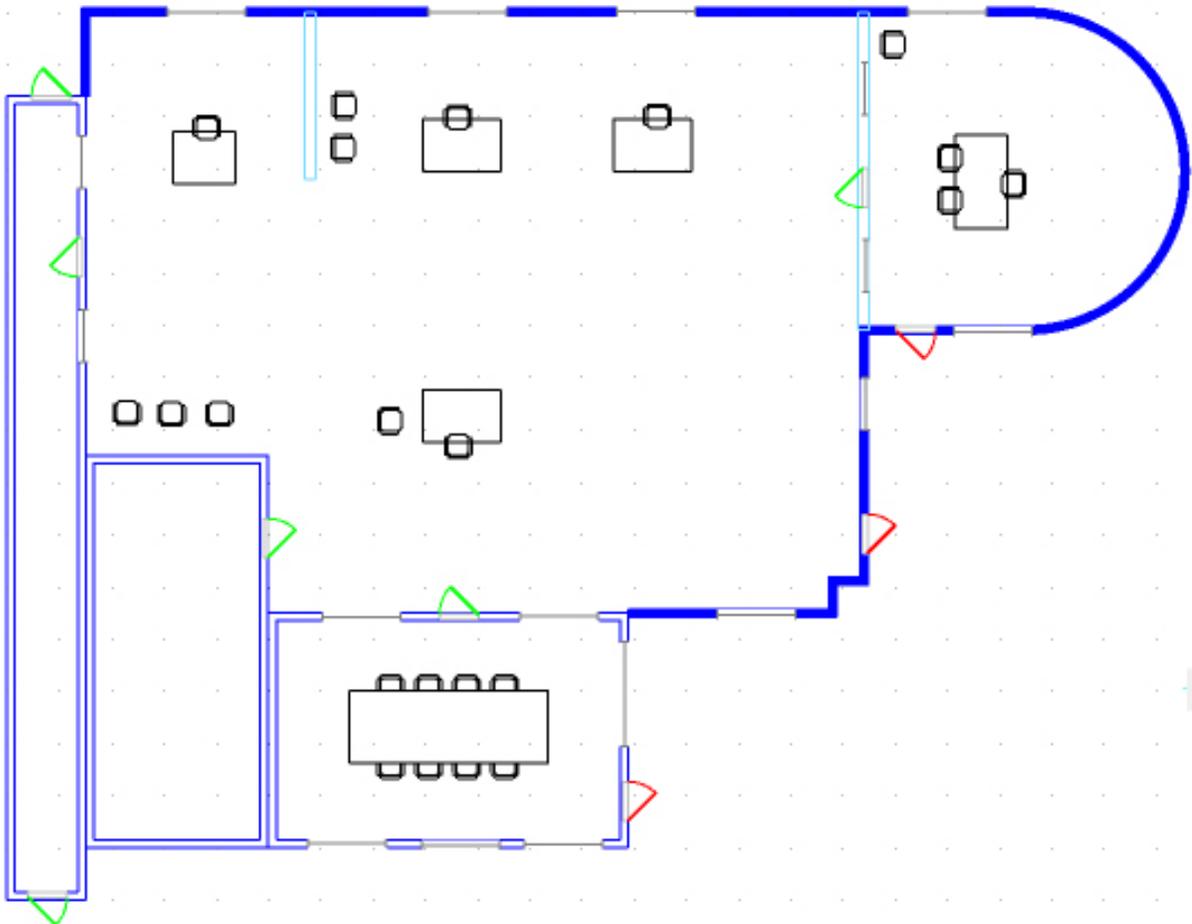
<b>Orientation</b>	<b>Rotation</b>	<b>X Scale Factor</b>	<b>Y Scale Factor</b>
Horizontal	0 degrees	Window width, like 1000mm	Wall width (100 or 150mm)
Vertical	90 degrees	Wall width of 100 or 150mm	Window width (1000mm)



Left: Horizontal window at 0 degrees, 1000 x 100mm

Right: Vertical window at 90 degrees, 100 x 1000mm

- Place desks in the executive office, the reception area, and elsewhere.
- Add at least one chair per desk.
- To place the same block many times, use the Copy or Pattern command. The Copy command is better for placing chairs and windows all over the drawing; the Pattern command is useful for placing chairs in a uniformly-spaced row.



## Extracting Blocks From Drawings

**Command:** ExportDrawing  
**Aliases:** wblock, w, dwgout, exportdwg  
**Menu Bar:** File | Export | Export DWG  
**Tool Matrix:** Insert | Export Drawing

When you create blocks in a drawing like *office plan.dwg*, you might come across this problem: how to use the blocks in another drawing? You cannot use the InsertBlock command, because it would insert the entire office floor plan as a block.

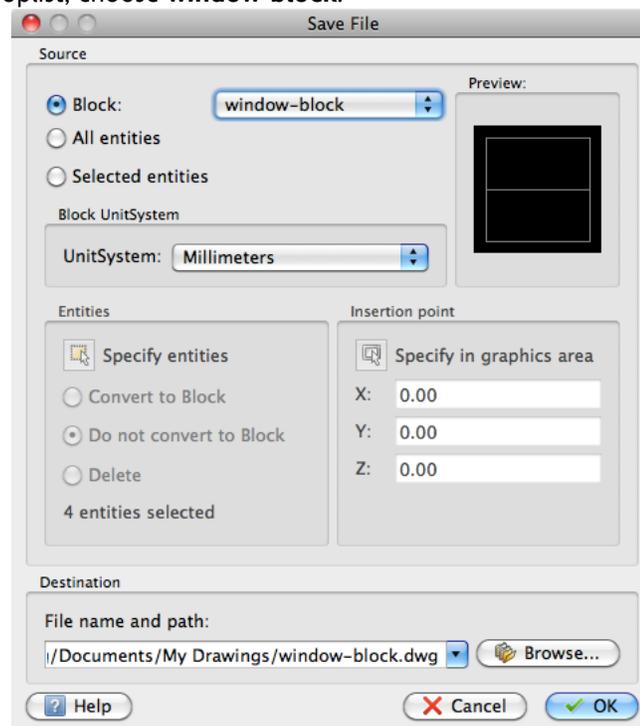
The solution is to extract the blocks into their own DWG files with the ExportDrawing command. This flexible command lets you extract individual entities, blocks, or the entire drawing (**All Entities** option).

### Export Block

---

In this tutorial, you extract the window-block block.

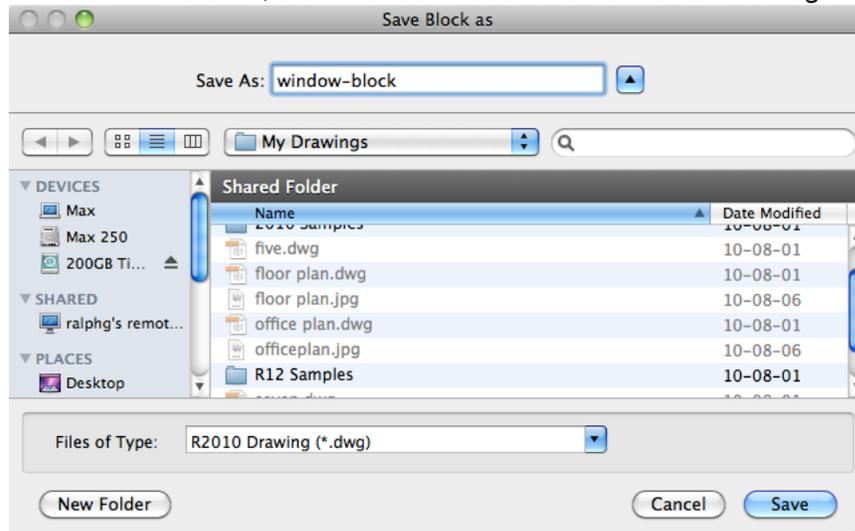
1. Enter the **ExportDrawing** command using one of the methods listed above. Notice the Save File dialog box.
2. Enter the following options to export the window block:
  - a. In the Source section, choose **Block**.
  - b. From the droplist, choose **window-block**.



3. The name of the exported drawing file does not need to match the name of the block. In fact, iCADMac gives the exported block a generic name, such as “NewBlock.Dwg.”

To change the name and to specify the folder in which to store the exported block, follow these steps:

- a. In the Destination section, click **Browse**. Notice the Save Entities As dialog box.



- b. If you wish, change the file name from “NewBlock.Dwg” to **Window-Block-Dwg**.
  - c. Choose the folder in which to store the exported block.
  - d. Click **Save** to return to the Save File dialog box.
4. Click **OK**.

iCADMac appears to do nothing, but when you check the folder, you should find the exported block under the name of *Window-Block.Dwg*.

You can now use the **InsertBlock** command’s **Browse** option to select and insert the file into other drawings.

## iCADLib Blocks Manager

The iCADLib Module enables you to collect in an organized way a series of blocks.

iCADMac includes more than 20,000 default blocks (architectural, electrical, mechanical, etc. ...).

In addition you can insert your own personal folders or download millions of blocks from the Traceparts and Cadenas websites.

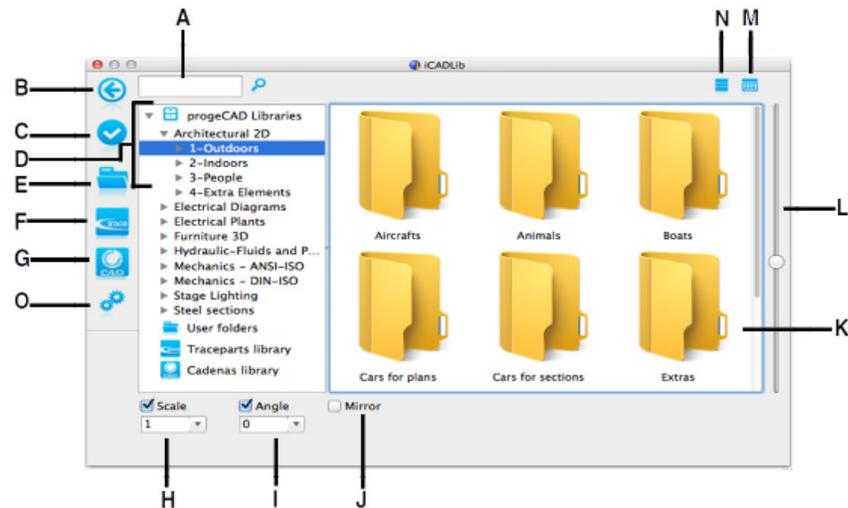
### Working with iCADLib

The iCADLib Module lets you collect a series of blocks in an organized way.

The function activates typing iCADLib and pressing Enter, via the Add-on menu or using the appropriate icon.

The blocks libraries can be arranged via the progeCAD Libraries, the User folder, the Traceparts and the Cadenas Libraries.

You can resize the New Library Manager form moving the mouse pointer to a corner and dragging it, another way is to choose to see only the main commands by clicking on the Reduce / Enlarge window.

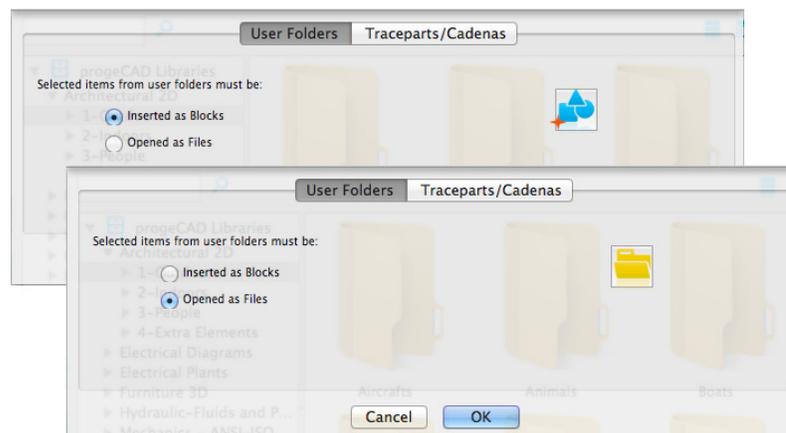


- A Write the name of the block to display it in the space next to it.
- B Click to Enlarge/Reduce iCADLib window.
- C Click to insert a block.
- D Click a library to display the categories and the contained blocks.
- E Click to choose a folder.
- F Click to visit the Traceparts website.
- G Click to visit the Cadenas website.
- H Specify the desired scale.
- I Specify the desired angle.
- J Click to enter the mirrored block.
- K Show the block preview.
- L Click to modify the preview zoom.
- M Click to show the block preview as icon.
- N Click to show the block preview as list.
- O Click to change the iCADLib settings.

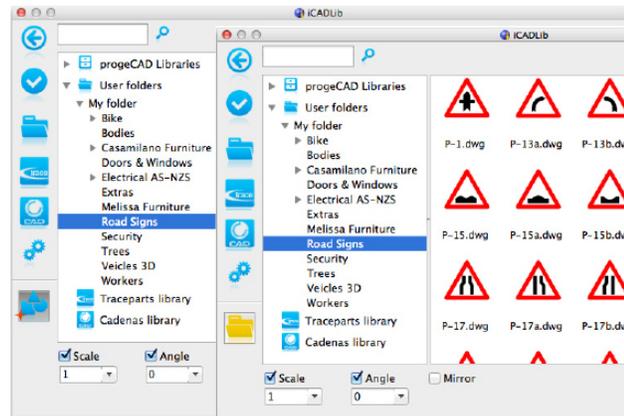
### Changing iCADLib settings

By clicking on Settings icon you can choose how to modify the User folder and the Traceparts/Cadenas Setting.

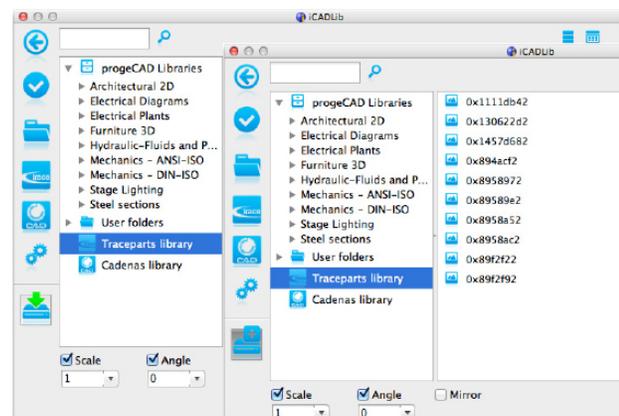
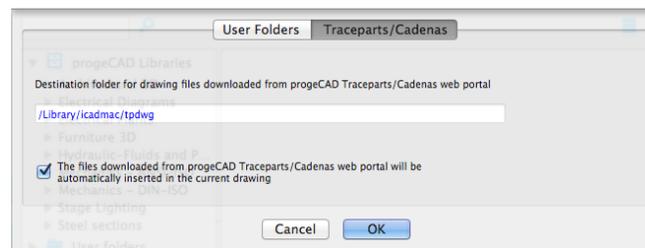
In the User folder tab you can choose if you want to insert the item you select as a block or open it as a file. You can click both on the text or on the icon on the right.



Alternatively you can click on the Insert as Block / Open as File button on the left, after selecting User Folders from the Libraries section.

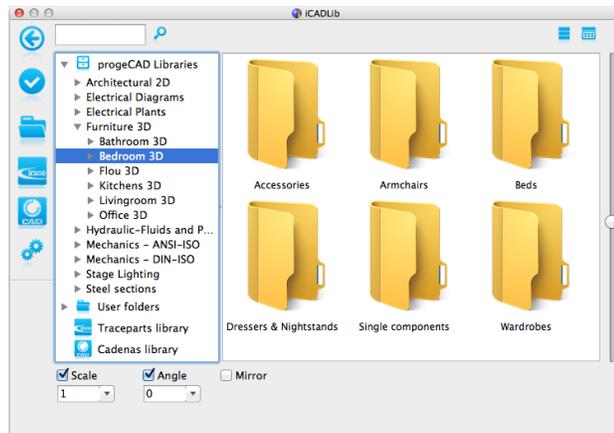


In the Traceparts/Cadenas tab you can choose a destination folder where to put the file downloaded from the Internet and if it will be automatically sent to iCADMac.



## progeCAD Libraries

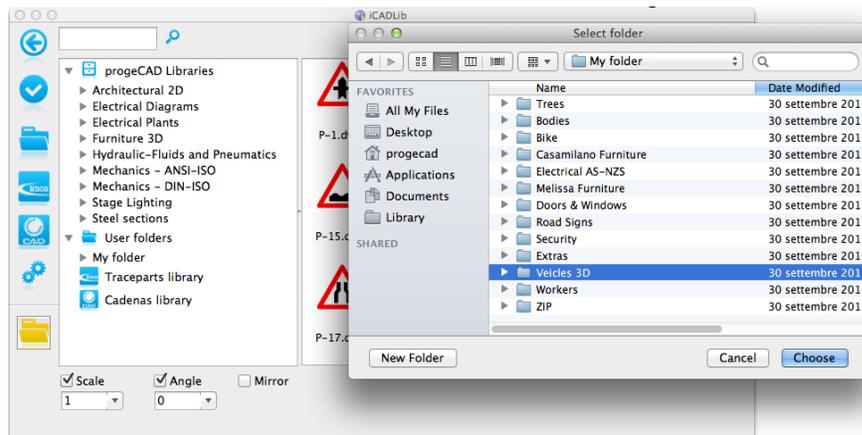
In the ProgeCAD Libraries section you find the default blocks already present when you install the software. These are organized in libraries (eg: Architectural, Electrical, Mechanical ...), categories (outdoors, indoors, people) and blocks. To insert a block into a drawing you need to select it and press the Insert icon or, simply, you can drag and drop it into the drawing.



NOTE: In the progeCAD Libraries you may not insert personal blocks.

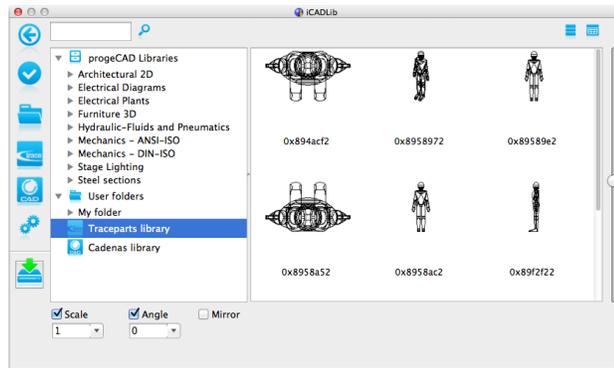
## User folder

In the User Folder section you can import personal folders containing blocks. To insert a new folder, you have to click on the Select Folder icon and navigate to it. The new library structure will reflect that of the selected folder. Each time you start iCADMac it will be possible to choose whether to delete or reload a previously imported library. You can import dwg and dxf format drawings to be used as blocks.



## Traceparts/Cadenas libraries

Clicking on the “Traceparts by progeCAD” or “Cadenas by progeCAD” icon you will be directed automatically to the homonymous sites from which you can download the blocks that you will find, after a completely free of charge registration. You will download a zip format file, you could save it everywhere. If you save it in the folder that you set in Settings section it will automatically unzip and add to the Traceparts or Cadenas library. Each time you start iCADMac it will be possible to choose if reload previously imported blocks.



## Summary

The use of blocks is a powerful way to increase your productivity by reducing the amount of drafting needed to complete a drawing. In the next chapter, you learn how to annotate drawings.

## Chapter 8

# Text and Attributes

### In This Chapter

- Placing notes in drawings
- Creating text styles
- Editing text
- Adding attributes to blocks

**Notes are added** to drawings to explain the designer's intent. For instance, it is common to label the rooms of floor plans with text. These text labels are also known as “callouts” or “annotations.” iCADMac supports two types of text — simple and paragraph:

**Single line text** consists of a single line of words, sometimes called a “string.” It is placed with the `-SimpleNote` command.

**Multiline text** consists of multiple lines of text arranged like paragraphs, sometimes called a “block” of text; it is placed with the `Note` command.

Make a mistake? You can edit simple and multiline text with the `EditAnnotation` command, and you can use the `SpellCheck` command to find *missteaks* you may have missed.

The look of text is controlled by *styles*, which specify the font and all other aspects of text.

In addition, it is common to attach text data to blocks so that the blocks can be counted by a spreadsheet.

In this chapter, you place text in the *office layout.dwg* drawing, and then change its look through styles. In addition, you learn how to attach attribute data and then export them to a spreadsheet for further analysis.

## Key Terms in This Chapter

**Annotation** — refers to notes placed in drawings.

**Attribute** — refers to textual data included with blocks.

**Attribute Definition** — defines attribute data.

**Attribute Data** — stores data attached to blocks.

**Bounding Box** — specifies the margins of paragraph text.

**Extract** — exports attribute data to a file.

**Fixed Attribute** — refers to a constant attribute value.

**Justification** — refers to horizontal positioning of text, such as left, right, or center.

**Name** (or Tag) — identifies attributes by name.

**String** — refers to a single line of text.

**Style** — specifies the font and other properties of text.

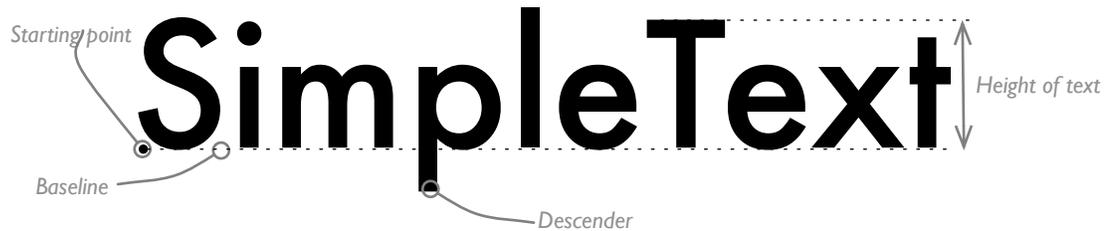
## Commands

<b>Command</b>	<b>Shortcuts</b>	<b>Menu Selection</b>
BlockAttributeOutput	attout	Tools   Attribute Export
EditAnnotation	ed	Modify   Object   Text
EditBlockAttribute	attedit	Modify   Object   Attribute   Single
EditNote	mtedit	Modify   Object   Text
MakeBlockAttribute	attdef	Draw   Block   Define Block Attributes
Note	t, mtext	Draw   Text   Multiline Text
-SimpleNote	text	Draw   Text   Single Line Text
TextStyle	style	Format   Text Style

## T Placing Simple Text

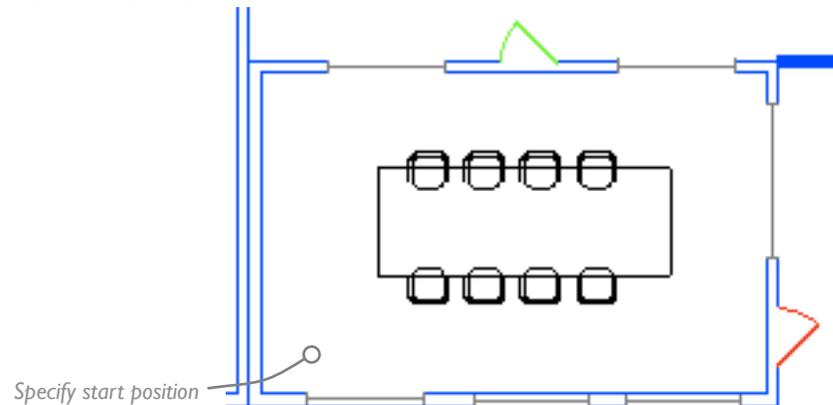
**Command:** -SimpleNote  
**Aliases:** text, dtext, snote, dt  
**Menu Bar:** Draw | Text | Single line text  
**Tool Matrix:** Draw | SimpleNote

In this tutorial, you label the reception area and the storage room with text. The figure below points out the most important parts of text:



Follow these steps:

1. Start the **-SimpleNote** command using one of the methods listed above.
2. In the command bar, iCADMac prompts you to pick a point in the drawing:  
 Command: -simplenote  
 Options: Justify, textSTyle or  
 Specify start position» (Pick a point in the drawing.)
3. Pick a point below the conference table.



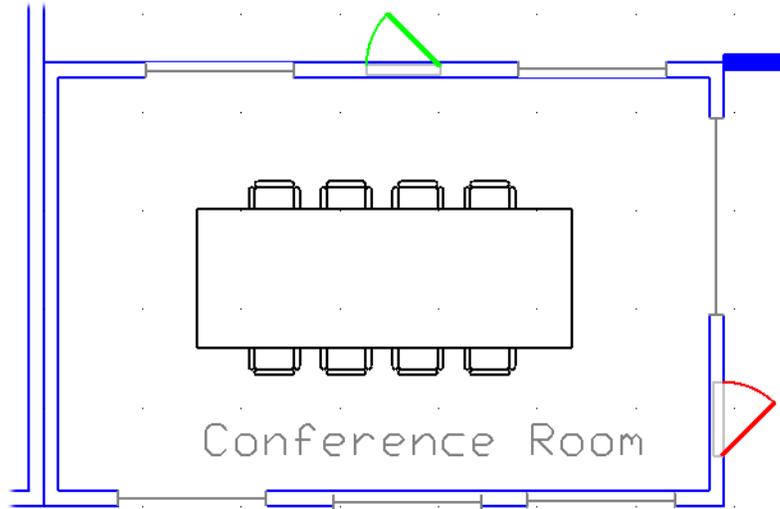
**TIPS** You may wish to zoom into the room to see it more clearly. You can zoom and pan during commands, such as during the -SimpleNote command, like this: just roll the mouse wheel until the drawing is a good size.

The dash prefix in -SimpleNote means that it operates in the Command window; the related SimpleNote command displays a dialog box.

4. iCADMac would like to know the height of the text. You can enter a height in millimeters, but for this tutorial instead pick two points:  
 Default: 0.20

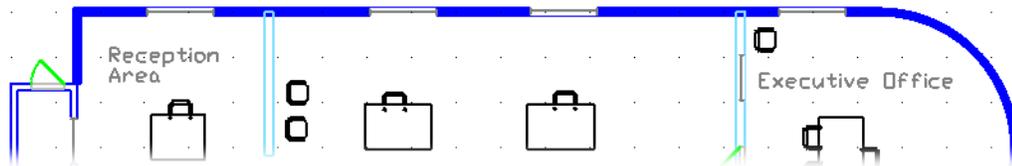
Specify height» (Pick two points to indicate the height.)

5. Text can be placed at any angle. For horizontal text, use the default value of 0:  
Default: 0  
Specify text angle» (Press return to accept the default angle.)
6. Finally, you get to specify the text. Type “Conference Room,” and then press **return**.  
Specify text» Conference Room (Press return to indicate the end of the line of text.)



7. To end the command, press **return** one more time:  
Specify text» (Press return to end the command.)

Go ahead, and repeat the **-SimpleNote** command to place text in the reception area and executive office, as show below.



You will find yourself pressing **return** repeatedly(!) for different reasons, as listed below.

- Pressing **return** at the ‘:’ prompt repeats the command:  
: (Press return to repeat the command.)  
Options: Justify, textStyle or  
Specify start position» (Pick a point in the reception area.)
- Pressing **return** at the following prompts accepts the default values stored by iCADMac:  
Default: 300.0  
Specify height» (Press return to accept default height.)  
Default: 0  
Specify text angle» (Press return to accept default angle.)
- Pressing **return** indicates the end of the line of text:  
Specify text» Reception (Press return to end line of text.)  
Specify text» Area (Press return to end line of text.)
- Pressing **return** ends the command:  
Specify text» (Press return to end the command.)

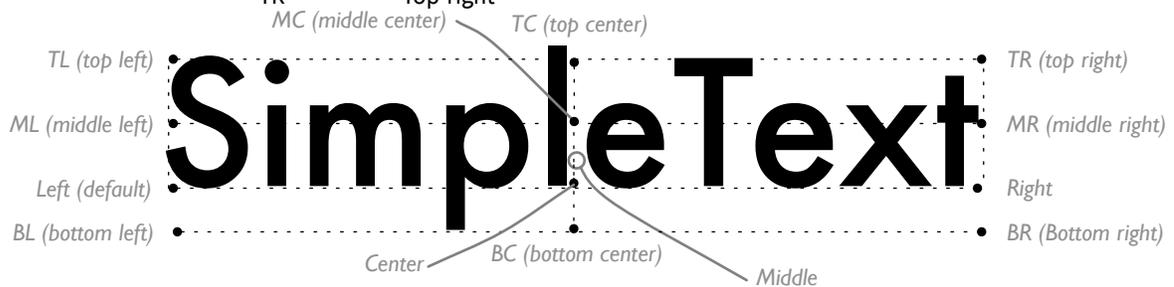
## Summary of -SimpleNote Command Options

Command: -simplenote  
 Options: Justify, textSTyle or  
 Specify start position»  
 Specify height»  
 Specify text angle»  
 Specify text»

**Justify** specifies the text justification:

Options: Align, BC, BL, BR, Center, Fit, Middle, MC, ML, MR, Right, TC, TL, or TR  
 Specify option» (Enter an option.)

Justify	Meaning
Align	Text fitted between two points; height is proportional to length of text
BC	Bottom center (See figure below.)
BL	Bottom left
BR	Bottom right
Center	Center of the text's baseline
Fit	Text fitted between two points
Middle	Center of the text's bounding box
MC	Middle center
ML	Middle left
MR	Middle right
Right	Right end of the text's baseline.
TC	Top center
TL	Top left
TR	Top right



**textSTyle** specifies the name of a style predefined by the TextStyle command:

Default: Standard  
 Options: ? to list or  
 Specify name» (Enter the name of a style, or type ? to list the styles stored in this drawing.)

See TextStyle command.

## Settings Text Styles

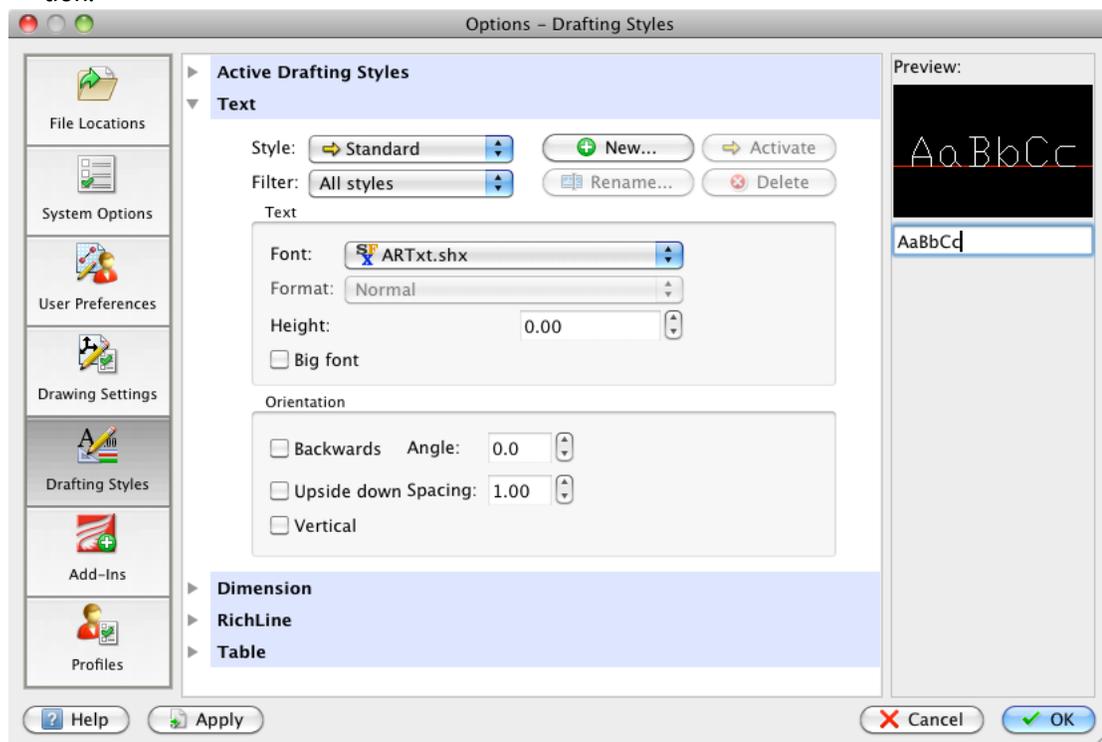
**Command:** TextStyle  
**Aliases:** st, style, txs  
**Menu Bar:** Format | Text Style  
**Tool Matrix:** Text | Text Style

The text is being drawn with a font named “Artxt,” a stick font that looks somewhat ugly. This font was popular in the early era of CAD because it could be displayed and printed quickly by the slow computers of the day, for it contains no curves. Today, computers are fast and so it is easy for them to display and print high quality fonts. iCADMac can use any font found on your computer.

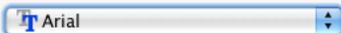
In CAD you do not, however, use fonts directly, as you would in a word processor; instead, you define fonts through text *styles*, which also define other properties, such as font orientation (whether the text should be drawn backwards or upside down). Text styles are defined through the TextStyle command. (iCADMac uses styles for other aspects of drawings, such as dimensions and tables.)

In this tutorial, you change the font from Artxt to Arial.

1. Start the **TextStyle** command. Notice that the Options dialog box opens to the Text section:



2. You can ignore all the text style properties, with the exception of the Font:
  - a. Click the droplist next to **Font**.
  - b. Scroll through the list of font names until you come to Arial.
  - c. Select **Arial**.

Font: 

## Summary of TextStyle Dialog Box Options

Command: `textstyle`

**Style** lists the text styles stored in the active drawing.

**New** names new text styles; displays the Create new TextStyle dialog box.

**Activate** activates the selected style, making it the default.

**Rename** renames the selected style.

**Delete** erases the selected style from the drawing; you cannot erase the style named Standard, or styles used by text in the drawing.

**Text Options**

**Font** lists the file names of TTF (TrueType) and SHX (shape) fonts available on your computer.

**Format** assigns normal, *italic*, **bold**, or **bold italic** format to TrueType fonts.

**Height** specifies the height; leave this value at 0 if you wish to specify the height during the `-SimpleNote` command. (The `Note`, `-Note`, and `SimpleNote` commands can always override the height.)

**Big Font** toggles the listing of SHX “big fonts” that accommodate extended character sets.

**Orientation Options**

**Backwards** draws text backwards.

**Angle** specifies the slant at which to draw the text; positive angles slant the text forward; negative angles backwards. Range is -84.9 to 84.8 degrees.

**Upside Down** draws text upside down.

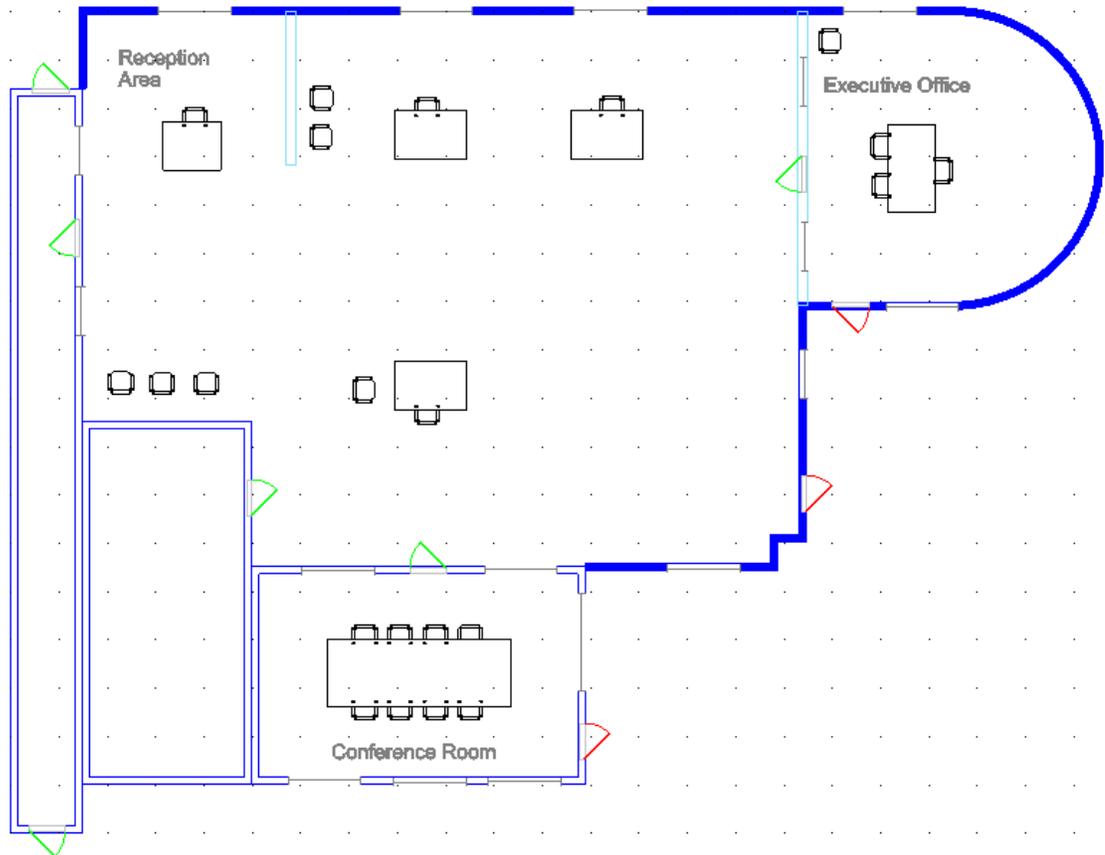
**Spacing** changes the aspect ratio; values larger than 1 expand the width of the text, while smaller values compress it.

**Vertical** draws certain SHX fonts vertically.

3. Notice that the Preview image updates to show the changed font. You can enter any text you wish to preview. For example, enter **Conference**:



4. Click **OK** to dismiss the dialog box. Notice that the strings of text in the drawing change from the stick ARtxt to the clean Arial font. From now on, all text uses this font — until you again change the style.



While it may seem indirect to use styles for specifying fonts, they allow for very quick global changes.

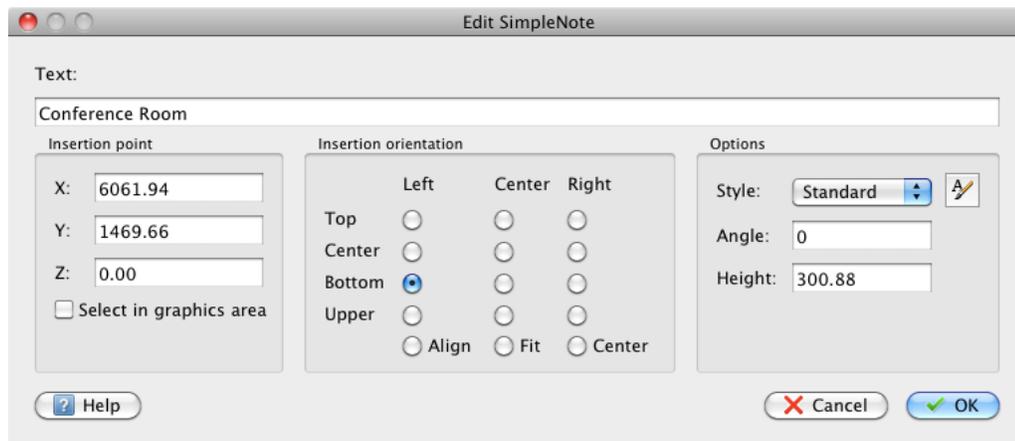
## Editing Text

**Command:** EditAnnotation  
**Aliases:** editttext, ddedit, ed, edanno  
**Menu Bar:** Modify | Object | Text  
**Shortcut:** *Double-click the text*

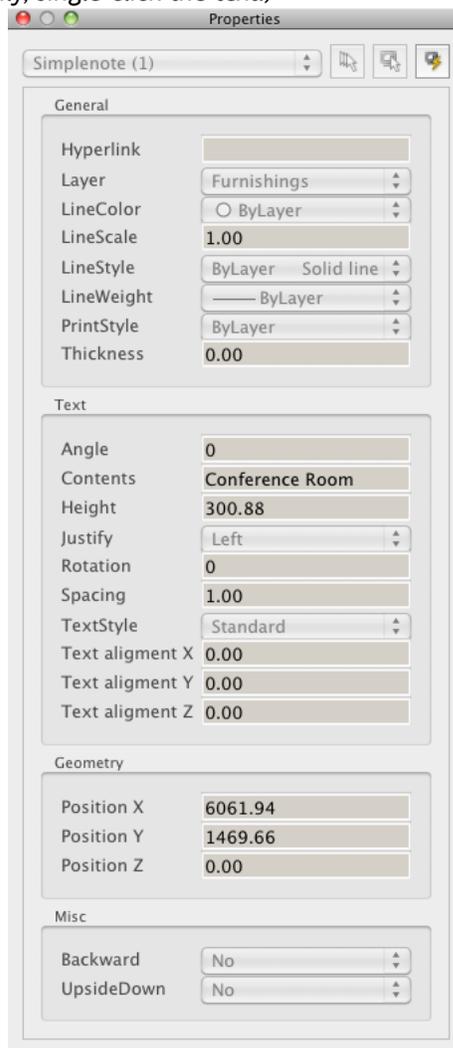
Text is edited with either the EditAnnotation command or the Properties palette — your choice. I find the dialog box displayed by the EditAnnotation command more straightforward, because it contains fewer options than does the Properties palette.

In this tutorial, you change the name of the conference room to “Board Room.”

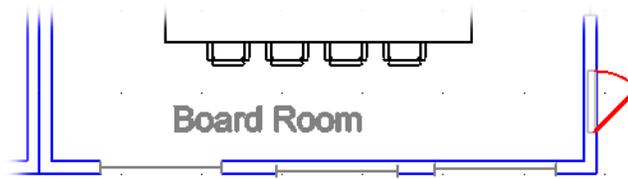
1. Double-click the text, “Conference Room.” Notice that iCADMac displays the Edit Simple-Note dialog box *and* has the Properties palette show the text properties.



(To access the palette only, single-click the text.)



2. In the Text field, change “Conference Room” to **Board Room**. (If you are using the Properties palette, then you make the change in the Contents field.)
3. Click **OK** to close the dialog box. (In the Properties palette, press **return** to affect the change.)



4. Save the drawing.

## T Placing Multiline Text

**Command:** Note  
**Aliases:** n, t, mtext, mt  
**Menu Bar:** Draw | Text | Multiline text  
**Tool Matrix:** Draw | MultiLine Text

The -SimpleNote command places just one line of text at a time. To enter entire paragraphs of text, use the Note command instead. It displays a dialog box in which you enter as much text as you need. The text is placed within an invisible rectangle called the “bounding box.” By stretching the bounding box, you define the margins of the text — left, right, and bottom — to make it fit a specific area.

You can edit and spellcheck notes, as well as apply styles, just as with simple text. But you can also override properties of notes, which lets you have text of different colors and sizes, bullets, underlining and boldfacing wherever you need.

In the following tutorial you add a paragraph of text to the lower right area of the drawing.

1. Start the **Note** command using one of the methods listed above.  
 Command: note
2. Notice that iCADMac prompts you to pick two points in the drawing. These two points define the corners of a bounding box. (Later, iCADMac will fit the text to the bounding box, starting at the top, extending as far down as necessary, and constrained by the sides.)

### Summary of Note Command Options

Command: note  
 Active TextStyle: “Standard” Text height: 63.956  
 Specify first corner»  
 Options: Angle, Height, Justify, Line spacing, textSTyle, Width or  
 Specify opposite corner»

**Angle** specifies the angle of the bounding box, rotated about the first corner.

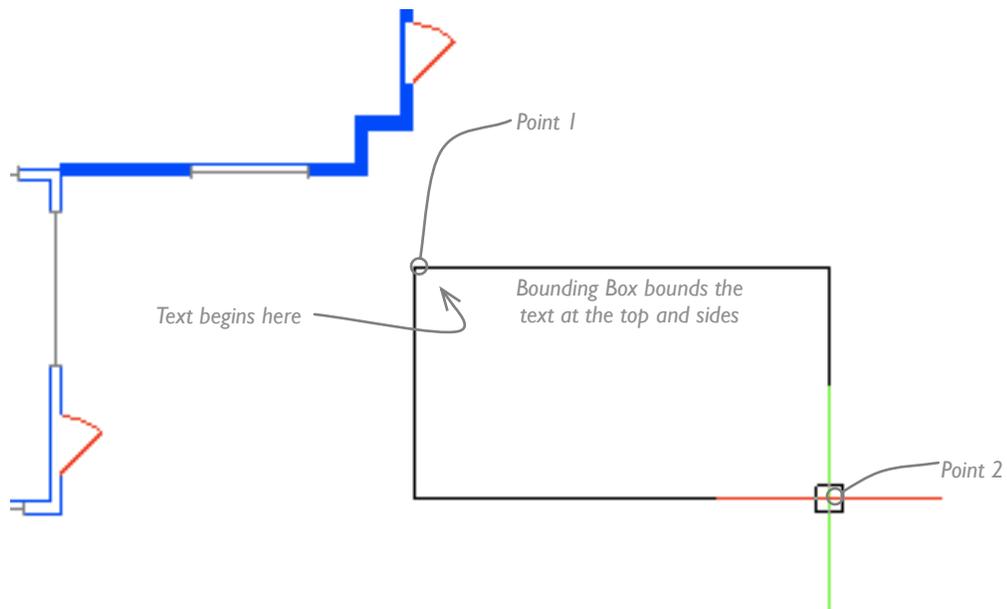
**Height** specifies the height of the text.

**Justify** specifies the text justification mode relative to the bounding box.

**Line spacing** specifies the spacing between lines of text.

**textSTyle** specifies the name of the text style.

**Width** specifies the width of the bounding box.



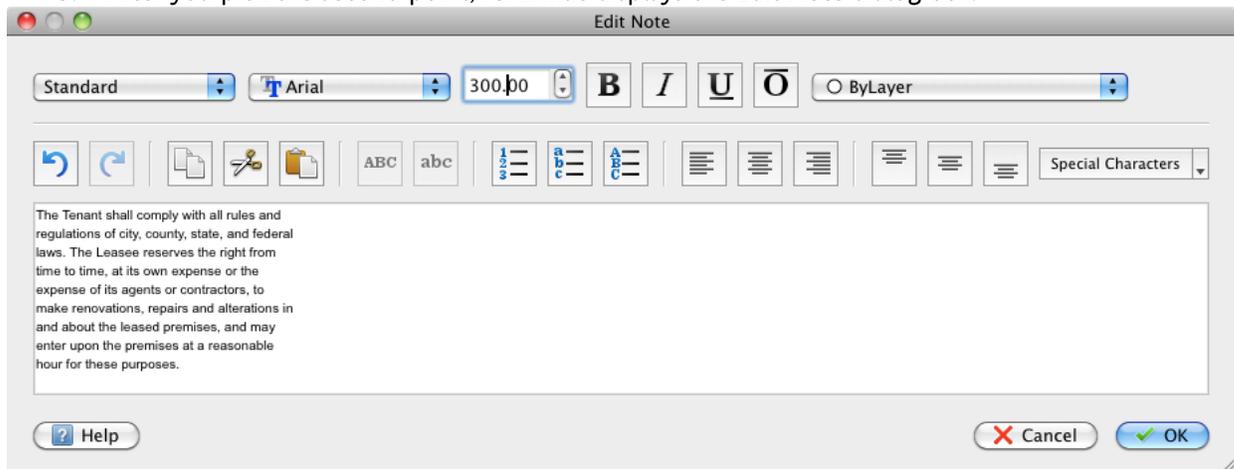
Active TextStyle: "Standard" Text height: 300.88

Specify first corner» (Pick a point for one corner of the bounding box.)

Options: Angle, Height, Justify, Line spacing, textStyle, Width or

Specify opposite corner» (Pick another point for the bounding box's opposite corner.)

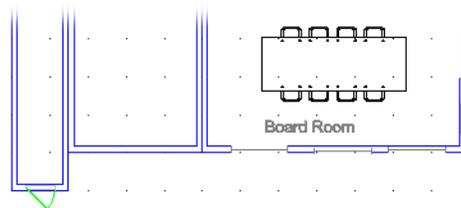
3. After you pick the second point, iCADMac displays the Edit Note dialog box.



Enter some text, such as the sample provided below.

The Tenant shall comply with all rules and regulations of city, county, state, and federal laws. The Leasee reserves the right from time to time, at its own expense or the expense of its agents or contractors, to make renovations, repairs and alterations in and about the leased premises, and may enter upon the premises at a reasonable hour for these purposes.

4. Click **OK**. Notice that the text is placed in the drawing.



The Tenant shall comply with all rules and regulations of city, county, state, and federal laws. The Lessee reserves the right from time to time, at its own expense or the expense of its agents or contractor, to make renovations, repairs and alterations in and about the leased premises, and may enter upon the premises at a reasonable hour for these purposes.

## Modifying Paragraph Text

**Command:** EditNote  
**Aliases:** mtedit, edn  
**Menu Bar:** Modify | Object | Text  
**Shortcut:** *Double-click the text*

You can click the text to modify it with the Properties palette, or double-click to edit the text with the Edit Note dialog box — just as with simple text.

In this tutorial, you selectively change the properties of some of the text.

1. Double-click the paragraph text. Notice that the Edit Note dialog box again contains the text you entered earlier.
2. Apply boldfacing to the first sentence, as follows:
  - a. *Select* the text to be boldfaced. To select text: (a) click at the start of the sentence, and then (b) drag the cursor to the period at the end of the sentence.)

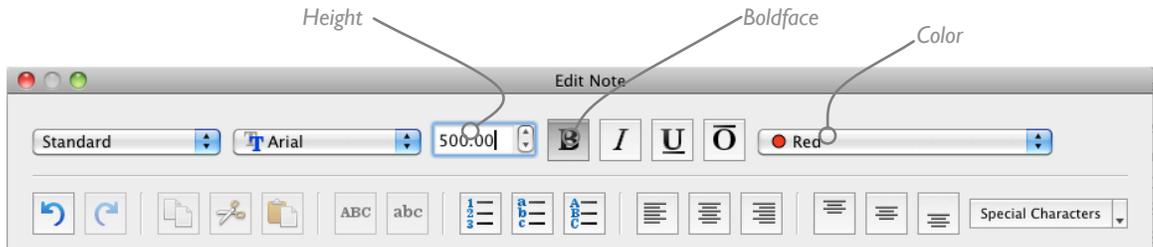
The Tenant shall comply with all rules and regulations of city, county, state, and federal laws. The Lessee reserves the right from time to time, at its own expense or the expense of its agents or contractor, to make renovations, repairs and alterations in and about the leased premises, and may enter upon the premises at a reasonable hour for these purposes.

- b. Click the **B** **Boldface** button. Notice that the text becomes bold.
- c. Click anywhere to remove the highlighting.

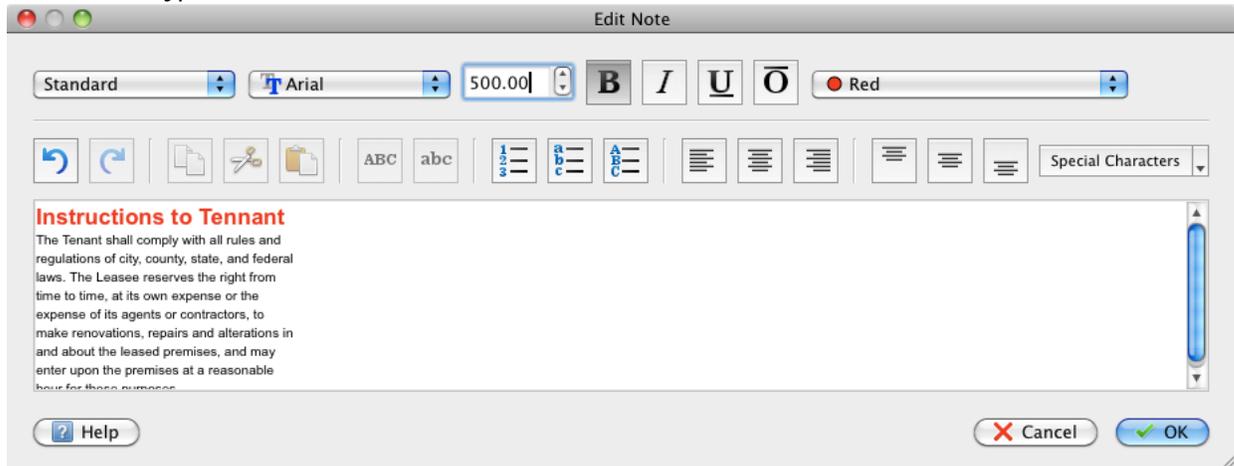
**The Tenant shall comply with all rules and regulations of city, county, state, and federal laws.** The Lessee reserves the right from time to time, at its own expense or the expense of its agents or contractor, to make renovations, repairs and alterations in and about the leased premises, and may enter upon the premises at a reasonable hour for these purposes.

3. To format text that looks like a title, follow these steps:
  - a. Position the cursor at the start of the text.
  - b. Press **return** to create a blank line.

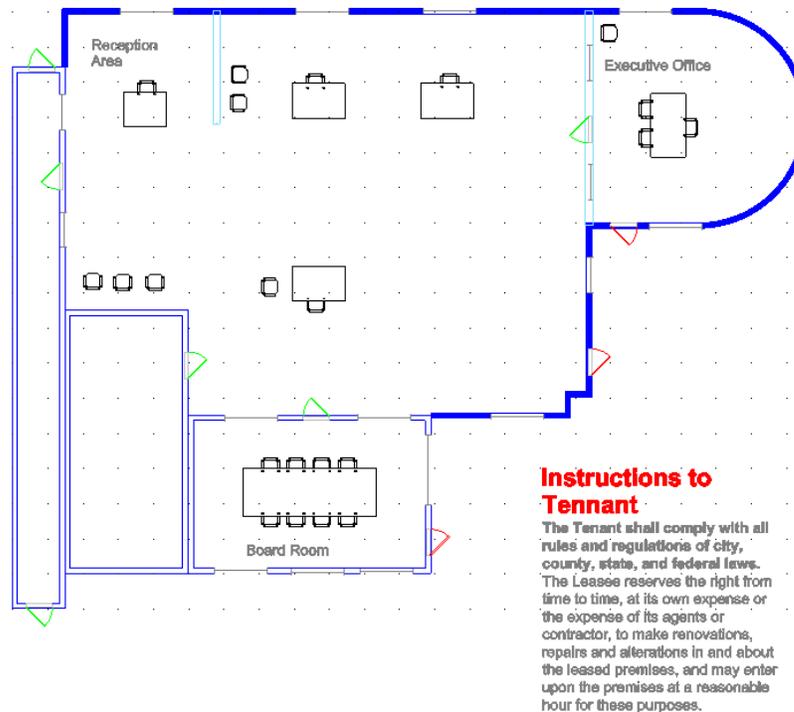
- c. Change the height to **500mm**:



- d. Change the color to **Red**.  
e. Type the title text, **Instructions to Tenant**.



4. Click **OK** to apply the changes, and then press **command+S** to save the drawing.



## Facilities Management

iCADMac has the ability to include data with blocks. It can be any kind of data, but in practice tends to be limited to descriptions of items. For instance, the block of a chair might be assigned its purchase price, model number, and owner.

The data can be extracted to a spreadsheet, thereby creating a list of the data of all chairs in the drawing. Formulas in the spreadsheet can add up the total number of chairs, total purchase price, and so on.

Naturally, this kind of data can be added to other blocks in the drawings — the doors, windows, and tables/desks. Keeping track of these things is known as “facilities management,” a field of study important to large companies, which lets them more easily track their capital assets.

The data attached to blocks are called “attributes.” Working with attributes in iCADMac involves three stages:

**Stage 1: Defining attributes** with names and default values.

**Stage 2: Attaching attributes** to blocks.

**Stage 3: Extracting attributes** to spreadsheets and data files.

Attributes cannot, unfortunately, be attached to any other kind of entity, such as circles or polylines. The workaround is to turn those entities into blocks.

It can get tedious working with many attributes in many blocks, so in this tutorial you attach only two attributes to the chair block.



---

### Stage 1: Defining Attributes

**Command:** MakeBlockAttribute

**Aliases:** att, attdef, mblkatt

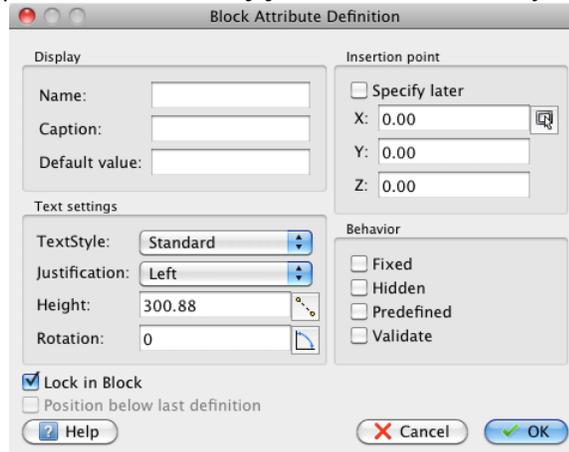
**Menu Bar:** Draw | Block | Define Block Attributes

Attributes are defined with the MakeBlockAttribute command. Every attribute consists of two parts: an identifying *name* (a.k.a. *tag*) and a *value*. Every block can store up to 255 attributes, and so the names allow iCADMac and spreadsheets (and you) to identify the values.

For example, a block has two values, 25 and 10-10-10. What do they mean? The name associated with each value tells us: Price (\$25.00) and Date (Oct 10, 2010).

For the tutorial involving the chair block, you define two attributes: the purchase price and its owner (who sits in it). Follow these steps:

1. Start the **MakeBlockAttribute** command. Notice that the Block Attribute Definition dialog box has many options, but fortunately you need to work with just a few of them.



### Summary of MakeBlockAttribute Dialog Box Options

Command: MakeBlockAttribute

#### Display Options

**Name** (or Tag) identifies the attribute; use letters, numbers, blanks, underscores, and dollar signs.

**Caption** (or Prompt) displays a prompt during block insertion; maximum of 256 characters.

**Default Value** specifies the default value for the attribute.

#### Insertion Point Options

**Specify later** prompts you for the insertion point after you close this dialog box; click the **Select in Graphics Area** button to pick a point in the drawing.

**X, Y, and Z** specify the coordinates of the insertion point.

#### Text Settings Options

**TextStyle** specifies the style for the attribute text; text styles are created and modified by the TextStyle command.

**Justification** specifies the justification mode.

**Height** specifies the height of the text; click the **Select in Graphics Area** button to pick two points in the drawing.

**Rotation** specifies the rotation angle; click the **Select in Graphics Area** button to pick two points in the drawing.

#### Behavior Options

**Fixed** gives the attribute the constant value defined by Default Value; the user cannot enter a different value.

**Hidden** hides the attribute text; the user can make hidden attributes visible with the DisplayBlockAttributes command.

**Predefined** uses the default value automatically; you are not prompted for a value when the block is being inserted. Values can be edited later using the EditBlockAttribute command.

**Validate** prompts you to verify that the value is correct.

**Lock in Block** prevents the attributes from being moved away from the block reference.

**Position Below Last Definition** places the next attribute definition below the previous one automatically.

2. Attributes are created one by one. The first one is “Purchase Price,” which you create like this:

- a. In the **Name** field, enter:

PurchasePrice

The *name* is the identifier for this attribute. Names cannot have spaces in them.

- b. In the **Caption** field, enter a prompt:

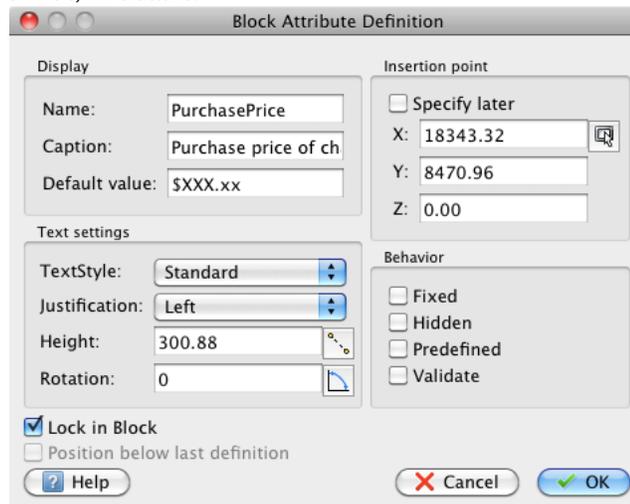
Purchase price of chair

The *caption* appears later in the Command window to remind users of the purpose of this attribute.

- c. In the **Default Value** field, enter:

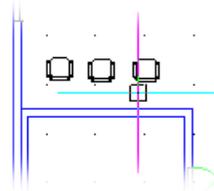
\$xxx.xx

The *default* contains either a common value or a format hint. Above, the \$xxx.xx hints at the format, in dollars.



3. In practice, attribute data are always located near their blocks. Click the **Specify Position** button, and then choose a point near a chair block in the drawing.

Options: press ESC to return to dialog or  
Specify position» (Pick a point near a chair block.)



4. You can ignore the other options in the dialog box. (Keep **Lock in Block** turned on, because it prevents users from moving the attribute away from the block.)

Click **OK**.

5. Notice that the attribute text appears near the block.



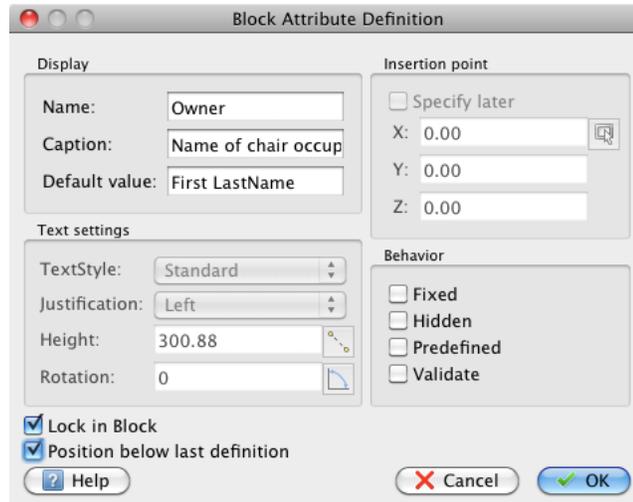
6. Press the **spacebar** to repeat the MakeAttributeBlock command to add the second attribute. The parameters are as follows:

Name	Owner
Caption	Name of chair occupant

Default Value First LastName

- You can have iCADMac automatically place the new attribute definition below the first one: turn on the **Position Below Last Position** option.

The Block Definition dialog box should look like this now, where the Insertion Point and Text Settings options are grayed out, because they copy the settings from the previous attribute definition.



- After you click **OK**, the second attribute appears:

PurchasePrice  
Owner

## Stage 2: Adding Attributes to Blocks

The text that you see in the drawing now is called the *attribute definition*; it is not the attribute data, like \$249.99. In this part of the tutorial, you attach the attribute definition to the chair block by redefining the block.

- To redefine the Office Chair block, you must first explode one of its insertions. Start the **Explode** command, and then choose the block next to the attribute definitions:  
 Command: explode  
 Specify entities» (Select the chair block.)  
 Specify entities» (Press return to end the command.)

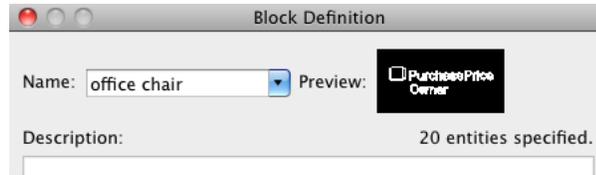
The chair looks no different, except that it is no longer a block. (You can move each line and arc individually — but don't!)

**TIP** This procedure of attaching attribute definitions also applies to entities that are not already blocks. The only change is that you don't need to explode them first, because they are not blocks yet; all of the following steps apply.

- With the block reduced to its constituent parts, you can now redefine the Office Chair, like this:
  - Start the **MakeBlock** command.
  - In the **Name** droplist, choose "Office Chair." By doing this, you redefine the block to include the attributes.

- c. In the Block Entities section, click the  **Select In Graphics Area** button.
- d. When the dialog box disappears, window-select the two attribute definitions and the 18 entities making up the chair, and then press **return** to return to the dialog box.
  - Select entities: (Select attributes and exploded chair.)
  - Select entities: (Press return to return to dialog box.)

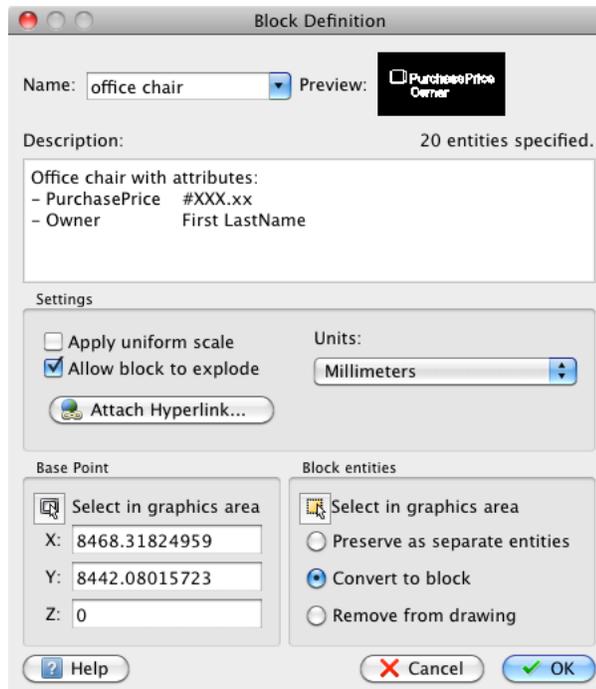
Notice that the preview window now shows the chair entities as well as the attribute definitions.



**TIP** The order in which you select the attributes determines the order in which their prompts are displayed later while inserting the block.

- e. Click the **Base Point** button, and then choose an insertion point on the chair.
- f. Change the **Block Entities** option to “Convert to Block.” This ensures that all existing blocks are replaced by the new definition.

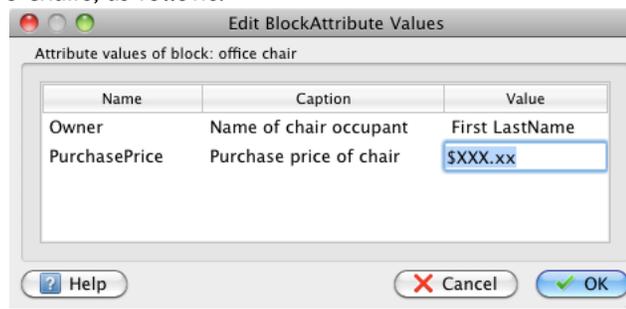
The dialog box should now look like the one illustrated below:



- g. Click **OK**.
3. Notice that iCADMac warns that you are engaging in a dangerous enterprise, the redefinition of existing blocks! Click **Yes**.



4. Now comes the tedious part: entering the Purchase Price and Owner data for each of the 18 chairs. In practice, this is a job that other people get paid to do; for this tutorial, only enter data for two chairs, as follows:



In the Edit Block Attribute Values dialog box, change the default values:

Name	Default Value	New Value
Owner	First LastName	<b>John Smith</b>
PurchasePrice	\$xxx.xx	<b>\$249.99</b>

5. Click **OK**. Notice that the block now displays attribute *data*.
6. Repeat this process for at least one other chair block.
7. Save your work.

**TIP** Don't worry if attribute data spill over on top of other entities. Since attributes are not meant to be seen in drawings, you can easily turn off their display through the DisplayBlockAttributes command, or through the menu:

1. From the **View** menu, choose **Display**, and then choose **Block Attribute**.
2. Choose **Off**. Notice that the attributes disappear from the drawing.

The **On** and **Off** options are straightforward.

The **Normal** option treats the display of attributes as defined by the Block Attribute Definition dialog box. One of the options is Hidden, which means the attribute is invisible by default. The Normal option displays only non-hidden attributes, while the **On** option displays all of them, including those marked as hidden.



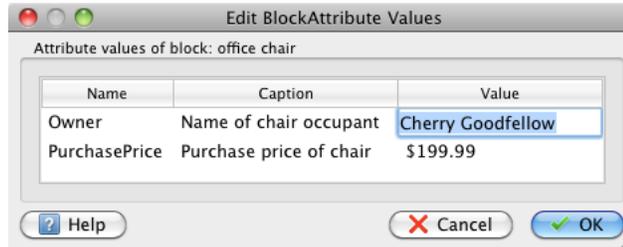
### Editing Attribute Data

**Command:** EditBlockAttribute

**Aliases:** ate, attedit, attxedit, ddatte, edblkatt

**Menu Bar:** Modify | Object | Attribute | Single

You can edit attributes with the EditBlockAttribute command. It displays the same Edit Block Attribute Values dialog box you saw earlier.



**TIP** When set to 1, the **AttDia** system variable causes the Insert command to display dialog boxes for inserting blocks and entering attribute values; when 0, Insert displays all prompts in the Command window.

### Stage 3: Extracting Attributes

**Command:** BlockAttributeOutput

**Alias:** attout

**Menu Bar:** Tools | Attribute Export

Attributes are of limited use in drawings; you can see them, but you cannot analyze them, such as adding their Price values together to arrive at a total. To do this, you must extract the data from the drawing, and then import them in a spreadsheet.

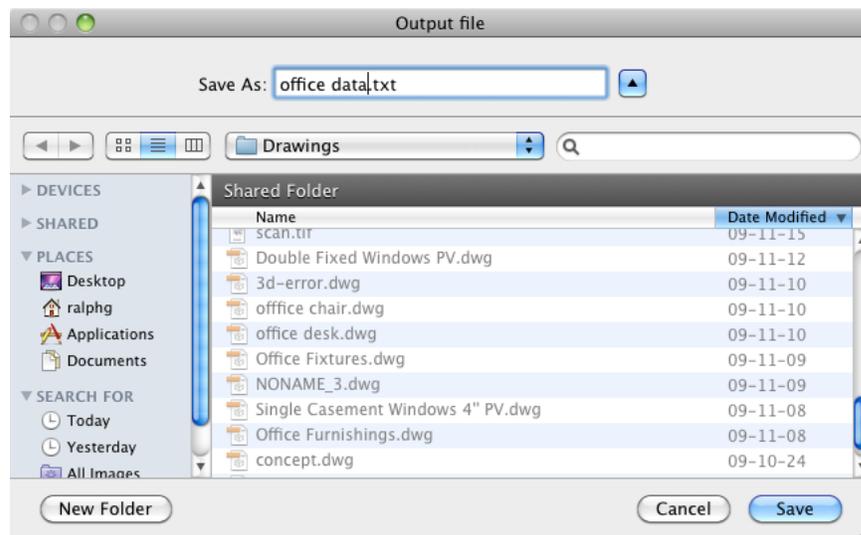
The process takes two steps:

- i. Export the attribute data with the BlockAttributeOutput command.
- ii. Open the data file in a spreadsheet program, such as Apple Numbers or OpenOffice Calc.

To export the block and attribute data to an external file, follow these steps:

1. Enter the **BlockAttributeOutput** command, and then select all entities in the drawing:  
Command: blockattributeoutput  
Select entities: all
2. iCADMac automatically filters out entities that do not contain attributes. Press **return** to end entity selection and continue.  
Select entities: (Press return to continue.)
3. In the Command Window, iCADMac reports the number of entities it found in the drawing:  
95 found, 93 total  
2 were not in drawing area.

In addition, it opens the Output File dialog box.



4. Enter a name for the data file, such as “office data.” (The data are stored in tab-delimited format, meaning that tabs separate each field of data.)
5. Click Save to export the attribute data.

### Opening the Data in a Spreadsheet

To open the data in a spreadsheet, follow these steps:

1. Start a spreadsheet program, such as Numbers from Apple or Calc from OpenOffice. First, I'll describe how to import the data into Numbers:
 

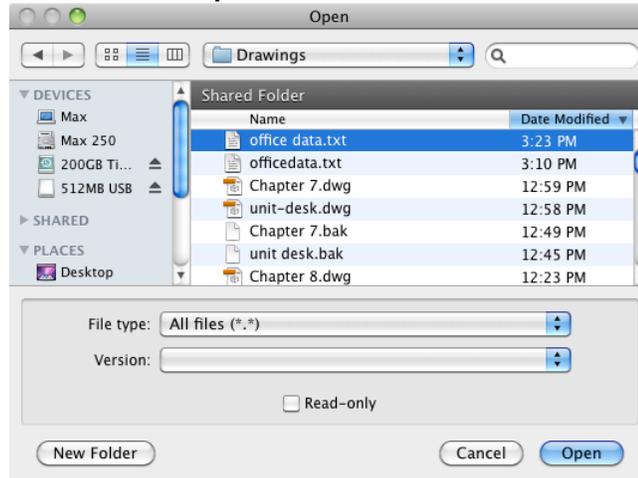
---

  2. To open the data file in Numbers, follow these steps:
    - a. Click **Open An Existing File**.
    - b. Navigate to the folder holding the *office data.txt* file, select it, and then click **Open**.
    - c. Notice that the data appear in the spreadsheet, segregated into columns.

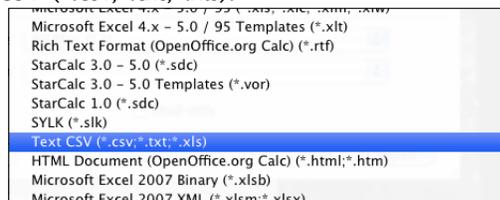
HANDLE	BLOCKNAME	Owner	PurchasePrice
294	office chair	<>	<>
295	office chair	<>	<>
296	office chair	<>	<>
297	office chair	<>	<>
298	office chair	<>	<>
299	office chair	<>	<>
29A	office chair	<>	<>
29B	office chair	<>	<>
2FA	door-100x750	<>	<>
2FB	door-100x750	<>	<>
2FC	door-100x750	<>	<>
2FD	door-100x750	<>	<>
2FE	door-100x750	<>	<>
2FF	door-100x750	<>	<>
300	door-100x750	<>	<>
301	door-100x750	<>	<>
302	door-100x750	<>	<>
30D	office chair	<>	<>
30E	office chair	<>	<>
30F	office chair	<>	<>
310	office chair	<>	<>
311	office chair	<>	<>

2. To open the data file in Calc, follow these steps instead:

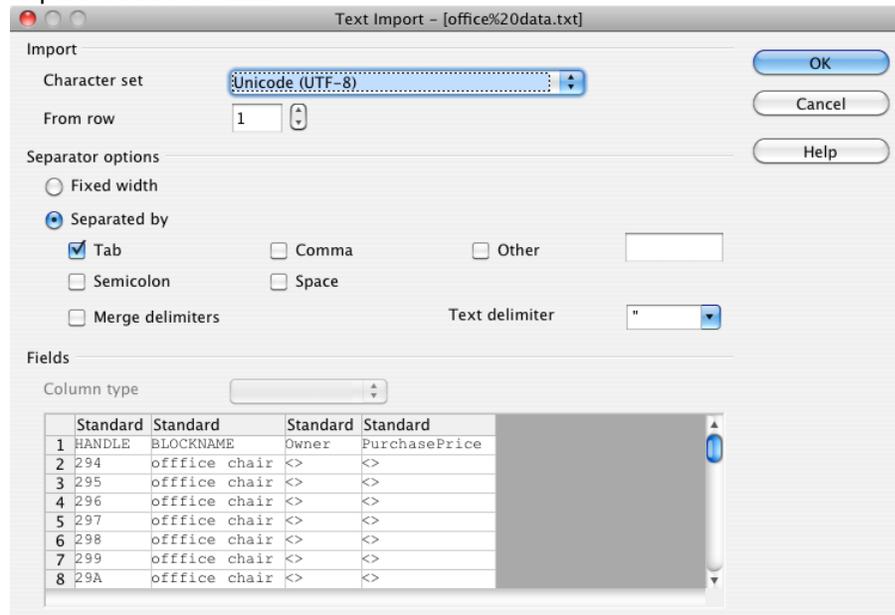
- a. From the **File** menu, choose **Open**.



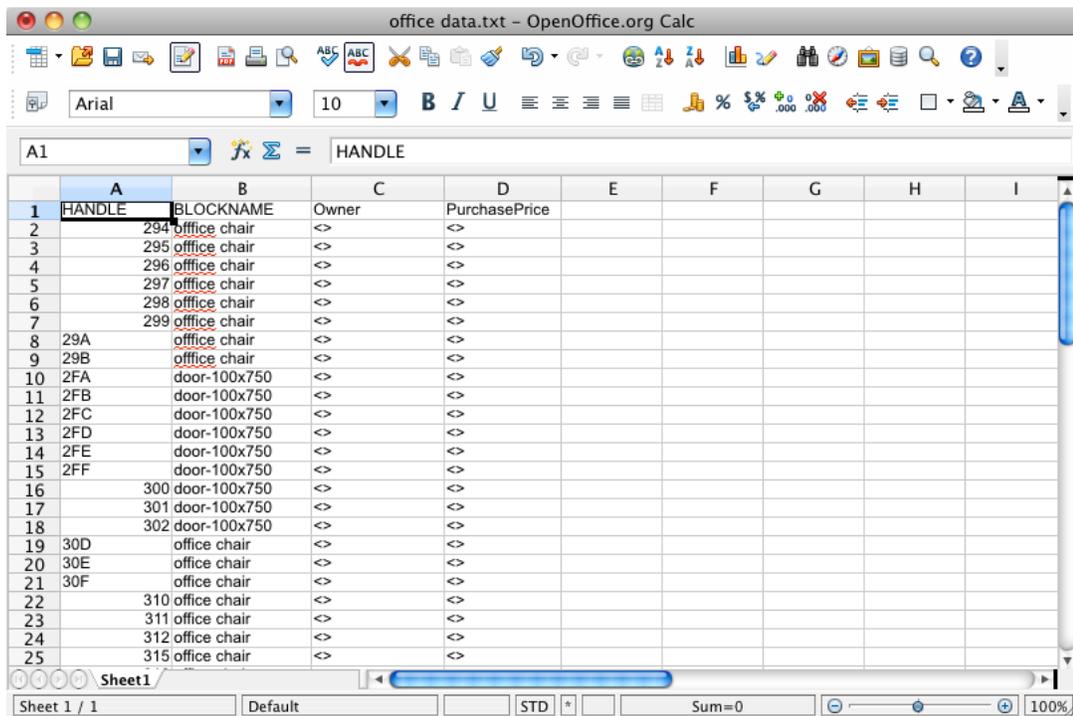
- b. Go to the folder that contains the *office data.txt* file, and then select it.
- c. This step is important; otherwise, the file will open in Writer instead of Calc:
- Click **File Type**.
  - Scroll down the droplist until you arrive at the section for spreadsheet formats.
  - Choose “Text CSV (\*.csv;\*.txt;\*.xls).”



- d. Click **Open**. Notice that Calc displays a dialog box confirming that each field of data is separated into columns.



- e. Click **OK**. The data are entered into the Calc spreadsheet automatically.



	A	B	C	D	E	F	G	H	I
1	HANDLE	BLOCKNAME	Owner	PurchasePrice					
2	294	office chair	<>	<>					
3	295	office chair	<>	<>					
4	296	office chair	<>	<>					
5	297	office chair	<>	<>					
6	298	office chair	<>	<>					
7	299	office chair	<>	<>					
8	29A	office chair	<>	<>					
9	29B	office chair	<>	<>					
10	2FA	door-100x750	<>	<>					
11	2FB	door-100x750	<>	<>					
12	2FC	door-100x750	<>	<>					
13	2FD	door-100x750	<>	<>					
14	2FE	door-100x750	<>	<>					
15	2FF	door-100x750	<>	<>					
16		300 door-100x750	<>	<>					
17		301 door-100x750	<>	<>					
18		302 door-100x750	<>	<>					
19	30D	office chair	<>	<>					
20	30E	office chair	<>	<>					
21	30F	office chair	<>	<>					
22		310 office chair	<>	<>					
23		311 office chair	<>	<>					
24		312 office chair	<>	<>					
25		315 office chair	<>	<>					

3. You can now apply spreadsheet functions to the data, such as counting, adding up, and multiplying.

## Summary

You now know how to add text to drawings, format the text through styles, and override styles with properties. Attributes can be useful for tracking inventory and creating parts lists.

In the next chapter, you learn how to place another type of annotation: dimensions.



## Chapter 9

# Placing Dimensions

### In This Chapter

- Adding dimensions to drawings
- Creating dimension styles

**Strictly speaking, CAD** drawings do not need to be dimensioned. This is because all entities are drawn full size, accurate to more than a dozen decimal places. If you want to know the length or diameter of things in iCADMac, use the GetDistance or Properties commands; when the drawing is on paper, use a scale ruler.

(Hand made drawings often included a warning, DO NOT SCALE. This meant that no one should use a scale ruler to measure distances on paper drawings, because the drafters could not guarantee that every pencil line was accurate. Instead, drafters added dimensions to annotate the size of every part.)

Still, it is useful to add dimensions to CAD drawings so that anyone can gauge the sizes of objects at a glance. In this chapter, you learn how to apply linear and radial dimensions. Linear dimensions measure distances, while radial ones measure radii and diameters. You also learn how to control the size and look of dimensions through styles.

## Key Terms in This Chapter

**Baseline dimension** — refers to dimensions that have a common starting extension line.

**Center mark** — indicates the centers of arcs and circles.

**Continued dimension** — refers to dimensions that carry on from the last dimension.

**Dimension style** — specifies the properties of dimensions.

**Dimension** — show lengths, distances, and angles between entities in drawings.

**Dimension line** — refers to the line that spans the distance between extension lines.

**Extension line** — refers to the two lines that indicate the distance being measured.

**Linear dimension** — shows horizontal, vertical, and rotated lengths of entities.

## Commands

Command	Shortcuts	Menu Selection
BaselineDimension	dimbase, dimbaseline	Dimension   Baseline
ContinueDimension	cdim, dimcontinue	Dimension   Continue
DiameterDimension	dimdia, dimdiameter	Dimension   Diameter
DimensionStyle	d, dimstyle	Format   Dimension Style
DimScale	...	... <sup>1</sup>
LinearDimension	dimlin, dimlinear	Dimension   Linear
ParallelDimension	dimali, dimaligned	Dimension   Aligned
RadialDimension	dimrad, dimradial	Dimension   Radius

<sup>1</sup> No menu selection.

## Linear Dimensions

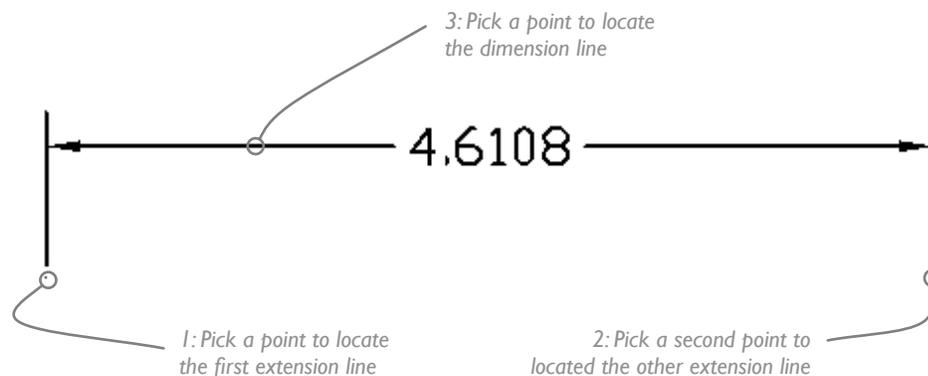
**Command:** LinearDimension  
**Aliases:** dimlim, dimlinear, dli, ldim  
**Menu Bar:** Dimension | Linear  
**Tool Matrix:** Dimension | Linear

*Linear* dimensions report the linear distance between two points, as shown by the following tutorial. The LinearDimension command draws horizontal, vertical, and rotated dimensions.

### Horizontal Dimensions

To draw a linear dimension, start iCADMac with a new drawing, and then follow these steps:

1. Start the LinearDimension command using one of the methods listed above.  
 Command: lineardimension  
 The prompts in the command window.
2. Pick a point in the drawing. This is the point from which the dimension begins its measurement:  
 Default: Entity  
 Options: Entity or  
 Specify first extension line position» (Pick a point.)
3. Hold down the **shift** key to keep the dimension horizontal.
4. Move the cursor, and then pick a second point in the drawing. This is the ending point for the dimension:  
 Specify second extension line position» (Hold down shift, and then pick a second point.)



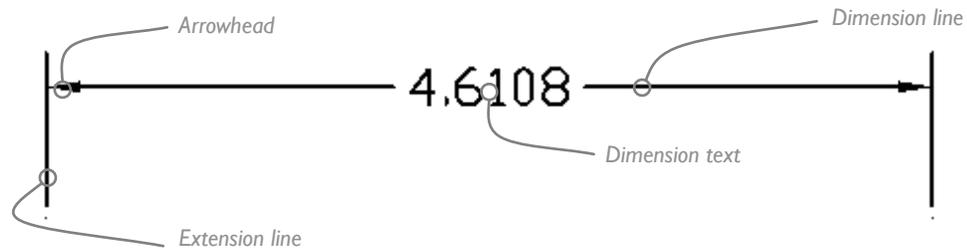
5. With the two pick points, you have defined the distance to be dimensioned.  
 There is one more item to specify: the one that locates the dimension line, which also determines the length of the extension lines.  
 Options: Angle, Horizontal, Note, Rotated, Text, Vertical or  
 Specify dimension line position» (Pick a third point.)

Notice that iCADMac reports the distance on the dimension and in the Command window.

Dimension Text: 4.6108

iCADMac draws all the elements of the dimension automatically; all you did was indicate the distance to be measured, and then locate the dimension line.

All linear dimensions are almost always made of the same entities illustrated by the following figure:



**Extension lines** point to the start and endpoints of the distance being measured.

**Dimension line** links the two extension lines.

**Arrowheads** point to the two extension lines.

**Dimension text** reports the distance between the two points.

#### Summary of LinearDimension Command Options

: lineardimension

Default: Entity

Options: Entity or

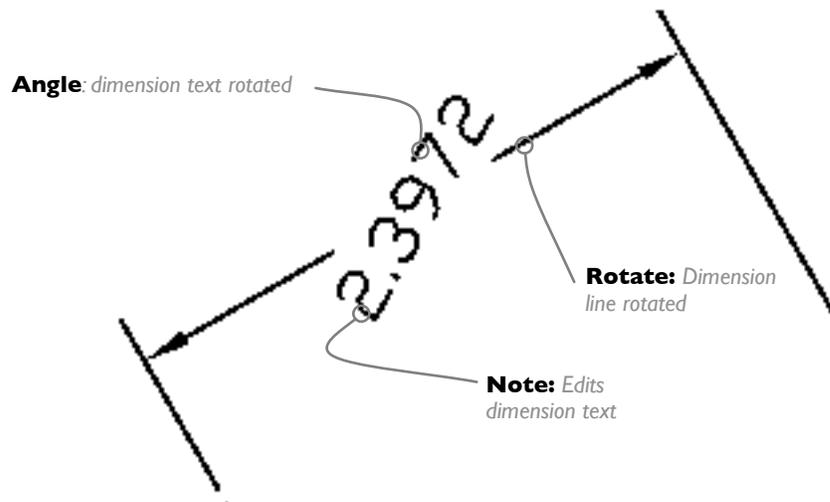
Specify first extension line position»

Specify second extension line position»

Options: Angle, Horizontal, Note, Rotated, Text, Vertical or

Specify dimension line position»

Dimension Text: 2.3972



**Entity** — dimensions the selected entity; extension line positions are not used.

**Angle** — specifies the angle of the dimension text; the default is 0 degrees:  
Specify text angle» (Enter angle, and then press return.)

**Horizontal** — forces the dimension line to be horizontal.

**Note** — customizes the dimension text with the Edit Note dialog box.

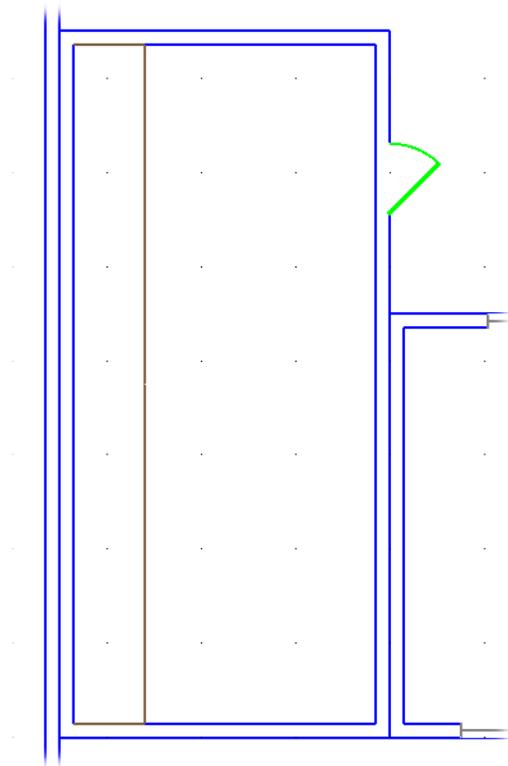
**Rotated** — specifies the angle of the dimension and extension lines:  
Specify dimension line rotation» (Enter angle, and then press return.)

## Vertical Dimensions

In the last tutorial, you drew a horizontal dimension, which is probably the most common type. iCADMac also draws other kinds of linear dimensions, including vertical, rotated, and aligned. In this next tutorial, you open the *office plan.dwg* file, and then add all of these kinds of linear dimensions.

1. Open the *office plan.dwg* file.
2. Ensure entity snaps are turned on. This ensures that dimensions are placed accurately.
3. With the **Layer** command, open the Layers Manager dialog box:
  - a. Create a new layer named “Dimensions.”
  - b. Choose a color that contrasts from the others in the drawing, such as brown, color #35.
  - c. Make the layer active.
  - d. Freeze unnecessary layers, such as Furnishings and Text.
4. Click **OK** to exit the dialog box.
5. Zoom in to the area of the storage room with the **Zoom Window** command.
6. To draw a vertical dimension, start the **LinearDimension** command. This single command places horizontal, vertical, and rotated dimensions.
7. Follow the prompts to dimension the inside length of the storage room:

Default: Entity  
Options: Entity or  
Specify first extension line position» (Pick one intersection.)  
Specify second extension line position» (Pick other intersection.)  
  
Options: Angle, Horizontal, Note, Rotated, Text, Vertical or  
Specify dimension line position» (Pick a point away from the wall.)  
Dimension Text : 7200.0000



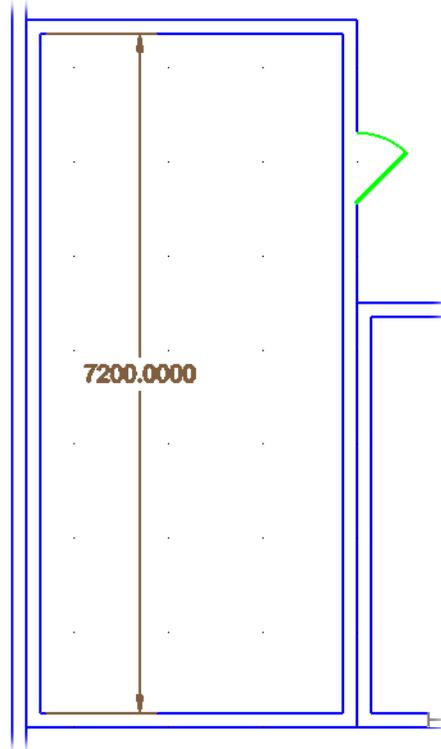
## Dimension Scale

System Variable: DimScale

You can see the dimension line, but not the text and arrowheads! Dimensions are like text; they need to be scaled large enough to see them. The dimension you drew had a scale of 1, which is too small for this large floor plan drawing. This size is suitable for drawings of things at millimeter scale, but our drawing is at the meter scale — 1000 times larger.

Use the DimScale system variable to change the dimension scale to 1000, making the dimensions a thousand times larger.

1. With the **Delete** command, erase the vertical dimension you drew. (DimScale is not retro-active.)
2. Enter **DimScale**, and then specify the scale factor at 1000, as follows:  
 Command: dimscale  
 Default: 1.00  
 Enter new value for DIMSCALE» 1000
3. Repeat the **LinearDimension** command to redimension the storage room.



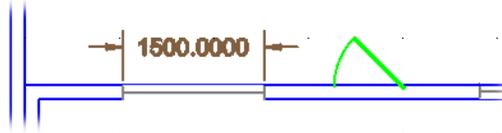
That's better!

### Dimensioning Entities

There is an alternative method to placing dimensions. Instead of picking two points in the drawing, you can select an entity to dimension. iCADMac locates the extension lines automatically; you just provide the location of the dimension line. In many cases, this approach is faster, as shown by the following tutorial:

1. Restart the **LinearDimension** command.
2. Enter 'e' at the prompt to switch to entity-dimensioning mode:  
 Command: lineardimension  
 Default: Entity  
 Options: Entity or  
 Specify first extension line position» e

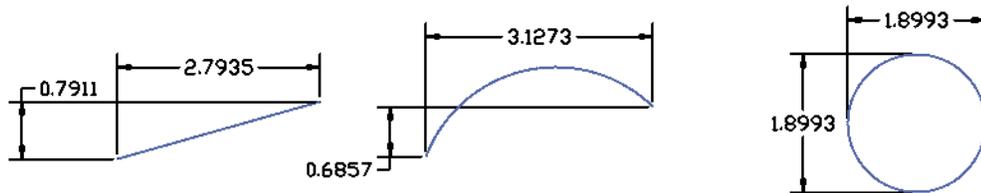
3. iCADMac prompts you to select an entity:  
Specify entity» (Pick a window block.)
4. Move the cursor, and then position the dimension line.  
Options: Angle, Horizontal, Note, Rotated, Text, Vertical or  
Specify dimension line position» (Pick a point to locate the dimension line.)  
Dimension Text : 1500.0000



Notice that iCADMac automatically found the two ends of the block, and placed the extension lines for you. The Entity option draws a horizontal or vertical dimension, depending on how you drag the cursor away from the entity at the “Specify dimension line location’ prompt:

- For horizontal dimensions, drag the cursor up or down.
- For vertical dimensions, drag the cursor left or right.

The figure below illustrates how a line, an arc, and a circle can be dimensioned through the LinearDimension command’s Entity option:



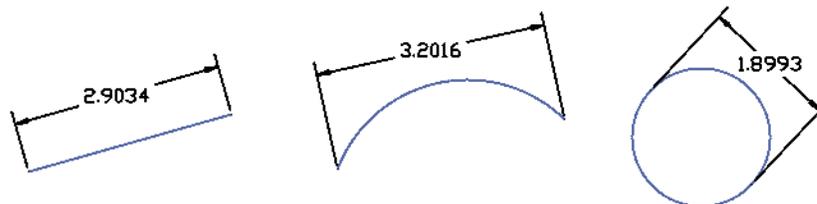
Ortho mode does not need to be turned on for these dimensions to be perfectly horizontal or vertical.



### Parallel Dimensioning

- Command:** ParallelDimension
- Aliases:** dal, dimali, dimaligned, pdim, paralleldim
- Menu Bar:** Dimension | Aligned
- Tool Matrix:** Dimension | Aligned

iCADMac also has the ParallelDimension command (a.k.a. *aligned* dimensioning), the Entity option of which places dimensions parallel to entities, as illustrated below. Notice how the lengths differ from those shown above by horizontal and vertical dimensions.



- For arcs, this command places dimensions parallel to the endpoints of arcs.
- For circles, this command places the dimension parallel to the first pick point.

This command ignores the setting of ortho mode.

## Continued Dimensions

**Command:** ContinueDimension  
**Aliases:** cdim, dco, dimcont, dimcontinue  
**Menu Bar:** Dimension | Continue  
**Tool Matrix:** Dimension | Continued

Once you draw one dimension, iCADMac can draw more in a perfectly straight line. This is accomplished with the ContinueDimension command, which “chains” additional ones. All you need to do is pick additional points for the second extension line; iCADMac places the dimension line at the same distance as previous ones. iCADMac does not draw the first dimension line, because it would overlap the second one of the previous dimension.

Usually, it is easier to see continued dimensions at work than to read about them. In the following tutorial, you place a string of dimensions along the top wall of the office layout.

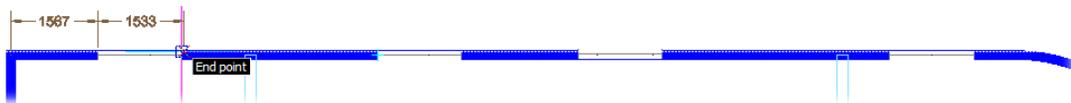
1. Currently, the dimension text displays four decimal places when no decimal places are required. To change the display accuracy from 1.0000 to 1, use the DimDec system variable, as follows:
  - a. Enter **DimDec** at the command prompt.  
 Command: dimdec
  - b. Type **0**, and then press **Enter**.  
 Default: 4  
 Enter new value for DIMDEC» 0
2. The first dimension of a continuous series is placed with the LinearDimension command; thereafter, the ContinueDimension command is used.

Enter the **LinearDimension** command to place the initial dimension, as illustrated below.



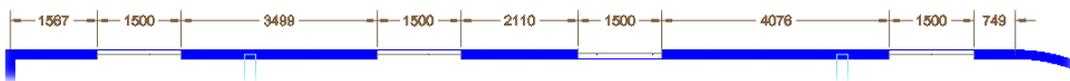
3. Start the ContinueDimension command using one of the methods listed above.

Command: continuedimension  
 Default: Select dimension  
 Options: Select dimension, Undo or  
 Specify second extension line position» (Pick a point.)  
 Dimension Text : 1500



You just pick the location of the second extension line, and iCADMac draws the rest.

4. The ContinueDimension command repeats automatically. Continue placing dimensions, and then press **return** to end the command.



Default: Select dimension  
 Options: Select dimension, Undo or  
 Specify second extension line position» (Pick another point.)  
 Dimension Text: 3489

...

Default: Select dimension  
 Options: Select dimension, Undo or

Specify second extension line position» (Press return to exit the command.)

This command continues linear dimensions created by the `LinearDimension`, `ParallelDimension`, and `ArcLengthDimension` commands; it does not work with radial dimensions made by `RadiusDimension` or `DiameterDimension`.

**TIP** To continue a dimension that was drawn earlier, use the `ContinueDimension` command's **Select dimension** option, which prompts you to select a linear dimension in the drawing:

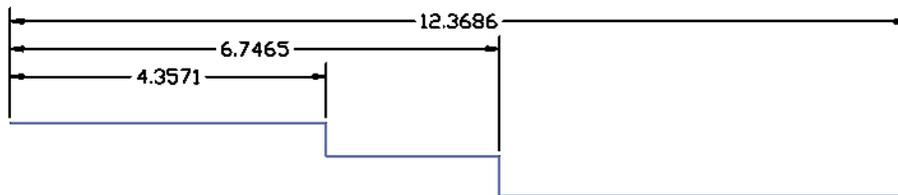
Specify dimension: (Pick a linear dimension.)



### Baseline Dimensions

**Command:** `BaselineDimension`  
**Aliases:** `basedim`, `bldim`, `dba`, `dimbase`, `dimbaseline`  
**Menu Bar:** Dimension | Baseline  
**Tool Matrix:** Dimension | Baseline

iCADMac also has the **BaselineDimension** command, which continues dimensions from a common *baseline*, as illustrated below. The “baseline” is the first extension line of the first dimension.



### Radial Dimensions

**Command:** `RadiusDimension`  
**Aliases:** `dimrad`, `dimradius`, `dra`, `rdim`  
**Menu Bar:** Dimension | Radius  
**Tool Matrix:** Dimension | Radius

Placing radial and diameter dimensions is like using the Entity option of linear dimensions: you pick a circle or arc, and iCADMac dimensions it immediately.

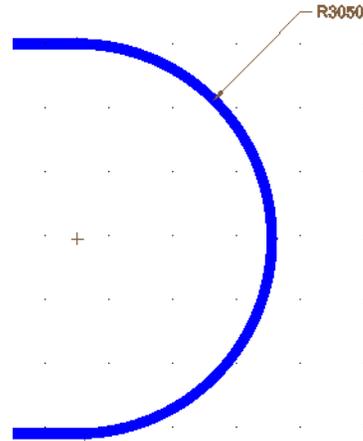
In the following tutorial, you use the `RadiusDimension` command to dimension the arc-shaped executive office:

1. Start the **RadiusDimension** command:  
Command: `radiusdimension`
2. Select the curved wall of the executive office.  
Specify curved entity» (Pick the arc.)

This command works with arcs, circles, and polylines; it does not work with ellipses or splines, because their curves are not of constant radius.

3. Locate the leader line. Notice that it can be placed inside or outside the arc; the standard is to place it outside of the entity.  
Options: Angle, Note, Text or  
Specify dimension position» (Pick a point.)

Dimension Text : 3050



### Diameter Dimensions

**Command:** DiameterDimension

**Aliases:** ddi, dimdia, dimdiameter

**Menu Bar:** Dimension | Diameter

**Tool Matrix:** Dimension | Dimension

iCADMac also has the DiameterDimension command, which dimensions the diameters of circles and arcs, as illustrated below.



**TIPS** Notice that both the RadiusDimension and DiameterDimension commands place a center mark (+) to indicate the center of the arc or circle. The **CenterMark** command draws the mark, as well.

For large-radii arcs, you can use the JoggedDimension command, which allows you to relocate the dimension's center point to a more convenient location.

Other useful dimensioning commands include AngularDimension (for measuring angles), Leader (for placing callouts), and Tolerance (for placing geometric tolerances).



### Modifying Dimension Styles

**Command:** DimensionStyle

**Aliases:** d, dimstyle, dimsty, dst, ddim

**Menu Bar:** Format | Dimension Style

**Tool Matrix:** Dimension | Dimension Style

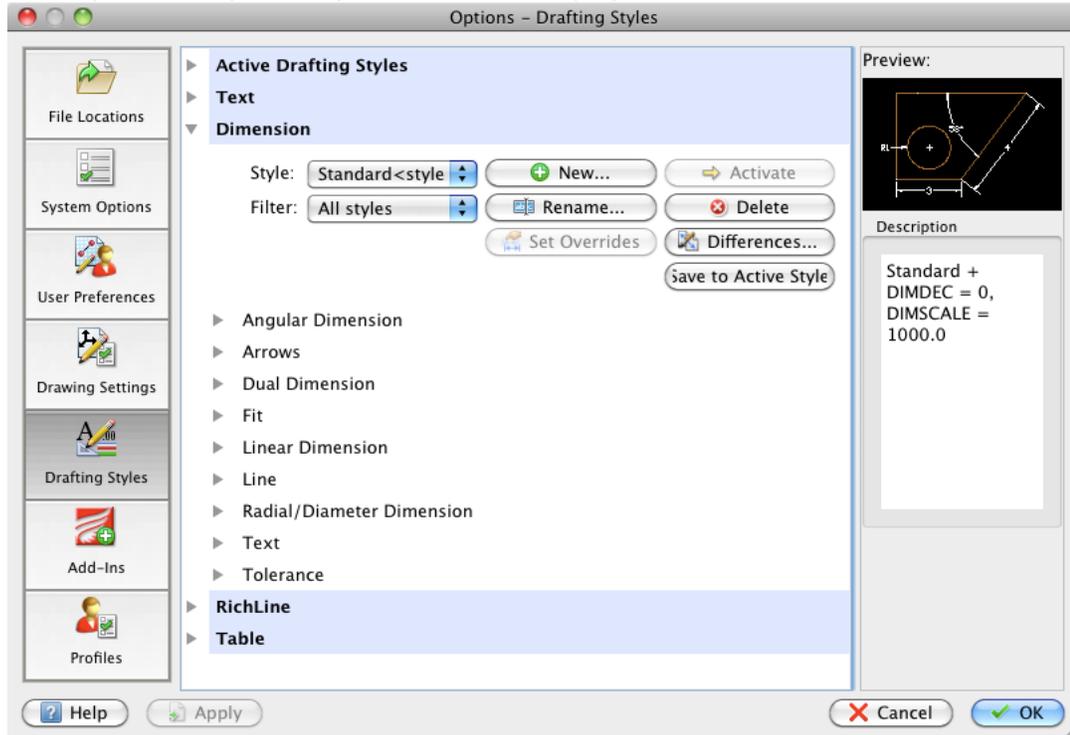
The look of dimensions is controlled globally by dimension styles, just as the look of text is controlled by text styles. The difference is that there are many, many more options for dimensions

than for text.

When you used the DimScale and DimDec commands earlier, you *overrode* their settings in the default dimension style found in every drawing, “Standard” dimension style.

There are three ways to access dimension styles: (a) through the DimensionStyle command’s dialog box, (b) through the Properties palette, and (c) at the command line. In this section, you’ll familiarize yourself with the DimensionStyle command and its dialog box.

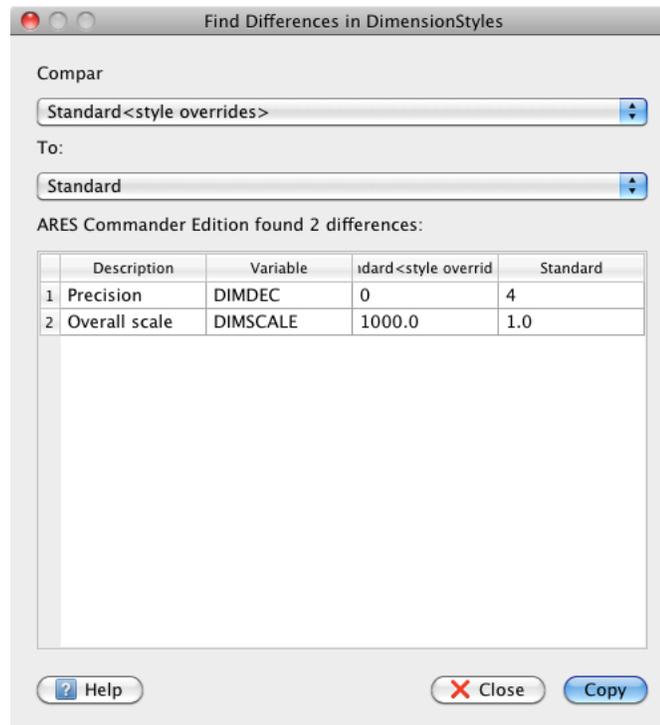
1. Enter the **DimensionStyle** command using one of the methods listed above. Notice that the Options dialog box is opened at the Drafting Styles section.



2. At the top are a number of buttons for creating new styles, renaming them, and so on — just as with text styles.

Notice that the name of the active style is “Standard<style overrides>.” This alerts you that the style has been overridden in one or more areas, which is what you did earlier with the DimScale and DimDec commands.

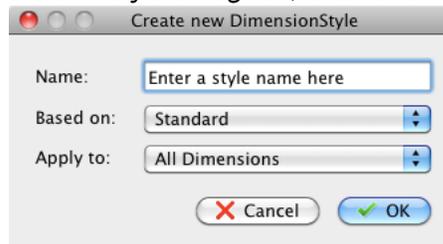
To see the difference between the Standard style and the overrides, click the **Differences** button. Notice that a new dialog box lists the differences between the two dimension styles. You can compare the differences between any two whose names you choose from the Compare and To droplists.



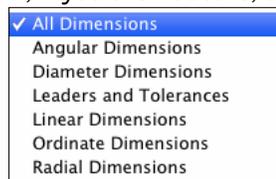
(Click **Copy** to copy the list of differences to the Clipboard; from there, you can paste the data into a word processor or spreadsheet.)

Click **Close** to return to the Options dialog box.

3. To create a new dimension style, you modify an existing one, like this:
  - a. Click **New**.
  - b. In the Create New Dimension Style dialog box, enter a name for the new style.



- c. From the **Based On** droplist, choose an existing style as the basis for the new one. For this tutorial, choose “Standard.”
  - d. In the **Apply To** droplist, decide if the new style applies to all dimensions or just certain types, as illustrated below; if you are not sure, choose **All Dimensions**.



- e. Click **OK**.
  - f. Back in the Options dialog box, change the dimension options as necessary. I show the names of all of options on the following pages.
  - g. Click **Activate**, and then **OK** to close the dialog box.

Any dimension you draw now takes on the new style; existing dimensions keep their old style. You can use the RebuildDimension command to update dimensions.

### Dimension Style Settings in iCADMac

The following figures illustrate all settings available for dimension styles. You access these settings through the DimensionStyle command.

▼ Angular Dimension

▼ Angular dimension settings

Format:

Precision:

Zeros display:

Hide leading zeroes

Hide trailing zeroes

▼ Arc length symbols

Before dimension text

Above dimension text

None

▼ Arrows

Start arrow:

End arrow:

Leader arrow:

Size:

▼ Dual Dimension

Show dual dimensions

Dual dimension settings

Format:

Precision:

Multiplier for converting units:

Round to the nearest:

Prefix:

Suffix:

Zeros display

Hide leading zeroes

Hide trailing zeroes

Hide if 0'

Hide if 0"

Insertion

After primary units

Below primary units

▼ Fit

▼ Geometry

When space is limited, move selected entity outside of extension lines:

Automatic

Arrows

Text

Text and arrows

Keep text between extension lines

Hide arrows

▼ Dimension text

When dimension text is not in the default position, move it:

Above the dimension line with leader

Above the dimension line without leader

Next to the dimension line

▼ Dimension scale

Scale factor:

Scale dimensions according to sheet

▼ Additional options

Dimension lines between extension lines

Specify text placement

▼ Linear Dimension

Format:

Precision:

Fractional display:

Decimal separator:

Round to the nearest:

Prefix:

Suffix:

Measurement scale:

Scale factor:

Zeros display:

Follow sheet dimensions

Hide leading zeroes  Hide if 0'

Hide trailing zeroes  Hide if 0"

▼ Line

▼ Dimension line settings

Style:

Weight:

Color:

Offset:

Distance past start arrow:

Hide:  Dimension line 1  
 Dimension line 2

▼ Extension line settings

Styles

Extension line 1:

Extension line 2:

Weight:

Color:

Offset:

Distance past dimension lines:

Hide:  Extension line 1  
 Extension line 2

Fixed length  
Length:

▼ Radial/Diameter Dimension

▼ Center mark display

None      Size:

As mark

As centerline

▼ Radius dimension jog

Angle:

▼ Text

▼ Text settings

Style:  

Color:

Fill:

Height:

Fractional scale:

Frame dimension text

▼ Text position

Horizontal:

Vertical:

Offset from &dimension lines:

▼ Text alignment

Use ISO standard

Align horizontally

Align with dimension lines

▼ Tolerance

Tolerance settings

Calculation:

Precision:

Maximum value:

Minimum value:

Scale:

Vertical text justification:

Zeros display:

Hide leading zeroes       Hide if 0'

Hide trailing zeroes       Hide if 0"

Dual dimension

Precision:

Zeros display:

Hide leading zeroes       Hide if 0'

Hide trailing zeroes       Hide if 0"

## Summary

This chapter showed how to scale, place, and format dimensions. In chapter 11, you learn about other forms of dimensions, such as ones with both metric and imperial text.

## Chapter 10

# PDF Format Import/Export

### In This Chapter

- Exporting your drawings in the PDF format
- PDF file conversion into DWG with Layer supported

**The PDF format** is a comfortable solution for publishing your documents that can be read through Acrobat Reader. Acrobat Reader is a currently used tool on the majority of computers and freely available for download on the Internet. iCADMac creates PDF files corresponding to the view chosen for printing. Your drawings can be delivered to customers and suppliers using this easy and reliable format.

The PDF file's parameters can easily be configured.

Drawings can be exported to create Layered PDF with the same Layers schema visible into the DWG

### To export a drawing to a PDF file

1 Do one of the following:

- Choose File > Export > Export PDF.
- Type export, press Enter and choose Portable Document Format (pdf).

2 Specify the name of the file you want to create.

3 Click Save.

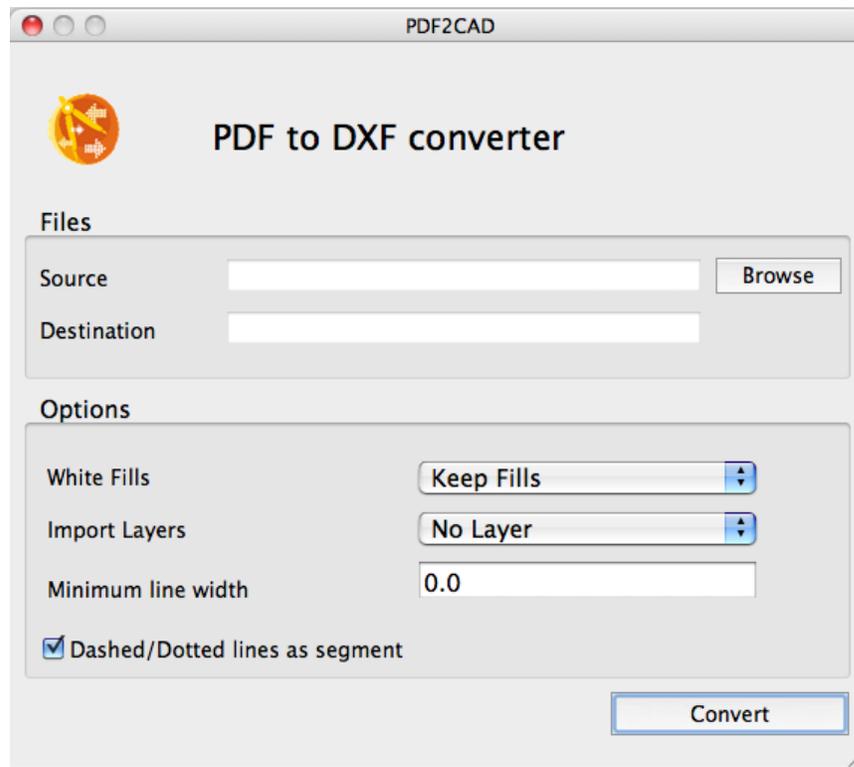
4 Choose the entity-selection method, and then create a selection set that contains the entities you want to export.

5 When you have finished selecting entities, press Enter.

6 Choose the options for how you want to export to the PDF file.

7 Click Export.

In this chapter, you also learn how to transform PDF files into DWG drawings with iCADMac.



## One click conversion

**Command:** PDF2CAD

**Menu Bar:** Add-Ons | Convert PDF to DXF

iCADMac converts vector PDF engineering drawings into the DWG/DXF format as editable objects. It accurately reproduces lines, shapes and text strings.

Thanks to our PDF2DWG module, you can create a DWG file by importing a PDF vector file and use it as a standard iCADMac drawing.

PDF files, having been edited on iPad, can be imported and converted to the DWG format for iCADMac. This can be a useful, time saving tool by importing PDF component drawings that you can readily find on the Internet.

Compatible with Layered PDF, it preserves the same Layers schema existing in the PDF iCADMac converts vector PDF engineering drawings into the DWG/DXF format as editable objects. It accurately reproduces lines, shapes and text strings.

1. Enter the **PDF2CAD** command.
2. In the dialog Pdf2CAD click Browse button to choose source file.
  - a. From the window Open, choose the PDF file to be converted in DXF and click the Open button:
  - b. The name and path of the generated DXF file will automatically appear in the box Destination
3. Specify the needed settings for:
  - Min. Line Width.
  - Dashed Line as segments.
  - Fill.

4. Click on the **Convert** button.

At the end of the conversion procedure, the resulting drawing will be automatically opened in iCADMac.

NOTE The name and path of the generated DXF file is the same of the original PDF.

## Summary

In this chapter you learned how to convert vector PDF files into DWG CAD -Editable format.

ATTENTION If the original PDF file doesn't contain vectorial information, the result of conversion will be a DXF with some attached raster images. The most common example of that result is a PDF made by the scanning of an image.

In the following chapter, you learn how to insert raster images into iCADMac, correct them for rotation and scale, and then trace over them to redraw the scanned paper drawings.



Part III

# 2D & 3D Mechanical Design



## Chapter 11

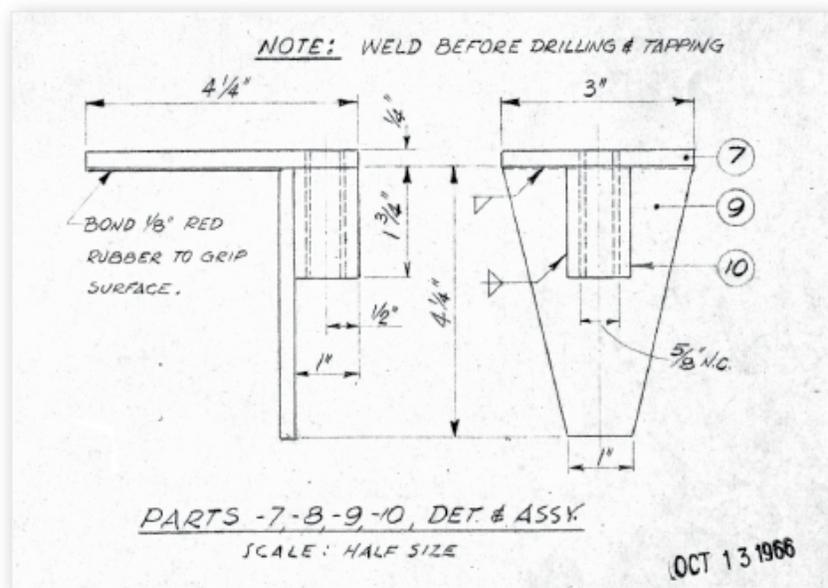
# Scan and Trace

### In This Chapter

- Scanning paper drawings
- Importing scanned image files
- Correcting skewed images
- Tracing over scanned drawings

**Some companies have** rooms filled with paper drawings created in the days before CAD. When these drawings need to be edited, the necessary manual drafting tools may no longer be available. In these cases, paper drawings are scanned, imported into CAD, and then traced over with regular drawing commands.

In this chapter, you learn how to insert raster images into iCADMac, correct them for rotation and scale, and then trace over them to recreate the drawing.



Hand-drafted drawing courtesy Herbert Grabowski

Paper drawings come in a variety of sizes, ranging from the small A4 (210 × 297mm) to the large A0 (841 × 1189mm) in metric or ISO/DIN sizes. In imperial units, standard paper sizes range from A (8.5 × 11") to E (36 × 48").

## Key Terms in This Chapter

- Chamfer** — replaces two intersecting lines with a single line.
- Dual Dimension** — reports dimensions in metric and imperial units.
- Displacement** — specifies the relative distance to move entities.
- Entity LineStyle Scale** — specifies the scale factor for individual entities in a drawing.
- Frames** — refers to the rectangles around images.
- Full Paths** — specifies the drive and full path to files.
- Global LineStyle Scale** — specifies the scale factor for all line styles in a drawing.
- Line Styles** — defines repeating pattern of dashes, dots, and gaps for entities.
- Move** — moves entities by a specified distance.
- Reference Images** — refers to the raster image files attached to drawings.
- Relative Paths** — describes the path from one file relative to a second file.
- Rotate** — turns entities by a specified angle.
- Scale** — resizes entities larger or smaller.
- Scanner** — converts paper drawings into raster files.

## Abbreviations

- A** — imperial size of standard letter paper, 8.5" x 11".
- A0** — metric size of largest paper sheets, 841 × 1189mm.
- A4** — metric size of standard letter paper, 210 × 297mm.
- DIN** — Deutsche Industrie Norm, German industrial standards.
- E** — imperial size of largest paper sheets, 36" x 48".
- ISO** — International Organization of Standards.
- TIF** — file extension of raster images stored in TIFF format (tagged image file format).

## Commands

<b>Command</b>	<b>Shortcuts</b>	<b>Menu Selection</b>
AttachImage	imageattach	Insert   Reference Image
Chamfer	cha	Modify   Chamfer
DisplayImageFrame	...	... <sup>1</sup>
LineStyle	lt, linetype	Format   Line Style
LtScale	...	...
Move	m	Modify   Move
Rotate	ro	Modify   Rotate
Scale	sc	Modify   Scale

<sup>1</sup> No menu selection.

Low cost, everyday office scanners, such as those built into desktop inkjet printers, are suitable for scanning A4- and A-size drawings. (The HP all-in-one printer-scanner shown at left is a typical example.)



Scanners for larger drawings are much more expensive, costing as much as \$10,000 for monochrome models and more for color ones, such as the floor-standing model from Canon-Oce illustrated at right.

An alternative to purchasing a scanner is to use a local scanning service bureau, which typically charges customers by the square foot.



## Importing Scanned

### Images

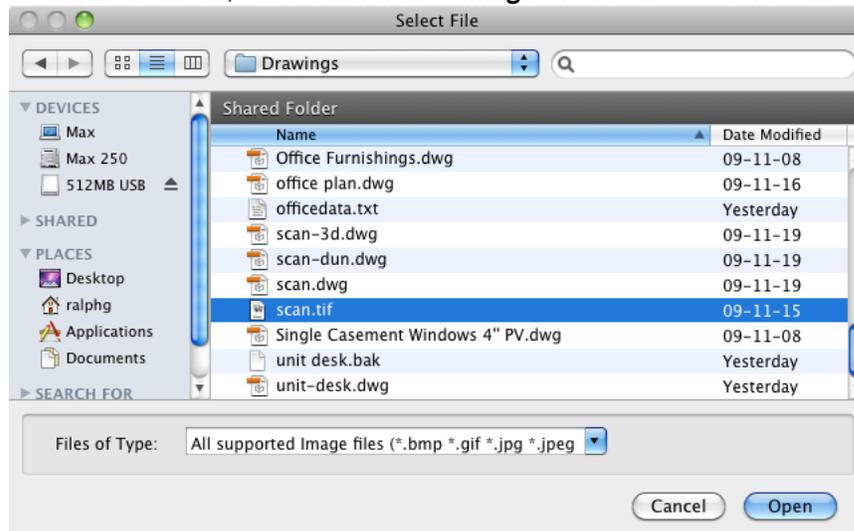
**Command:** AttachImage

**Aliases:** atimg, attaching, iat, imageattach

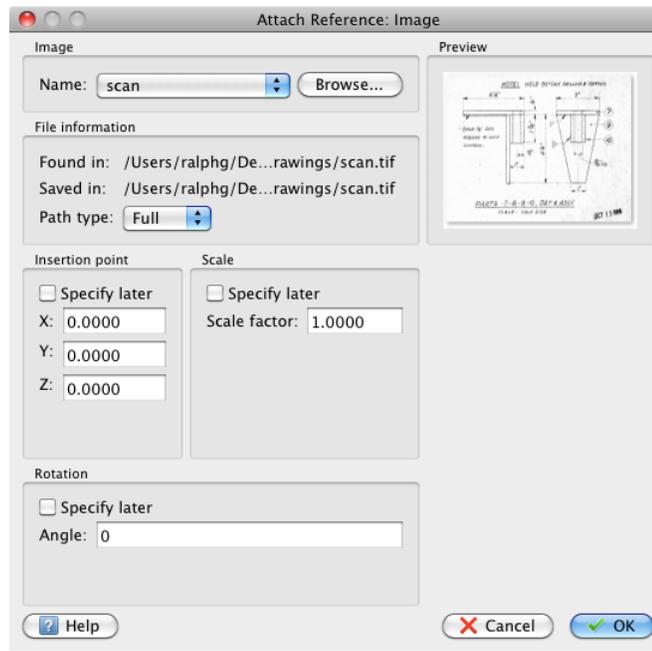
**Menu Bar:** Insert | Reference Image

Our paper drawing is dimensioned in inches, so you should start the new CAD drawing with Imperial units. (You will learn how to apply dual Imperial-metric dimensions later in this chapter.)

1. Enter the **New** command, and then choose the *standard.dwt* template file.
2. To place the scanned image into the drawing, follow these steps:
  - a. From the **Insert** menu, choose **Reference Image**. Notice the Select File dialog box.



- b. Choose the *scan.tif* file, and then click **Open**.
3. Notice that the Attach Reference Image dialog box looks very similar to the one for inserting blocks.



Make the following changes to the dialog box:

- a. Change Path Type to **None**.

#### Summary of AttachImage Dialog Box Options

Command: attachimage  
 (Displays Select File dialog box; choose an image file.)  
 (Displays Attach Reference Image dialog box.)

**Name** displays the file name of the selected image; click the droplist to choose from among previously-attached images.

**Browse** selects image file names from the Select File dialog box. Choose from among *.bmp*, *.gif*, *.tif*, *.jpg*, and *.png* file types.

**Path Type** specifies how iCADMac stores the path to the image file:

- **Full** stores the complete drive and path name.
- **Relative** stores the relative path to the image file.
- **None** stores the file name only, no path.

**Insertion point** specifies the location of the lower left corner of the image:

- **Specify later** prompts you for the insertion point in the Command window after the dialog box is closed; allows for interactive placement.
- **X, Y, Z** specify the x, y, and x coordinates for the image.

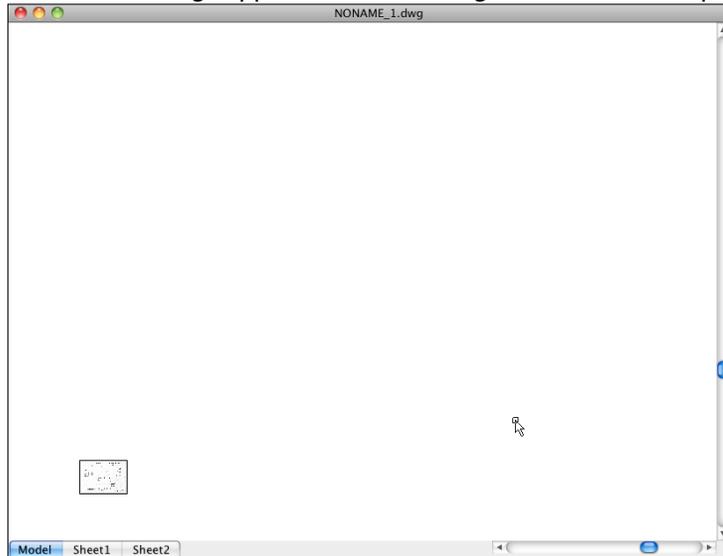
**Scale** specifies the size of the image:

- **Specify later** prompts you for the scale factor in the Command window after the dialog box is closed; allows for interactive sizing.
- **Scale factor** specifies the size of the image; values larger than 1 increase the size; those between 0 and 1 shrink the size.

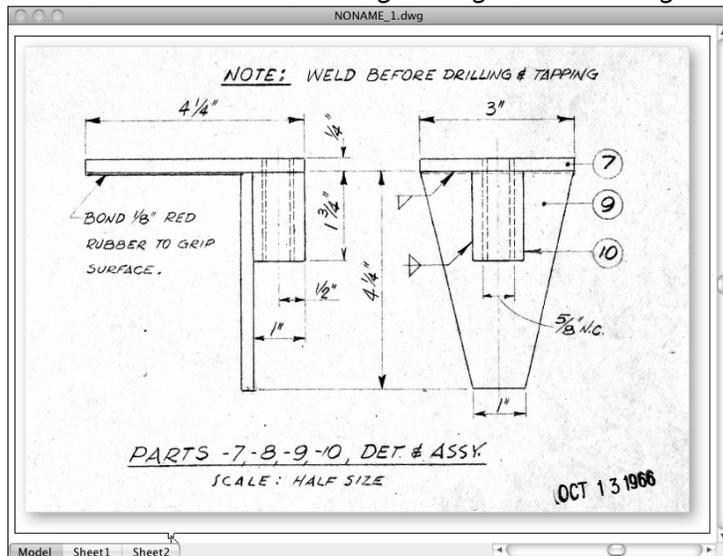
**Rotation** specifies the angle of the image:

- **Specify later**: prompts you for the angle in the Command window after the dialog box is closed; allows for interactive rotation.
- **Angle** specifies the angle of rotation; positive angles rotate the image counterclockwise, starting from the positive x axis (0 degrees).

- b. Ensure the Insertion Point is set to **0,0,0**.
  - c. Ensure that the Scale is set to **1**.  
(In both cases, the **Specify later** option should be turned off.)
4. Click **OK**. Notice that the image appears in the drawing, but that it looks quite small.



5. Use the **Zoom Fit** command to make the image as large as the drawing window.



6. Save the drawing as *scan.dwg* with the **SaveAs** command.

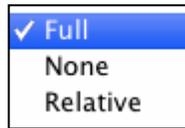
**TIP** When you picked the Insert | Reference Image menu item, iCADMac executed the AttachImage command, the one that displays dialog boxes. EditImage is the analogous command that displays prompts in the command window. A third command, References, lets you control images with a palette.

When images are placed in a drawing, they are called “attached” or “referenced” images. This is because images are not a part of the drawing; the drawing contains only a reference to the image file, specifically the path, such as “C:\Users\Documents\My Scans.” Each time iCADMac opens this drawing, it refers to the path to find the image, and then loads the file.

Sometimes iCADMac cannot load the file, because the image file was moved out of the folder, or because the drawing was opened on another computer that lacks the image file.

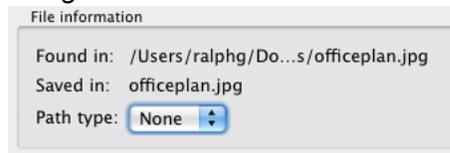
### About Path Types

iCADMac remembers paths to source image files in these ways:



**Full** remembers everything from the drive name to the file name. This option is best when the image files are stored on a computer different from the drawing, because the full path can include the name of the networked computer. This option is the worst when the DWG file is moved to a different computer, because the path instantly invalid.

**None** remembers only the image's file name. This option is best when the image file is in the same folder as the drawing file.



**Relative** remembers the relative path between the DWG file and the image file. Use this path type if the drawing and image are moved (together) among computers; keep the image files in a folder inside the folder holding the DWG file. This option does not work when the DWG and image files are on different computers; it also does not work when the drawing is not yet saved, because iCADMac does not know the drawing's location.

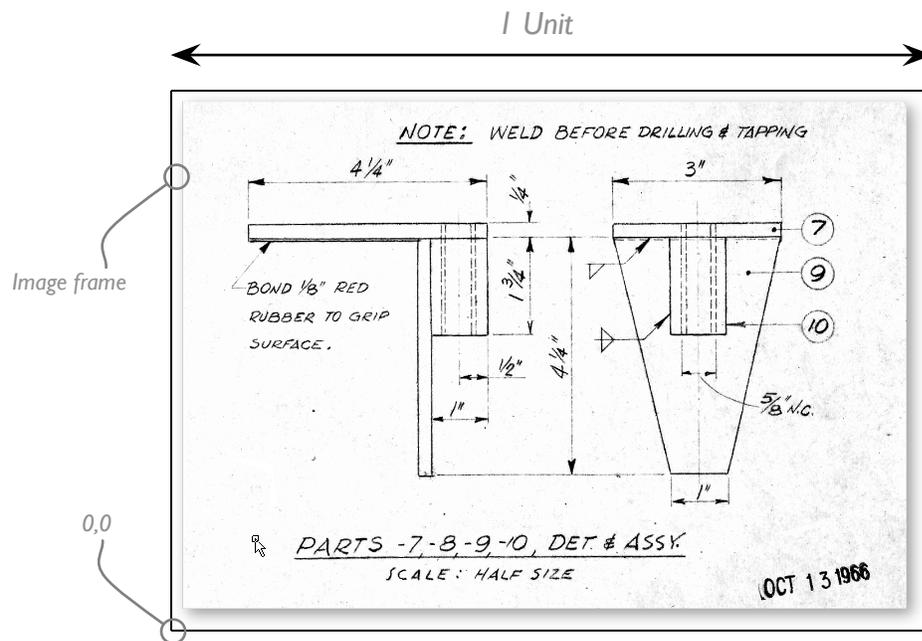
### Examining the Attached Image

Now that the scanned image is in the drawing, look at some of its characteristics:

#### Which Image Format to Use?

iCADMac supports a number of image file types; given a choice, choose TIFF format (.tif files).

<b>Extension</b>	<b>Name</b>	<b>Benefits; Drawbacks</b>
BMP	Bitmap	Output by some programs; creates very large files.
GIF	Graphics Interchange Format	Popular last decade; limited to 256 colors.
TIF	Tagged Image Format	Used in desktop publishing; no limitations.
JPG	Joint Photographic Group	Extremely small file sizes through high compression, which creates distortions.
PNG	Portable Network Graphics	Designed to be a distortion-free alternative to JPG, but not common.



The lower left corner is at 0,0 (insertion point).

The width is 1 unit (1 inch, in this case).

The image is surrounded by a black rectangle, called the “frame.” To select the image, you must click on the frame; you cannot select the image by clicking on the image!

**TIP** You can turn the image frames off and on with the **DisplayImageFrame** command. It is normal to turn off the frames prior to plotting so that the excess black rectangles do not appear on the paper. When frames are turned off, however, you can no longer select the image; for this reason the frame is usually left on at all other times.

When paper drawings are scanned, the image can end up being at the wrong scale and not perfectly straight. You fix these two problems in the next section.



## Rotating Entities

**Command:** Rotate

**Alias:** ro

**Menu Bar:** Modify | Rotate

**Tool Matrix:** Modify | Rotate

Paper drawings are often not aligned precisely when they are scanned. You can use the Rotate command to straighten them. (Many other editing commands also work with reference images, such as Copy, Move, Pattern, and Erase.)

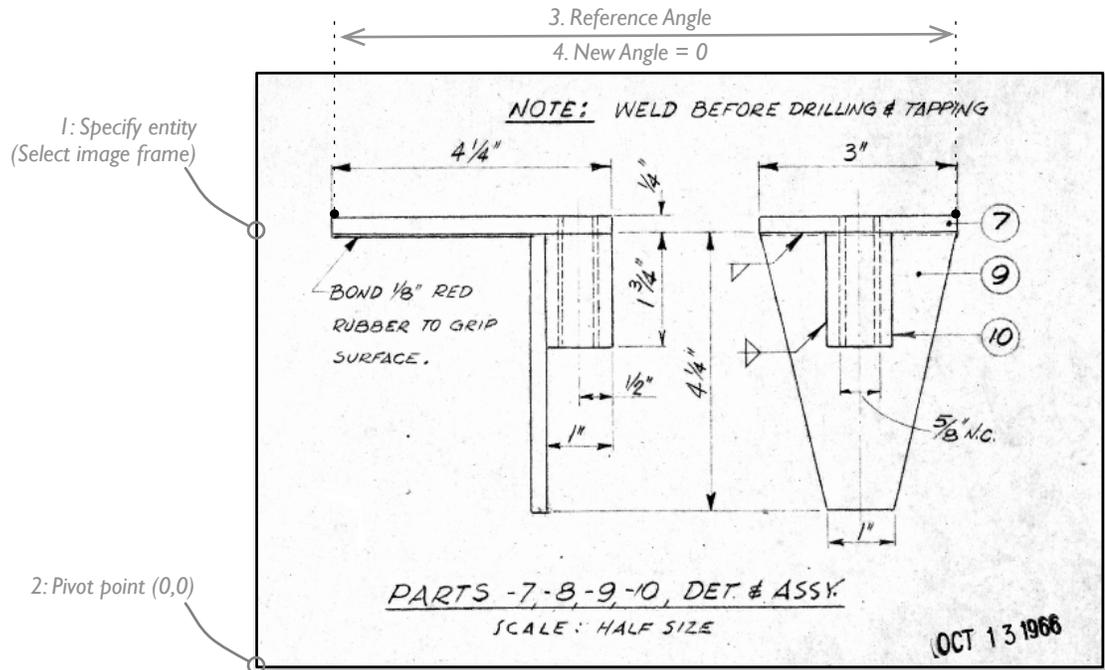
To rotate the attached image, follow these steps:

1. Start the **Rotate** command using one of the methods listed above:  
Command: rotate  
Active positive angle in CCS: DIRECTION=counterclockwise BASE=0
2. Select the image by clicking its frame, or by entering “l” (short for *last* object drawn), and then pressing **return** to end entity selection:  
Specify entities> l

1 found, 1 total  
Specify entities» (Press return to continue.)

3. The *pivot point* indicates the spot about which the image is rotated. For this tutorial, it doesn't matter much where it is located, but 0,0 is a convenient spot.

Specify pivot point» 0,0



4. To specify the rotation angle, use the **Reference** method. (The word *reference* in this command is unrelated to the image being a reference image.) This method is a bit tricky; I usually end up trying it a couple of times before getting it right. In summary, here is how the Reference method works:

**Step 1: Specify Reference Angle.** Pick two points in the drawing that indicate the *current* angle. For this tutorial, pick the endpoints of any line that should be *horizontal*, but isn't.

**Step 2: Specify New Angle.** Enter the angle you want; for this tutorial, the angle is 0, because you want horizontal-looking lines to become horizontal.

(You use this Reference system later to scale the image to the correct size.)

Follow the above steps to straighten out the image:

- a. Enter **r** to specify the Reference option:

Options: Reference or  
Specify rotation angle» r

- b. Pick two points to indicate the current angle. In this case, pick a point at either end of the two horizontal lines illustrated above.

Default: 0  
Specify reference angle» (Pick left end of the 4-1/4" line.)  
Specify second point» (Pick right end of the 3" line.)

- c. Now enter the angle that this horizontal line should be, 0 degrees:

Default: 0  
Specify new angle» 0

5. Notice that the image rotates slightly; the hand-drafted content of the image is now orthogonal. The image frame, however, now looks slightly crooked. You can turn off the frame with the `DisplayImageFrame` command. Enter **n**:

Command: displayimageframe  
 Default: Above image  
 Confirm: Display image frames?  
 Specify No, Above image, or Below image» n

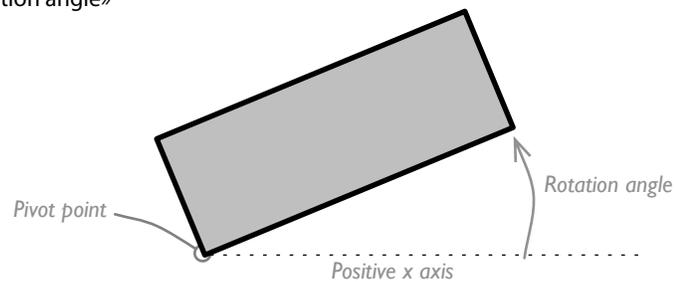
**TIP** You cannot use entity snaps with images, because images are made up of small squares (called “pixels,” short for picture elements) of varying shades of gray. They are not vector entities, and so there is nothing to which to snap.

The figure below is a zoomed-in view of the image in the upper left corner; lines, text, and arrowheads are clumps of pixels that fool the eye into seeing lines and text.



#### Summary of Rotate Command Options

Command: rotate  
 Active positive angle in CCS: DIRECTION=counterclockwise BASE=0  
 Specify entities»  
 Options: Reference or  
 Specify pivot point»  
 Specify rotation angle»



**Pivot Point** specifies the point about which the selected entities rotate.

**Rotation Angle** specifies the angle by which to rotate the entities; measures angle from the positive x axis (0 degrees).

**Reference** changes the starting angle (overrides 0 as the starting angle):  
 Specify reference angle» (Pick two points, or enter an angle.)

Use the **Units** command to change the default direction (counterclockwise) and base point (0,0).



## Scaling Entities

**Command:** Scale  
**Alias:** sc  
**Menu Bar:** Modify | Scale  
**Tool Matrix:** Modify | Scale

The scanned image is just one inch wide, and so you need to change it to its actual size. This is done with the Scale command. You can use the 4-1/4"-long line as the reference; you tell iCADMac you want the 4-1/4"-long line to be 4-1/4" long.

Even though the drawing was originally drafted at half-scale, you can ignore this historical fact, because in CAD you use actual dimensions for the drawing. (In the days of hand drafting, the part would have been drawn at half-size so that it would fit the paper.)

To correct the size of the reference image, follow these steps:

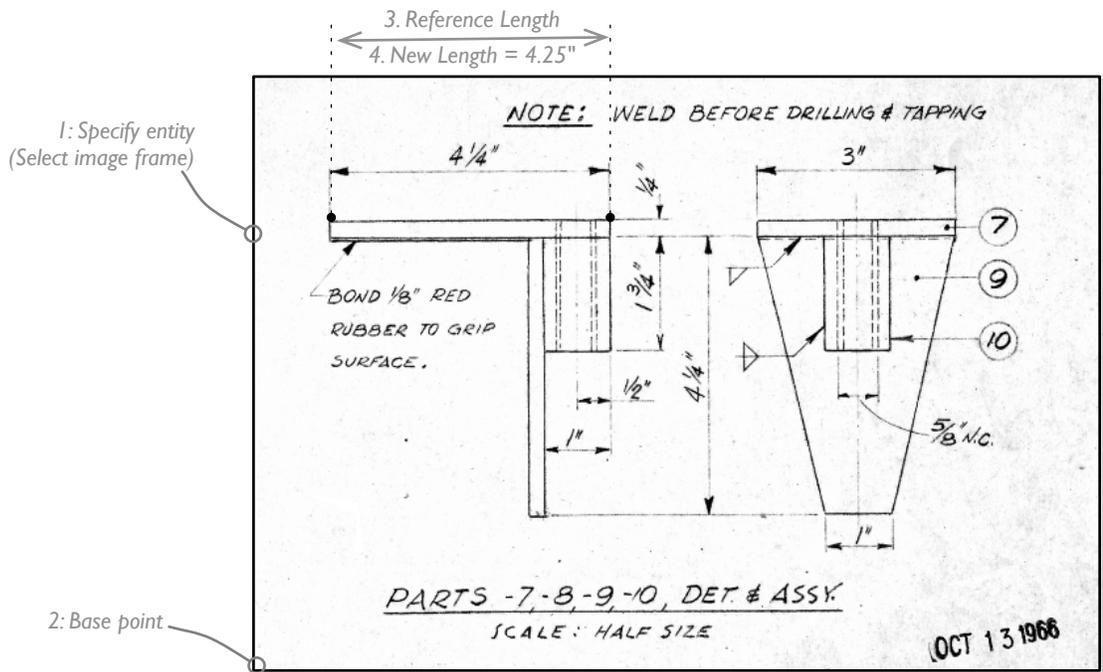
1. Start the **Scale** command using one of the methods listed above, and then select the image again with the **I** option.

Command: scale  
 Specify entities» I  
 1 found, 1 total  
 Specify entities» (Press return to continue.)

(If you turned off the frame, then you have to use the Last option to select the frame, since you can no longer pick it directly.)

2. The *base point* is the same as the Rotate command's pivot point, 0,0.

Specify base point» 0,0



3. Again, you use the Reference option. This time, you pick two points that represent the future (or *true*) length, and then enter the length you want it to be:

Options: Reference or  
 Specify scale factor» r  
 Default: 1.0000  
 Specify reference length» (Pick one end of the 4-1/4" line.)

Specify second point» (Pick the other end of the line.)  
 Default: 1.0000  
 Options: Points or  
 Specify new length» 4.25

**TIP** For more accurate picking of the ends of the 4-1/4" line, scroll the mouse's wheel to zoom in closer to the line.

5. Notice that the image has become much larger. Use the **Zoom Fit** command to refit the image to the drawing window.
6. To prove to yourself that the image is now at the correct angle and size, use the GetDistance command:
  - Command: getdistance
  - Specify start point» (Pick one end of the 4-1/4" line.)
  - Specify end point» (Pick the other end of the line.)

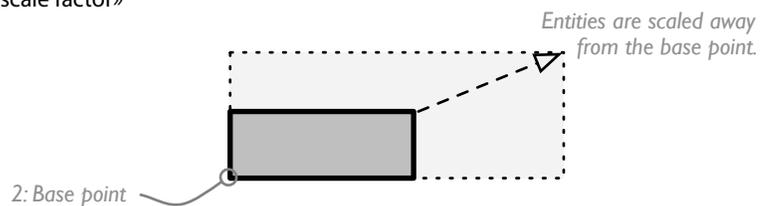
iCADMac reports that the distance is (almost) 4.25 units long, and that the angle is 0 degrees:

Distance = 4.2837, Angle in XY Plane = 0, Angle from XY Plane = 0  
 Delta X = 4.2837, Delta Y = 0.0000, Delta Z = 0.0000

7. Press **command+S** to save your work.

#### Summary of Scale Command Options

Command: scale  
 Specify entities»  
 Specify base point»  
 Options: Reference or  
 Specify scale factor»



**Base Point** is the point relative to which the selected entities are scaled.

**Scale Factor** specifies the amount by which to size the entities:

- Values larger than 1 increase the size; e.g. 2 doubles the size.
- Values between 0 and 1 decrease the size; e.g. 0.5 halves the size.

**Reference** scales without you needing to know the scale factor:

Specify reference line» (Pick two points.)

When the base point is not on the entity, iCADMac determines the scale factor from the distance between the base point to all points on the entity.

## Tracing Over Scanned Images

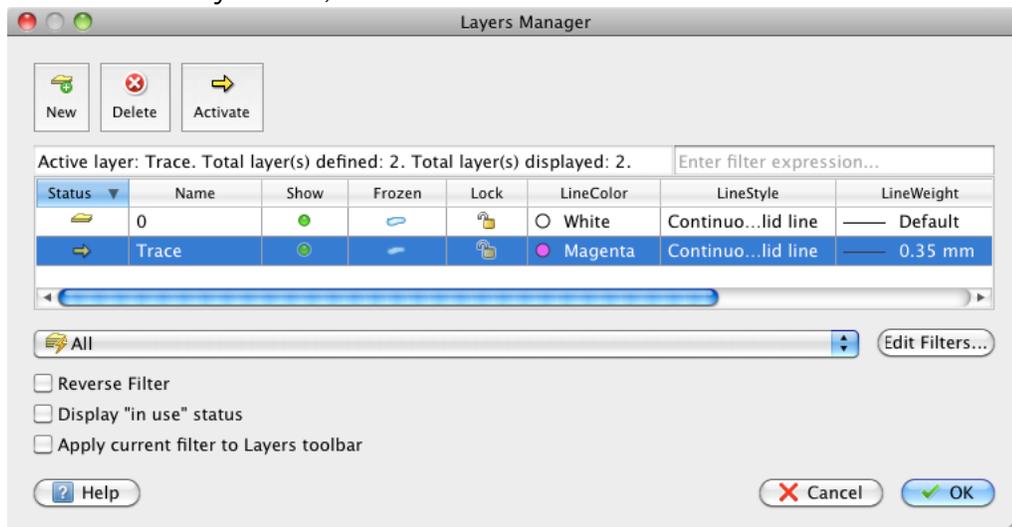
To “convert” the paper drawing to vectors, you trace over the image. There is a difficulty in this: paper drawings were created by hand, and so contain inherent errors. As I noted earlier in Chapter 9, the “DO NOT SCALE” warning meant that we should not measure distances on the paper drawing with a ruler.

Because paper drawings are inexact, you employ the following strategy to trace over them with lines and other CAD entities: begin the tracing at one corner, and then use the drawing’s printed dimensions to determine the size of CAD entities.

### Layers and Rectangles

In this tutorial, you create a new layer for tracing over the scan, and then use several drawing and editing commands to replicate the paper drawing.

1. Create a new layer, as follows:
  - a. Enter the **Layer** command to open the Layers Manager dialog box.
  - b. Click **New**, and then rename the new layer “Trace.”
  - c. To make the CAD entities stand out, change the LineColor of layer Trace to **Magenta**.
  - d. Change the LineWeight to a heavier value, such as **0.35mm**.
  - e. To make the layer active, click **Activate**.



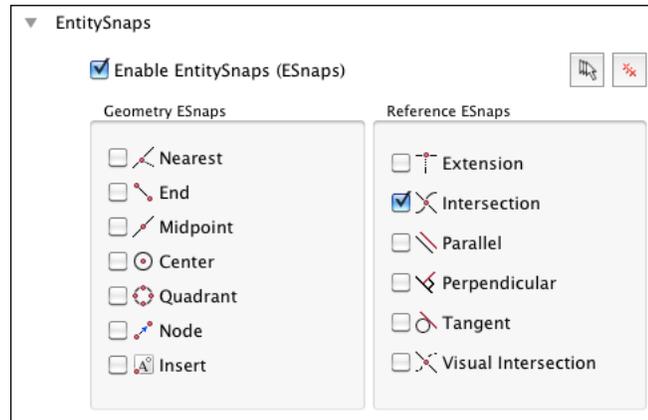
- f. Click **OK** to close the dialog box.
2. On the status bar, click **LWT** to turn on lineweights.

Together with the magenta color, the heavier lineweight makes the lines from the new CAD drawing easier to see over top of the scanned image.

## Drawing the Side View

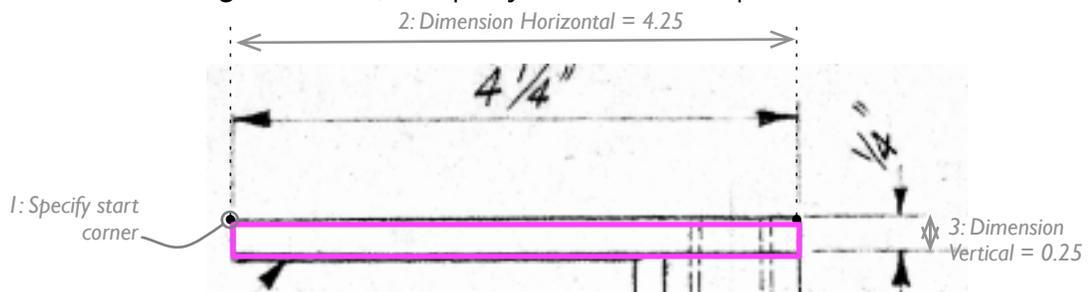
The scanned drawing has two views, a side view on the left, and a front view on the right. The drawing of each view contains a number of rectangles, and so it is efficient to use the Rectangle command for the most of the drafting.

1. The rectangles connect with each other at intersections, so it makes sense to turn on INTersection entity snap, as follows:
  - a. On the status bar, right-click the **ESnap** button.
  - b. From the shortcut menu, choose **Settings**.
  - c. In the Options dialog box, turn on the following options:
    - **Enable EntitySnaps (ESnaps)**
    - **Intersection**



- d. Click **OK** to close the dialog box. On the status bar, the ESnap button should be become a dark gray color.
2. To create a CAD drawing accurately, you need to choose one point in the scanned image from which to start, and then employ the dimensions written on the drawing. Your start point is at the upper left corner, as shown in the figure below.

Start the **Rectangle** command, and specify the **Dimensions** options.

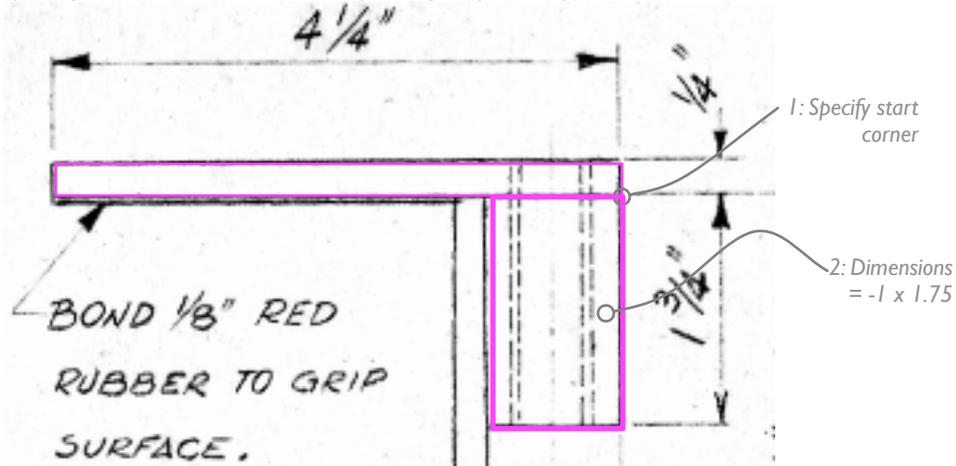


```
Command: rectangle
Options: Chamfer, Elevation, Fillet, Thickness, line Width or
Specify start corner» (Pick the point indicated in the drawing above.)
Options: Area, Dimensions, Rotation or
Specify opposite corner» d

Default: 10.0000
Specify horizontal dimension» 4.25
Default: 10.0000
Specify vertical dimension» -.25
```

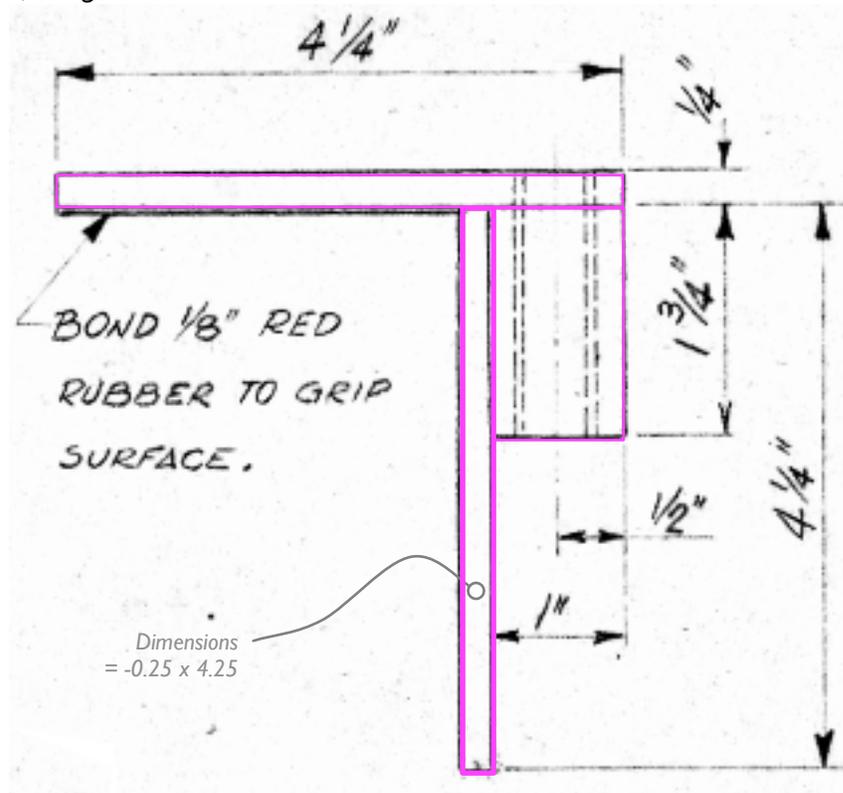
Using a negative value for the rectangle's vertical dimension draws it downwards from the start corner. Notice that the "perfectly" drawn CAD rectangle does not match the hand-drawn one; this is to be expected.

3. Attach the next rectangle to the first one, ignoring the inaccuracies in the handmade drawing. The dimensions indicate this rectangle is 1.75" tall by 1" wide; enter negative values (-1.75 and -1) to draw the rectangle downwards and to the left from the start point. Entity snap ensures the new rectangle begins precisely at an intersection of the first one:



Command: rectangle  
 Specify start corner» (Pick the intersection of the first rectangle.)  
 Specify opposite corner» d  
 Specify horizontal dimension» -1  
 Default: 10.0000  
 Specify vertical dimension» -1.75

4. Press the **spacebar** to repeat the **Rectangle** command, and then draw the third rectangle, using dimensions of **-0.25** and **-4.25**.



5. The 1/8" red rubber strip is the final rectangle, whose dimensions can be worked out as **-3** and **-0.125**. Draw it with the **Rectangle** command.
6. Press **command+S** to save your valuable work.

The dashed lines are drawn in a later part of this chapter.



## Drawing the Front View

**Command:** Move

**Alias:** m

**Menu Bar:** Modify | Move

**Tool Matrix:** Modify | Move

The front view is similar in that it consists of two rectangles. But it differs from the side view, because the rectangles are centered on each other, and there are some angled lines.

Drawing one rectangle centered on another is tricky. There are several approaches you could take, but I feel the easiest one is known as “draw’n move”: draw it in a convenient place, and then move it into position.

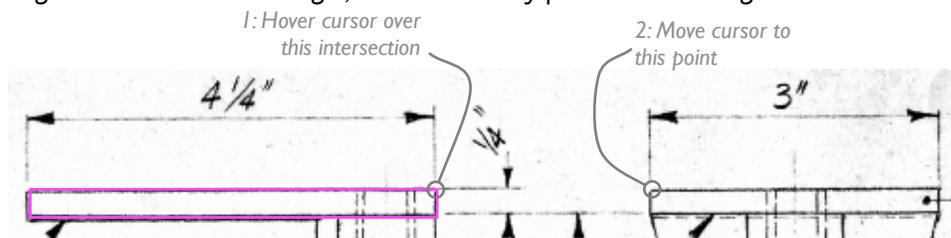
The other issue is that the front view should be aligned precisely with the side view. The solution is to draw the initial rectangle, and then use *entity tracking* to line it up with a rectangle in the side view.

Both techniques are illustrated in this tutorial.

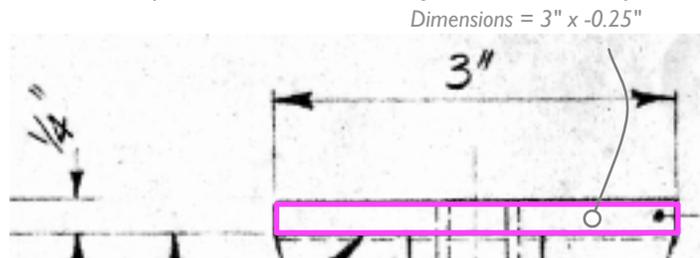
1. Turn on ortho by clicking **Ortho** on the status bar. Turn on entity tracking by clicking **ETrack**. Both buttons should look gray.



2. Use entity tracking to position the front view's first rectangle relative to those in the side view:
  - a. Start the **Rectangle** command.
  - b. At the 'Specify start corner' prompt, move the cursor so that it hovers over the upper right corner of the rectangle, as illustrated by point #1 in the figure below.



- c. Move the cursor horizontally to point #2, and then click. (They may be hard to see, but markers indicate the entity tracking snap points: a cyan + appears over point #1, and a magenta x over point #2.)
- d. Use the **Dimensions** option to size the rectangle: **3** horizontal by **-.25** vertical.



3. Turn off **Ortho** mode.

**TIP** The scan image is quite busy. When you find the image getting in the way of your drafting, freeze its layer, as follows:

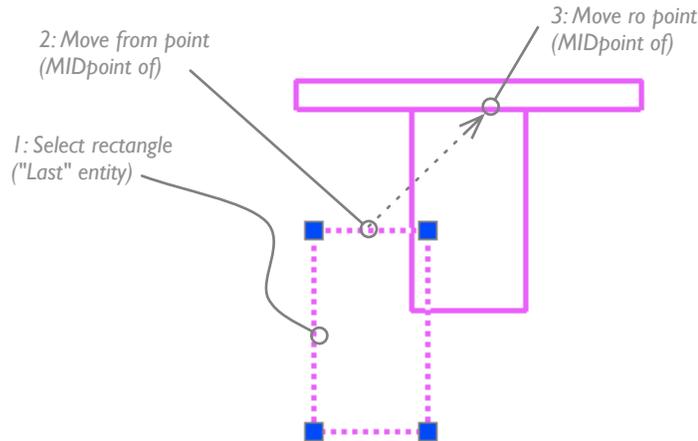
- a. In the Layers dialog box, choose layer 0.

Status ▾	Name	Show	Frozen	Lo
	0			
	Trace			

- b. Click the  puddle (a melted icicle!) icon next to layer 0.

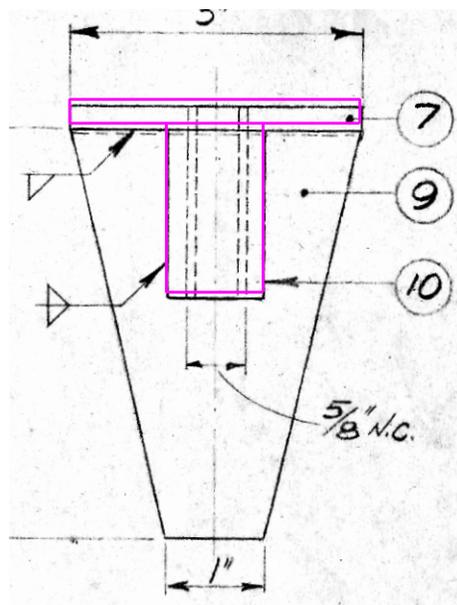
Notice that the scanned image disappears, and that the puddle icon turns into something that looks like  icicles; this indicates the layer is frozen (can be neither seen nor edited).

4. The second rectangle is centered below the first one. In this case, you draw the rectangle “anywhere,” and then move it into place:
- Draw a rectangle with dimensions **1** and **-1.75**. Its location in the drawing does not matter.
  - Start the **Move** command with one of the methods noted above:  
Command: move
  - Use the **Last** entity selection mode to select the rectangle you just drew:  
Specify entities> |  
Specify entities> (Press return to continue.)



- Use **MIDpoint** entity snap to grab the rectangle at point #2, as illustrated above:  
Default: Displacement  
Options: Displacement or  
Specify from point> mid  
of (Pick the middle of the rectangle’s top line.)
- Repeat **MIDpoint** entity snap to grab the other rectangle at point #3:  
Options: Enter to use from point as displacement or  
Specify destination> mid  
of (Pick the middle of the rectangle’s bottom line.)

Notice that the moved rectangle snaps into place.

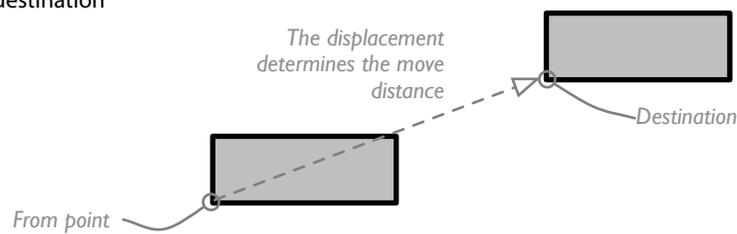


5. Press **command+S** to save the drawing.

**TIP** As an alternative, you could have used the **Copy** command to copy the existing 1x1.75 rectangle from the side view to the front view.

#### Summary of Move Command Options

Command: move  
 Specify entities»  
 Options: Displacement or  
 Specify from point»  
 Options: Enter to use from point as displacement or  
 Specify destination



**Displacement** specifies the relative distance to move the selected entities.

**From Point** specifies the starting point to determine the move distance; can use x, y or x, y, z coordinates.

**Enter** uses the distance between the original (0,0) and the 'From point' coordinates to determine the distance to move entities.



## Chamfering Entities

**Command:** Chamfer

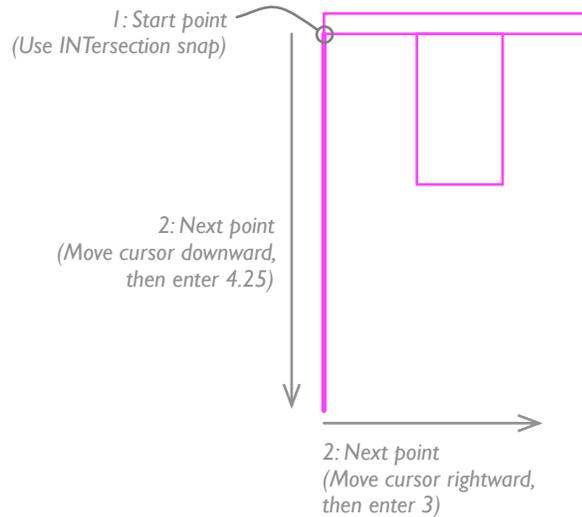
**Alias:** cha

**Menu Bar:** Modify | Chamfer

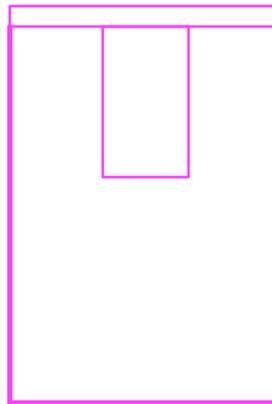
**Tool Matrix:** Modify | Chamfer

The trapezoid shaped part can be drawn by several methods. I feel the easiest is to draw three lines with direct distance entry, and then use the Chamfer command to create the angles.

1. Turn on **Ortho** mode, and then start the **Line** command.
2. Draw three lines using direct distance entry:
  - a. Move the mouse downwards. Notice that ortho mode keeps the line perfectly vertical.
  - b. On the keyboard, enter the length of the line, **4.25**, and then press **return**. Notice that iCADMac draws the line.



- c. Move the mouse to the right. Enter **3**, and then press **return**.
- d. Move the mouse upwards, and then enter **4.25** again.
- e. Press **return** twice to end the Line command. The three lines are in place.



A chamfer is like a fillet, except that two lines are connected with a straight line instead of an arc. Arc fillets are defined by radii; chamfer lines are defined by distances or by angles:



In this case, chamfer-by-distance is the easier approach. By examining the drawing's dimensions, you can determine that the two chamfer distances are 4.25 and 1.

3. Start the **Chamfer** command with one of the methods listed above.

Command: chamfer

- a. At the 'Specify first line' prompt, enter the **Multiple** option, because you have two chamfers to perform.

(Trim mode) Active chamfer Dist1 = 0.0000, Dist2 = 0.0000

Options: Angle, Distance, mEthod, Multiple, Polyline, Trim mode, Undo or

Specify first line» m

- b. To specify the chamfer distances, enter the **Distance** option:

Specify first line» d

Default: 0.0000

Specify first distance» 4.25

Default: 4.2500

Specify second distance» 1

- c. Now you chamfer the first pair of lines. Remember that the first chamfer distance (4.25) applies to the first line you pick (the vertical line):

Specify first line» (Pick 4.25"-long line.)

#### Summary of Chamfer Command Options

Command: chamfer

(Trim mode) Active chamfer Dist1 = 0.0000, Dist2 = 0.0000

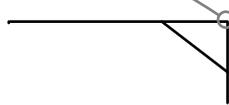
Options: Angle, Distance, mEthod, Multiple, Polyline, Trim mode, Undo or

Specify first line»

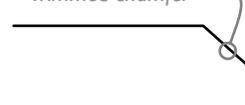
Options: Shift + select to apply corner or

Specify second line»

Untrimmed chamfer



Trimmed chamfer



**Angle** specifies a length and an angle of the chamfer.

**Distance** specifies two lengths for the chamfer; enter 0 for both to clean up intersecting lines.

**mEthod** toggles the default chamfer method: length and angle, or two lengths.

**Multiple** repeats the command to chamfer more than two entities.

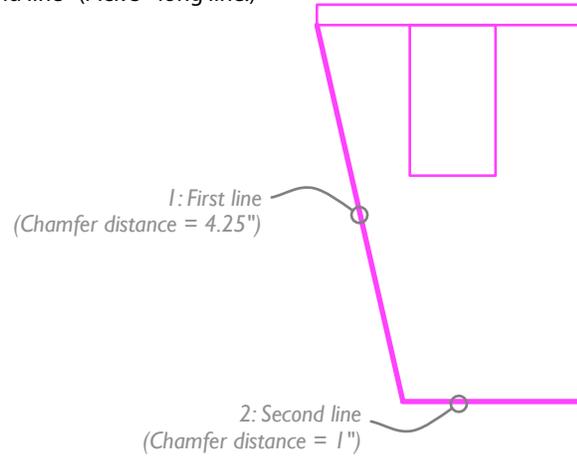
**Polyline** chamfers all vertices of polyline-based entities, such as polylines, polygons, and rings.

**Trim mode** toggles trimming of entities. When the mode is on, portions of entities beyond the chamfer are cut off; when off, the entities are preserved.

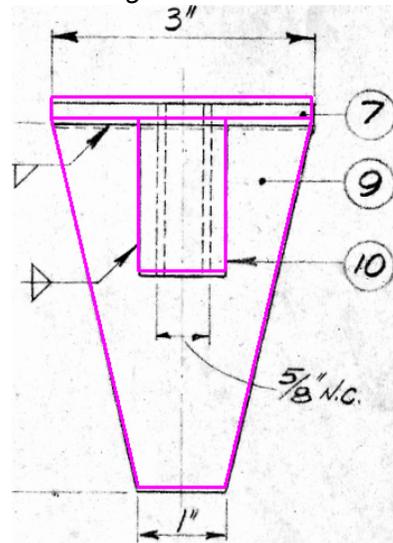
**Undo** undoes the previous chamfer when Multiple mode is active.

**Shift+select** applies chamfer distances of 0, overriding the default chamfer distances.

Options: Shift + select to apply corner or  
Specify second line» (Pick 3"-long line.)



- d. Repeat for the other side:  
Specify first line» (Pick 4.25"-long line.)  
Specify second line» (Pick 3"-long line.)
- e. Press **return** to end the Chamfer command:  
Specify first line» (Press return.)
4. Press **command+S** to save the drawing.



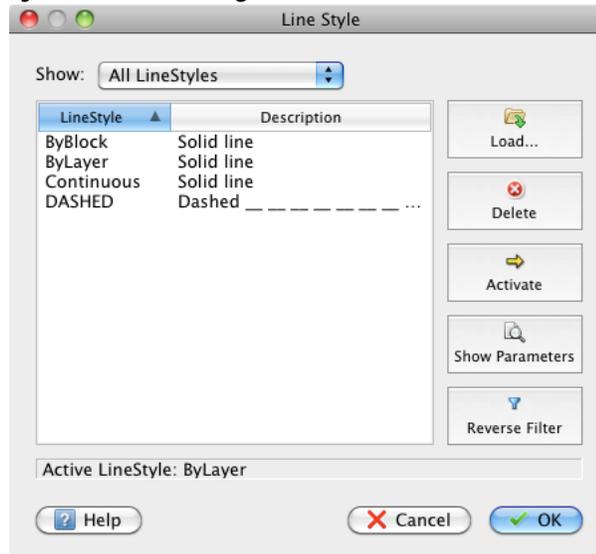
### Drawing Line Styles

**Command:** LineStyle  
**Aliases:** lstyle, linetype, ltype, lt  
**Menu Bar:** Format | LineStyle  
**Properties:** LineStyle

Some of the lines in the handmade drawing are dashed. Traditionally, dashes represent hidden lines. In this case, the dashed lines represent holes through the steel parts.

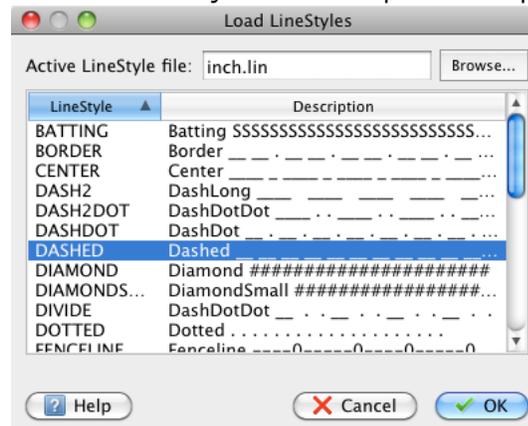
In iCADMac, dashed lines are drawn with the usual commands, such as Line and Offset. But two initial steps are required: (1) the dashed line style must be loaded into the drawing with the LinStyle command, and (2) the active line style (linetype) must be set to Dashed.

1. Follow these steps to load the Dashed line style into the drawing:
  - a. Start the **LineStyle** command using one of the methods listed above.



Notice the Line Style dialog box. It lists the names of line styles currently stored in the drawing — ByBlock, ByLayer, and Continuous are all solid lines.

- b. Click **Load**. Notice the list of line style names and pattern samples.



- c. Choose **DASHED**, and then click **OK** to load the line style into the drawing.
  - d. Back in the Line Styles dialog box, choose **DASHED**, and then click **Activate**.
  - e. Click **OK** to close the dialog box.

With the needed line style loaded, you can now determine how best to draw the dashed lines. I suggest using the Offset command.

From the drawing, you can determine the offset distances. The  $5/8$ " N.C. notation means that the hole is  $5/8$ " in diameter. (N.C. is the abbreviation used by the American National standard indicating a "coarse thread.")

## Summary of Line Style Dialog Box Options

Command: linestyle  
(Displays the Line Style dialog box.)

**Show** determines which line style names to display in the dialog box:

- **All LineStyles** lists all line styles loaded into the current drawing.
- **LineStyles in Use** lists those line styles used by entities and layers.
- **References LineStyles** listed line styles used by referenced drawings.

**Load** opens the Load LineStyles dialog box; choose one or more to load into the drawing.

**Delete** removes the selected line style from the drawing; in-use line styles cannot be erased.

**Activate** makes the selected line style active.

**Show Parameters** expands the dialog box to show rarely-used options:

**Name** edits the line style's name.

**Description** edits the line style's description.

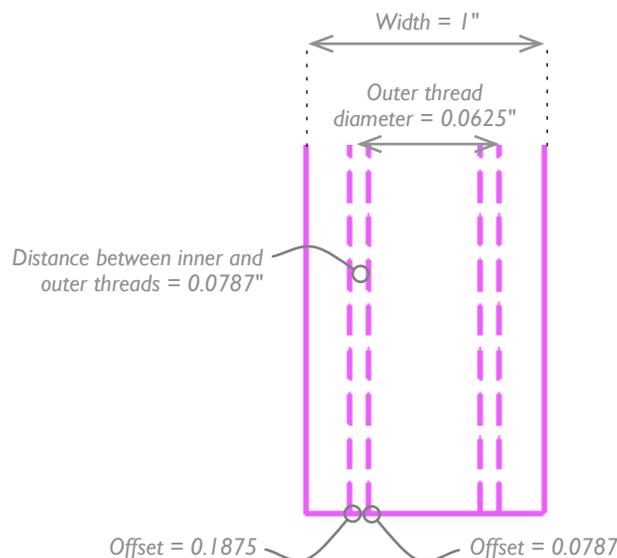
**Global Scale** specifies the scale factor for all line styles in the drawing.

**Entities Scale** specifies the scale factor for new entities relative to the global scale factor.

**Scale Based on Sheet's Units** applies the Viewport scale factor to line styles; when off, line styles have

the same scale factor in model and sheet modes.

A 5/8" diameter is 0.625" in decimal notation, and represents the outer diameter of the hole; the inner diameter is determined by applying a 60-degree line to the thread's pitch distance. I won't make you to go through the mathematics, and so I have illustrated the distances below:



Thus, the offset distances are:

0.1875"  
0.0787"

2. Use the **Explode** command to explode the rectangle so that it can be offset.

Command: explode  
Specify entities» (Select the rectangle.)  
Specify entities» (Press return.)

3. Start the **Offset** command, and then specify the offset distances. First, do the two lines that are offset by 0.1875”.

Command: offset

Specify distance» .1875

Specify source entity» (Pick line A.)

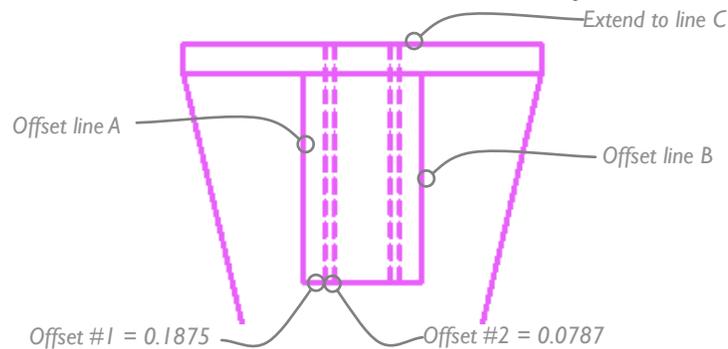
Specify side for destination» (Pick inside the rectangle.)

Specify source entity» (Pick line B.)

Specify side for destination» (Pick inside the rectangle.)

Specify source entity» (Press return to exit.)

- Repeat the command to do the two inner lines that are offset by 0.0787.



4. The lines need to extend to the topmost line, which is accomplished with the **Extend** command:

Command: extend

Specify boundary edges» (Pick line B.)

Specify boundary edges» (Press return to end entity selection.)

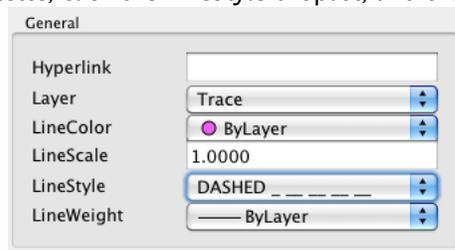
Specify segments to extend» (Pick the dashed lines.)

Specify segments to extend» (Press return to exit the command.)

5. Change the line style to Dashed, as follows:

- a. Select the four lines.

- b. In the Properties palette, click the LineStyle droplist, and then choose **DASHED**.



Notice that the lines change from solid to dashed, as illustrated above.

6. Use the **Copy** command to copy the four dashed lines from the front view to the side view.

*Hints:* Select them with Window selection mode, and then move them with INTersection entity snap.

Command: copy

Specify entities» w

Specify first corner» (Pick a point.)

Specify opposite corner» (Pick another point to encompass the four lines.)

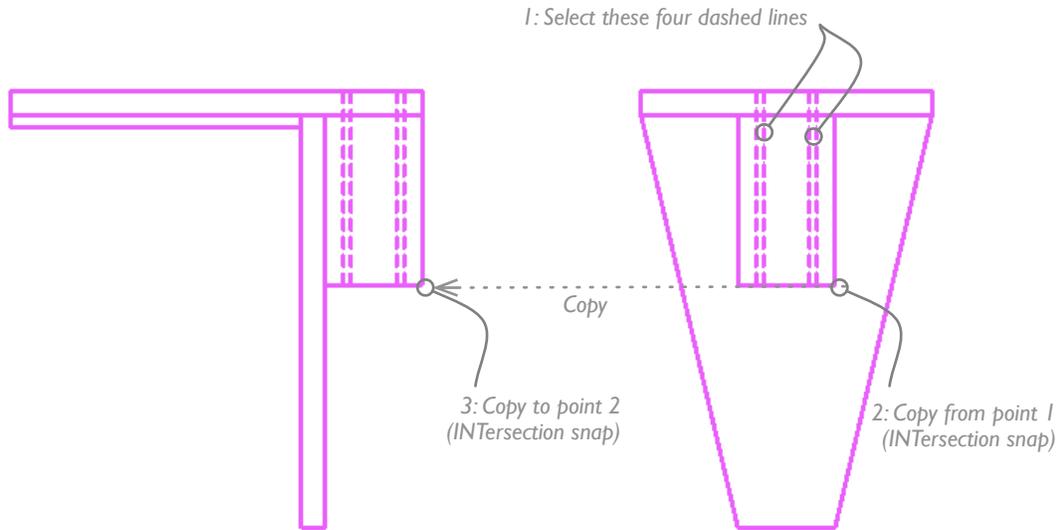
Specify entities» (Press return to continue.)

Specify from point» int

of (Pick point #1.)

Specify second point» int

of (Pick point #2.)  
Specify second point» (Press return to end the command.)



7. If the dashed lines are too close or too far apart, use **LtScale** to change this:

Command: ltscal  
Default: 1.0000  
Enter new value for LTSCALE» 0.5

Values over 1 make the line styles larger, while values between 0 and 1 make them smaller.

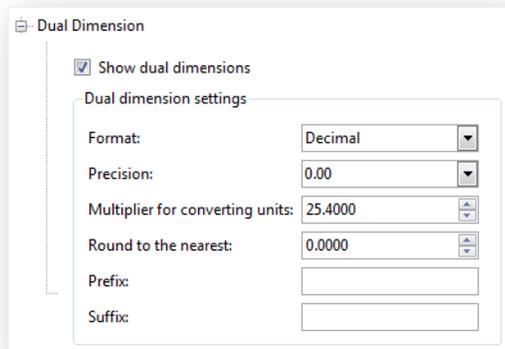
8. Save the drawing.

## Dual Dimensioning

The final step is to dimension the drawing. The hand drawn one was drafted in 1966 using inches. You redrew it in inches as well; most countries, however, use metric units. iCADMac is able to place dimensions with both Imperial and metric units through the DimensionStyle command.

Here's how:

1. Enter the **DimensionStyle** command.
2. In the Options dialog box, open the **Dual Dimension** node.



3. Turn on **Show Dual Dimensions**.

The other settings are already appropriate for adding metric measurements to the dimensions through the value of 25.4 used for the **Multiplier for Converting Units**.

4. Click **OK**.

5. It's a good idea to place dimensions on a layer of their own. Use the **Layer** command to create a new layer with these parameters:

Name	Dims
LineColor	8 (gray)
LineWeight	0.00mm
LineStyle	Continuous

Make the **Dims** layer active.

6. Start the **LinearDimension** command.

Command: lineardimension

a. Pick two points to locate the extension lines...

Specify first extension line position» (Pick one end of a line.)

Specify second extension line position» (Pick other end of the line.)

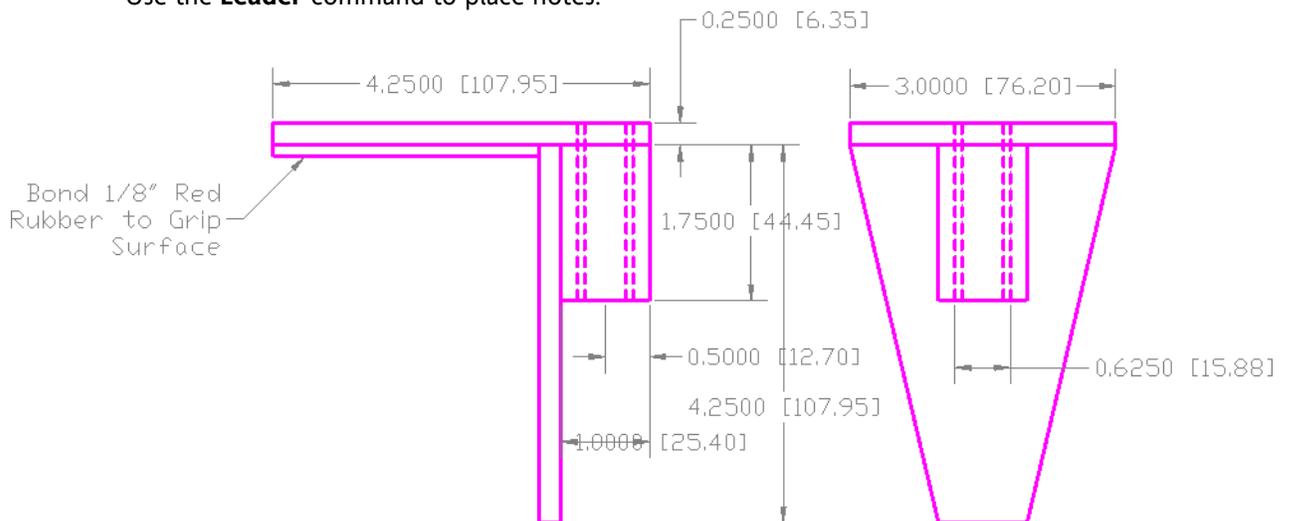
b. ... and then locate the dimension line.

Specify dimension line position» (Pick a point.)

Dimension Text : 3.0000

7. Apply the other dimensions, copying the ones shown on the scanned image.

Use the **Leader** command to place notes.



8. Save the drawing.

If you wish, print the completed drawing.

## Summary

In this chapter you learned how to “convert” scanned-in paper drawings to CAD drawings, as well as how to apply imperial-metric dimensions to entities.

In the following chapter, you convert this 2D drawing into an *isometric* drawing, one that looks as if it is three-dimensional.

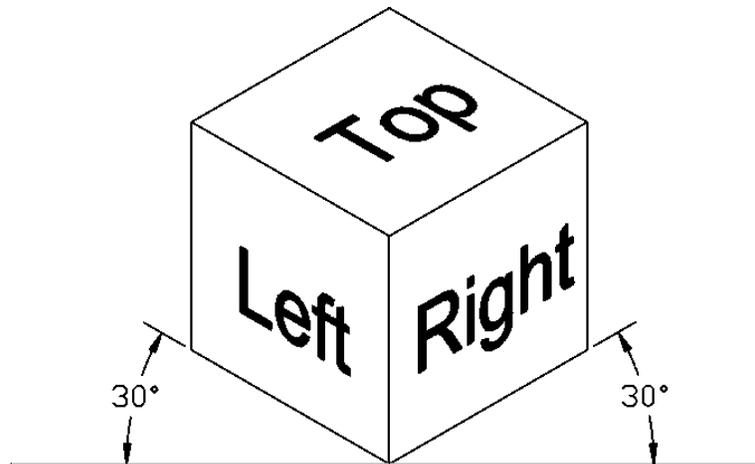
## Chapter 12

# Isometric Drafting

### In This Chapter

- Customizing template files
- Preparing iCADMac for isometric drawings
- Defining text and dimension styles
- Drawing in isometric mode

**Isometric drawings** are like fake three-dimensional drawings. They are most commonly used for piping schematics, but can also be used to represent architectural and mechanical designs. Isometric drawings clearly illustrate three sides of 3D models: left, top, and right. The figure below illustrates a cube drawn in an isometric view, with the three sides labeled.



Lines and text are drawn in three isometric planes. (Isometric planes are often named “isoplanes.”) The curious thing is that text and dimensions must be slanted by 30 degrees to look correct. You learn how to create isometric text and dimension styles in this chapter.

## Key Terms in This Chapter

**Isometric Drawings** — refers to the 2D drawings that show the left, front (right), and top view simultaneously.

**Isometric Planes** — refers to the three drawing planes of isometric drawings, also known as “isoplanes.”

**Oblique Angle** — refers to the angle by which text is slanted forwards or backwards.

**Rotate Angle** — refers to the angle by which text is rotated.

## Abbreviations

**fn+F5** — shortcut keystroke for switching between isometric planes.

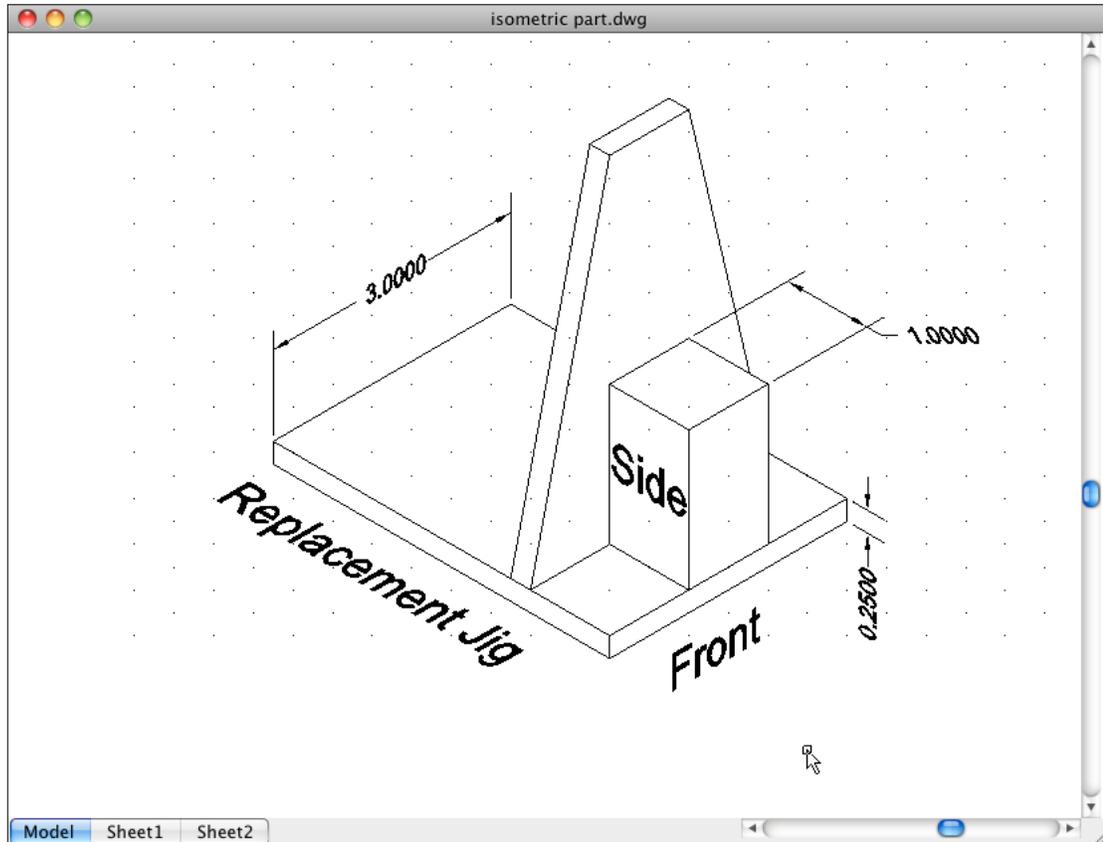
**Iso** — short for “isometric.”

## Commands

<b>Command</b>	<b>Shortcut</b>	<b>Menu Selection</b>
EditDimension	...	Dimension   Oblique
Isoplane	<b>fn+F5</b>	...

iCADMac has an isometric drawing mode, but “hides” it inside the Snap command. When isometric mode is turned on, iCADMac displays an isometric cursor and changes the grid display, snap mode, and ortho mode to work correctly, as illustrated below. (Other commands are not, however, adapted to isometric mode, such as Trim and Mirror.)

In this chapter’s tutorials, you learn how to set up text and dimensions styles suitable for isometric drafting. This chapter’s completed isometric drawing is below:



## Setting Up an Isometric Template

In this tutorial, you set up a template drawing that can be reused for future isometric drawings.

(Isometric drawings have nothing to do with the ISO template and other files, except that both are based on the Greek word for “same.” ISO is short for the International Standards Organization.)

1. Start a new drawing with the *Standard.dwt* template drawing.
2. Turn on isometric drafting mode, as follows:
  - a. Start the **Snap** command, and then specify the **Grid Style** option:
    - Command: snap
    - Default: 0.5000
    - Options: Grid style, OFF, ON, ORIENTATION, SPACING or Specify horizontal and vertical spacing» g
  - b. At the prompt, enter ‘i’ to specify isometric mode:
    - Default: Rectangular
    - Options: Isometric, or Rectangular
    - Specify option» i
  - c. Change the spacing to 0.25 units:

Default: 0.5000  
Specify isometric snap spacing» .25

3. On the status bar, click the following buttons to turn on the related modes:

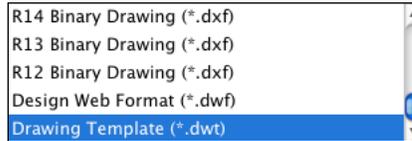
**Snap    Grid    Ortho    ESnap    LWT**

4. The CS icon is not useful for isometric drafting, so turn it off with the **CSIcon** command:

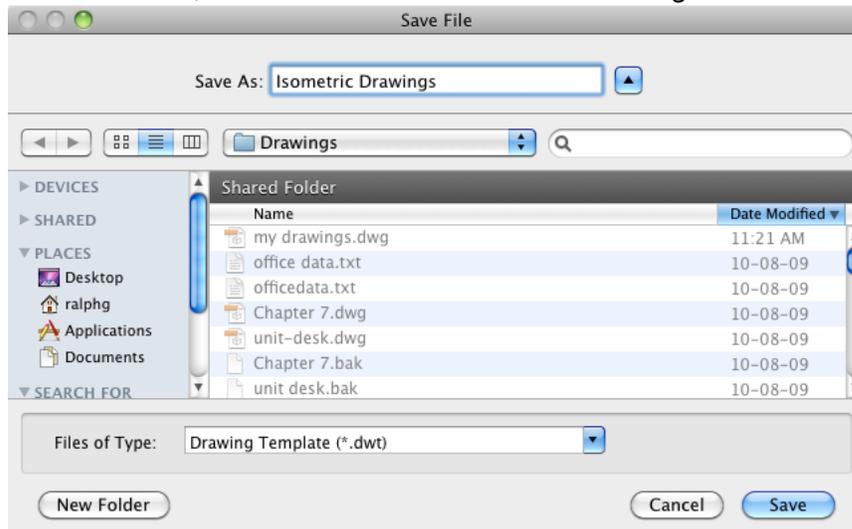
Command: csicon  
Default: ON  
Options: All, No origin, Off, ON or ORigin  
Specify option» off

5. Save the drawing as a template file, as follows:

- a. Enter the **SaveAs** command.  
b. From the **Files As Type** droplist, choose “Drawing Template (\*.dwt)” — you’ll find it at the very end of the list:



- c. In the **Save As** field, enter the file name of “Isometric Drawings.”



- d. Click **Save**.

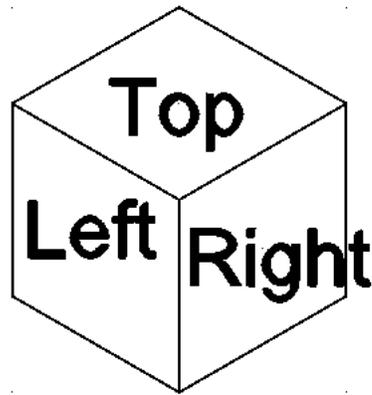
At this point, the drawing should look like this:

Isometric grid is visible.  
No UCS icon is visible.  
Snap, Grid, Ortho, and other status bar buttons turned on.

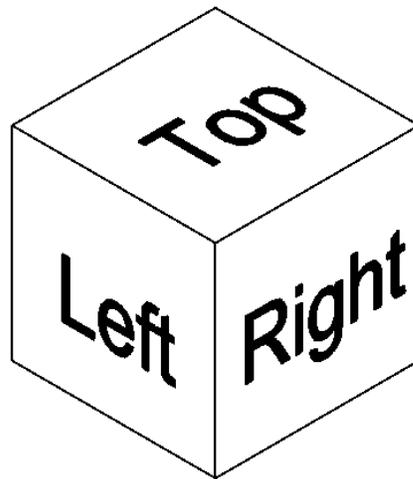
In the next parts of this tutorial, you create text and dimension styles. They needed for isometric drawings.

## Isometric Text Styles

For text to look “correct” in isometric drawings, the letters need to be slanted. The figure below illustrates the problem: when regular text is placed in an isometric drawing, it looks unaligned.



The solution is to slant text by 30 degrees in two directions, obliques and rotated. *Obliques* means each letter is slanted, *rotated* means the entire word is slanted, as well. Below, I have applied both angles to the text, making it look correct.



There is a second problem, however: each isometric plane requires a different combination of 30-degree angles — some positive, some negative — as shown by the table below:

Isoplane	Style Command Oblique Angle	Text Command Rotate Angle
Left	-30°	-30°
Top	30°	-30°
Right	30°	30°

The obliquing angle is set in the TextStyle command, and can be stored in the template drawing. Two text styles are needed, one for the left isoplane's -30° oblique angle, and another for the top and right isoplane's 30° angle.

The rotation angle is set by the SimpleNote and Note commands. It cannot be set in the template drawing, but is employed later when placing text in the isometric drawing.

In the *Isometric drawing.dwt* template drawing, create the two styles of isometric text using the following obliquing angles:

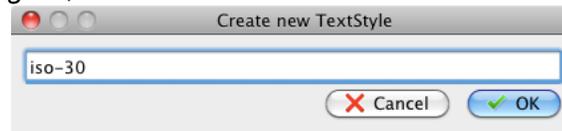
Text Style Name	Oblique Angle	Used for...
iso-30	-30	Left isoplane

iso+30

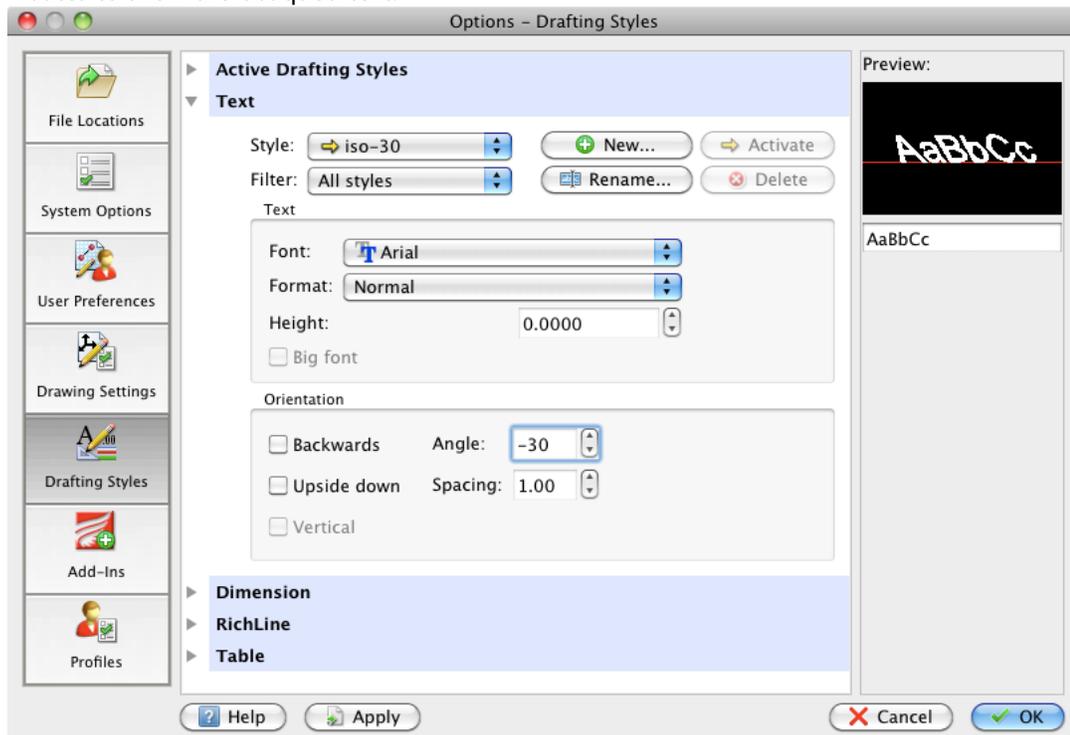
30

Top and right isoplanes

1. Enter the **TextStyle** command, and then click **New** to create the new text style for the left isoplane (-30 degrees):



2. Enter "iso-30" and then click **OK**.
3. Back in the Options dialog box, change the font to something nicer than ARTxt.shx. From the **Font** droplist, choose "Arial.ttf."
4. In the Orientation section, change Angle to **-30**. Notice that the Preview thumbnail updates to show the obliques text.



5. Create the second isometric text style using these settings:
  - a. Click **New**.
  - b. Enter "iso+30" for the style name, and then click **OK**.
  - c. Change the Angle to **30** degrees. (Continue using Arial as the font.)
6. Click **OK** to exit the dialog box. The drawing looks no different, but now contains two new text styles.
7. Save the drawing again as a template; you *cannot*, however, use the Save command or press command+S, because iCADMac would then save the file as a regular DWG file. Instead, you have to follow the steps of the previous tutorial:
  - a. Enter the **SaveAs** command.
  - b. From the **Save As Type** droplist, choose "Drawing Template (\*.dwt)."
  - c. Click **Save** to save the template file.
  - d. Click **Replace** when iCADMac asks if you want to replace *isometric drawings.dwt*:



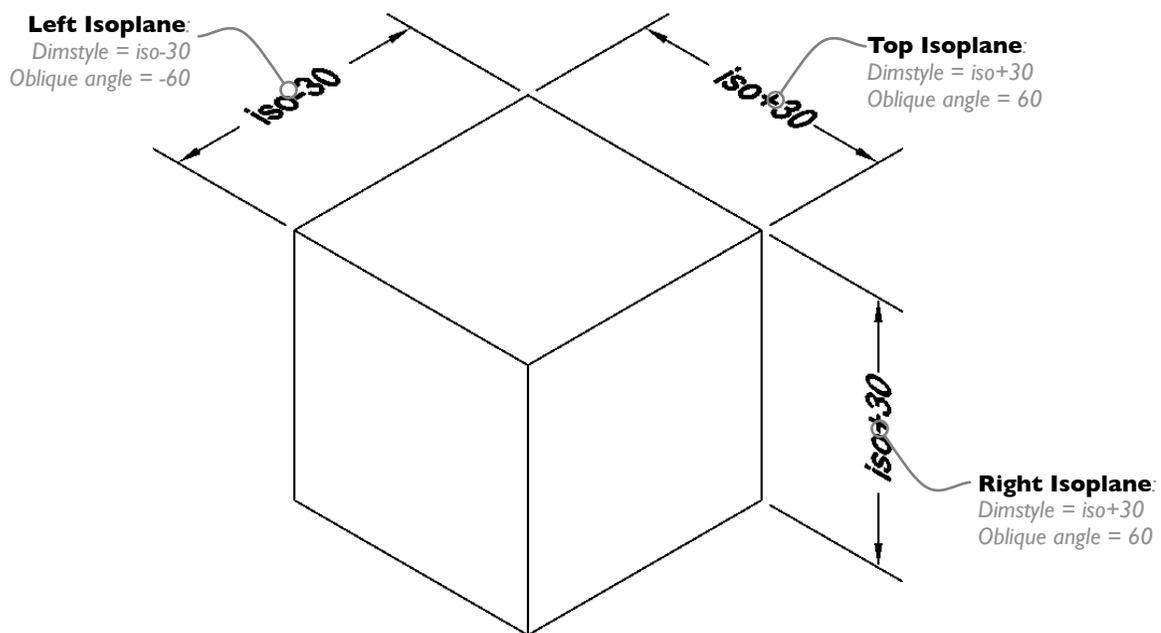
## Isometric Dimension Styles

With the isometric-friendly text styles in place, you can now go ahead and create the isometric dimension styles. These need dimension text that's slanted, just like isometric text; in fact, the dimension styles use the two text styles you created in the previous tutorial.

(Later, when drawing the isometric dimensions, you will use the `ParallelDimension` command, because it automatically aligns dimensions with slanted entities drawn at 30 degrees. In addition, dimensions in the top isoplane need their extension lines slanted by 60 degrees. The slanting is done during the `EditDimension` command's `Oblique Angle` option.)

Here are the parameters you will need for placing isometric dimensions:

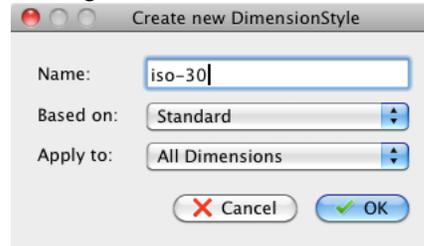
Isoplane	DimensionStyle command uses these text styles:	EditDimension command's Oblique option applies these angles:
Left	iso-30	-60°
Top	iso+30	60°
Right	iso+30	60°



Continue in the *Isometric drawing.dwt* template to create the two styles for isometric dimensions, as follows:

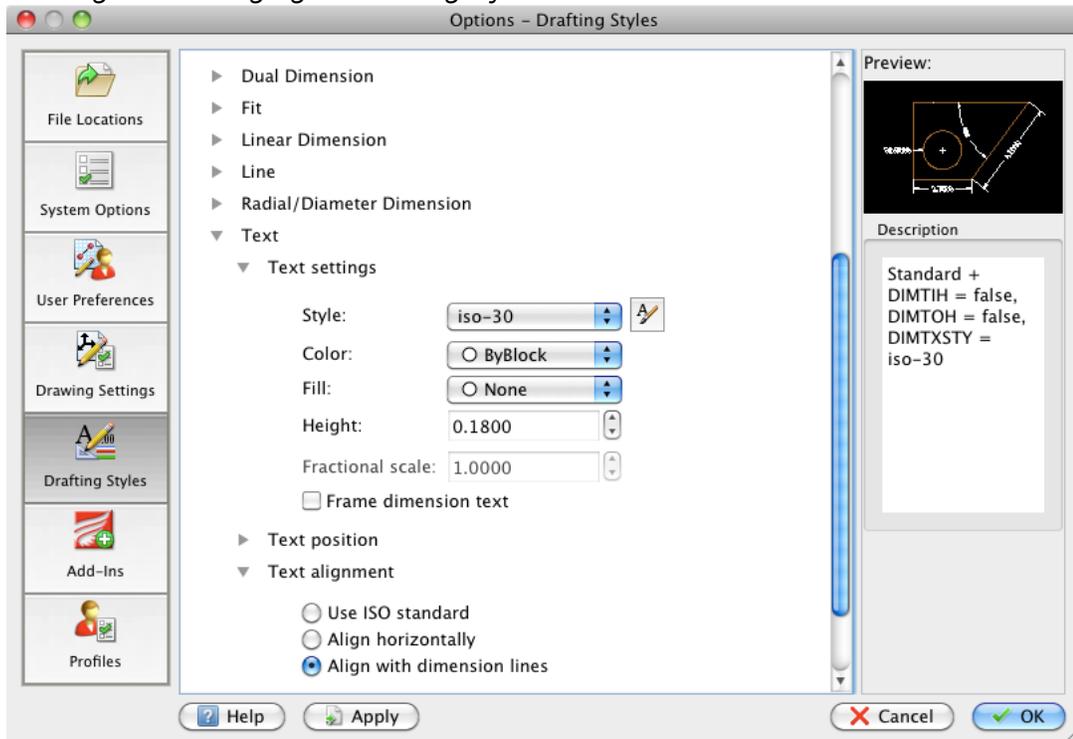
Dimension Style Name	Text Style Name	Used for...
iso-30	iso-30	Left isoplane
iso+30	iso+30	Top and right isoplanes

1. Start the **DimensionStyle** command, and then click **New** to create the new dimension style for the left isoplane (-30 degrees):



2. Enter “iso-30” for **Name**, and then click **OK**.
3. In the Options dialog box, open the **Text** node, and then open the **Text Settings** node.
4. Change **Style** to **iso-30**.
5. Open the **Text Alignment** node, and then choose the **Align with dimension line** option.

The figure below highlights the changes you need to make.



6. Repeat to create the second isometric dimension style using these settings:
 

<b>Name</b>	iso+30	(Click <b>New</b> to create the new style.)
<b>Style</b>	iso+30	
7. Click **OK** to exit the dialog box.
8. The template drawing is complete. Save it one more time with the **SaveAs** command. (Remember to choose “Drawing Template (\*.dwt)” from the **Save As Type** droplist.)

The *Isometric drawings.dwt* template file now contains all settings needed to begin drafting in

isometric mode.

## Drawing in Isometric Mode

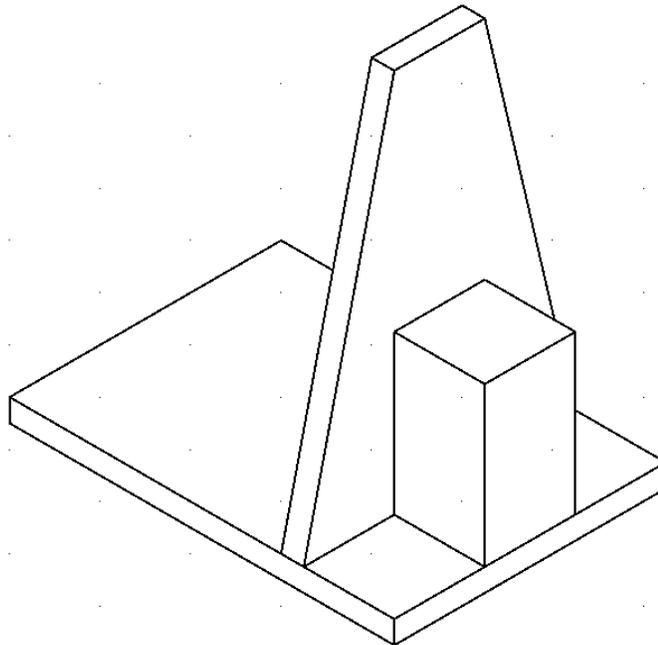
In this tutorial, you recreate the 2D drawing from the last chapter as a 3D-looking isometric drawing. (I assume that iCADMac is still running from the last tutorial.)

1. Enter the **New** command, and then choose *Isometric drawings.dwt* in the Specify Template dialog box.
2. Click **Open**. Notice that iCADMac creates a new drawing with the generic name *NON-AME\_1.dwg* (or something similar).
3. With the **Save** command, save the drawing by the name of *Isometric part.dwg*.

You are now ready to start drawing in isometric mode.

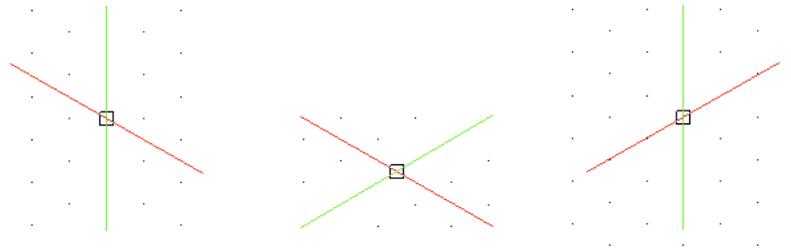
### Drawing Isometric Boxes

You will draw the part upside-down, so the details can be seen more clearly. The finished drawing is illustrated below.



1. Enter the **line** command:  
Command: line

Notice that the cross hair cursor becomes isometric. One or both cross hairs are slanted by 30 degrees. The style of slant tells you which of the three isometric drawing planes is current: left, top, and right (or front). The cursor shape for each isoplane is shown below:



Left to right: Cross hair cursors for left, top, and right isoplanes.

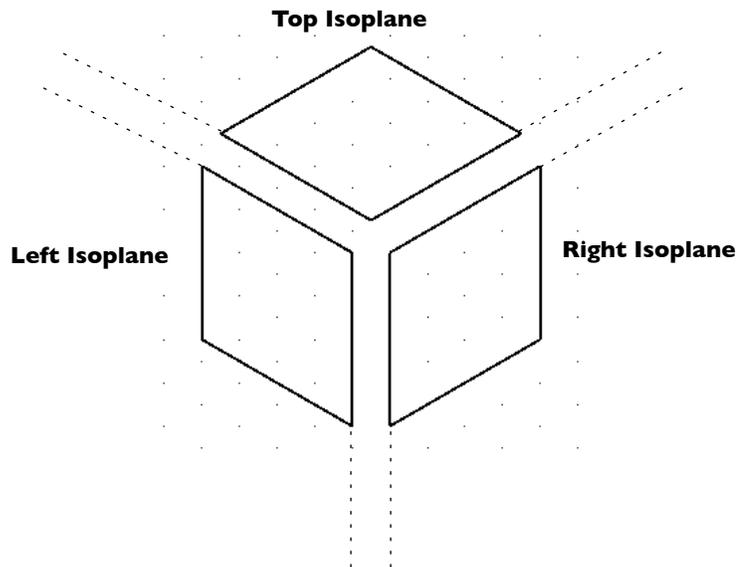
You switch between isoplanes by pressing the **fn+F5** function key. Each time you do, iCADMac reports the name of the current plane:

**fn+F5** <IsometricPlane Top>

**fn+F5** <IsometricPlane Right>

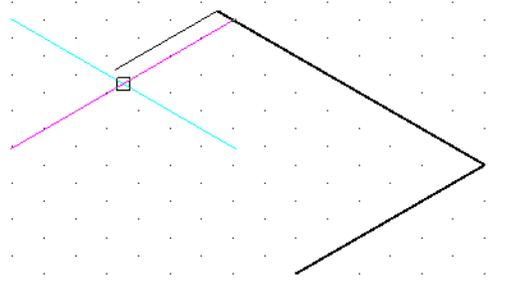
**fn+F5** <IsometricPlane Left>

In each plane, you are limited to drawing in two directions only, as summarized by the figure below:



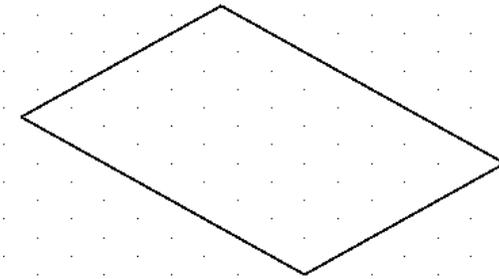
This means you will be pressing **F5** frequently throughout the following tutorials!

2. Press **fn+F5** until iCADMac reports the top isoplane:
  - fn+F5** <IsometricPlane Top>
3. The first entity to draw is the top plate, which is 3" wide by 4.25" long. Continuing with the Line command, begin drawing at the center of the bottom portion of the drawing area. (The exact location is not important.)
  - Specify start point» (Pick a point in the lower center of the drawing.)
4. Move the cursor to the right. Notice that ortho mode forces the line to travel in the isoplane, 30 degrees up from the x axis.
5. Enter the length, **3**, and then press **return**.
  - Specify next point» (Move cursor to the right.) 3
6. Move the cursor up, and then enter the next length, **4.5**. (Remember to press **return**.)
  - Specify next point» (Move cursor up.) 4.25



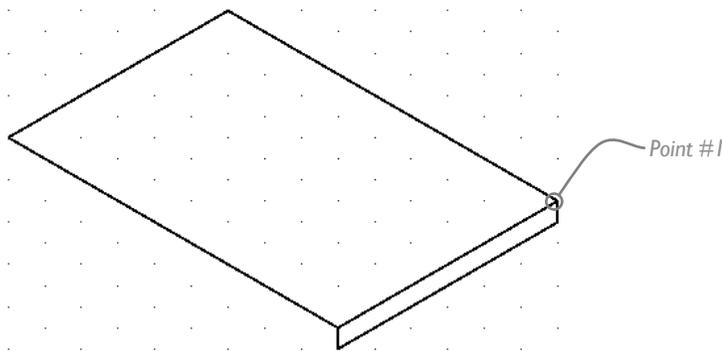
7. Move cursor left, and then enter **3**:  
Specify next point» 3

8. Close the “rectangle” by typing ‘c’:  
Specify next point» c



9. To draw the right side of the plate, first switch to the next isoplane, right:  
**fn+F5** <IsometricPlane Right>

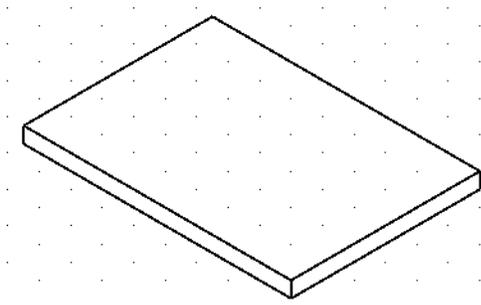
10. Press the **spacebar** to restart the Line command, and then draw the 0.25” thick sides on the right isoplane, as follows:  
: (Press spacebar to repeat command.)  
LINE Specify start point» (Pick point #1.)



Specify next point» (Move cursor downwards.) 0.25  
Specify next point» 3  
Specify next point» 0.25  
Specify next point» (Press return to end the command.)

11. To draw the left side of the plate, switch to the left isoplane with **fn+F5**:  
**fn+F5** <IsometricPlane Left>

12. Press the **spacebar** to restart the Line command, and then draw the 0.25” thick sides on the left isoplane. You now have a rectangular plate that should look like the one in the figure below.



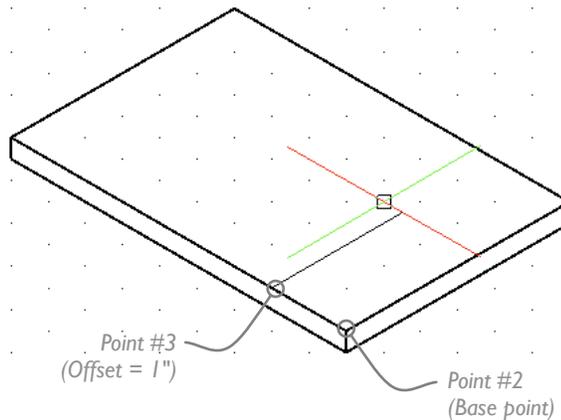
13. Press **command+S** to save your work.

### Drawing Irregular Boxes

Drawing rectangles and squares isometrically is fairly straightforward, as the above tutorial showed. The difficulty comes with drawing irregularly-shaped objects; the workaround is to draw a rectangular outline, and then trim it to recreate the shape.

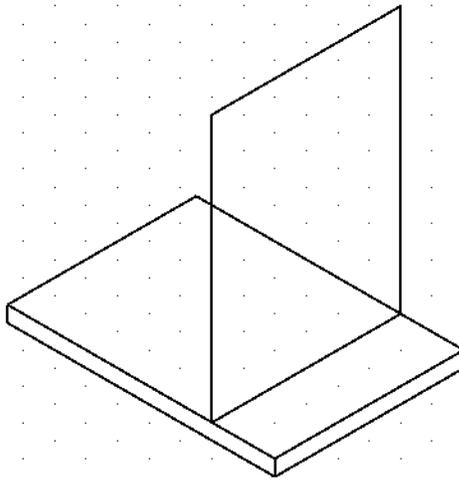
In this tutorial, you draw the trapezoid-shape. It is 3" wide by 4.25" tall.

1. The trapezoid is located one inch from the front edge of the base plate. Use From entity snap to begin the line, as follows:  
Command: line
2. Enter 'from' to indicate the offset distance, and then pick point #2:  
Specify start point» from  
Base point (Pick point #2.)
3. Move the cursor to the left-top, and then specify the offset distance, 1:  
(Offset) (Move cursor towards point #3.) 1

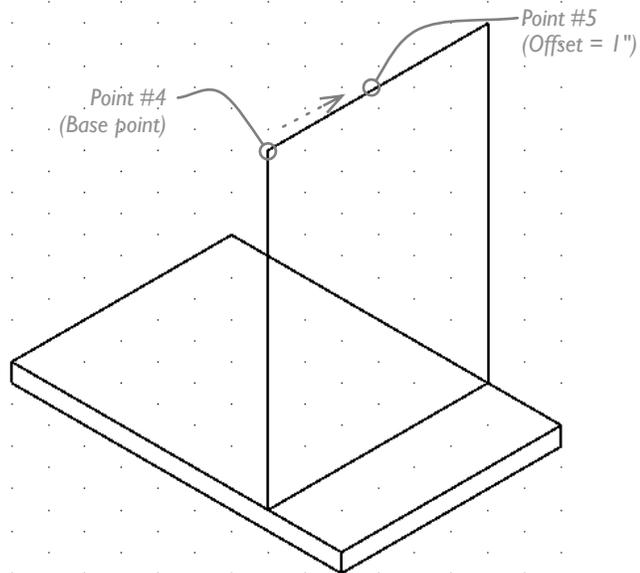


4. Press **fn+F5** until the drawing is in the right isoplane.  
Specify next point» **fn+F5** <IsometricPlane Right>
5. You can now draw the rest of the rectangle as per the earlier tutorial.  
Specify next point» 4.25  
Specify next point» 3  
Specify next point» 4.25  
Specify next point» (Press return.)

The result should look like the figure below.



6. Now you need to trim the rectangle to create the trapezoid. The top is 1" wide. Use **From** again to position the next line 0.25" long in the top isoplane:



Command: (Press return to repeat the Line command, and then press **fn+F5**.)

LINE **fn+F5** <IsometricPlane Top>

Specify start point» from

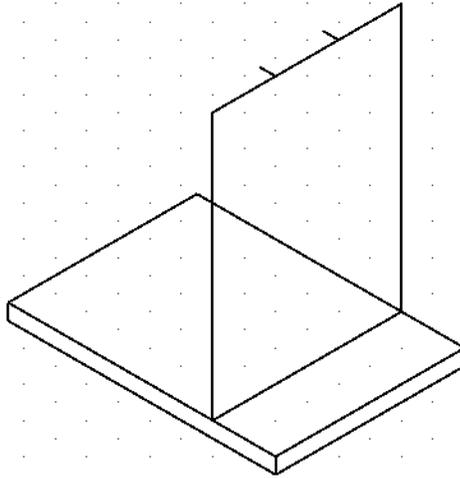
Base point (Pick point #4.)

(Offset) (Move cursor to point #5.) 1

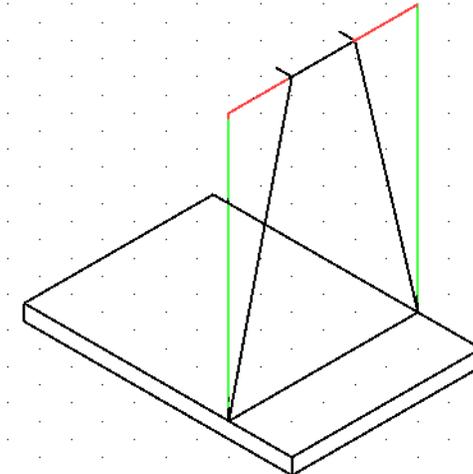
Specify next point» .25

Specify next point» (Press return to end the command.)

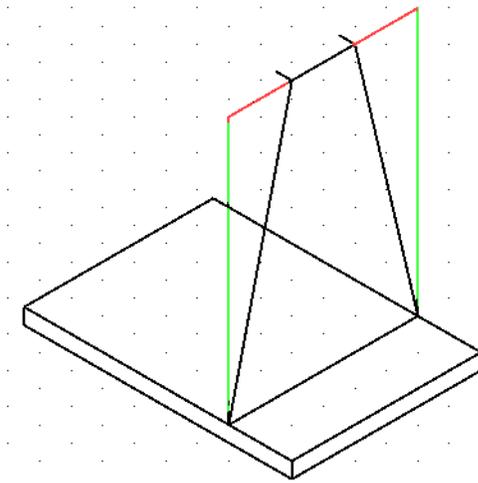
7. Repeat to draw the other 0.25"-long line.



8. With the two stubby lines in place, you can add lines to form the trapezoid, as follows:
  - a. Turn off ortho mode by clicking the **Ortho** button on the status bar.
  - b. With the **Line** command, draw the two diagonal lines shown in black by the figure below.

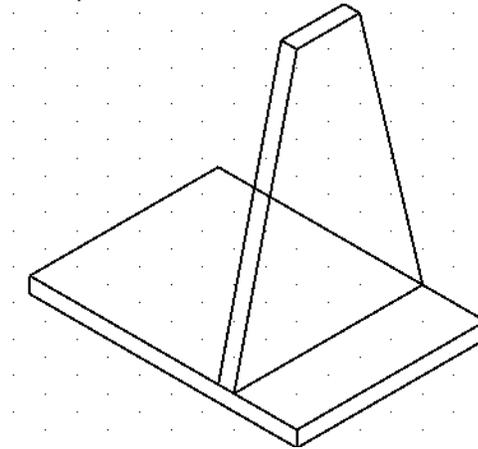


9. Now you trim and erase some of the lines to complete the trapezoid:
  - a. Start the Trim command.  
Command: trim
  - b. Press **return** to select all entities as cutting edges.  
Specify cutting edges ...  
Options: Enter to specify all entities or  
Specify entities» (Press return to select all entities in the drawing.)
  - c. Remove the line segments shown in red by the figure below:  
Specify segments to remove» (Select one segment.)  
Specify segments to remove» (Select other segment.)  
Specify segments to remove» (Press return to end the command.)

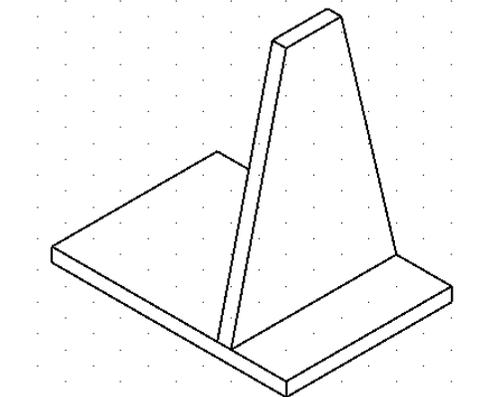


d. Use the **Erase** command to remove the segments shown in green. (These cannot be removed by the Trim command.)

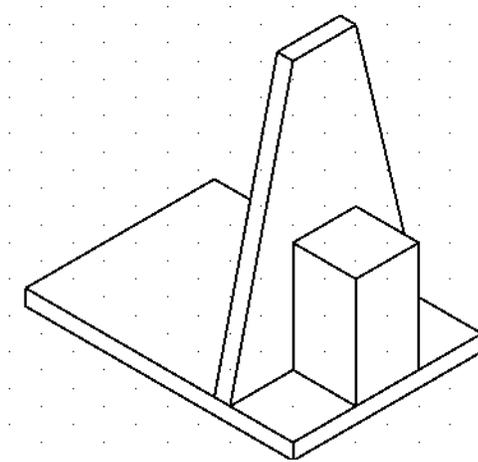
10. Draw the remainder of the trapezoid with the **Line** command.



11. Use the **Trim** command to remove the hidden lines.



12. To complete the drawing, use the techniques you learned above to draw the one remaining box. It has a 1"-square base and is 1.75" tall. The result should look like this:



13. Press **command+S** to save your work to disc.

## Placing Isometric Text

Adding text to isometric drawings takes these steps:

- i. Select an isoplane in which to work.
- ii. Start the -SimpleNote or Note command, and then specify the parameters appropriate to the isoplane:  
Specify the style name appropriate to the current isometric plane.  
Enter the appropriate rotation angle.

Use the following table to determine which style names and rotation angles to use for each isoplane:

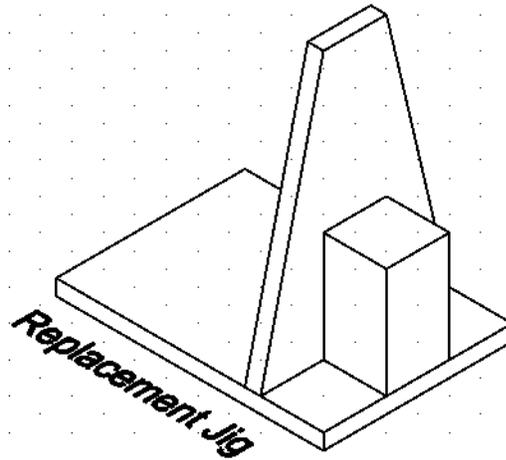
Isoplane	Text Style Name	Rotation Angle
Left	iso-30	-30°
Top	iso+30	-30°
Right	iso+30	30°

In the following tutorial, you add text to the top isoplane:

1. Press **fn+F5** until you reach the top isoplane:  
Command: **fn+F5** <IsometricPlane Top>
2. Start the -SimpleNote command, and then enter 'st' to specify the text style:  
Command: -simplenote  
Options: Justify, textSTyle or  
Specify start position» st
3. Specify the **iso+30** style name.  
Default: Standard  
Options: ? to list or  
Specify name» iso+30
4. Pick a point in the drawing to start the text, and then specify a suitable height, such as 0.35":  
Specify start position» (Pick a point.)  
Specify height» 0.35

5. From the table above, you know that the angle needs to be -30 degrees:  
Specify text» -30
6. Finally, enter some text, and then press **return** twice.  
Specify text» Replacement Jig  
Specify text» (Press return.)

The result should look similar to the figure illustrated below.



## Placing Isometric Dimensions

Adding dimensions to isometric drawings is much more complex than adding text, unfortunately. It takes these steps:

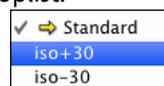
- i. Select an isoplane in which to work.
- ii. Specify the dimension style appropriate to the isoplane.
- iii. Use the ParallelDimension command to place linear dimensions.
- iv. After dimensions are placed, use the Dimension | Oblique menu to adjust the angle of extension lines.

Use the following table to determine which dimension styles to use for each isoplane:

Isoplane	DimStyle Name	EditDimension Oblique Angle
Left	iso-30	-60°
Top	iso+30	60°
Right	iso+30	60°

In this tutorial, you place a dimension in the top isoplane.

1. Before starting the ParallelDimension command for placing isometric dimensions, you need to set the dimension style name. This is done through the DimensionStyle command:
  - a. Start the **DimensionStyle** command.
  - b. Choose **iso+30** from the Style droplist.

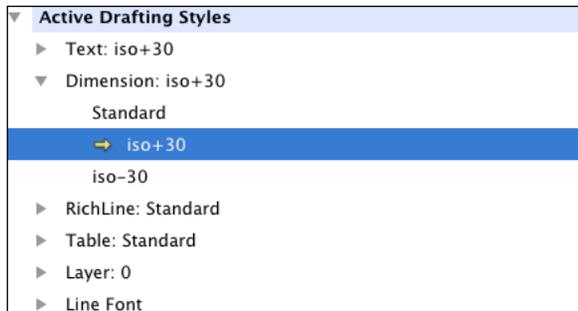


- c. Click **Activate** to make it the active dimension style.

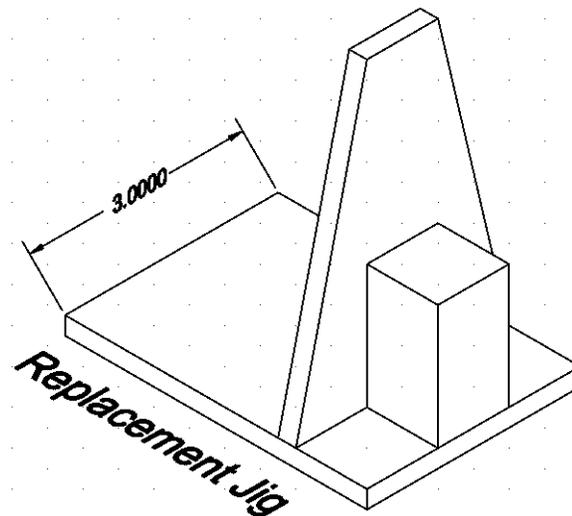


- d. Click **OK** to close the dialog box.

**TIP** Here is another way to change the active dimension style: (a) enter the **Styles** command, (b) open the **Dimension** section, (c) double-click the style name to make it active, and then (d) click **OK** to close the dialog box. See figure below.



2. Ensure the active isoplane is set to Top.
3. Start the **ParallelDimension** command:



Command: paralleldimension  
 Default: Entity  
 Options: Entity or  
 Specify first extension line position» e  
 Specify entity» (Pick a line.)  
 Options: Angle, Note, Text or  
 Specify dimension line position» (Position the dimension line.)  
 Dimension Text : 3.0000

4. After all dimensions are placed, use the **Dimension | Oblique** menu item to adjust the angle of extension lines. (You could enter the EditDimension command and its Oblique

Angle option, but the menu pick is faster, because it takes you directly to the obliquing option, which is more convenient.)

For this dimension, adjust the extension lines by -60 degrees, as follows:

Command: `_EDITDIMENSION`

Default: Home

Options: Angle, Home, Move, New, Oblique or

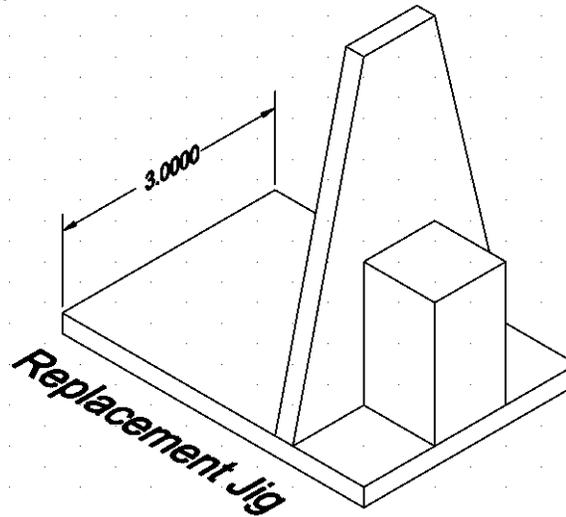
Specify option» `_O`

Options: Enter to exit or

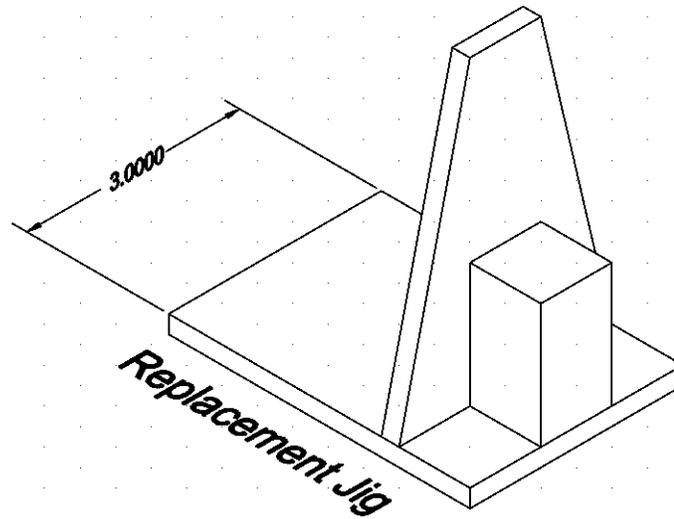
Specify oblique angle» `-60`

Specify entities» (Choose the dimension.)

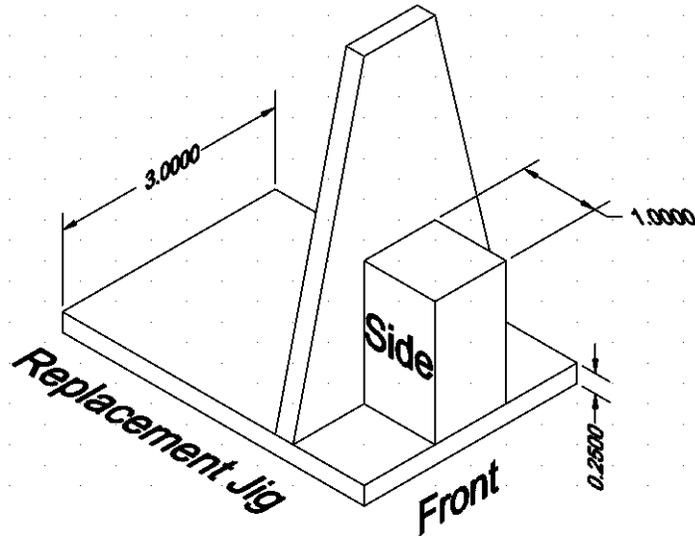
Specify entities» (Press return to end the command.)



**TIP** You get a different — but equally valid — result when you enter -60 degrees, as illustrated below.



5. If you wish, add more isometric text and dimensions to the drawing.



6. Press **command+S** to save your work, and then exit iCADMac.

## Summary

In this chapter you learned how to simulate the look of a three-dimensional drawing through the use of isometric drafting. You learned how to oblique the angles of text and dimensions to make them look “correct.”

In the next chapter, you turn this design into a real 3D drawing.

## Chapter 13

# Solid Modeling

### In This Chapter

- Understanding solid modeling
- Drawing 3D solid primitives
- Applying Boolean operations
- Analyzing solid models

**Solid modeling** is sometimes considered the ultimate in CAD design, for it creates precise 3D models that can be analyzed as if they were physical prototypes.

There are two primary approaches to creating 3D solid models:

- Draw 3D primitives, and then combine them into complex shapes.
- Draw 2D entities, and then revolve or extrude them into 3D models.

This chapter works with 3D primitives, which include boxes, cylinders, and wedges. iCADMac' capabilities in solid modeling include the following:

**Solid primitives:** box, wedge, cone, sphere, cylinder, torus, pyramid, and planar surface.

**Solid construction:** extrude, sweep, revolve, and loft.

**Boolean operations:** union (join together), subtract (remove one from the other), and intersect (common volumes).

**Solid editing:** change the color and location of edges and faces.

## Key Terms in This Chapter

**Arcball** — refers to the visual aid for rotating 3D viewpoints with the RollView command.

**Boolean** — refers to the editing operations that join or remove 3D solid entities.

**Cube** — refers to a 3D solid box with equal sides.

**Extrusion** — refers to a closed 2D entity thickened into a 3D solid.

**Primitives** — refers to the basic 3D solid entities, such as box, sphere, and cylinder.

**Solid Models** — refers to 3D entities with solid interiors.

**Subtract** — removes one or more 3D solid entities from one or more other such entities.

**Swivel** — moves 3D viewpoint left and right.

**Taper Angle** — specifies the angle at which an extrusion thickens.

**Tilt** — moves 3D viewpoint up and down.

**Union** — joins two or more 3D solid entities into a single entity.

## Commands

<b>Command</b>	<b>Shortcuts</b>	<b>Menu Selection</b>
Box	...	Draw   Modeling   Box
Cylinder	cy	Draw   Modeling   Cylinder
Extrude	ext	Draw   Modeling   Extrude
MassProp	...	Tools   Inquiry   Region/Mass Properties
PlanView	plan, pview	... <sup>1</sup>
Polyline3D	3dpoly, 3p	Draw   3D Polyline
RollView	3do, orbit	View   3D Orbit
Subtract	su	Modify   Solid Editing   Subtract
Union	uni	Modify   Solid Editing   Union

<sup>1</sup> No menu selection.

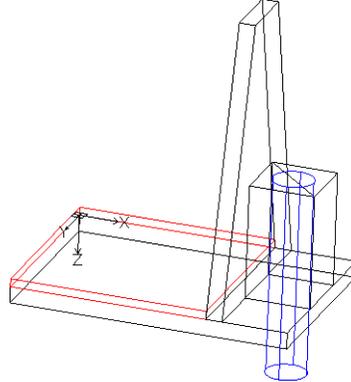
## Drawing 3D Solid Primitives

**Command:** Box

**Menu Bar:** Draw | Modeling | Box

**Tool Matrix:** Modeling | Box

In the this tutorial, you create a 3D solid model of the 2D drawing from earlier chapters.



1. Start iCADMac with a new drawing.
2. Enter the **Box** command, and then notice that iCADMac prompts you in the Command window.  
Command: box
3. Place the first corner of the box at the drawing's origin (0,0,0), as follows:  
Options: Center or  
First corner of box» 0,0,0
4. The box is 4.25" long by 3" wide...  
Options: Cube, Length or

### Summary of Box Command Options

Command: box  
Options: Center or  
First corner of box» (Enter coordinates or pick a point.)

Options: Cube, Length or  
Other corner» (Enter coordinates or pick a point.)

Options: 2Point or  
Height» (Enter a height or pick a point.)

**Center** — specifies the location of the cube's center.

**Cube** — creates a cube of equal sides; prompts you for the length to apply to all sides:  
Length» (Enter a length or pick two points.)

**Length** — specifies the length and width of the base:  
Length» (Enter a length or pick two points.)  
Width» (Enter another length or pick two points.)

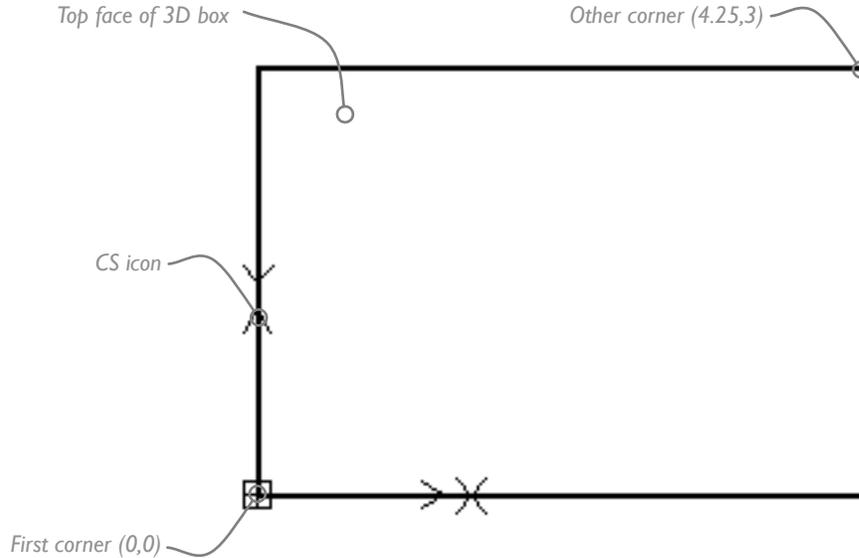
**2Point** — specifies the height by picking two points anywhere in the drawing:  
Specify first point» (Enter coordinates or pick a point.)  
Specify second point» (Enter coordinates or pick a point.)

Other corner» 4.25,3

5. ... and 0.25" high.

Options: 2Point or  
Height» .25

The figure below shows the top face of the box, along with the CCS icon in the lower left corner. (The CCS icon is useful when working in 3D space.)



6. Use the **Save** command to save the drawing by the name "3d model.dwg."

## Viewing in 3D

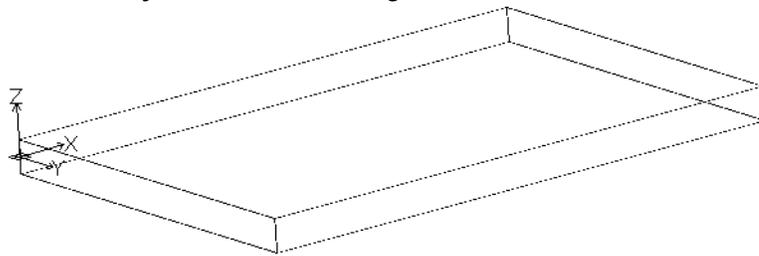
**Command:** RollView  
**Aliases:** 3dorbit, 3do, orbit  
**Menu Bar:** View | 3D Orbit  
**Shortcut:** Shift+middle button

The box was drawn in 2D, just like all earlier drawings. To see it in three dimensions, you need to rotate the viewpoint.

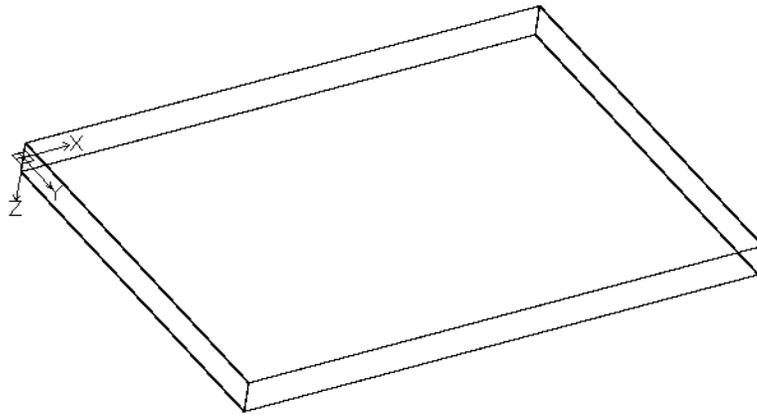
1. To view the box in three dimensions, enter the **RollView** command. Notice the rollview cursor

Command: rollview

- Move the cursor outside the box.
- Hold down the *left* mouse button. (Or hold down the trackpad button.)
- Drag the cursor. As you do, the box changes from 2D to 3D.



2. Use the cursor to flip the box over. You know you have succeeded when the Z of the CCS icon faces downwards.



3. When you are satisfied with the 3D view, press **esc** or **return** to exit the command.

Press ESC or ENTER to exit. (Press return.)

**TIP** To return to the 2D viewpoint, enter the **PlanView** command, and then press **return** twice:

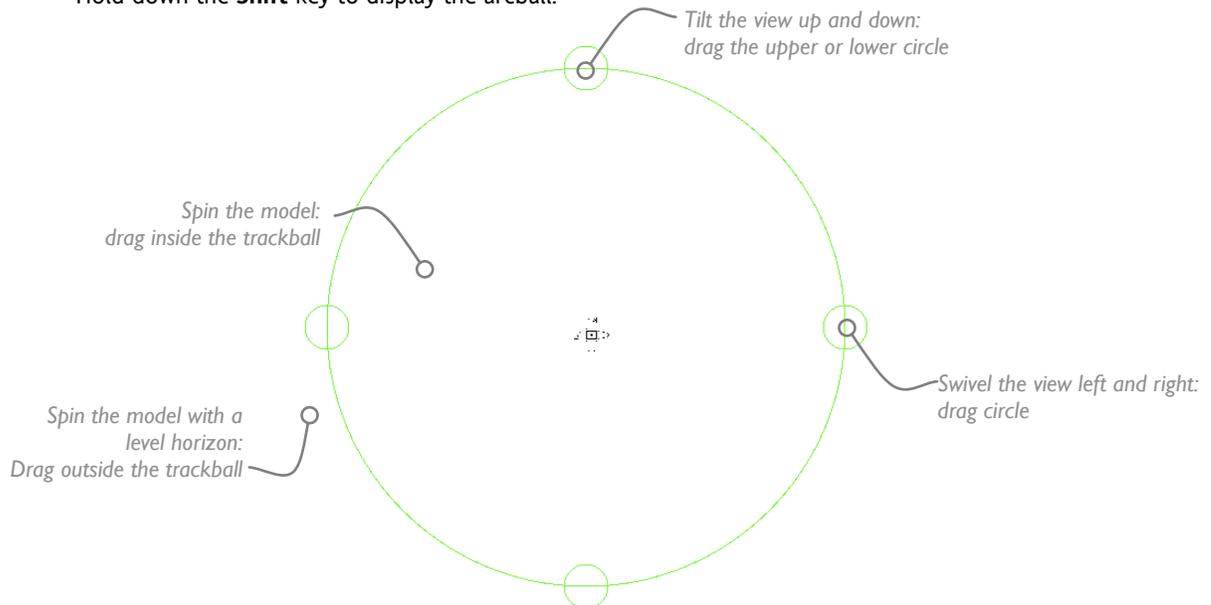
Command: planview (Press return.)

Default: Active

#### Summary of RollView Command Options

Command: rollview

Hold down the **Shift** key to display the arcball:



#### Action

#### How

Spin model around the viewpoint...	...by dragging <b>inside</b> the arcball with left mouse button.
Rotate viewpoint with level horizon...	...by dragging <b>outside</b> the arcball.
Tilt viewpoint up and down...	...by dragging circle <b>at top or bottom</b> of the arcball.
Swivel viewpoint left and right...	...by dragging circle <b>at left or right</b> of the arcball.
Zoom in and out....	...by rolling the mouse's roller wheel forward and back.

Options: Active ccs, CCs, or World  
Specify option» (Press return.)

## Adding More Solids

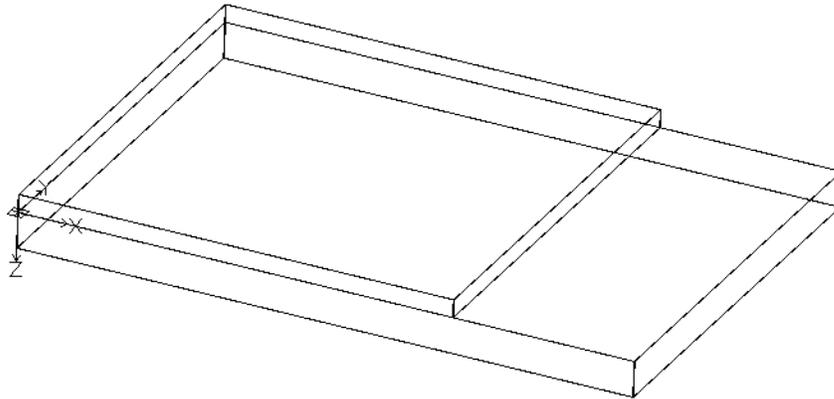
The part has three more solids that need to be drawn. You'll use the following commands for these remaining parts:

- i. **Box** command: Red rubber grip is 3x3" square by 0.125" thick.
- ii. **Extrude** command: Trapezoid is 3" (top) x 1" (bottom) x 4.25" (tall) by 0.25" thick.
- iii. **Box** command: Square post with hole is 1x1" square by 1.75" tall.

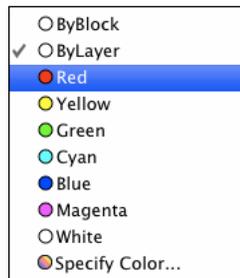
There is also a hole in the part. Holes are made of cylinders, that are subtracted from other solids; this takes place in a later tutorial.

1. The red rubber grip is drawn just like the first box, except that the height is *negative* to draw it downwards in the negative z-direction:

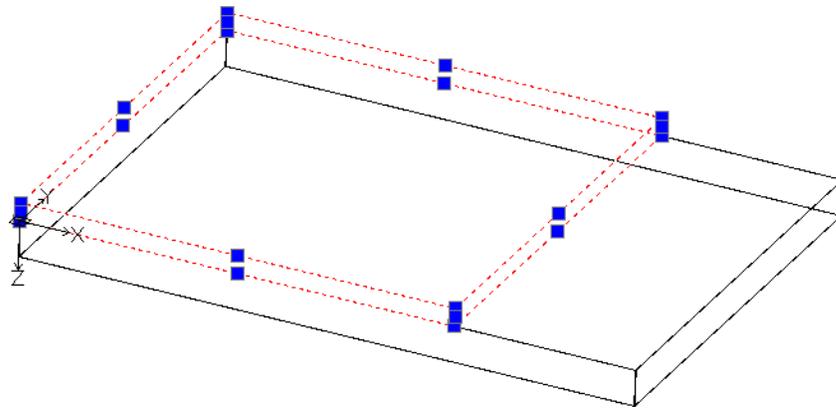
Command: box  
Options: Center or  
First corner of box» 0,0  
  
Options: Cube, Length or  
Other corner» 3.3  
  
Options: 2Point or  
Height» -0.125



2. Change the color of the box to red:
  - a. Select the "rubber mat" box.
  - b. From the Properties palette, click the **Color** droplist.
  - c. Select **Red**.



Notice that the box turns red.



- d. Press **esc** to unselect the box.

### Extruding 2D Objects to Create Solids

**Command:** Extrude

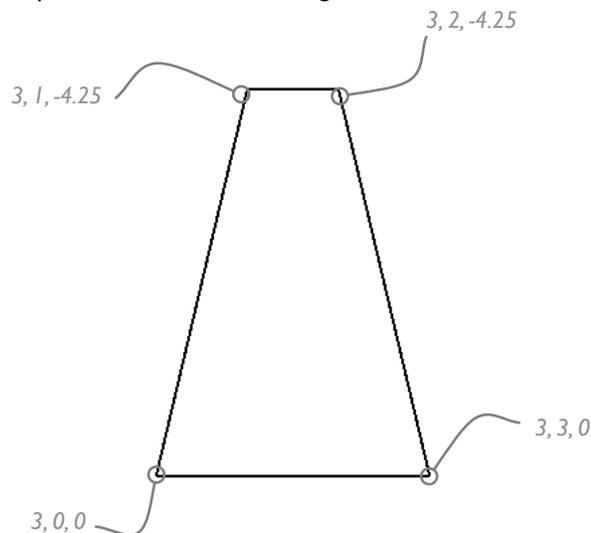
**Alias:** ext

**Menu Bar:** Draw | Modeling | Extrude

**Tool Matrix:** Modeling | Extrude

There are often several ways to solve a design problem with 3D modeling, as is the case with the trapezoid-shaped part. One approach is to draw a box, and then subtract the triangular portions. Another is to draw a 2D outline of the shape, and then *extrude* (thicken) it into a 3D solid — which is the approach you take in this tutorial.

1. The corners of the trapezoid have the following coordinates:



2. Use the **Line** or **Polyline3D** command to draw the outline. (You cannot use the Polyline command, because it works with 2D coordinates only.)

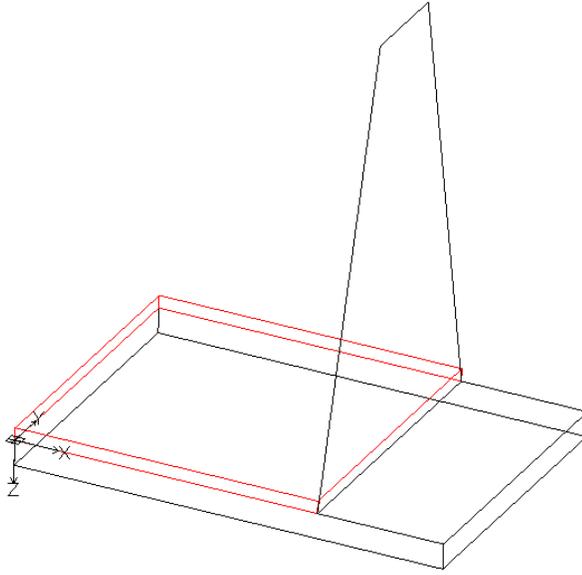
Command: polyline3d  
Specify start point» 3,0,0

Options: Undo, Enter to go back or  
Specify vertex position» 3,3,0

Options: Undo, Enter to exit or  
Specify next vertex position» 3,2,-4.25

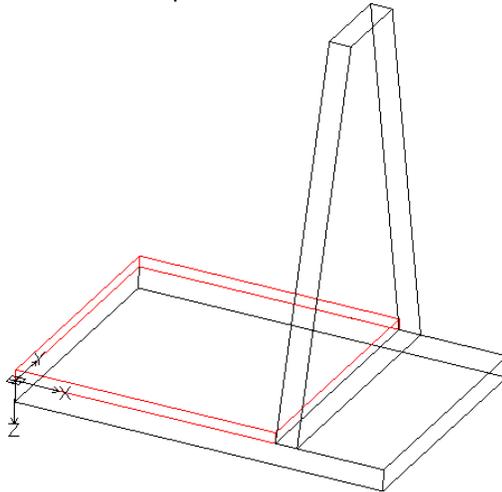
Options: Close, Undo, Enter to exit or  
Specify next vertex position» 3,1,-4.25

Options: Close, Undo, Enter to exit or  
Specify next vertex position» c



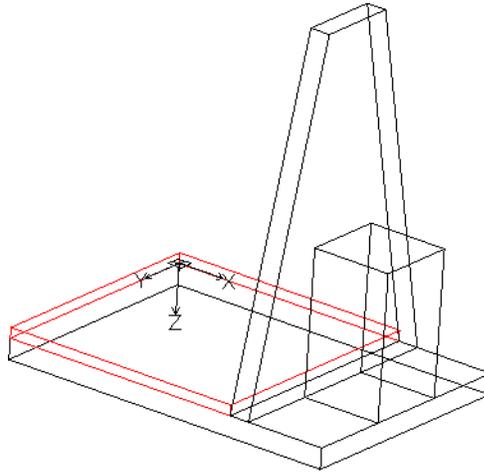
3. “Fatten” the trapezoid by applying the Extrude command, as follows:
  - a. Start the **Extrude** command by one of the methods listed above.  
Command: extrude
  - b. Select the trapezoidal polygon.  
Specify entities» (Pick trapezoid.)  
Specify entities» (Press return to end entity selection.)
  - c. Specify the extrusion distance, 0.25”:  
Options: Path, Taper angle or  
Specify extrusion height» .25

Notice that iCADMac turns the 2D trapezoid into one that is 3D.



4. Add the square box, which has a 1x1” base and is 1.75” tall. It is located at the center of the end of the main plate. Place it with the coordinates listed below, and then use the **Length** option to specify its size:  
Command: box

First corner of box» 3.25,2  
 Options: Cube, Length or  
 Other corner» I  
 Length» -1  
 Width» -1  
 Height» -1.75



5. Press **command+S** to save your work.



## Drawing Cylinders

**Command:** Cylinder

**Alias:** cy

### Summary of Extrude Command Options

Command: extrude

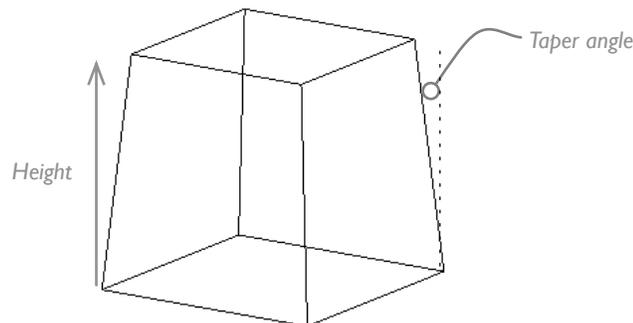
Active wire frame density: ISOLINES=4

Specify entities» (Pick one or more closed 2D entities.)

Specify entities» (Press return to end entity selection.)

Options: Path, Taper angle or

Specify extrusion height» (Enter an option, or pick two points.)



**Path** — selects an open 2D entity that defines the extrusion path:

Specify extrusion path» (Pick an entity.)

**Taper angle** — specifies the angle at which the extrusion narrows or expands.

Specify angle of taper for extrusion» (Enter an angle.)

**Extrusion height** — specifies the extrusion distance.

**Menu Bar:** Draw | Modeling | Cylinder

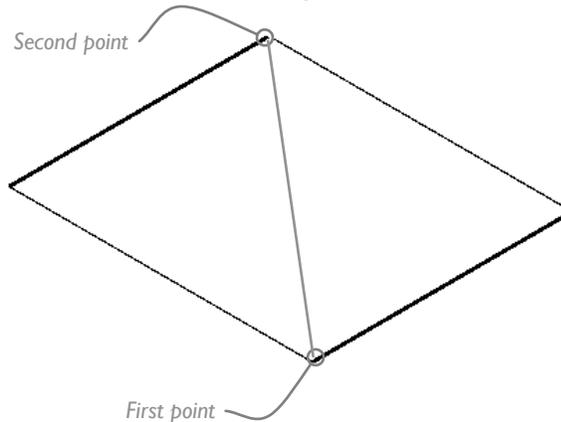
**Tool Matrix:** Modeling | Cylinder

The square box has a hole drilled through it. Making holes in solid models takes two steps:

- i. Draw a cylinder the same size as the hole.
- ii. *Subtract* the cylinder from the rest of the model.

The tricky part is centering the cylinder in the square box. The easiest way to accomplish this is to draw a line that spans the diagonal of the box top, and then start the cylinder at the midpoint of the line. This temporary line is also known as a “construction” line.

1. On the status bar, ensure **ESnap** is turned on. You will need the endpoint and midpoint entity snaps.
2. Start the **Line** command, and then draw a diagonal line, as illustrated in the figure below.

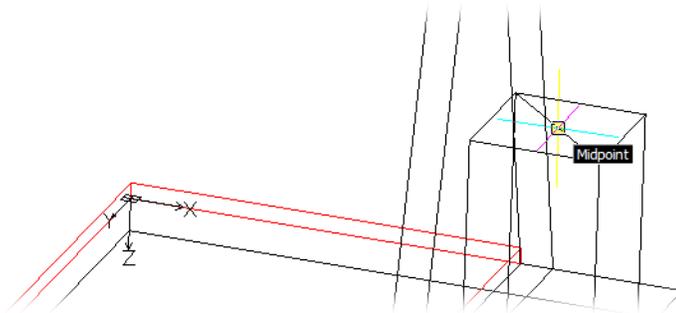


3. Start the **Cylinder** command using one of the methods listed above.

Command: cylinder

- a. Pick the midpoint of the line:

Options: 3P, 2P, TTR, TTT, Elliptical or  
Center point for base of cylinder» (Pick the midpoint of the line.)

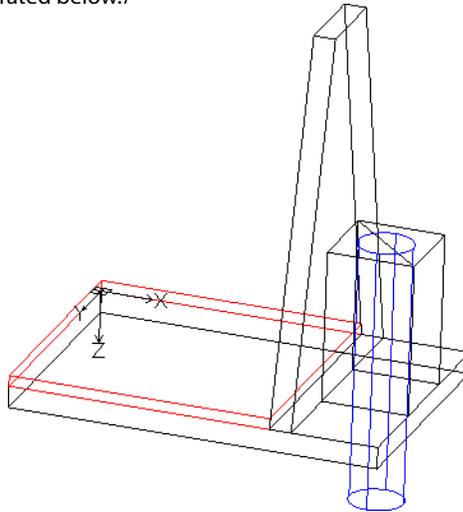


- b. Specify the 0.625” diameter of the cylinder using the **Diameter** option:

Options: Diameter or  
Radius for base of 1» d  
Specify diameter for base of cylinder» 0.625

- c. The length does not matter, because it will be erased in the next step. It only matters that the cylinder is longer than the combined length of the boxes, or 2”.

Options: Center of other end or  
 Height of 1» (Drag the cursor so that the cylinder extends below  
 the boxes, as illustrated below.)



4. Erase the construction line, as it is no longer needed.

## Applying Boolean Operations

Parts of solid models can be added and subtracted from each other, creating new parts. These operations are known as “booleans,” named after George Boole who invented this logic. The operation to join two parts is known as “union”; removing one part from another is “subtract.”



### Subtracting Solids

**Command:** Subtract  
**Alias:** su  
**Menu Bar:** Modify | Solid Editing | Subtract  
**Tool Matrix:** Modeling | Subtract

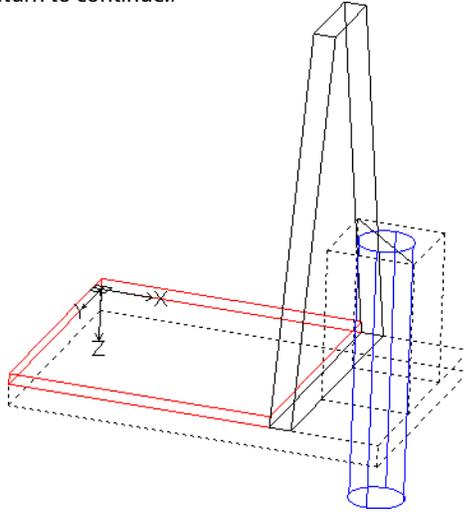
To create the hole, subtract the cylinder from the model. It can be a bit tricky to understand how the Subtract command works — which entities are being subtracted and which subtracted from:

- i. First, select the entities from which the cylinder will be subtracted, the boxes in this tutorial.
  - ii. Then, select the entities that will be subtracted, the cylinder in this case.
1. Start the **Subtract** command using one of the methods listed above.  
 Command: subtract
  2. Select the two boxes through which the cylinder passes, shown with dashed lines in the figure below:  
 Specify solids and regions to subtract from...  
 Specify entities» (Select the square box.)  
 1 found, 1 total

Specify entities» (Select the baseplate.)

1 found, 2 total

Specify entities» (Press return to continue.)



The figure shows the cylinder in blue to distinguish it from the rest of the model.

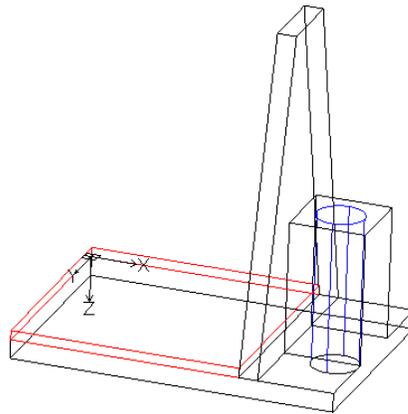
3. Select the cylinder:

Specify solids and regions to subtract...

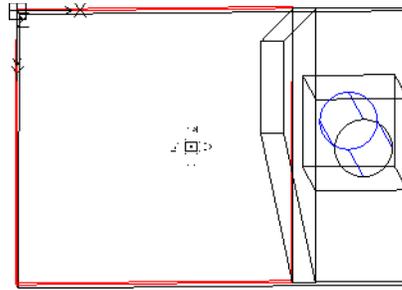
Specify entities» (Select the cylinder.)

1 found, 1 total

Specify entities» (Press return to end the command.)



4. To show that the hole now exists in the model, use the **RollView** command to twist the model around.



## Joining Solids

**Command:** Union

**Alias:** uni

**Menu Bar:** Modify | Solid Editing | Union

**Tool Matrix:** Modeling | Union

All of the parts are welded together, except for the red rubber mat. You can simulate the welding process through the Union command: it joins two or more solid partse. (Unbeknownst to you, the Subtract command had unioned the two solids through which the hole passed.)

To join two or more solid entities, follow these steps:

1. Enter the **Union** command.  
Command: union
2. Select the solid entities to join:  
Specify entities» (Select all entities in the drawing — except the red rubber mat.)  
Specify entities» (Press return to end the command.)

**TIP** You can use the **Intersect** command to remove all solids except for the volume in common.

3. Press **command+S** to save the drawing.



## Analyzing Solid Models

**Command:** MassProp

**Menu Bar:** Tools | Inquiry | Region/Mass Properties

Solid models are unique in CAD, because some of their physical properties can be determined. The MassProp command is short for “mass properties,” and it reports on the volume, moments of inertia, and so on.

1. Enter the **MassProp** command.  
Command: massprop
2. Select all the entities in the *3d model* drawing:  
Specify entities» (Select all solid model parts.)  
3 found, 3 total  
Specify entities» (Press return.)
3. Press function key **fn+F2** to switch to the Command Window, and then review the report:

```

----- SOLIDS -----
Mass: 7.5739
Volume: 7.5739
Bounding box X: 0.0000 -- 4.2500
Y: 0.0000 -- 3.0000
Z: -4.2500 -- 0.2500
Centroid: X=2.5566 Y=1.5000 Z=-0.5949
Moments of inertia: X=32.1967 Y=70.3458 Z=80.7330
Products of inertia:
XY: 29.0446
YZ: -6.7589
ZX: -15.0347
Radii of gyration: X=2.0618 Y=3.0476 Z=3.2649
Principal moments and X-Y-Z directions about centroid:
I: 9.7138 along [0.7864,0.0000,-0.6177]
J: 18.1623 along [0.0000,1.0000,0.0000]
K: 16.9498 along [0.6177,0.0000,0.7864]

```

The sole problem is that you cannot assign a density to represent the materials, whether steel or wood. This means that the mass is always equal to the volume, because density is set to 1.

### Summary of MassProp Report

**Mass** equals the volume, because density = 1.

**Volume** is the 3D space occupied by the 3D solid.

**Bounding Box** reports the x, y, z coordinate triple of a 3D box enclosing 3D solids or bodies.

**Centroid** reports the x, y, z coordinates of the center of mass of 3D solids and bodies.

**Moments of Inertia** are  $Mass * Radius^2$  about each axis.

**Products of Inertia** are  $Mass * Distance$  (of centroid to y, z axis) \*  $Distance$  (of centroid to x, z axis).

**Radii of Gyration** are  $(MomentOfInertia / Mass)^{1/2}$

**Principal Moments** are the axes along which the mass is symmetrical.

## Summary

This concludes the tutorial. You can save the drawing, and then exit iCADMac.

In the next chapter, you learn how to set up multiple views of this 3D drawing through the use of viewports.



## Chapter 14

# Multiple Viewports, Multiple Drawings

### In This Chapter

- Viewing 3D models from multiple viewpoints in view tiles
- Creating overlapping viewports in sheet mode
- Attaching referenced drawings

**Viewing a 2D** plan is straightforward because the entire drawing is always visible from plan view. Viewing 3D models is tougher; you need to see all six sides — top, bottom, left, right, front, and back — but only three at most are visible at a time.

In the last chapter, you used the RollView command to rotate your viewpoint around the 3D model interactively. An alternative is to view the 3D model from multiple viewpoints at the same time, albeit from static ones.

The ViewTiles command splits the iCADMac drawing area into two or more windows, known as “view tiles.” Each view tile displays a different viewpoint; each can also have settings independent of the others, such as whether snap or grid is turned on, as well as have different zoom or pan settings.

In addition to multiple views of one model, a drawing can display multiple drawings by attaching additional *.dwg* files. These are known as “reference files,” and act somewhat independently of the primary drawing.

## Key Terms in This Chapter

**Sheet Mode** — displays drawings as if they were on a sheet of paper; also known as “paper space” or “layouts.”

**Reference Files** — refers to additional drawing files attached to the active drawing.

**View Tiles** — refers to tiled windows in model mode.

**Viewports** — refers to overlapping windows in sheet mode.

## Abbreviations

**command+R** — moves the focus from one view tile or viewport to the next.

**command+Tab** — switches focus between open drawings.

## Commands

Command	Shortcuts	Menu Selection
References	xr, xref	Tools   References
Viewport	mv, mview	... <sup>1</sup>
-Viewport	-vports	...
ViewTiles	viewports, vports	View   Viewports   Viewports

<sup>1</sup> No menu selection.

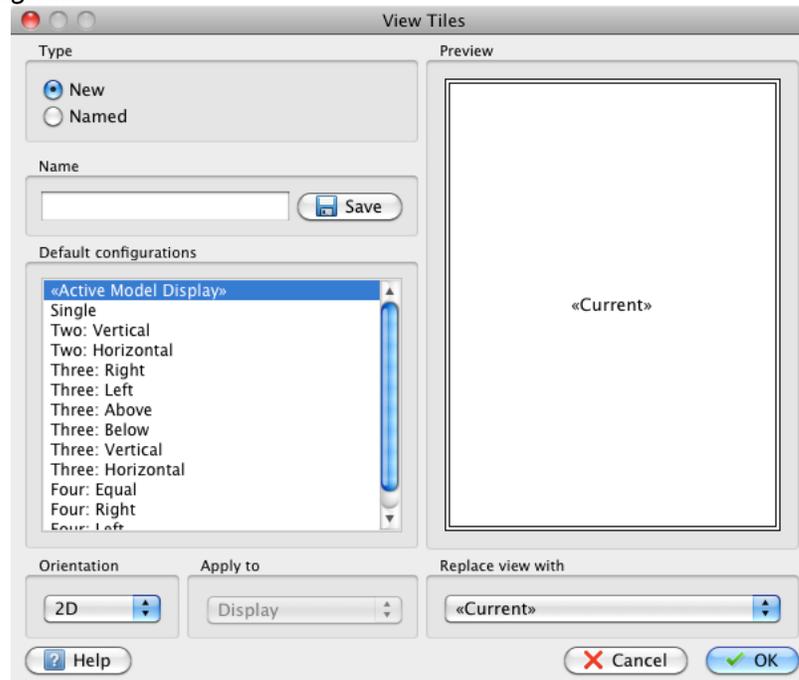
## Multiple Viewports

- Command:** ViewTiles  
**Aliases:** viewports, vports, vtiles  
**Menu Bar:** View | Viewports | Viewports  
**Shortcut:** command+R  
**Tool Matrix:** View Tiles | View Tiles Manager

In the following tutorial, you split the drawing area into four, and then set each *view tile* to display last chapter's 3D model in one of the four standard engineering views: front, right, top, and isometric.

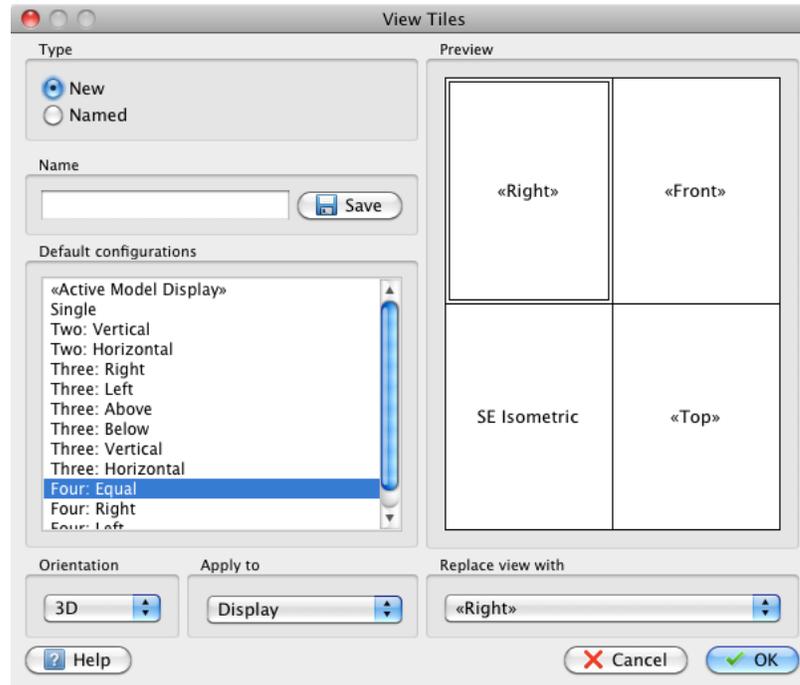
If necessary, start iCADMac, open the *3d model.dwg* file, and then follow these steps:

1. Enter the **ViewTiles** command using one of the methods listed above. Notice the View Tiles dialog box.



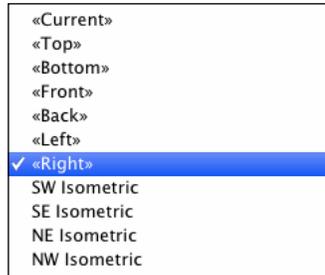
2. This dialog box has many controls. Here is how to split the drawing area into multiple tiles:
  - a. From the Orientation droplist, choose **3D**. This ensures the 3D model is seen from multiple viewpoints.
  - b. In the Default Configurations list, choose **Four: Equal**. This splits the drawing area into four equal-sized view tiles.

Notice that the Preview area shows the names of the four viewpoints. These four views are standard for engineering drawings. See the figure on the next page.

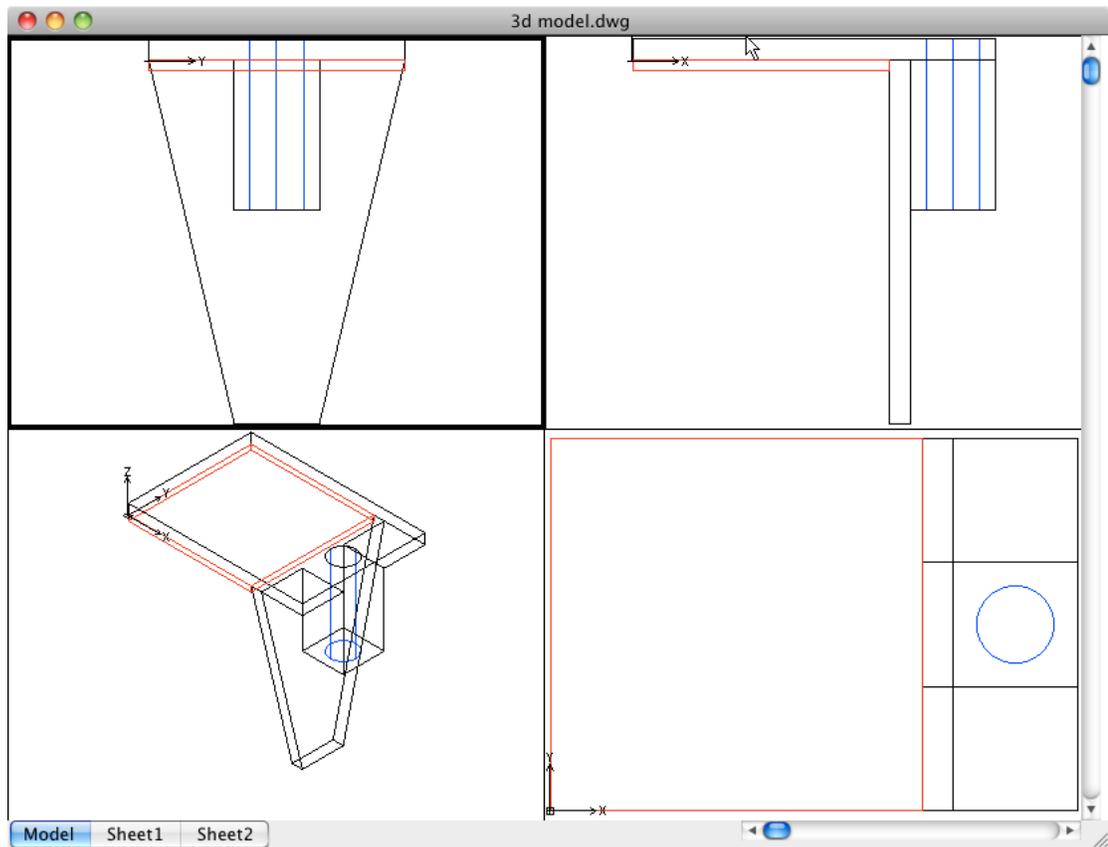


**TIP** The **Replace View With** droplist lets you change the viewpoint of each tile:

1. In the Preview area, click on a view tile.
2. From the **Replace View With** droplist, choose a different viewpoint name.
3. Repeat for the other view tiles.



3. Click **OK**. Notice that the drawing area splits into four, and that each view tile shows the 3D model from a different viewpoint.



(If you see the model only partially, or not at all, then use the **Zoom Fit** command in each view tile.)

4. You can edit the model in each view tile. To switch to a different view tile, simply click in it, or press **command+R**. The *active* view tile has the heavy border.

Drawing aid settings, such as snap, grid, and even as hidden-line removal (HideView command), can be set independently for each view tile.

View tiles have their limitations, however. They can only *tile*, which means they cannot overlap each other. You can only print the active view tile, not all four at once. The solution is described next.

## Sheets

A second way to see multiple viewpoints is with *sheets*, which are designed for printing, and prove more flexible than view tiles. Indeed, a sheet looks like a piece of paper onto which one or more *viewports* are opened.

Like view tiles, viewports show any part of any 2D or 3D drawing.

Viewports differ from view tiles, however, because they are entities in their own right. You can edit the rectangular boundary of viewports, for example resize, move, and copy. You can apply properties that are unique to viewport boundaries, such as locking them or specifying the rendering style. Indeed, viewports don't even have to be rectangular: they can be any shape.

Whereas a drawing can have just one set of view tiles, the drawing can have up to 256 sheets, each customized for a specific printer. For instance, you can have one sheet set up for large-format color printers, another for small-format monochrome, and so on.

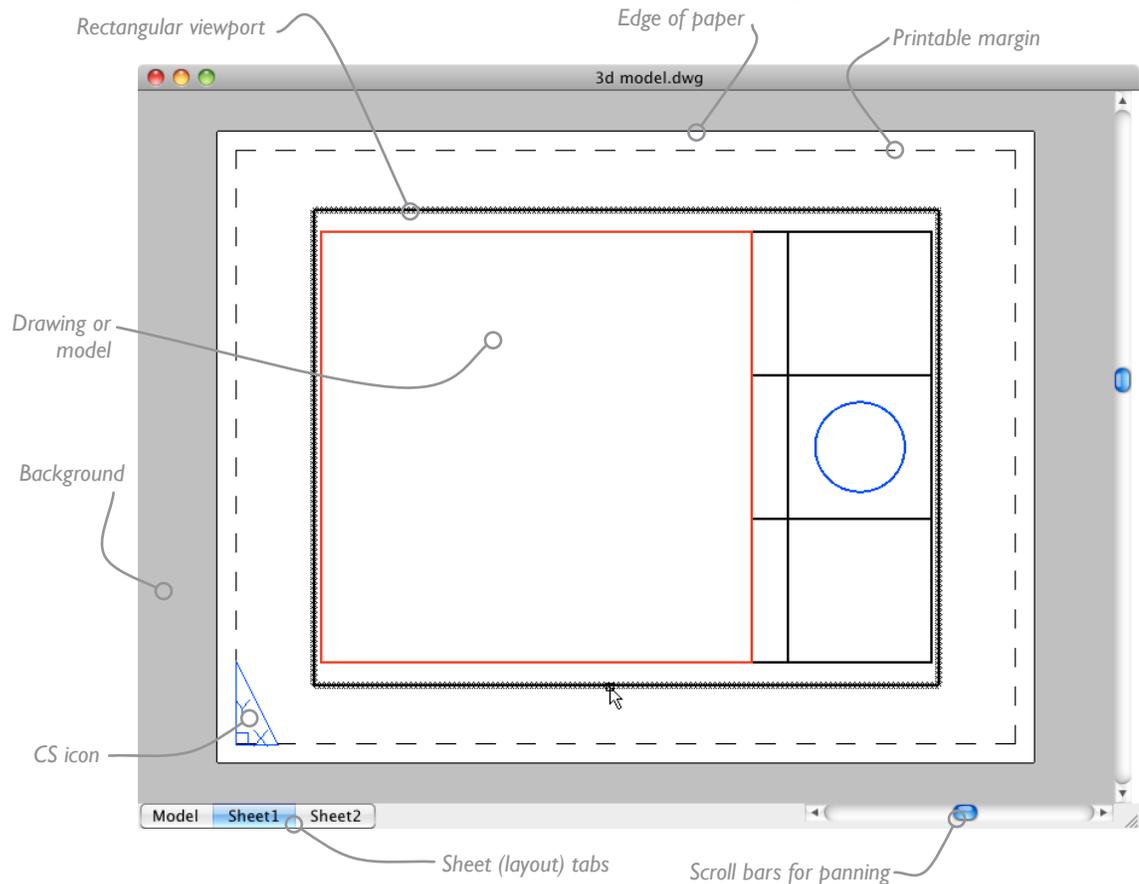
**TIP** “Paper space” and “layouts” are other names for sheets.

In the following tutorial, you switch the drawing from model to sheet view, and then open multiple overlapping viewports.

1. Below the drawing area are three *tabs* labelled Model, Sheet1, and Sheet2. Click on **Sheet1**.



2. Notice that the drawing area changes its look, and that the plan view of the 3D model appears. This is known as *sheet mode*; its parts are labelled in the figure below.



**Edge of Paper** shows the sheet of paper used by the currently-assigned printer. In many cases, the white rectangle represents a standard A- or A4-size sheet.

**Printable Margin** is indicated by the dashed rectangle. Outside of it, the printer cannot print, and so it makes sense to keep the model within the dashed rectangle.

**Background** is shown in gray; it too is an unprintable area.

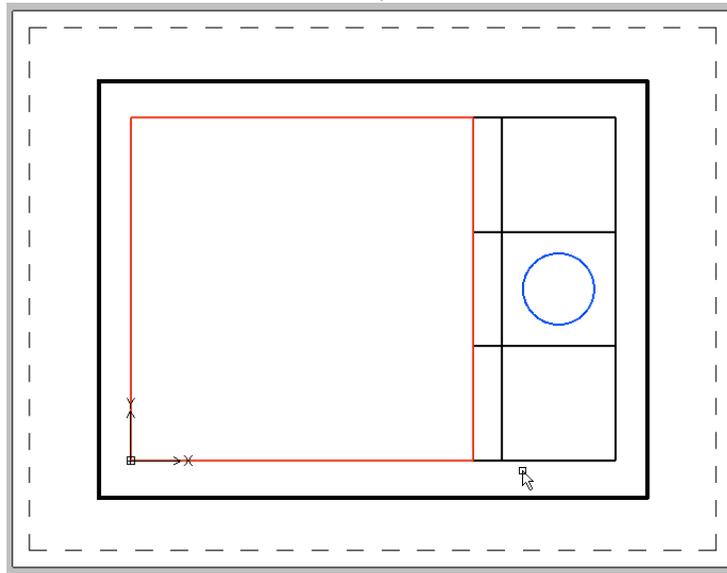
**Viewport** is the black rectangle that shows the 3D model or 2D drawing. Sheet mode can have multiple overlapping viewports.

**CS Icon** looks like the triangle used for hand drafting, which reminds you that the drawing is currently in sheet mode. It performs no other useful function.

3. Just as you could in model mode's view tiles, you can edit in sheet mode. Here's how:
  - a. On the status bar, notice the SHEET button. It reminds you that the drawing is currently in sheet mode.



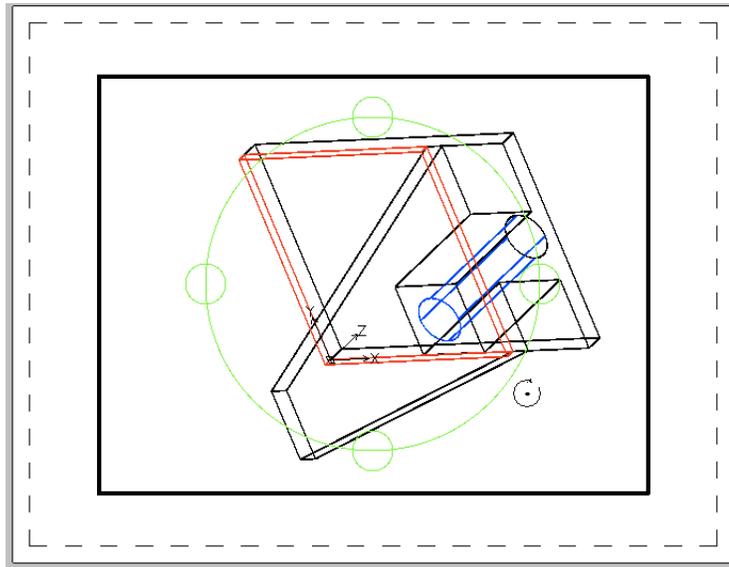
- b. Move the cursor inside the viewport, and then double-click. Notice the changes that take place:
    - The drawing is surrounded by a heavy black rectangle.
    - The CS icon moves to inside the viewport, and reverts to its familiar 2D look.



- The button on the status bar changes to MODEL.



- c. Use the **RollView** command to change the viewpoint of the model. Notice that the area outside of the viewport remains in place. The content of the viewport is independent of the rest of the sheet.



- d. To switch back to sheet mode, double-click outside the viewport (or click the **MODEL** button).

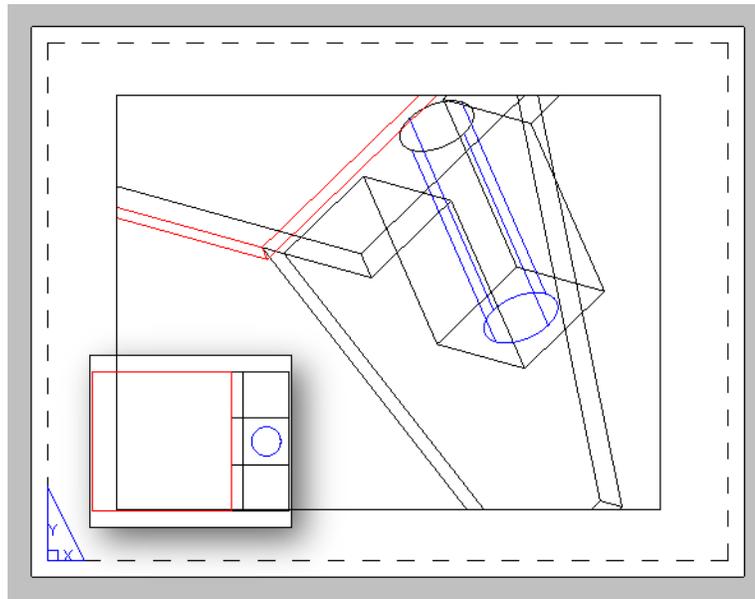
### Creating Additional Viewports

---

**Command:** Viewport  
**Aliases:** mv, mview

When you enter sheet mode for the first time, iCADMac creates the first viewport automatically. You create additional ones with the Viewport command, as follows:

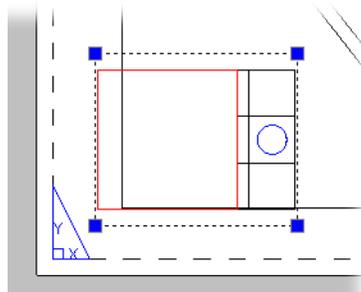
1. Enter the **Viewport** command using one of the methods listed above.  
 Command: viewport
2. Pick a point on the sheet:  
 Default: Fit  
 Options: 2, 3, 4, Entity, Fit, Lock, OFF, ON, Polygonal, Restore, SHaded view or  
 Specify start corner» (Pick a point.)
3. Pick a second point, and then notice that the model appears inside the rectangle formed by your two pick points. In the figure below, I highlighted the second viewport (seen in lower left corner).  
 Specify second corner» Pick another point to define the rectangle.)



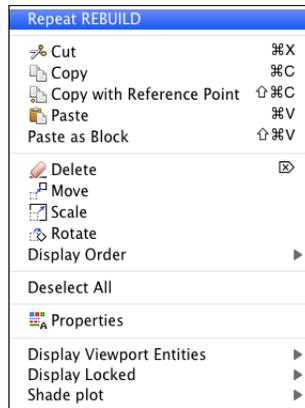
4. You can edit the rectangular boundary of the viewport, because it is an entity. For instance, you can resize, copy, and delete it. A viewport boundary has the same properties as other entities, so you can use the Properties palette to change the layer, color, line style, and so on. Note that these editing and property changes work only when the sheet is in paper mode.

Here are some of the actions you can apply to viewport boundaries:

- a. Make sure the drawing is in *paper* mode by double-clicking outside the viewport boundary.
- b. Select a viewport boundary. Notice that it reacts like other entities: the rectangular board is highlighted (dashed) and grips appear.



- c. Grab the boundary with the cursor, and then drag it. Notice that it moves around the sheet.
  - d. You can resize the rectangular boundary by dragging a grip.
  - e. To make a copy, use the **Copy** command.  
To delete the viewport, use the **Erase** command.  
To bring back an erased viewport, use the **U** command.
5. You can apply properties to the viewport, as follows:
  - a. Select a viewport, and then right-click.
  - b. Notice the shortcut menu. Near the bottom are a number of options specific to viewports:



**Display Viewport Entities** turns on and off the display of the contents of the selected viewport. When off, the viewport is blank (empty):

- **Yes** displays the viewport contents (default setting).
- **No** hides the viewport contents.

**Display Locked** prevents editing of the viewport when on:

- **Yes** locks the viewport, making its contents uneditable.
- **No** unlocks the viewport, allowing editing (default setting).

**Shade Plot** displays the content of the viewport in one of several shaded modes:

- **As Displayed** displays the contents of the viewport in their default mode. For instance, if the model was in hidden-line mode before you created the viewport, it displays now in hidden-line mode.
- **Wireframe** displays the content of the viewport in wireframe mode (no hidden-line removal or rendering).
- **Hidden** displays the contents in hidden-line mode. Entities hidden by other entities are not displayed.
- **Rendered** displays the contents as a rendering.

**TIPS** Viewport boundaries do not need to be rectangular; they can be any closed shape. To create a circular viewport, follow these steps:

1. Draw a circle with the **Circle** command.
2. Start the **Viewport** command, and then choose the **Entity** option.
3. Choose the circle at the 'Select clipping entities' prompt.

#### Summary of Viewport Command Options

Command: viewport

Default: Fit

Options: 2, 3, 4, Entity, Fit, Lock, Off, ON, Polygonal, Restore, SHaded view or

Specify start corner» (Pick a point.)

Specify second corner» (Pick another point.)

**2, 3, 4**— specifies the number of tiled viewports; after they are created, you can move and resize them.

**Entity** — converts the selected entity into a clipping boundary.

**Fit** — creates a single viewport that fits inside the margins.

**Lock** — locks the viewport to prevent editing, moving, and so on.

**Off, ON** — toggles the visibility of the contents of the selected viewport.

**Polygonal** — specifies the vertices of a polygonal (non-rectangular) viewport.

**Restore** — returns the viewport to its default settings.

**SHaded view** — changes the display mode of the viewport, from wireframe to hidden-line removal or shaded.

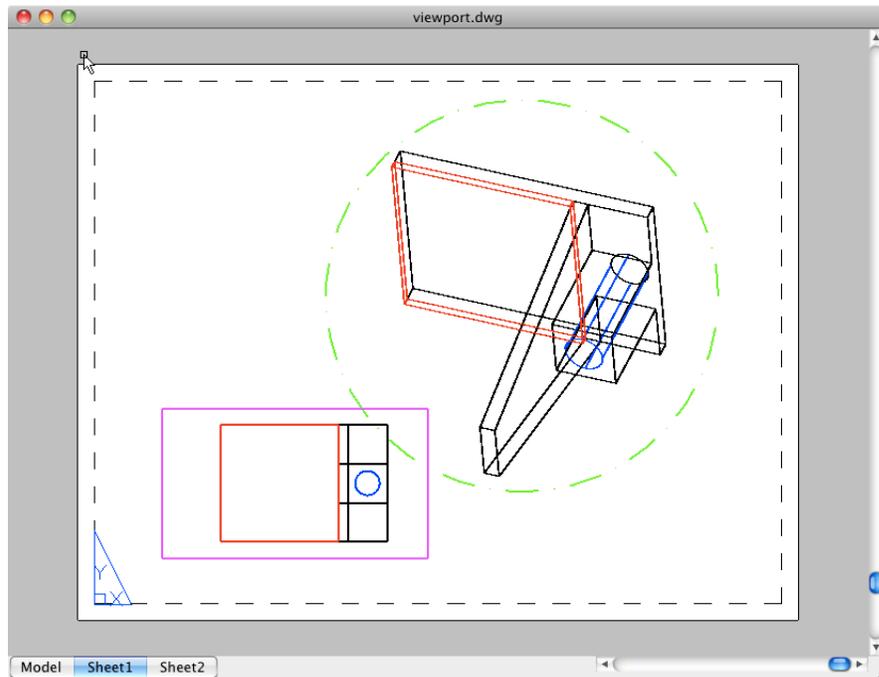
Command: viewport

Default: Fit

Options: 2, 3, 4, Entity, Fit, Lock, OFF, ON, Polygonal, Restore, SHaded view or

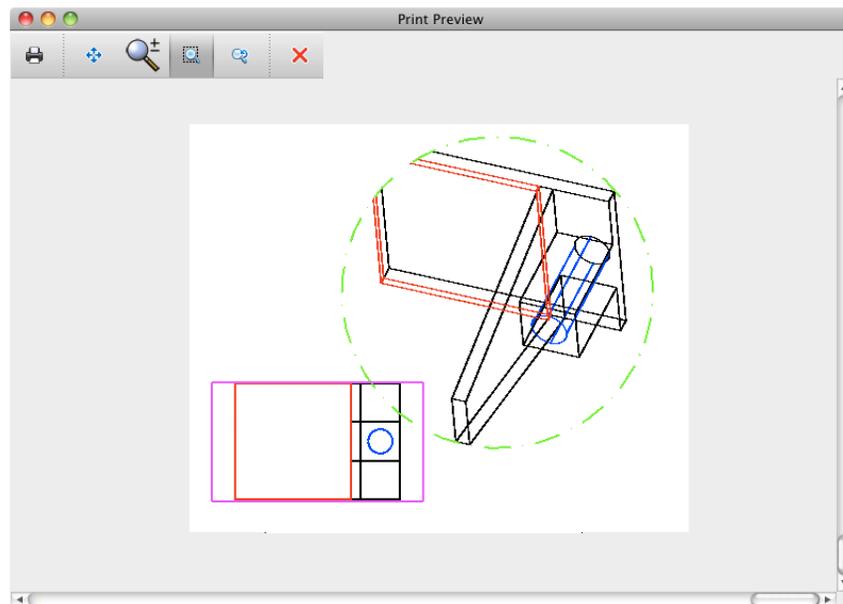
Specify start corner» e

Specify clipping entities» (Choose the circle.)



In the figure above, I changed the properties of the two viewport boundaries, selecting other colors and line types.

When you print the drawing in sheet mode, you get what you see, as illustrated by the Print Preview window below.



## Attaching Multiple Drawings

**Command:** References

**Aliases:** er, externalreferences, refs, xlink, xr, xref, image, im

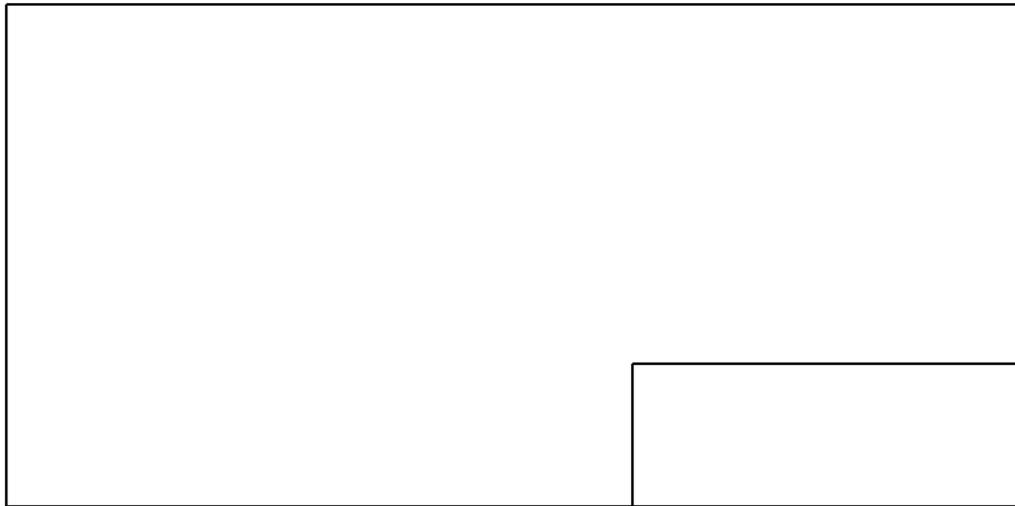
**Menu Bar:** Tools | References

**Tool Matrix:** Reference | References Manager

In the previous tutorials, you learned how to create multiple views of the same drawing. iCADMac also lets you view more than one drawing at the same time. Through the References command, you can attach one or more drawings to be displayed at the same time.

To see how this is accomplished, you first create a simple drawing border in the following tutorial, and then attach it to the 3D model drawing:

1. Start a new drawing with the **New** command.
2. Use the **Rectangle** command to draw a simple drawing border and title block, as illustrated below. (The size does not matter, since the drawing will be scaled later.)



3. With the **Save** command, name the drawing as *border.dwg*, and then exit the drawing with the **Close** command.

4. Back in the *3d model.dwg* drawing, collapse the four viewports into one with the **-Viewport** command, as follows:

Command: -viewport

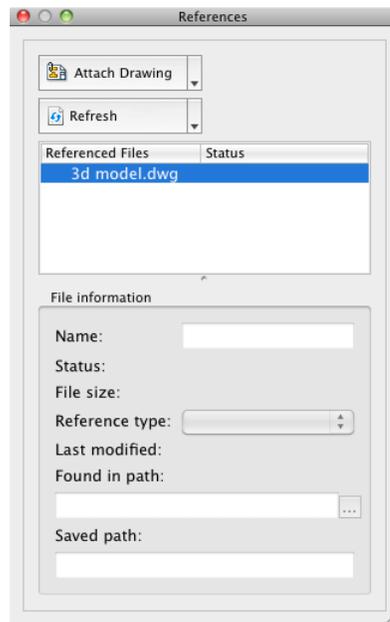
Default: 3

Options: ? to list, 2, 3, 4, Delete, Join, Restore, Save, or Single

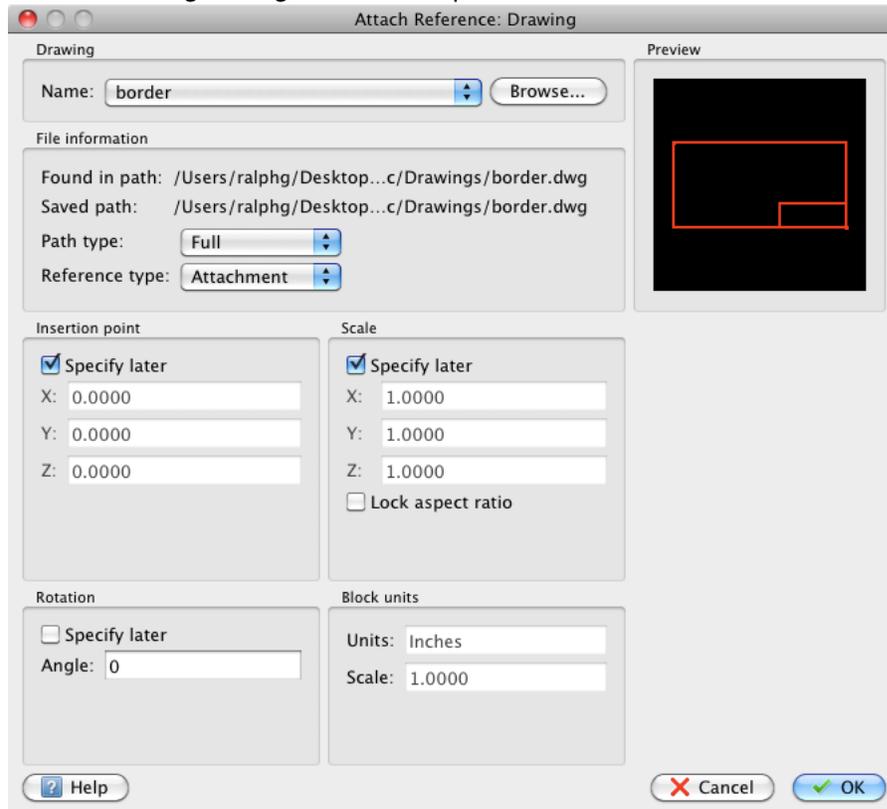
Specify option» si

(You can also restore a single view tile in model space with the ViewTiles command.)

5. Start the **References** command using one of the methods listed above. Notice the Reference palette.



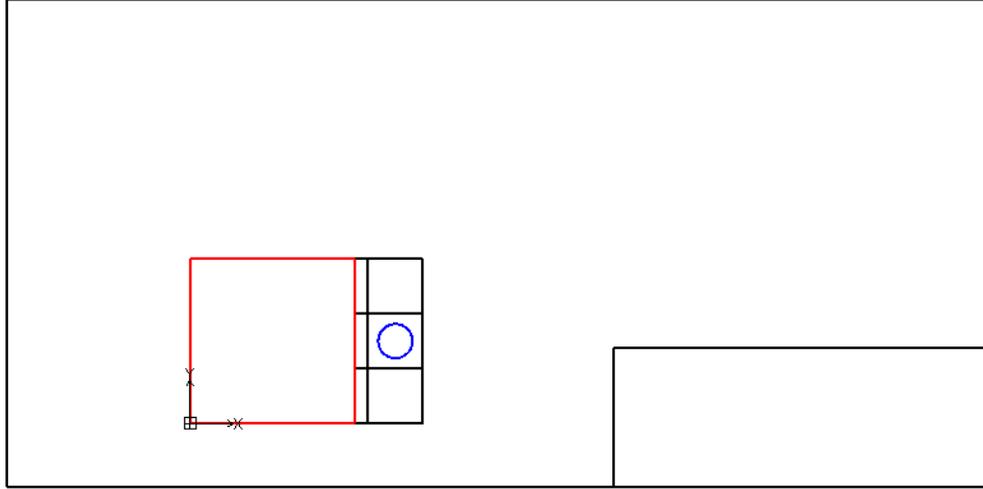
6. To attach a drawing, click  **Attach Drawing**. (This command also attaches raster images.) Notice the Select File dialog box.
7. Choose *border.dwg* and then click **Open**.
8. Notice the Attach Reference Drawing dialog box. It operates much the same way as the Attach Reference Image dialog box from Chapter 10.



Make the following changes to the settings in the dialog box:

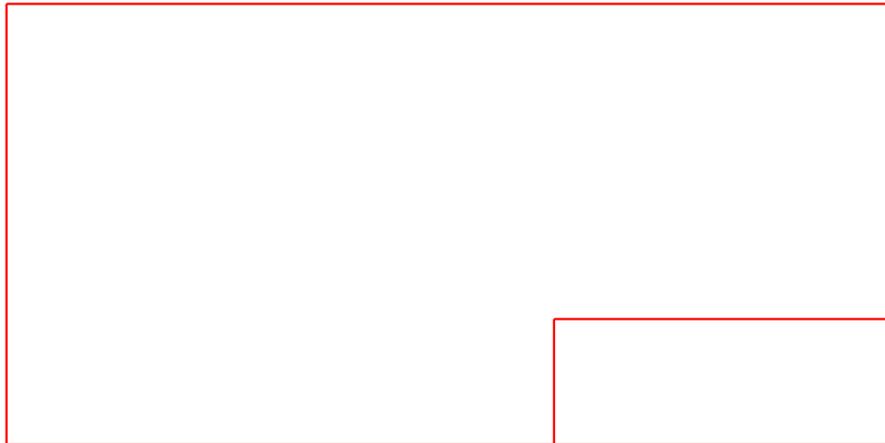
Insertion point	Specify Later
Scale	Specify Later
Rotation	0

9. Click **OK** to exit the dialog box, and then follow the prompts in the Command window:
  - Specify position» (Pick a point in the lower left corner of the drawing.)
  - X scale or specify opposite corner» (Pick a second point in the upper right corner of the drawing.)
  - Default: 1.0000
  - Specify Y scale» (Press return.)



Attached, the *border.dwg* drawing acts like a block. Yet it is independent of the *3d model* drawing. Another drafter can continue working on *border.dwg*, changing it. When you update the *border.dwg* in the 3D model, you see the changes.

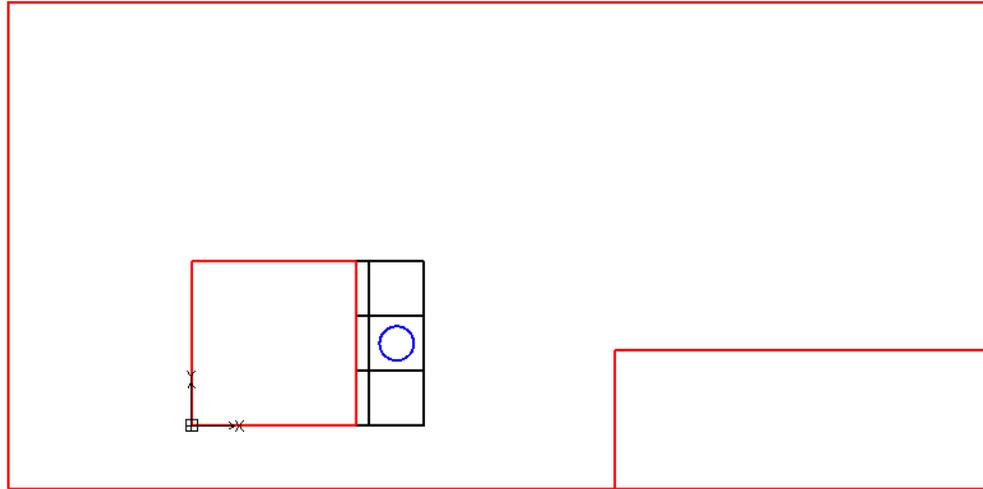
1. Open *border.dwg*, and then change the color of its rectangles to red.



2. Save the drawing.
3. Return to the *3d model* drawing. (Press **F9** to see all open documents, and then choose *3d model*.)
4. Reload the drawing by following these steps:
  - a. In the References palette, select **border**,
  - b. Click the Refresh droplist.
  - c. In the droplist, choose **Reload All**.



Notice that the drawing border turns red.



5. You can attach drawings in sheet mode as well by following these steps:
  - a. Click **Sheet1**.
  - b. Run the **References** command, and then place *border.dwg* on the sheet.

**TIP** You can apply a limited number editing commands to attached reference files, such as Move and Rotate.

This concludes the tutorials.

## Summary

This concludes the *Inside iCADMac for Macintosh* book. The appendices that follow contain summaries of information that you may find helpful for working with iCADMac.



Part **IV**

# Appendices



## Appendix A

# Alphabetical Command Summary

### In This Appendix

- Brief descriptions of command names, listed in alphabetical order

**i**CADMac works with the commands listed on the following pages in alphabetical order.

A - (dash) prefix indicates commands that display their prompts at the command line, such as -Layer; the equivalent command names without the dash display dialog boxes, such as Layer.

The following commands are available only in the Windows version of CADs, and so are not available in iCADMac for Macintosh.

- ConvertOLE
- EnterPoint
- ExportEmf
- InsertObject
- OpenOLE
- ResetOLE

The WindowsCalc command is renamed OSCalc in iCADMac for Macintosh.

**A**

**About** displays information about the program.

**ActivateLayer** activates the layer of a specified entity.

**Align** aligns entities with other entities in 2D and 3D space.

**Align3D** aligns entities with other entities in 2D and 3D space by specifying a source and a destination plane.

**AngleDimension** draws angle dimensions.

**Arc** draws arcs.

**AreaBoundary** or **-AreaBoundary** draws polyline or region entities of enclosed areas.

**ARender** displays the current drawing rendered in a separate window.

**AttachDrawing** attaches referenced drawings to the current drawing.

**AttachImage** attaches images to the current drawing.

**AttachLink** attaches hyperlinks to entities or areas.

**AutoRebuild** controls automatic drawing rebuilds.

**B**

**BaselineDimension** continues linear, angular, or ordinate dimensions from the baseline of the previous or selected dimension.

**BlipMode** enables and disables display of marker blips.

**BlockAttributeOutput** writes block attribute values of selected blocks to text files.

**Box** draws three-dimensional solid boxes.

**BoxText** turns boxed text display and print mode on and off.

**C**

**CCS** sets, saves, and restores custom coordinate systems.

**CenterMark** draws center marks or center lines of circles and arcs.

**Chamfer** bevels entities.

**Check** evaluates the integrity of drawings and corrects errors.

**Circle** draws circles.

**Clean** or **-Clean** removes unused named references from drawings, such as blocks and layers.

**ClipboardCopy** copies entities to the Clipboard.

**ClipImage** defines clipping boundaries to suppress the display of portions of referenced images.

**ClipReference** defines clipping boundaries to suppress display of portions of blocks or references.

**ClipViewport** clips the display of tiled views on sheets with non-rectangular shapes.

**Close** exits the current drawing, but not the program.

**CloseAll** exits all open drawings, but not the program.

**CommandHistory** opens the command history in a window.

**Cone** draws three-dimensional solid cones.

**ContinueDimension** continues linear, angular, or ordinate dimensions from the second extension line of the previous or a selected dimension.

**ConvertPrintStyles** converts print styles between color-dependent and named.

**Copy** duplicates entities.

**Copy@** copies entities with a specified reference point to the Clipboard.

**CopyHistory** copies the command history to the Clipboard.

**CSIcon** sets the visibility and position of the coordinate symbol (CS icon).

**CSStyle** manages custom coordinate systems (CCS).

**Customize** customizes user interface elements, such as menus, toolbars, and shortcuts.

**Cut** copies entities to the Clipboard and deletes the entities.

**Cylinder** draws three-dimensional solid cylinders.

## D

**Delete** removes entities from drawings.

**DeleteLayer** deletes entities and the layer of selected entities.

**DetachImage** detaches referenced images attached to drawings.

**DetachLink** detaches hyperlinks that are attached to entities or areas.

**Dim** switches from command mode to dimension mode.

**Dim1** executes one dimensioning command and returns to the command prompt.

**DiameterDimension** draws diameter dimensions for circles and arcs.

**DimensionStyle** or **-DimensionStyle** creates and modifies dimension styles.

**DirectionalLight** inserts directional distant lights for renderings.

**DisplayBlockAttributes** defines block attribute visibility.

**DisplayImageFrame** determines whether frames are displayed at the borders of referenced images.

**DisplayOrder** changes the display order of entities and inserted images in the drawings.

**DisplayQuality** controls the resolution of circular and curved entities.

**DraftingOptions** sets drafting options, such as snap, grid, ortho, entity snaps, and entity tracking.

**DraftingStyles** manages text styles, dimension styles, richline styles, and table styles; sets styles for use with subsequent commands.

**DrawingBounds** sets and controls the drawing bounds.

**DrawingSettings** sets the behavior of drawing commands, the appearance of certain drawing entities, the unit system of the current drawing, and manages Custom Coordinate Systems (CCSs).

## E

**EdgeMesh** draws 3D polygon meshes based on four adjoining edges.

**EditAnnotation** edits items, such as single-line text, dimension text, and attribute definitions.

**EditBlockAttribute** or **-EditBlockAttribute** modifies block attributes.

**EditBlockAttributeDefinition** modifies existing block attribute definitions.

**EditDimension** edits dimension text and extension lines.

**EditDimensionText** moves and rotates dimension text.

**EditHatch** or **-EditHatch** modifies hatches and fills.

**EditImage** attaches and detaches images to and from the drawing, and manages image references.

**Note** edits multiline text.

**EditPolyLine** edits polylines, polyline vertices, and 3D polygon meshes.

**-EditRichLine** adds and deletes control points in RichLines.

**EditSolid** edits bodies, faces and edges of three-dimensional solid objects.

**EditTable** changes text in a table cell.

**EditTolerance** modifies tolerances (datum indicators and basic dimension notation).

**EditVertex** edits 2D polyline vertices.

**Ellipse** draws ellipses and elliptical arcs.

**-EmbedDrawing** embeds dependent objects and styles of referenced drawings in the drawing.

**EntityGrips** sets preferences for editing with grips.

**EntityGroup** or **-EntityGroup** creates user-defined selection sets of entities.

**EntitySnap** or **-EntitySnap** sets running entity snap modes.

**Exit** exits the software.

**Explode** breaks complex objects into their component entities.

**Explorer** displays the operating system's file explorer.

**Export** saves entities in other file formats.

**ExportBMP** saves the drawing as a raster BMP file (device-independent Bitmap).

**ExportDrawing** or **-ExportDrawing** writes entities, blocks, or the entire drawing to a new drawing file.

**ExportDXF** exports the drawing as a vector DXF file (Drawing Interchange Format).

**ExportJPG** saves selected entities to a raster JPEG file.

**ExportPDF** saves the current view of the graphics display to a PDF file (Portable Document Format).

**ExportPNG** saves selected entities to a raster PNG file (Portable Network Graphics).

**ExportSAT** exports solid modeling objects from the drawing to an ACIS SAT file (Save As Text).

**ExportSLD** exports the current view to a vector SLD file (slide).

**ExportSVG** exports the current view to a vector SVG file (Scalable Vector Graphics).

**Extend** extends entities to bounding edges defined by other entities.

**ExtractBlockAttribute** extracts block attribute values of selected blocks to text files based on a template file defining the data to write.

**Extrude** extrudes entities as 3D solids or surfaces.

**F**

**Face** draws 3D faces.

**FileLocations** displays, sets, and edits default paths and/or file names.

**FileManager** displays the Finder file browser.

**FillArea** fills areas with a solid color or color gradient.

**Fillet** rounds entities.

**Find** finds and replaces text in notes, annotations, and dimension text.

**Flip** mirrors and deletes the original entity.

**FreezeLayer** freezes the layers of selected entities.

**FullScreen** displays the graphics area as a full screen (without toolbars and palettes).

**G**

**GetArea** calculates the area and perimeter of entities or of defined areas.

**GetDistance** measures the distance and angle between two points.

**GetProperties** displays information about selected entities.

**GetStatus** displays status information about settings and memory usage of the drawing.

**GetTime** displays time-related statistics about the drawing and lets you toggle (on or off), or reset a timer.

**GetXY** displays the coordinate values of a location.

**Gravity** determines the size of the entity snap target box.

**Grid** turns the grid display on or off and sets other grid options.

**H**

**Hatch** or **-Hatch** fills a selected boundary with a pattern.

**Help** displays online help.

**HideCommandHistory** closes the command history window.

**HideCommandWindow** hides the command prompt area.

**HideFullScreen** returns from full screen display mode to normal display mode.

**HideLayer** turns off the layer of the selected entity.

**HideLightlist** hides the lighting palette.

**HideProperties** hides the properties palette.

**HideReferences** hides the references palette.

**HideView** generates a hidden line view.

**Hyperlink** or **-Hyperlink** adds hyperlinks to entities or modifies existing hyperlinks.

**I**

**ImageQuality** determines the display quality of images attached to the drawing.

**ImportDWF** inserts DWF files.

**ImportDXF** opens the selected DXF file in a separate drawing window.

**ImportSAT** reads ACIS SAT files.

**InfiniteLine** draws infinite construction lines.

**InsertBlock** or **-InsertBlock** inserts blocks or another drawing into the current drawing.

**InsertShape** inserts shapes from SHX shape files into the drawing.

**Interfere** checks interferences between solid models.

**Intersect** creates regions or 3D solids from the intersection of regions or 3D solids.

**IsolateLayer** isolates the layers of selected entities.

**IsometricGrid** controls the isometric plane (left, right, or top) when isometric snap is used.

## J

**JoggedDimension** draws jogged radius dimensions for circles and arcs.

## L

**Language** sets the language to communicate with the software.

**Layer** or **-Layer** controls layers and layer properties.

**Leader** draws leader lines that connect annotations to drawing entities.

**Light** places lights in drawings.

**Lightlist** displays the lighting palette.

**Line** draws straight line segments.

**LinearDimension** draws linear dimensions.

**LineColor** or **-LineColor** sets the color for new entities.

**LineStyle** sets the global line style scale factor.

**LineStyle** or **-LineStyle** creates, loads, and sets line styles.

**LineWeight** or **-LineWeight** sets line weight options.

**LoadApplication** loads applications for use in the program.

**LoadMenu** adds menu or customization files to customization sources.

**LoadScript** executes a sequence of commands from scripts.

**LoadShape** loads shape files for use by the Shape command.

**LockLayer** locks the layers of selected entities.

## M

**MakeBlock** or **-MakeBlock** makes blocks from selected entities.

**MakeBlockAttribute** or **-MakeBlockAttribute** makes block attribute definitions.

**MarkDivisions** places evenly spaced point entities or blocks along the length of entities.

**Mask** covers existing entities with a blank area.

**MassProp** calculates and reports the mass properties of 3D solids and regions.

**MatchLayer** changes the layer of selected entities to match a destination layer.

**Mesh** draws 3D polygon meshes.

**Mirror** draws mirror image copies of entities.

**Mirror3D** draws mirror images of entities about a plane in 3D space.

**Model** toggles from a sheet tab to the model tab.

**ModelMode** toggles from sheet workspace to a model workspace on sheets.

**Modify** changes the properties of drawing entities.

**ModifyProperties** changes the layer, color, line style, linescale, and thickness of entities.

**Move** displaces entities a specified distance in a specified direction.

**MoveDimensionText** moves dimension text.

## N

**New** starts new drawing files.

**NontargetLight** places free lights in drawings (lights without a targets).

**Note** or **-Note** draws text entities.

## O

**ObliqueDimension** modifies the orientation of existing linear dimensions.

**Offset** offsets linear entities in parallel orientation.

**Open** opens an existing drawing file.

**OpenReference** opens referenced drawings in a new drawing window.

**OpenWebpage** displays the specified Web site with the computer's default browser.

**Options** configures program operating parameters.

**OrdinateDimension** draws ordinate point dimensions.

**OSCalc** opens the Macintosh calculator.

**Ortho** constrains the pointer so it moves parallel to the axes of the current coordinate system.

**OverrideDimensionStyle** overrides dimension style variables.

## P

**PageLayout** sets print and plot defaults.

**Pan** or **-Pan** moves the drawing display in the active view tile. (Pan performs dynamic pan.)

**PanDown** pans down.

**PanLeft** pans left.

**PanRight** pans right.

**PanUp** pans up.

**ParallelDimension** draws aligned linear dimensions.

**Paste** inserts text or entities from the Clipboard.

**PasteAsBlock** inserts entities from the Clipboard as a block.

**Pattern** or **-Pattern** draws multiple copies of entities in a pattern.

**Pattern3D** draws linear or circular patterns of entities in 3D space.

**PauseScript** pauses scripts.

**PlaneSurf** draws rectangular planar surfaces.

**PlanView** sets plan view to construction plane.

**Point** draws point entities.

**PointFormat** defines the type and size of point entities.

**PointLight** places point lights in drawings.

**PolyFace** draws 3D polyface meshes, vertex by vertex.

**Polygon** draws equilateral closed polylines.

**Polyline** draws 2D polylines.

**Polyline3D** draws 3D polylines of straight line segments in 3D space.

**Preview** shows a preview before printing the drawing.

**Print** or **-Print** plots the drawing to a plotter, printer, or file.

**PrintStamp** prints or plots a stamp on the plot.

**PrintStyle** or **-PrintStyle** manages named print styles.

**Profiles** stores and manages individual profiles based on specified settings.

**Properties** displays the Properties palette to change drawing entity properties.

**PropertyPainter** applies properties from one entity to selected entities.

**Pyramid** draws three-dimensional solid pyramids.

## R

**RadiusDimension** draws radial dimensions for circles and arcs.

**Ray** draws semi-infinite construction lines.

**Rebuild** rebuilds the drawing and refreshes the active view tile.

**RebuildAll** rebuilds the drawing and refreshes all view tiles.

**RebuildDimension** updates dimensions with the active dimension style.

**Recover** repairs damaged drawings.

**Rectangle** draws a rectangular polyline.

**Redefine** restores built-in commands that have been undefined using the Undefine command.

**Redo** reverses the effects of a previous U or UndoN command.

**RedoN** reverses last or all previous U and UndoN commands.

**References** or **-References** displays the references palette.

**Refresh** refreshes the display of the active view tile.

**RefreshAliases** reloads the modified alias commands map file.

**RefreshAll** refreshes the display of all currently-open view tiles.

**Region** converts an entity enclosing an area into a region.

**RelateDimension** reestablishes the relation of specified dimensions to entities.

**Rename** or **-Rename** changes the names of objects.

**Repeat** repeats the specified command.

**ReplaceDimensionText** modifies the dimension text value.

**ResetDimensionText** undoes the movement of dimension text.

**ResumeScript** resumes an interrupted script.

**Revolve** draws 3D solids or surfaces by revolving 2D objects about an axis.

**RevolvedMesh** draws rotated surfaces about a specified axis.

**RichLine** draws richlines.

**RichLineStyle** draws and manages richline styles.

**Ring** draws filled circles or rings.

**RollView** controls 3D view points.

**Rotate** rotates entities about base points.

**Rotate3D** moves entities about 3D axes.

**RotateDimensionText** rotates dimension text.

**RuledMesh** draws ruled surfaces between two curves.

## S

**Save** saves the drawing under the current file name or a specified name.

**SaveAll** saves all open drawings.

**SaveAs** saves an unnamed drawing with a file name or renames the current drawing.

**Scale** enlarges or reduces specified entities equally in the X, Y, and Z directions.

**ScriptN** repeats script execution.

**Section** creates a cross section based on the intersection of a plane and 3D solids.

**Select** places selected entities in the 'Previous' selection set.

**SelectAll** selects all drawing entities including those not visible.

**SelectionFilter** determines a list of requirements that drawing entities must meet to be included in a selection set.

**SetVariable** lists or sets the values of system variables.

**Shade** shades the drawing mode.

**ShadeView** determines the shade mode.

**Sheet** creates, copies, renames, and deletes sheet tabs (layout modes).

**SheetMode** toggles from a model workspace to sheet workspace on sheets.

**ShowLayers** turns on all layers.

**ShowLicense** displays the license agreement.

**SimpleNote** displays text on screen as you enter it.

**Sketch** draws freehand lines.

**Slice** slices 3D solids with a plane or surface.

**SmartCalculator** performs mathematical and trigonometric computations and unit conversions.

**SmartLeader** draws leader lines and annotations.

**SmartOpen** opens a drawing specified at the command prompt.

**SmartNew** starts a new drawing with the default template drawing.

**SmartSelect** draws entity selections based on filtering criteria.

**Snap** restricts pointer movements and pointing in the drawing to specified intervals.

**Solid** draws solid-filled 2D faces.

**Sphere** draws three-dimensional solid spheres.

**Spline** draws quadratic or cubic spline (NURBS) curves.

**SpotLight** inserts spot lights into drawings.

**Stretch** moves or stretches entities.

**Subtract** creates a composite region or a 3D solid by subtraction.

**Sweep** creates solid primitives or surfaces by sweeping two dimensional entities along a path.

**SystemOptions** determines general operating options.

## T

**Table** or **-Table** draws tables in drawings.

**TableStyle** draws and manages table styles.

**TabulatedMesh** draws tabulated surfaces from a path curve and a direction vector.

**TargetpointLight** places targeted point lights.

**TextStyle** or **-TextStyle** creates or modifies text styles, and sets the active text style.

**ThawLayers** thaws all layers.

**Thicken** creates 3D solids by thickening surfaces.

**ToActiveLayer** changes the layers of specified entities to the active layer.

**Tolerance** draws tolerances (datum indicators and basic dimension notation).

**Toolbars** displays and hides toolbars.

**Torus** draws three-dimensional toroidal solids.

**Trace** draws traces.

**Trapezoid** draws trapezoidal 2D polylines.

**Trim** trims entities at a cutting edge defined by other entities.

## U

**U** reverses the most recent command.

**UiProfile** manages user interface profiles.

**Undefine** disables built-in commands.

**Undelete** restores deleted entities.

**UndoLayer** restores previous layer settings.

**UndoN** reverses the effect of multiple commands.

**Union** creates composite regions or solids by addition.

**UnisolateLayer** undoes layer isolation.

**UnrelateDimension** dissolves the relation of specified dimensions to entities.

**UnitSystem** or **-UnitSystem** sets coordinate and angle display formats and precision.

**UnlockLayer** unlocks locked layers.

**UserPreferences** defines drafting options, mouse options, and alias commands.

## V

**VerifyDimensions** verifies all measurements of related dimensions in the drawing.

**ViewDirection** sets the viewing direction for a 3D view of the drawing.

**Viewport** or **-Viewport** draws and controls viewports on sheet tabs.

**ViewportLayer** controls layer visibility in viewports on sheets.

**Views** or **-Views** saves, restores, and manages user-defined model and sheet views, and presets views.

**ViewSlide** displays captured slides in the drawing area.

**ViewTiles** draws and controls single or multiple model view tiles or sheet viewports.

## W

**Wedge** draws three-dimensional solids with a sloped face tapering along the X axis.

**Weld** joins two entities into a single entity.

**Windows** cascades and tiles windows, and arranges icons.

## Z

**Zoom** increases or decreases the visible part of the drawing.

**ZoomBack** zooms to previous view.

**ZoomDynamic** zooms the drawing in real-time (dynamic zoom).

**ZoomFactor** zooms by a factor.

**ZoomFit** zooms to the drawing extents.

**ZoomIn** zooms in to the drawing.

**ZoomOut** zooms out from the drawing.

**ZoomWindow** zooms to a region of the drawing specified by a box.

**ZPlane** sets elevation and thickness of new drawing entities.

## #

**3D** draws 3D polygon mesh objects: boxes, cones, cylinders, dishes, domes, pyramids, spheres, tori,

wedges, or meshes.

**3PointAngleDimension** draws angular dimensions based on three points.

**4PointAngleDimension** draws angular dimensions based on four points.

## Appendix B

# Alphabetical System Variable Summary

### In This Appendix

- Brief descriptions of system variables, with names listed in alphabetical order

**i**CADMac uses system variables to control the state of drawings, options in commands, and aspects of the user interface. The following pages list the system variables in alphabetical order. You can view and modify the values of system variables with SetVariable command.

Some system variables are included only for compatibility with other CAD programs, such as AutoCAD and progeCAD; they have no effect on iCADMac. Some system variables report values only; you cannot change their value. These are called “read-only” variables.

**(r/o)** indicates the system variable is read-only.

Some system variables are present in iCADMac for legacy reasons, and they provide compatibility with older software. These variables are not included in this appendix.

**A**

**AcadPrefix** (r/o) specifies paths to support files used by the program.

**AcisOutVer** controls ACIS version number; values are 15, 16, 17, 18, 20, 21, 30, 40, or 70.

**AcisSaveAsMode** controls how ACIS entities (3D solids, bodies, regions) are exploded for export.

**ActDb** (r/o) returns ID number of active drawing database.

**AFlags** controls default attribute display mode.

**AngBase** controls direction of zero degrees relative to CCS.

**AngDir** controls rotation of positive angles.

**AnnoAllVisible** toggles display of annotative objects at current scale.

**ApBox** toggles display of aperture box cursor.

**Aperture** controls entity snap aperture in pixels.

**ApState** reports state of ApBox system variable.

**Area** (r/o) reports area measured by Area command.

**ArLineDisp** determines whether edges of faces and bodies are displayed by the real-time render window (ARender command).

**AttDia** controls user interface for entering attributes.

**AttMode** controls display of attributes.

**AttReq** toggles whether attribute values are prompted for during insertion.

**AuditCtl** toggles creation of *.adt* audit log files.

**AUnits** specifies type of angular units.

**AUPrec** specifies number of decimal places displayed by angles.

**AutoNew** determines whether iCADMac starts new drawings based on a default template drawing, or asks the user to specify the template file.

**AutoNewName** specifies the name for new drawings; default is "noname\_*n*," where *n* is incremented.

**B**

**BackgroundPlot** toggles background printing.

**BackZ** controls location of back clipping plane offset from target plane.

**BigFontAlt** specifies the replacement Bigfont file to substitute for Bigfont used in the current drawing when it is not available.

**BindType** specifies how xref names are converted when bound or edited.

**BlipMode** toggles display of blip marks.

**C**

**CAnnoScale** names current annotative scale for current viewport.

**CDate** (r/o) specifies current date and time in format *yyyymmdd.hhmmssdd*.

**CeColor** specifies current color.

**CeLtScale** specifies current linetype scaling factor.

**CeLType** specifies current linetype.

**CeLWeight** specifies current linewidth in millimeters.

**ChamferA** specifies current value of first chamfer distance.

**ChamferB** specifies current value of second chamfer distance.

**ChamferC** specifies current value of chamfer length.

**ChamferD** specifies current value of chamfer angle.

**ChamMode** toggles chamfer input mode.

**CircleRad** specifies most-recent circle radius.

**CLayer** specifies name of current layer.

**CleanScreenState** (r/o) reports whether full screen (cleanscreen) mode is active.

**CliState** (r/o) reports whether command line palette is active.

**CmdActive** (r/o) reports type of command currently active.

**CmdEcho** toggles LISP command display.

**CmdLnText** specifies command prompt; default is :.

**CmdNames** (r/o) reports name of command currently active.

**CMLJust** specifies richline (multiline) justification mode.

**CMLScale** specifies scale of overall richline width.

**CMLStyle** specifies current richline style.

**Compass** toggles display of 3D compass.

**Coords** specifies coordinate display style.

**CPlotStyle** specifies current print style.

**CProfile** (r/o) specifies name of current profile (*.arg*) file.

**CTab** specifies name of current tab.

**CTableStyle** specifies name of current table style name.

**CursorMode** specifies look of cross hair cursor.

**CursorSize** specifies cursor size as a percentage of viewport size.

**CVPort** specifies current viewport number.

## D

**Date** (r/o) reports current date in Julian format, such as 2448860.54043252.

**DbIClkEdit** toggles editing by double-clicking objects.

**DBMod** (r/o) reports how drawing has been modified.

**DctMain** controls code for spelling dictionary.

**DebugMode** controls graphic card optimization; used for support reasons.

**DefaultLighting** toggles distant lighting.

**DefaultLightingType** toggles between new (1) and old (0) types of lighting.

**DefLPIStyle** reports default print style for layer 0.

**DefPIStyle** reports default print style for new objects.

**DefPIStyleTable** specifies the default print style for new entities and layers in drawing files that use the R14 (or earlier) DWG or DXF format.

**DelObj** toggles deletion of source objects used to create solids and regions.

**DiaStat** (r/o) reports whether user exited dialog box by clicking OK or Cancel.

**DispSilh** toggles silhouette display of 3D solids.

**Distance** (r/o) reports distance last measured by GetDistance command.

**DonutId** specifies inside diameter of donuts.

**DonutOd** specifies outside diameter of donuts.

**DragMode** controls drag mode.

**DrawOrderCtrl** specifies how draw order behaves.

**DwfFrame** toggles the frame around DWF overlays.

**DwfOsnap** toggles esnapping of DWF frames.

**DwgCodePage** (r/o) reports the same value as SysCodePage.

**DwgName** (r/o) reports current drawing's file name, such as "noname\_0.dwg."

**DwgPrefix** (r/o) reports drawing's drive and folder, such as "d:\my drawings\."

**DwgTitled** (r/o) reports whether the drawing has been given a file name.

**DxfPrec** specifies the default precision when saving drawings as DXF files.

**DynaSnap** specifies visibility of entity snap cues and tooltips.

**DynaSnapSize** specifies the size of entity snap icons displayed when moving the cursor over geometric features of objects.

### **Dimensions**

**DimADec** specifies angular dimension precision.

**DimAlt** toggles alternate units.

**DimAltD** specifies alternate unit decimal places.

**DimAltF** specifies alternate unit scale factor.

**DimAltRnd** specifies rounding factor of alternate units.

**DimAltTD** specifies decimal places of tolerance alternate units.

**DimAltTZ** specifies display of zeros in alternate tolerance units.

**DimAltU** specifies display of alternate units.

**DimAltZ** specifies display of zeros in alternate units.

**DimAPost** specifies prefix and suffix for alternate text.

**DimArcSym** specifies location of arc symbol.

**DimAssoc** specifies how dimensions are created.

**DimASz** specifies default arrowhead length.

**DimAtFit** specifies how text and arrows are fitted when there is insufficient space between extension lines.

**DimAUnit** specifies format of angular dimensions.

**DimAZin** specifies display of zeros in angular dimensions.

**DimBlk** specifies name of arrowhead block.

**DimBlk1** specifies name of first arrowhead's block.

**DimBlk2** specifies name of second arrowhead block.

**DimCen** specifies how center marks are drawn.

**DimClrD** specifies color of dimension lines.

**DimClrE** specifies color of extension lines and leaders.

**DimClrT** specifies color of dimension text.

**DimDec** specifies number of decimal places for primary tolerance.

**DimDLE** specifies length of dimension line extension.

**DimDLI** specifies increment distance of continued dimension lines.

**DimDSep** specifies decimal separator (a single character).

**DimExe** specifies distance of extension line above dimension line.

**DimExO** specifies extension line origin offset.

**DimFit** specifies how dimension text is fitted around extension lines.

**DimFrac** specifies fraction format.

**DimFXL** specifies default length of fixed extension lines.

**DimFxlOn** toggles fixed-length extension lines.

**DimGap** specifies gap between text and dimension line.

**DimJogAng** specifies default angle for jogged dimension lines.

**DimJust** specifies positioning of horizontal text.

**DimLdrBlk** specifies name of block used for leader arrowheads.

**DimLFac** specifies linear unit scale factor.

**DimLim** toggles display of dimension limits.

**DimLtEx1** specifies linetype for first extension line.

**DimLtEx2** specifies linetype for second extension line.

**DimLtype** specifies linetype name for dimension line.

**DimLUnit** specifies dimension units (except angular).

**DimLwD** controls dimension line's lineweight.

**DimLwE** specifies extension lines' lineweights.

**DimPost** specifies default prefix or suffix for dimension text.

**DimRnd** specifies rounding value for dimension distances.

**DimSAh** toggles whether separate arrowhead blocks are used.

**DimScale** specifies overall dimension scale factor.

**DimSD1** toggles display of first dimension line.

**DimSD2** toggles display of second dimension line.

**DimSE1** toggles display of first extension line.

**DimSE2** toggles display of second extension line.

**DimSho** toggles dimension updates while dragging.

**DimSOXD** toggles display of dimension lines outside of extension lines.

**DimStyle** (r/o) specifies name of current dimension style.

**DimTAD** specifies vertical position of text.

**DimTDec** specifies number of decimal places for primary tolerances.

**DimTFac** specifies scale factor for tolerance text height.

**DimTFill** toggles background fill color for dimension text.

**DimTFillClr** specifies background color for dimension text.

**DimTIH** toggles alignment of text placed inside extension lines.

**DimTIX** toggles placement of text inside extension lines.

**DimTM** specifies value of minus tolerance.

**DimTMove** specifies how dimension text is moved.

**DimTOFL** toggles placement of dimension lines.

**DimTOH** toggles text alignment when outside of extension lines.

**DimTol** toggles generation of dimension tolerances.

**DimTolJ** specifies vertical justification of tolerances.

**DimTP** specifies value of plus tolerance.

**DimTSz** specifies size of oblique tick strokes.

**DimTVP** controls vertical position of text when DimTad = 0.

**DimTxSty** specifies name of text style.

**DimTxt** specifies text height.

**DimTZin** specifies display of zeros in tolerances.

**DimUnit** specifies units used by dimensions.

**DimUPT** specifies user-positioned text.

**DimZIN** specifies display of zero in feet-inches units.

## E

**EdgeMode** toggles edge mode for Trim and Extend commands.

**Elevation** specifies current elevation relative to current CCS.

**ErrNo** (r/o) reports error numbers from Lisp and other APIs.

**ErState** (r/o) reports display of external references palette.

**Expert** specifies display of prompts.

**ExplMode** toggles whether Explode command explodes non-uniformly scaled blocks.

**ExtMax** (r/o) specifies upper-right coordinates of drawing extents.

**ExtMin** (r/o) specifies lower-left coordinates of drawing extents.

**ExtNames** specifies format of named objects.

## F

**FaceTRatio** specifies aspect ratio of facets on rounded 3D bodies.

**FaceTRres** specifies smoothness of shaded and hidden-line objects.

**FcTemplate** specifies default template drawing used by New command.

**FileDia** toggles user interface for file-access commands.

**FilletRad** specifies current fillet radius.

**FillMode** toggles fill of 2D solids, wide polylines, traces, fills, and hatches.

**FontAlt** specifies font used for missing fonts.

**FontMap** specifies name of font mapping file.

**FrontZ** (r/o) reports front clipping plane offset.

## G

**GridDisplay** determines grid display (sum of bitcodes).

**GridMode** toggles display of grid.

**GridUnit** specifies x, y spacing of grid.

**GripBlock** toggles display of grips in blocks.

**GripColor** specifies color of unselected grips.

**GripHot** specifies color of selected grips.

**GripHover** specifies grip color when cursor hovers.

**Grips** toggles display of grips.

**GripSize** specifies size of grips.

## H

**Handles** (r/o) reports whether entity handles (not grips) can be accessed by applications.

**Highlight** toggles entity selection highlighting.

**HPAng** specifies current hatch pattern angle.

**HpAssoc** toggles associativity of hatches.

**HpBound** specifies whether polylines or regions are created by Hatch and Boundary commands.

**HpDouble** toggles double hatching.

**HpDrawOrder** specifies draw order of hatches and fills.

**HpGapTol** specifies largest gap allowed in hatch boundaries.

**HpInherit** toggles how the PropertyPainter command copies hatch origin from source entity to destination entities.

**HpName** specifies default hatch name.

**HpObjWarning** specifies maximum number of hatch boundaries that can be selected before program warns users.

**HpOrigin** specifies default origin for hatch objects.

**HpOriginMode** specifies default hatch origin point.

**HpScale** specifies current hatch scale factor.

**HpSeparate** specifies whether the Hatch command creates a single hatch, or separate hatches when applied to multiple boundaries.

**HpSpace** specifies default spacing of user-defined hatches.

## I

**IniFilename** stores the name of an application initialization file used for development support.

**InsBase** specifies default insertion base point relative to current CCS for Insert and Xref commands.

**InsName** specifies default block name.

**InsUnits** specifies drawing units of blocks dragged into drawings.

**InsUnitsDefSource** specifies source drawing units value.

**InsUnitsDefTarget** specifies target drawing units.

**ISaveBak** toggles creation of *.bak* backup files.

**IsoLines** specifies number of contour lines on 3D solids.

## L

**Language** specifies the language used by iCADMac' user interface (2 = English).

**LastAngle** (r/o) reports end angle of last-drawn arc.

**LastAttachFolder** (r/o) reports the path to the folder from which the last externally referenced drawing file was attached.

**LastOpenFolder** (r/o) reports the path to the folder from which the last drawing file was opened.

**LastPoint** reports x, y, z coordinates of last-entered point.

**LastPrompt** (r/o) reports last string on command line.

**LightGlyphDisplay** toggles display of light glyph.

**LightingUnits** specifies type of lighting used.

**LightListState** (r/o) reports whether the Lighting palette is open.

**LimCheck** toggles drawing limits checking.

**LimMax** specifies upper right drawing limits.

**LimMin** specifies lower left drawing limits.

**Locale** (r/o) reports ISO language code used by the operating system.

**LocalRootPrefix** (r/o) specifies path to local root folder.

**LockUi** specifies position and size of toolbars and palettes.

**LogFileMode** toggles writing command prompts to *.log* file.

**LogFileName** (r/o) specifies name of *.log* logging file.

**LogFilePath** specifies path to *.log* logging file.

**LogInName** (r/o) reports user's login name.

**LTScale** specifies current linetype scale factor.

**LUnits** specifies current linear units display.

**LUPrec** specifies current decimal places (or inverse of smallest fraction) of linear units.

**LwDefault** specifies default lineweight, in millimeters.

**LwDisplay** toggles whether line weights are displayed.

**LwDispScale** specifies the factor by which line weights are scaled on screen.

**LwUnits** toggles units used for line weights.

## M

**MaxHatchDensity** specifies maximum density of hatching.

**MButtonPan** toggles behavior of wheel mouse.

**Measurement** toggles current drawing units.

**MenuEcho** specifies menu and prompt echoing.

**MirrText** toggles text handling by Mirror command.

**ModeMacro** displays text strings on the status bar.

**MStopOnErr** toggles whether macros halt on error.

**MyDocumentsPrefix** (r/o) reports path to *"/Users/username/Documents"* folder of currently logged-in user.

## O

**OffsetDist** specifies current offset distance.

**OffsetGapType** specifies how polylines reconnect when segments are offset.

**OpenFilterIndex** presets the default drawing file format for the Open command by setting the index number for the Files of Type droplist of the Open dialog box.

**OpmState** toggles whether Properties palette is active.

**OrthoMode** toggles orthographic mode.

**OsMode** specifies current entity snap modes.

**OsOptions** determines when entities with negative z values are entity-snapped.

## P

**PageSetupManager** determines whether the Page Layout dialog box is displayed with the creation of new sheets.

**PanScale** specifies scale factor when panning left, right, up, and down.

**PaperUpdate** toggles how program plots layouts when paper size differs from plotter's default size.

**PDMode** specifies point display style.

**PDSize** specifies point display size.

**PEditAccept** toggles display of EditPolyline command's 'entity selected is not a polyline' prompt.

**PEllipse** toggles entity used for creating ellipses.

**Perimeter** (r/o) reports perimeter calculated by last GetArea command.

**PickAdd** toggles meaning of Shift key on selection sets.

**PickAuto** toggles selection set mode.

**PickBox** specifies size of selection pickbox.

**PickDrag** toggles selection window mode.

**PickFirst** toggles command-selection mode.

**PickStyle** specifies how groups and associative hatches are selected.

**Platform** (r/o) reports name of operating system.

**PLineGen** toggles polyline linetype generation.

**PLineType** specifies automatic conversion and creation of 2D polylines by Polyline command.

**PLineWid** specifies current polyline width.

**PolarAddAng** specifies user-defined polar angles.

**PolarAng** specifies increment of polar angle.

**PolarDist** specifies polar snap increment when SnapStyl is set to 1 (isometric).

**PolarMode** specifies polar and entity snap tracking.

**PolySides** specifies default number of polygon sides.

**PrevCmd** (r/o) reports the name of the command currently executing (meant for use by application development).

**PreviewEffect** specifies visual effect when previewing selected entities.

**PreviewFilter** specifies exclusion of objects from selection previewing.

**Product** (r/o) reports name of software.

**Program** (r/o) reports name of software's executable file.

**ProjMode** specifies projection mode for Trim and Extend commands.

**PsLtScale** toggles paper space linetype scaling.

**PStyleMode** toggles plot color matching mode of drawing.

**PStylePolicy** (r/o) reports whether entity color is associated with its plot style.

**PUcsBase** (r/o) reports name of CCS defining origin and orientation of orthographic CCS settings; in paper space only.

## Q

**QTextMode** toggles quick text mode.

**R**

**RasterPreview** (r/o) toggles creation of bitmap preview image.

**RecoveryMode** specifies recording of drawing recovery information after program failure.

**RefEditName** specifies reference file name when in reference-editing mode.

**RegenMode** toggles regeneration mode.

**S**

**SaveFile** (r/o) reports automatic save file name.

**SaveFilePath** specifies path for automatic save files.

**SaveFilterIndex** presets the default drawing file format for the SaveAs command by setting its index number for the Files of Type droplist of the SaveFile dialog box.

**SaveName** (r/o) reports drawing's save-as file name.

**SaveRoundtrip** determines whether entities unknown to R14 DWG format are stored in R14 DWG drawing files.

**SaveTime** specifies automatic save interval, in minutes.

**ScreenSize** (r/o) reports current viewport size, in pixels.

**SelectBox** specifies the size of the square pickbox cursor when selecting entities; same as the PickBox variable.

**SelectionPreview** specifies selection preview.

**SerNumber** (r/o) reports the serial number of the user's iCADMac licence of iCADMac.

**ShapeAlt** specifies the SHX shape file to substitute for references to unavailable shapes found in the current drawing.

**ShortcutMenu** specifies display of shortcut menus.

**ShowHyperlinkCursor** toggles the display of the hyperlink cursor and tooltip, which appear when the cursor moves over entities with hyperlinks.

**ShowLayerUsage** toggles layer-usage icons in Layers dialog box.

**ShpName** specifies default shape name.

**SketchInc** specifies Sketch command's recording increment.

**SkPoly** toggles sketch line mode.

**SnapAng** specifies rotation angle for snap and grid.

**SnapBase** specifies current origin for snap and grid.

**SnapIsoPair** specifies current isometric drawing plane.

**SnapMode** toggles snap mode.

**SnapStyl** toggles snap style.

**SnapType** toggles snap for current viewport.

**SnapUnit** specifies x, y spacing of snap distances.

**SolidCheck** toggles solids validation.

**SplFrame** toggles display of frames on polylines and meshes.

**SplineSegs** specifies number of line segments that define splined polylines.

**SplineType** specifies type of spline curve.

**SurfTab1** specifies density of m-direction surfaces and meshes.

**SurfTab2** specifies density of n-direction surfaces and meshes.

**SurfType** specifies smoothing of surfaces by EditPolyline command.

**SurfU** specifies surface density in m-direction.

**SurfV** specifies surface density in n-direction.

**SysCodePage** (r/o) reports system code page; set by operating system.

## T

**TDCreate** (r/o) reports date and time that drawing was created.

**TDInDwg** (r/o) reports duration since drawing was loaded.

**TDuCreate** (r/o) reports universal date and time when drawing was created.

**TDUpdate** (r/o) reports date and time of last update.

**TDUsrTimer** (r/o) reports decimal time elapsed by user-timer.

**TDuUpdate** (r/o) reports universal date and time of last save.

**TextSize** specifies default height of text.

**TextStyle** specifies default name of text style.

**Thickness** specifies default entity thickness.

**TileMode** toggles view mode.

**ToolTips** toggles display of tooltips.

**TraceWid** specifies current width of traces.

**TrackPath** controls display of polar and entity snap tracking alignment paths.

**TrimMode** toggles use of trims during Chamfer and Fillet commands.

## U

**UcsAxisAng** specifies default angle for rotating CCS (custom coordinate system) around an axis.

**UcsBase** specifies name of CCS that defines origin and orientation of orthographic CCS settings.

**UcsFollow** toggles view displayed with new CCSs.

**UcsIcon** specifies display of CCS icon.

**UcsName** (r/o) reports name of current CCS view.

**UcsOrg** (r/o) reports origin of current CCS.

**UcsOrtho** specifies whether related orthographic CCS settings are restored automatically.

**UcsView** toggles whether current CCS is saved with a named view.

**UcsVp** toggles whether CCS in active viewports remains fixed (locked) or changes (unlocked) to match CCS of current viewport.

**UcsXDir** (r/o) reports x-direction of current CCS.

**UcsYDir** (r/o) reports y-direction of current CCS.

**UndoCtl** (r/o) reports settings that control Undo actions.

**UndoMarks** (r/o) reports number of undo marks.

**UndoZoomPanGrouping** determines whether zoom and pan operations are grouped as a single operation.

**UnitMode** toggles type of units display.

**UserI1** thru **UserI5** specify five user-definable integer variables.

**UserR1** thru **UserR5** specify five user-definable real variables.

**UserS1** thru **UserS5** specify five user-definable string variables; these values are not saved.

## V

**ViewCtr** (r/o) reports x, y, z coordinates of center of current view.

**ViewDir** (r/o) reports current view direction relative to CCS.

**ViewSize** (r/o) reports height of current view in drawing units.

**ViewTwist** (r/o) reports twist angle of current view.

**VisRetain** specifies xref drawing's layer settings.

## W

**WorldUcs** (r/o) toggles matching of world coordinates with CCS.

**WorldView** toggles view during the RollView and ViewDirection commands.

**WriteStat** (r/o) reports whether *.dwg* file is read-only.

## X

**XClipFrame** toggles visibility of external reference clipping boundaries.

**XEdit** specifies whether the current drawing can be edited in-place when being referenced by another drawing.

**XFadeCtl** specifies faded display of entities not being edited in-place.

**XrefType** specifies whether externally-referenced drawings are attached as Attachments or Overlays.

## Z

**ZInScale** specifies zoom factor for ZoomIn command.

**ZoomFactor** specifies zoom level via mouse wheel.

**ZoomWheel** switches zoom direction when mouse wheel is rotated forward.

**ZOutScale** specifies zoom factor for ZoomOut command.



## Appendix C

# Command Aliases

### In This Appendix

- Aliases sorted by command name
- Aliases sorted by alias name

**iCADMac** allows you to enter abbreviations for command names, such as **l** for the Line command. The abbreviations are known as “aliases.” Aliases allow you to enter commands quickly, and allow iCADMac to work more like AutoCAD and other CAD systems. For example, you can enter AutoCAD’s **cui** command and that will execute iCADMac’s Customize command.

On the following pages, equivalent AutoCAD commands are shown in **boldface**.

Aliases are stored in the *alias.xml* file. iCADMac can read alias definition files from AutoCAD (.*pgp*) and progeCAD (.*ica*). You can customize aliases through the User Preferences section of the DraftingOptions command’s dialog box.

This appendix lists aliases in two ways: sorted by iCADMac command names and by alias names.

## Aliases Sorted by iCADMac Commands

### iCADMac Command Aliases

#### A

activatelay	actlay, <b>laymcur</b>
<b>align</b>	al
<b>align3d</b>	al3
angledimension	aldim, angledim, dan, dimang, <b>dimangular</b>
<b>arc</b>	a
arclengthdimension	dar, <b>dimarc</b>
areaboundary	ab, -ab, <b>boundary</b> , bo, bpoly
-areaboundary	-bo, <b>-boundary</b>
arender	<b>render</b>
attachdrawing	xa, <b>xattach</b>
attachimage	ating, attachimg, iat, <b>imageattach</b>
attachlink	atlnk, <b>attachurl</b>
autorebuild	<b>regenauto</b>

#### B

baselinedimension	basedim, bldim, dba, dimbase, <b>dimbaseline</b>
blockattributeoutput	attout, battout
boxtext	<b>qtext</b> , btext

#### C

ccs	<b>ucs</b>
centermark	cm, dce, <b>dimcenter</b>
<b>chamfer</b>	cha
check	<b>audit</b> , chk
<b>circle</b>	c
clean	cl, pu, <b>purge</b>
-clean	-pu, <b>-purge</b>
clipimage	iclip, <b>imageclip</b>
clipboardcopy	<b>copyclip</b>
clipreference	clip, xc, <b>xclip</b>
clipviewport	<b>vpclip</b>
commandhistory	cmdhist, <b>textscr</b>
commandwindow	cmdwin, <b>commandline</b>
continuedimension	cdim, dco, dimcont, <b>dimcontinue</b>
convertprintstyles	<b>convertpstyles</b>
<b>copy</b>	co, cp
copy@	<b>copybase</b> , cp@

### iCADMac Command Aliases

copyhistory	<b>copyhist</b>
csicon	<b>ucsicon</b>
csstyle	css, uc, <b>ucsman</b>
customize	<b>cui</b> , cust
cut	<b>cutclip</b>
<b>cylinder</b>	cyl

#### D

delete	del, e, <b>erase</b>
deletelay	dellay, <b>laydel</b>
detachimage	<b>imagedetach</b> , dimage
detachlink	<b>detachurl</b>
diameterdimension	ddi, dimdia, <b>dimdiameter</b>
dimensionstyle	d, dimsty, <b>dimstyle</b> , dst
-dimensionstyle	<b>-dimstyle</b>
directionallight	<b>distantlight</b>
displayblockattributes	<b>attdisp</b> , dsatt, dispblkatt
displayimageframe	<b>imageframe</b> , iframe
displayorder	dr, <b>draworder</b> , do
displayquality	<b>viewres</b>
draftingoptions	dop, ds, <b>dsettings</b> , se
drawingbounds	bounds, <b>limits</b>

#### E

edgemesh	<b>edgesurf</b>
editannotation	<b>ddedit</b> , ed, edanno, edittext, textedit
editblockattribute	ate, <b>attedit</b> , attxedit, ddatte, edblkatt
-editblockattribute	-ate, atte, <b>-attedit</b> , -edblkatt, qattedit
editblockattributedefinition	attdedit, edblkattdef
editdimension	ded, dimed, <b>dimedit</b> , editdim
editdimensiontext	dimted, <b>dimtedit</b> , editdimtxt
edithatch	<b>hatchedit</b> , he
-edithatch	<b>-hatchedit</b>
editimage	-im, <b>-image</b>
editlength	edlen, <b>lengthen</b> , len
editnote	edn, <b>mtedit</b>
editpolyline	edpl, edpline, pe, <b>pedit</b> , polyedit

**iCADMac Command Aliases**

editrichline	editrline, edrl, <b>mledit</b>
-editrichline	<b>-mledit</b>
edittable	edtbl, <b>tabledit</b> , tableedit
edittolerance	edittol, edtol, toledit
editvertex	editvtx, vtxedit
<b>ellipse</b>	el
-embeddrawing	<b>-xbind</b> , -embeddwg
enterpoint	entpt
entitygrips	egrips, gr
entitygroup	g, <b>group</b> , egroup
-entitygroup	-g, -group
entitysnap	es, esnap, os, <b>osnap</b>
-entitysnap	-es, -esnap, -os, <b>-osnap</b>
exit	<b>quit</b>
<b>explode</b>	x
<b>export</b>	exp
exportbmp	<b>bmpout</b>
exportdrawing	dwgout, w, <b>wblock</b>
-exportdrawing	-exportdwg, -w, <b>-wblock</b>
exportdxf	<b>dxfout</b>
exportjpg	<b>jpgout</b>
<b>exportpdf</b>	pdfout
exportpng	pngout
exportsat	<b>acisout</b> , satout
exportsld	<b>mslide</b> , sldout
exportsvg	svgout
<b>extend</b>	ex
extractblockattribute	<b>attext</b>
-extractblockattribute	<b>-attext</b>
<b>extrude</b>	ext

**F**

face	<b>3dface</b> , 3f
fillarea	<b>gradient</b>
<b>fillet</b>	f
flip	fl
freezelayer	<b>layfrz</b> , frzlay
fullscreen	<b>cleanscreenon</b> , fscreen

**G**

getarea	aa, <b>area</b> , ga
getdistance	di, <b>dist</b> , gd, getdist
getproperties	getprops, gp, <b>list</b>
getstatus	gs, <b>status</b>
gettime	gt, <b>time</b>
getxy	gxy, <b>id</b>
gravity	<b>aperture</b>

**iCADMac Command Aliases****H**

<b>hatch</b>	bh, bhatch, h
<b>-hatch</b>	-bhatch, -h, qhatch
hidecommandhistory	<b>graphscr</b> , hidecmdhist
hidecommandwindow	<b>commandlinehide</b> , hidecmdwin
hidefullscreen	<b>cleanscreenoff</b> , hfscreen
hidelay	hidelay, <b>layoff</b>
hidelightlist	<b>lightlistclose</b>
hideproperties	hideprops, prclose, <b>propertiesclose</b>
hidereferences	hiderefs, xrefclose
hideview	hi, <b>hide</b> , hview, qhide
horizontaldimension	dimhor, hdim

**I**

<b>imagequality</b>	iquality
importdwf	<b>dwfattach</b> , dwfin
importdxf	<b>dxfin</b>
importsat	<b>acisin</b> , satin
infinetiline	il, iline, xl, <b>xline</b>
insertblock	i, <b>insert</b>
-insertblock	-i, <b>-insert</b> , qinsert
insertblockn	insblock, <b>minsert</b>
insertshape	inshape, insshape, <b>shape</b>
<b>interfere</b>	inf
<b>intersect</b>	in
isolatelayer	isolay, <b>layiso</b>
isometricgrid	isogrid, <b>isoplane</b>

**J**

joggeddimension	<b>dimjogged</b> , djo, jog, jogdim
-----------------	-------------------------------------

**L**

<b>layer</b>	la
<b>-layer</b>	-la, qlayer
<b>leader</b>	lead
<b>line</b>	l
lineardimension	dimlin, <b>dimlinear</b> , dli, ldim
linecolor	col, colour, <b>color</b> , lc, lcolor
-linecolor	<b>-color</b> , -lcolor
linescale	lscale, lts, <b>ltscale</b>
linestyle	<b>linetype</b> , lstyle, lt, ltype, qlinetype
-linestyle	loadltype, <b>-linetype</b> , -lt, -ltype
lineweight	lw, <b>lweight</b>
-lineweight	<b>-lweight</b>

**iCADMac Command Aliases**

loadapp	ap, <b>appload</b>
loadmenu	lmenu, <b>menu</b>
loadscript	lscript, scr, <b>script</b>
loadshape	<b>load</b>
locklayer	<b>laylck</b> , llay, lcklay
<b>M</b>	
makeblock	b, <b>block</b> , mbck, partdef, bmake
-makeblock	-b, <b>-block</b>
makeblockattribute	att, <b>attdef</b> , mblkatt
-makeblockattribute	-att, <b>-attdef</b> , qattdef
markdivisions	div, <b>divide</b> , mdiv
marklengths	me, <b>measure</b> , mlen
mask	<b>wipeout</b>
matchlayer	<b>laymch</b> , mlay
mesh	<b>3dmesh</b>
<b>mirror</b>	mi
<b>mirror3d</b>	3dmirror, mi3d
modelmode	mm, ms, <b>mspace</b>
modify	-ch, <b>change</b> , mod, qpropedit
modifyproperties	<b>chprop</b> , modprops
<b>move</b>	m
movedimensiontext	dimtmove, movedimtxt
<b>N</b>	
nontargetlight	<b>freespot</b>
note	mt, <b>mtext</b> , n, t
-note	<b>-mtext</b> , qmtext
<b>O</b>	
obliquedimension	dimobl, obliquedim, odim
<b>offset</b>	o
openreference	<b>xopen</b>
openwebpage	<b>browser</b> , oweb
<b>options</b>	op, config
ordinatedimension	dimord, <b>dimordinate</b> , dor, orddim, ordinatedim
overridedimensionstyle	dimover, <b>dimoverride</b> , dov, overdims
oscalc	<b>cal</b> , wcalc
<b>P</b>	
pagelayout	<b>pagesetup</b>
<b>pan</b>	p, dpan
<b>-pan</b>	-p
pandynamic	pdy, <b>rtpan</b>
paralleldimension	dal, dimali, <b>dimaligned</b> , pdim, paralleldim
paste	<b>pasteclip</b>

**iCADMac Command Aliases**

pattern	ar, <b>array</b> , pat
-pattern	-ar, qarray
pattern3d	3a, <b>3darray</b> , pat3d
pausescript	<b>delay</b>
planview	<b>plan</b> , pview
<b>point</b>	po, pt
pointformat	<b>ddptype</b>
<b>polygon</b>	pol, pgon
polyline	pl, <b>pline</b>
polyline3d	<b>3dpoly</b> , 3p, pl3, pline3d
<b>preview</b>	pre
print	<b>plot</b>
-print	<b>-plot</b>
printstamp	pstamp
printstyle	<b>plotstyle</b> , pstyle
-printstyle	-plotstyle
<b>properties</b>	ch, mo, pr, props
propertypainter	ma, <b>matchprop</b> , paint
<b>Q</b>	
<b>quit</b>	exit
<b>R</b>	
radiusdimension	dimrad, <b>dimradius</b> , dra, rdim, radiusdim
rebuild	re, <b>regen</b>
rebuildall	rea, <b>regenall</b>
rebulddimension	dimupd, rebulddim, dimupdate
rectangle	rec, rect, <b>rectang</b>
redon	<b>mredo</b>
references	er, externalreferences, refs, xlink, xr, <b>xref</b> , image, im
-references	-xr, <b>-xref</b> , qxref, qxlink
refresh	r, <b>redraw</b>
refreshall	ra, <b>redrawall</b>
<b>region</b>	reg
relatedimension	<b>dimreassociate</b> , dre, redim
<b>rename</b>	ren, rn
<b>-rename</b>	-ren
repeat	<b>multiple</b>
replacedimensiontext	dimtnew, replacedimtxt
replaceneu	rnew
replaceopen	ropen
resetdimensiontext	dimthome, resetdimtext
resumescript	<b>resume</b>
<b>revolve</b>	rev
revolvedmesh	<b>revsurf</b>
richline	ml, <b>mline</b> , rl
richlinestyle	<b>mlstyle</b> , rls, rlstyle, rlinestyle

**iCADMac Command Aliases**

ring	do, <b>donut</b>
rollview	3do, <b>3dorbit</b> , orbit
<b>rotate</b>	ro
<b>rotate3d</b>	ro3d
rotatedimensiontext	dimtrot, rodimtext
ruledmesh	revmesh, <b>rulesurf</b>

**S**

save	<b>qsave</b>
<b>scale</b>	sc
scriptn	<b>rscript</b>
<b>section</b>	sec
selectionfilter	fi, <b>filter</b> , sf
setvariable	set, <b>setvar</b>
shadeview	sha, shademode, <b>-shademode</b> , sview
sheet	<b>layout</b> , lo
-sheet	<b>-layout</b>
sheetmode	ps, <b>pspace</b> , sm
showlayers	<b>layon</b>
simplenote	dtext, <b>text</b> , snote, dt
simplepolygon	spoly, ngon
<b>slice</b>	sl
smartbmp	qbmpout
smartcalculator	calc, smartcalc, quickcalc, qc
smartleader	le, <b>qlleader</b>
smartnew	<b>qnew</b>
smartopen	qopen
smartselect	ss
<b>snap</b>	sn
<b>solid</b>	so
spellcheck	<b>spell</b>
<b>spline</b>	spl
split	br, sp, <b>break</b>
<b>stretch</b>	s
<b>subtract</b>	su
switcharea	tlmode, switch

**T**

<b>table</b>	tb
<b>tablestyle</b>	ts
tabulatedmesh	tabmesh, <b>tabsurf</b>
targetpointlight	<b>targetpoint</b>
textstyle	st, <b>style</b> , txs
-textstyle	<b>-style</b>
thawlayers	<b>laythw</b> , thawlay
<b>thickness</b>	th
<b>tilemode</b>	ti, tm
toactivelayer	<b>laycur</b>
<b>tolerance</b>	tol

**iCADMac Command Aliases**

toolbars	<b>tbconfig</b>
<b>torus</b>	tor
<b>trim</b>	tr

**U**

undeleter	<b>oops</b> , undel
undolayer	<b>layerp</b>
undon	<b>undo</b>
<b>union</b>	uni
unisolatelayer	<b>layuniso</b>
unitsystem	un, <b>units</b>
-unitsystem	-un, <b>-units</b>
unitsystem	unitsys, us
unlocklayer	<b>layulk</b> , unla
unrelatedimension	dda, <b>dimdisassociate</b> , undim, unrelatedim

**V**

verticaldimension	dimver, vdim
viewdirection	-vp, <b>vpoin</b> t, vdirect
viewport	mv, <b>mview</b>
-viewport	qviewport, <b>-vports</b>
viewportlayer	<b>vplayer</b>
views	v, <b>view</b>
-views	-v, <b>-view</b> , qview
viewslide	<b>vslide</b>
viewtiles	<b>viewports</b>

**W**

<b>wedge</b>	we
weld	j, <b>join</b>
windows	<b>syswindows</b>

**Z**

<b>zoom</b>	z
zoomback	zb
zoomdynamic	dzoom, zoomdyn, zd
zoomfactor	zfa
zoomfit	zf
zoomin	zi
zoomout	zo
zoomwindow	zoomarea, zw
zplane	elev

**#**

3pointangledimension	dima3p, dim3ap, 3padim
4pointangledimension	dima4p, dim4ap, 4padim

## Aliases Sorted by Alias Names

Alias Name iCADMac Command

### A

a	arc
aa	getarea
ab	areaboundary
-ab	areaboundary
acisin	importsat
acisout	exportsat
actlay	activatelayer
al	align
al3	align3d
aldim	angledimensiond
angledim	angledimension
ap	loadapp
aperture	gravity
appload	loadapp
ar	pattern
-ar	-pattern
area	getarea
array	pattern
atdwg	attachdrawing
ate	editblockattribute
-ate	-editblockattribute
atimg	attachimage
atlnk	attachlink
att	makeblockattribute
-att	-makeblockattribute
attachdwg	attachdrawing
attachimg	attachimage
attachurl	attachlink
attdedit	editblockattributedefinition
attdef	makeblockattribute
-attdef	-makeblockattribute
attdisp	displayblockattributes
atte	-editblockattribute
attedit	editblockattribute
-attedit	-editblockattribute
attext	extractblockattribute
-attext	extractblockattribute
attout	blockattributeoutput
attxedit	editblockattribute
audit	check

### B

b	makeblock
-b	-makeblock

Alias Name iCADMac Command

basedim	baselinedimension
battout	blockattributeoutput
bh	hatch
bhatch	hatch
-bhatch	-hatch
bldim	baselinedimension
block	makeblock
-block	-makeblock
bmake	makeblock
bmpout	exportbmp
bo	areaboundary
-bo	-areaboundary
boundary	areaboundary
-boundary	-areaboundary
bounds	drawingbounds
bpoly	areaboundary
br	split
break	split
browser	openwebpage
btext	boxtext

### C

c	circle
cal	windowscal
calc	smartcalculator
cdim	continuedimension
ch	properties
-ch	modify
cha	chamfer
change	modify
chk	check
chprop	modifyproperties
cl	clean
cleanscreenoff	hidefullscreen
cleanscreenon	fullscreen
clip	clipreference
cm	centermark
cmdhist	commandhistory
cmdwin	commandwindow
co	copy
col	linecolor
color	linecolor
-color	-linecolor
colour	linecolor
commandline	commandwindow
commandlinehide	hidecommandwindow

**iCADMac Command Aliases**

config	options
convertpstyles	convertprintstyles
copybase	copy@
copyclip	clipboardcopy
copyhist	copyhistory
cp	copy
cp@	copy@
css	csstyle
cui	customize
cust	customize
cutclip	cut
cyl	cylinder

**D**

del	delete
delay	pausescript
dellay	deletelayer
detachurl	detachlink
di	getdistance
dispbkatt	displayblockattributes
dist	getdistance
distantlight	directionallight
div	markdivisions
divide	markdivisions
do	draworder
donut	ring
dop	draftingoptions
dpan	pan
dr	displayorder
draworder	displayorder
ds	draftingoptions
dsatt	displayblockattributes
dsettings	draftingoptions
dt	simplenote
dtext	simplenote
dwfattach	importdwf
dwgdout	exportdrawing
dxfin	importdxfin
dxfout	exportdxfin
dzoom	zoomdynamic

**Dimensions**

d	dimensionstyle
dal	paralleldimension
dan	angledimension
dar	arclengthdimension
dba	baselinedimension
dce	centermark
dco	continuedimension

**iCADMac Command Aliases**

dda	unrelatedimension
ddedit	editannotation
ddi	diameterdimension
ddptype	pointformat
ded	editdimension
dim3ap	3pointangledimension
dim4ap	4pointangledimension
dima3p	3pointangledimension
dima4p	4pointangledimension
dimage	detachimage
dimali	paralleldimension
dimaligned	paralleldimension
dimang	angledimension
dimangular	angledimension
dimarc	arclengthdimension
dimbase	baselinedimension
dimbaseline	baselinedimension
dimcenter	centermark
dimcont	continuedimension
dimcontinue	continuedimension
dimdia	diameterdimension
dimdiameter	diameterdimension
dimdisassociate	unrelatedimension
dimed	editdimension
dimedit	editdimension
dimhor	horizontaldimension
dimjogged	joggeddimension
dimlin	lineardimension
dimlinear	lineardimension
dimobl	obliquedimension
dimord	ordinatedimension
dimordinate	ordinatedimension
dimover	overridedimensionstyle
dimoverride	overridedimensionstyle
dimrad	radiusdimension
dimradius	radiusdimension
dimreassociate	relatedimension
dimsty	dimensionstyle
dimstyle	dimensionstyle
-dimstyle	-dimensionstyle
dimted	editdimensiontext
dimtedit	editdimensiontext
dimthome	resetdimensiontext
dimtmove	movedimensiontext
dimtnew	replacedimensiontext
dimtrot	rotatedimensiontext
dimupd	rebuilddimension
dimupdate	rebuilddimension
dimver	verticaldimension

**iCADMac Command Aliases**

djo	joggeddimension
dli	lineardimension
dor	ordinatedimension
dov	overridimensionstyle
dra	radiusdimension
dre	relatedimension
dst	dimensionstyle

**E**

e	delete
ed	editannotation
edanno	editannotation
edblattdef	editblockattributedefinition
edblkatt	editblockattribute
-edblkatt	-editblockattribute
edblkattdef	editblockattributedefinition
edgesurf	edgemesh
editdim	editdimension
editdimtxt	editdimensiontext
editrline	editrichline
edittol	edittolerance
editvtx	editvertex
edlen	editlength
edn	editnote
edpl	editpolyline
edpline	editpolyline
edrl	editrichline
edtbl	edittable
edtol	edittolerance
egrips	entitygrips
egroup	entitygroup
el	ellipse
elev	zplane
-embeddwg	-embeddrawing
entpt	enterpoint
er	references
erase	delete
es	entitysnap
-es	-entitysnap
esnap	entitysnap
-ensap	-entitysnap
ex	extend
exp	export
exportdwg	exportdrawing
-exportdwg	-exportdrawing
ext	extrude
externalreferences	references

**iCADMac Command Aliases****F**

f	fillet
fi	selectionfilter
filter	selectionfilter
fl	flip
freespot	nontargetlight
frzlay	freezelay
fscreen	fullscreen

**G**

g	entitygroup
-g	-entitygroup
ga	getarea
gd	getdistance
getdist	getdistance
getprops	getproperties
gp	getproperties
gr	entitygrips
gradient	fillarea
graphscr	hidecommandhistory
group	entitygroup
-group	-entitygroup
gs	getstatus
gt	gettime
gxy	getxy

**H**

h	hatch
-h	-hatch
hatchedit	edithatch
-hatchedit	-edithatch
hdim	horizontaldimension
he	edithatch
hfscreen	hidefullscreen
hi	hideview
hide	hideview
hidecmdhist	hidecommandhistory
hidecmdwin	hidecommandwindow
hidelay	hidelay
hideprops	hideproperties
hiderefs	hidereferences
hview	hideview

**I**

i	insertblock
-i	-insertblock
iat	attachimage
iclip	clipimage
id	getxy

**iCADMac Command Aliases**

iframe	displayimageframe
il	infinetiline
iline	infinetiline
im	references
-im	editimage
image	references
-image	editimage
imageattach	attachimage
imageclip	clipimage
imagedetach	detachimage
imageframe	displayimageframe
in	intersect
inf	interfere
insblock	insertblockn
insert	insertblock
-insert	-insertblock
inshape	insertshape
insshape	insertshape
iquality	imagequality
isogrid	isometricgrid
isolay	isolatelay
isoplane	isometricgrid

**J**

j	weld
jog	joggeddimension
jogdim	joggeddimension
join	weld
jpgout	exportjpg

**L**

l	line
la	layer
-la	-layer
laycur	toactivelayer
laydel	deletelayer
layerp	undolayer
layfrz	freezelay
layiso	isolatelay
laylck	locklayer
laymch	matchlayer
laymcur	activatelay
layoff	hidelay
layon	showlayers
layout	sheet
-layout	-sheet
laythw	thawlayers
layulk	unlocklayer
layuniso	unisolatelay

**iCADMac Command Aliases**

lc	linecolor
lcklay	locklayer
lcolor	linecolor
-lcolor	-linecolor
ldim	lineardimension
le	smartleader
lead	leader
len	editlength
lengthen	editlength
lightlistclose	hidelightlist
limits	drawingbounds
linetype	linestyle
-linetype	-linestyle
list	getproperties
lly	locklayer
lmenu	loadmenu
lo	sheet
load	loadshape
loadltype	-linestyle
lscale	linescale
lscript	loadscript
lstyle	linestyle
lt	linestyle
-lt	-linestyle
lts	linescale
ltscale	linescale
ltype	linestyle
-ltype	-linestyle
lw	lineweight
lweight	lineweight
-lweight	-lineweight

**M**

m	move
ma	propertypainter
matchprop	propertypainter
mbck	makeblock
mbkatt	makeblockattribute
mdiv	markdivisions
me	marklengths
measure	marklengths
menu	loadmenu
mi	mirror
mi3d	mirror3d
minsert	insertblockn
ml	richline
mly	matchlayer
mledit	editrichline
-mledit	-editrichline

**iCADMac Command Aliases**

mle	marklengths
mline	richline
mlstyle	richlinestyle
mm	modelmode
mo	properties
mod	modify
modprops	modifyproperties
movedimtxt	movedimensiontext
mredo	redon
ms	modelmode
mslide	exportsld
mSPACE	modelmode
mt	note
mtedit	editnote
mtxt	note
-mtxt	-note
multiple	repeat
mv	viewport
mview	viewport

**N**

n	note
ngon	simplepolygon

**O**

o	offset
obliquedim	obliquedimension
odim	obliquedimension
oops	undelete
op	options
orbit	rollview
orddim	ordinatedimension
ordinatedim	ordinatedimension
os	entitysnap
-os	-entitysnap
osnap	entitysnap
-osnap	-entitysnap
overdims	overridimensionstyle
oweb	openwebpage

**P**

p	pan
-p	-pan
pagesetup	pagelayout
paint	propertypainter
paralleldim	paralleldimension
partdef	makeblock
pasteclip	paste
pat	pattern

**iCADMac Command Aliases**

pat3d	pattern3d
pdfout	exportpdf
pdim	paralleldimension
pdY	pandynamic
pe	editpolyline
pedit	editpolyline
pgon	polygon
pl	polyline
pl3	polyline3d
plan	planview
pline	polyline
pline3d	polyline3d
plot	print
-plot	-print
plotstyle	printstyle
-plotstyle	-printstyle
pngout	exportpng
po	point
pol	polygon
polyedit	editpolyline
pr	properties
prclose	hideproperties
pre	preview
propertiesclose	hideproperties
props	properties
ps	sheetmode
pspace	sheetmode
pstamp	printstamp
pstyle	printstyle
pt	point
pu	clean
-pu	-clean
purge	clean
-purge	-clean
pview	planview

**Q**

qarray	-pattern
qattdef	-makeblockattribute
qattedit	-editblockattribute
qbmpout	smartbmp
qc	smartcalculator
qhatch	-hatch
qhide	hideview
qinsert	-insertblock
qlayer	-layer
qleader	smartleader
qlinetype	-linestyle
qmtxt	-note

**iCADMac Command Aliases**

qnew	smartnew
qopen	smartopen
qpropedit	modify
qsave	save
qtext	boxtext
quickcalc	smartcalculator
qview	-views
qviewport	-viewport
qxlink	-references
qxref	-references

**R**

r	refresh
ra	refreshall
radiusdim	radiusdimension
rdim	radiusdimension
re	rebuild
rea	rebuildall
rebuilddim	rebuilddimension
rec	rectangle
rect	rectangle
rectang	rectangle
redim	relatedimension
redraw	refresh
redrawall	refreshall
refs	references
reg	region
regen	rebuild
regenall	rebuildall
regenauto	autorebuild
ren	rename
-ren	-rename
render	arender
replacedimtxt	replacedimensiontext
resetdimtext	resetdimensiontext
resume	resumescript
rev	revolve
revmesh	ruledmesh
revsurf	revolvedmesh
rl	richline
rlinestyle	richlinestyle
rls	richlinestyle
rlstyle	richlinestyle
rn	rename
rnew	replaceneu
ro	rotate
ro3d	rotate3d
rodimtext	rotatedimensiontext
ropen	replaceopen

**iCADMac Command Aliases**

rscript	scriptn
rtpan	pandynamic
rulesurf	ruledmesh

**S**

s	stretch
satin	importsat
satout	exportsat
sc	scale
scr	loadscript
script	loadscript
se	draftingoptions
sec	section
set	setvariable
setvar	setvariable
sf	selectionfilter
sha	shadeview
shademode	shadeview
-shademode	shadeview
shape	insertshape
sl	slice
sldout	exportsld
sm	sheetmode
smartcalc	smartcalculator
sn	snap
snote	simplenote
so	solid
sp	split
spell	spellcheck
spl	spline
spoly	simplepolygon
ss	smartselect
st	textstyle
status	getstatus
style	textstyle
-style	-textstyle
su	subtract
svgout	exportsvg
sview	shadeview
switch	switcharea
syswindows	windows

**T**

t	note
tabledit	edittable
tableedit	edittable
tabmesh	tabulatedmesh
tabsurf	tabulatedmesh
targetpoint	targetpointlight

**iCADMac Command Aliases**

tb	table
tbconfig	toolbars
text	simplenote
textedit	editannotation
textscr	commandhistory
th	thickness
thawlay	thawlayers
ti	tilemode
time	gettime
tlmode	switcharea
tm	tilemode
tol	tolerance
toledit	edittolerance
tor	torus
tr	trim
ts	tablestyle
txs	textstyle

**U**

uc	csstyle
ucs	ccs
ucsicon	csicon
ucsman	csstyle
un	unitsystem
-un	-unitsystem
undel	undelele
undim	unrelatedimension
undo	undon
uni	union
units	unitsystem
-units	-unitsystem
unitsys	unitsystem
unla u	nlocklayer
unrelatedim	unrelatedimension
us	unitsystem

**V**

v	views
-v	-views
vdim	verticaldimension
vdirect	viewdirection
view	views
-view	-views
viewports	viewtiles
viewres	displayquality
-vp	viewdirection
vpclip	clipviewport
vplayer	viewportlayer
vpoint	viewdirection

**iCADMac Command Aliases**

vports	viewtiles
-vports	-viewport
vslide	viewslide
vtiles	viewtiles
vtxedit	editvertex

**W**

w	exportdrawing
-w	-exportdrawing
wblock	exportdrawing
-wblock	-exportdrawing
we	wedge
wipeout	mask

**X**

x	explode
xa	attachdrawing
xattach	attachdrawing
-xbind	-embeddrawing
xc	clipreference
xclip	clipreference
xl	infinetiline
xline	infinetiline
xlink	references
xopen	openreference
xr	references
-xr	-references
xref	references
-xref	-references
xrefclose	hidereferences

**Z**

z	zoom
zb	zoomback
zd	zoomdynamic
zf	zoomfit
zfa	zoomfactor
zi	zoomin
zo	zoomout
zoomarea	zoomwindow
zoomdyn	zoomdynamic
zw	zoomwindow

**#**

3a p	attn3d
3darray	pattern3d
3dface	face
3dmesh	mesh
3dmirror	mirror3d

**iCADMac Command Aliases**

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3do	rollview
3dorbit	rollview
3dpoly	polyline3d
3f	face

**iCADMac Command Aliases**

---

3p	polyline3d
3padim	3pointangledimension
4padim	4pointangledimension



## Appendix D

# Keyboard Shortcuts

### In This Appendix

- Keyboard shortcuts

**y**ou can execute common commands through keyboard shortcuts. Once memorized, shortcuts are the fastest ways to enter commands. For example, to save the drawing quickly, press **command+S**.

Note that you need to hold down the **fn** key on the Mac keyboard to make function keys execute iCADMac functions.

You customize shortcuts and overrides through the Customize command's Keyboard tab.

## Keyboard Shortcuts

Keyboard Shortcut	Action	Command Executed
Customizable shortcuts:		
command+0	Maximizes drawing area (Clean Screen)	FullScreen, HideFullScreen
command+1	Toggles Properties palette	Properties, HideProperties
command+9	Toggles command prompt area	...
command+A	Selects all non-frozen objects in current viewport	SelectAll
command+B	Toggles snap mode	...
command+C	Copies entities to Clipboard	ClipboardCopy
command+shift+C	Copies to Clipboard with reference point	Copy@
command+G	Toggles grid display	...
command+K	Displays Hyperlink dialog box	Hyperlink
command+L	Toggles ortho mode	...
command+N	Displays New dialog box	New
command+O	Displays Open dialog box	Open
command+P	Displays Print dialog box	Print
command+Q	Exits iCADMac	Exit
command+R	Switches to next viewport	...
command+S	Saves drawing	Save
command+shift+S	Displays Save As dialog box	SaveAs
command+V	Pastes from Clipboard	Paste
command+shift+V	Pastes entities from Clipboard as a block into drawing	PasteBlock
command+X	Cuts entities to Clipboard	Cut
command+shift+Z	Redoes last undo	Redo
command+Z	Undoes last command	U
delete	Erases entities	Delete
Function Keys:		
fn+F1	Displays Help window	Help
fn+F2	Shows and hides Command window	...
fn+F3	Toggles entity snap	-ESnap
fn+F5	Switches to next isoplane	IsometricGrid
fn+F6	...	...
fn+F7	Toggles grid display	...
fn+F8	Toggles ortho mode	...
fn+F9	Toggles snap mode	...
fn+F10	Toggles polar mode	...
fn+F11	Toggles entity tracking	...
fn+F12	...	...
esc	Cancels current command	...
shift	Temporary ortho override	Ortho

## Appendix E

# Button Clicks & Double Clicks

### In This Appendix

- Mouse buttons
- Double-click actions

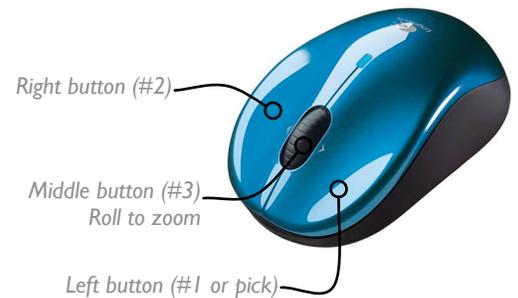
**W**hen you click a mouse or trackpad button, iCADMac executes a command or displays a shortcut menu.

When you double-click certain entities, iCADMac usually executes the Properties command so that you can modify the properties of the entity. In a few cases, a different command is executed, as listed by the table on a following page.

You can change the actions of buttons through the Customize command — with the exception of Button #1 (left), which always picks entities and buttons.

## Mouse Buttons

Button	Action
1 (left)	Picks entities; cannot be customized.
2 (right)	Displays context-sensitive shortcut menu
3 (center)	Pans the drawing
<i>Shift+buttons:</i>	
2	Displays entity snap menu
3	Orbits drawing transparently
<i>Command+button:</i>	
2	Displays entity snap menu
<i>Shift+Command+buttons:</i>	
2	Displays entity snap menu
3	Orbits drawing transparently
<i>Double-click:</i>	
1	(See double-click section)
2	Zooms drawing to fit viewport
<i>Roller wheel:</i>	
Up	Zooms in
Down	Zooms out



## Macintosh Multi-touch Compatibility

iCADMac for Macintosh understands some of the multi-touch gestures available through the MacBook's trackpad and the MagicMouse. The following actions work with the trackpad; mouse actions typically requires one less finger.

Trackpad Gesture	Action in iCADMac
Single-finger click	Pick
Two-finger click	Right-click (usually displays a context-sensitive shortcut menu)
Two-finger drag up	Zoom in at the cursor
Two-finger drag down	Zoom out at the cursor
Four-finger drag	Switch between drawings (displays Expose)

## Double Click Actions

Position the cursor over an entity, and then double-click the left mouse button to execute the following commands. Most double-clicked entities display the Properties palette. Those that do not are segregated in the list below.

### **Entity Double-clicked Command Executed**

---

#### *Left button:*

AttBlockRef	EditBlockAttribute
Hatch	EditHatch
Lwpolyline	EditPolyline
Mtext	EditAnnotation
Polyline	EditPolyline
Tolerance	EditTolerance
Text	EditAnnotation
AttDef	EditAnnotation

#### Arc Properties

Body	Properties
Circle	Properties
Dimension	Properties
Ellipse	Properties
Helix	Properties
Leader	Properties
Light	Properties

#### Line Properties

Point	Properties
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#### Ray Properties

Region	Properties
Shape	Properties
Solid	Properties
Table	Properties
Trace	Properties
Wipeout	Properties
XLine	Properties

ExtrudedSurface	Properties
-----------------	------------

LoftedSurfaceProperties

3dFace	Properties
--------	------------

3dSolid	Properties
---------	------------

#### *Middle button:*

...

Zooms drawing to fit viewport



**i**CAD Mac

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