inside
progeCAD

A Tutorial for New Users

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Copyright Information

contents

2 • progeCAD Guided Tour 15

9 • Printing Drawings 110

10 • Creating Symbols 135

Welcome to progeCAD

Welcome to progeCAD!

By learning how to work with progeCAD, you are joining over a million others who use the computer-aided design software to draft drawings. It is exciting software, because it has many of the features of CAD software that costs ten times as much — yet it is very similar to the market leader, AutoCAD.

About progeCAD

progeCAD is based on IntelliCAD, and comes from progeSOFT (www.progesoft.com).

One of the main benefits to progeCAD is its GUI, designed to have an "AutoCAD-like" feeling.

Features and Add-ons Exclusive to progeCAD

Here is a summary of features and enhancements new to the latest release of progeCAD:

ALE, Block Library Management
progeCAD Professional contains the full version of ALE, progeSOFT's powerful module for block management. More than 11,000 blocks are included, with more delivered on a regular basis.
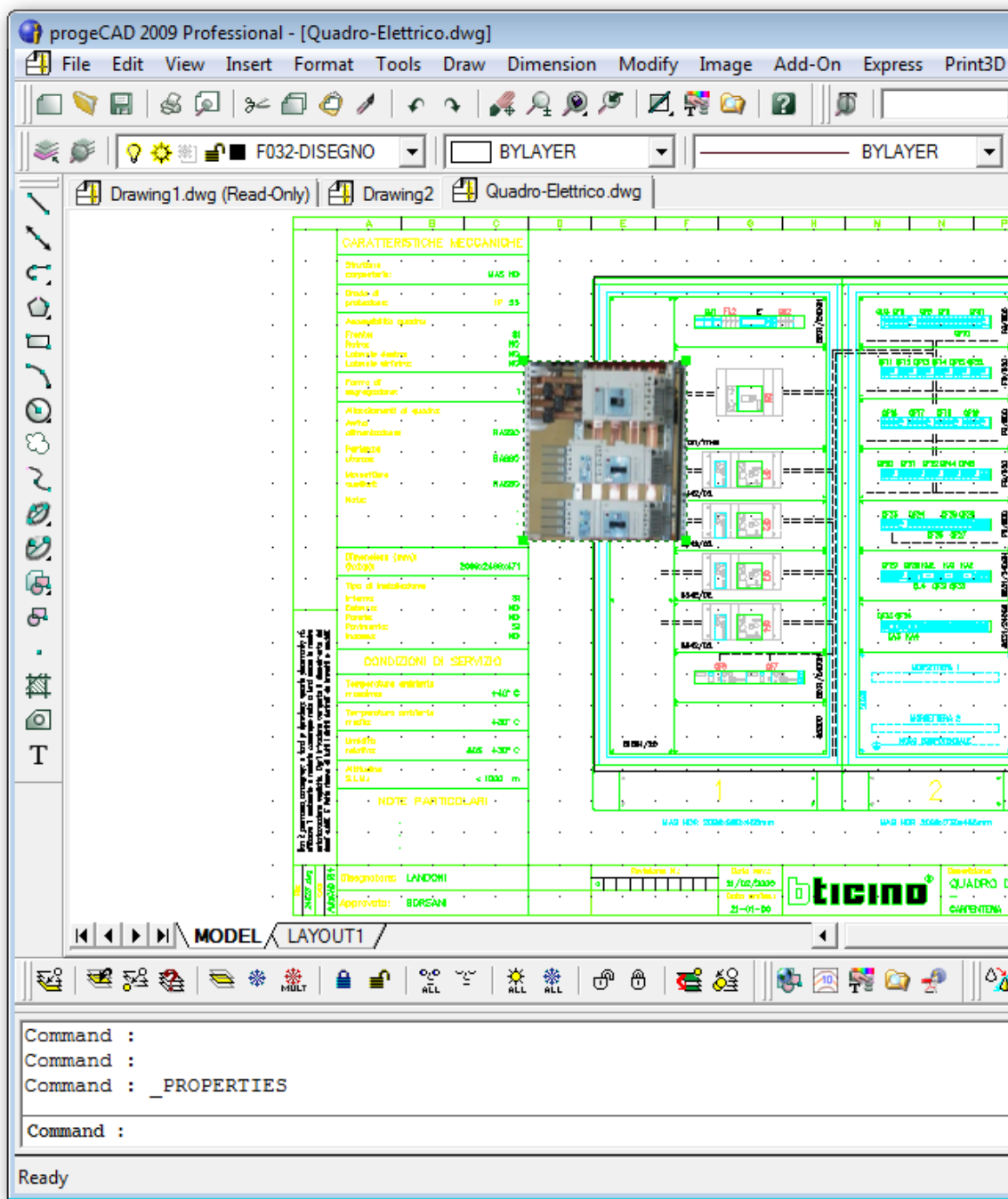
Raster to Vector
progeCAD 2009 Professional includes WinTopo raster to vector tool.

Convert PDF to DXF
progeCAD 2009 Professional automatically adds a virtual printer, which can produce PDF

prints.
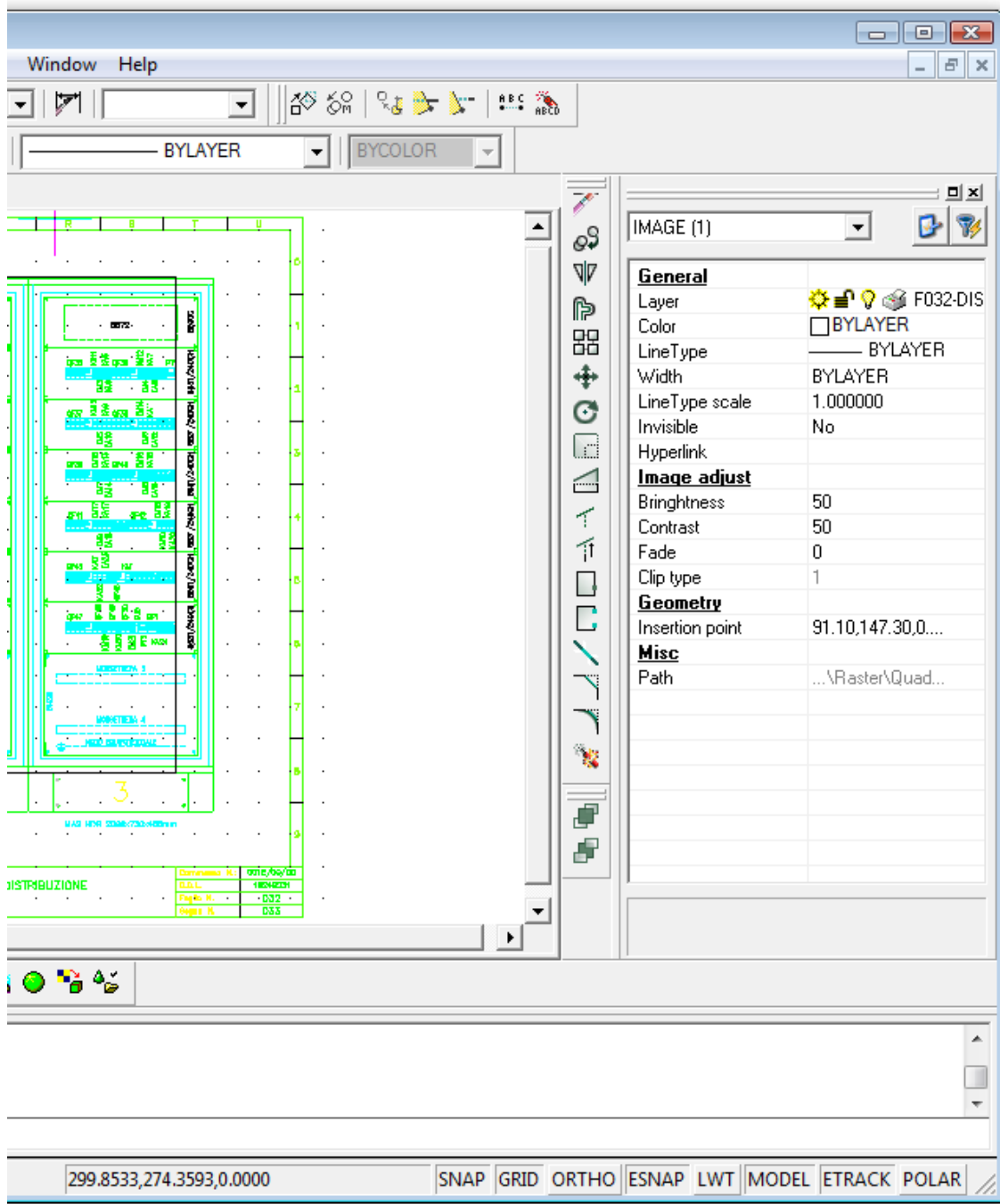
O2C Converter

progeCAD Professional contains o2c features, where 3D models are easily exported and managed with the o2c tool. o2c is the best tool for 3D model presentations.

Google Earth (new)

Export your models to Google Earth.

Layer Filters
AutoCAD-like layers management with layers filters support.

Drawing Tabs (new)
Open multiple drawings in tabs across the top of the editor window.

Layout Paper Preview
Shows you how your drawing will look when printed.

One-click Single Line Text Edit
Click single-line text with your mouse and edit it immediately.

eTransmit
Compact and transmit your DWG drawing files by email.

Graphic Purge
Interactive interface for the Purge command.

Find
Find and replace text and block attributes in drawings.

Spell Check
Spell checker is updated to include more accurate comparisons of misspelled words.

Advanced Data Extraction
An easy-to-use wizard for exporting data from attributes and blocks.

Qdim
Quick dimensioning by simply selecting the objects to dimension and placing the dimensions

New Express Tools
These powerful tools improve your productivity:

- Create perspective views.
- Replace block with xref.
- Justify text.
- Delete duplicate objects.

Standard GUI
More comfortable, standard graphic interface, new toolbars, and an AutoCAD-like feeling.

Properties Palette
AutoCAD-like interface for modifying properties: color, layer, text height, lineweights, linetype, geometry, and so on.
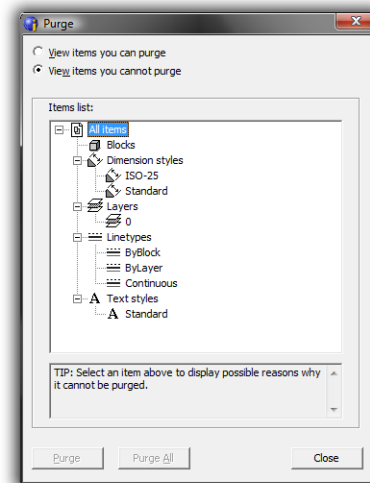
Layer Manager
Save and restore layer configurations.

progeSOFT DWG Explorer
Windows Explorer-style window for browsing drawngs and and inserting blocks.

progeSOFT New Wizard
Initial configuration template of new designs based on prototypes or predefined values.

Bitmap Clipping
Clip and polygonal clip of images.

Polar Tracking
Keeps the line to exact angles automatically.

Snap 'From'
The smart tool completing the snap family.

Print 3D
Output your 3D models to a rapid prototyping service bureau, and receive your part back within 2-3 days.

JPG and PDF print
Export drawing to JPG and PDF formats.

ECW and Jpeg 2000 raster format
The standard wavelet compression format optimized for aerial and satellite imagery.

BAttMan
Local editing of block attributes.

EAttEdit
Advanced attribute editing.

progeSOFT Wblock Manager
Tool for the improved creation of blocks on disk.

Extended Hatches
More than 300 new hatch patterns available.

progeSOFT Rendering
Advanced rendering module with multiple lights management and the creation/editing of materials. Includes reflection, mirror, transparency, smooth shadows, customizable size bitmap output.
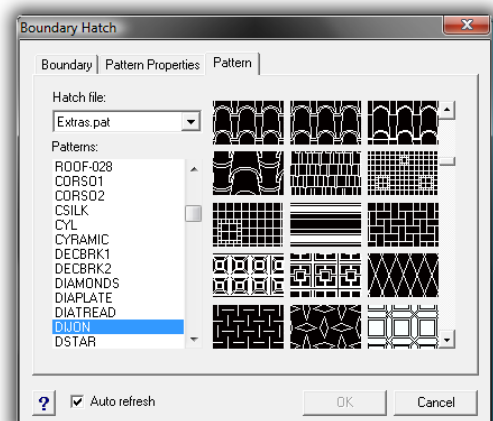
Advanced Layout Features

- Hide on layout.
- Shade on layout.
- Selection of model space with double-click.
- Layout tabs.

Redline and Markup Support

Support for redlining, commenting, markup, and collaboration with progeCAD Viewer DWG, the low cost viewer program from progeSOFT.

Asiatic Fonts

Chinese, Korean and Japanese improved compatibiliy and new fonts.

progeCAD is distributed in the United States by iCADsales (www.icadsales.com). In addition, iCADsales offers industry-specific add-ons that run within progeCAD

- progeEARTH for civil/survey work.
- progeCAM Basic and Professional for CNC machining, using progeCAM Link to transfer geometry directly to the toolpath generator, Dolphin Partmaster.
- progeOFFICE suite for interfacing with office apps (progeTABLE for spreadsheet files, progeWORD for word processing documents, and progeDATABASE for database files, and progeFIELD).

About This eBook

Inside progeCAD is based on progeCAD 2009, a commercial version of progeCAD. This book is divided into three major sections.

Section 1 introduces you to progeCAD by taking you on a guided tour. (If you know AutoCAD, then you are probably already familiar with progeCAD. Feel free to skip ahead to Chapter 4, "progeCAD for AutoCAD Users" for an overview of the differences.)

Section 2 takes you through a tutorial that designs the floor plan of a one-bedroom apartment.

Section 3 consists of an appendix that summarizes user interface elements.

progeCAD Guided Tour

In this chapter, you learn how to start progeCAD, and then take a guided tour. In addition, you learn the basics of working with any CAD system.

Starting progeCAD

You can start progeCAD in any of these ways:

- Double-click the progeCAD icon found on your computer's desktop.
- Or, on the Windows taskbar, click the **Start** button, and then select **Programs | progeCAD | progeCAD 2009**.
- Or, in Windows Explorer, double-click the name of a *.dwg* file. (This option works only when progeCAD is specified to work with *.dwg* files.)

In This Chapter

- Starting progeCAD
- Control Menu
- Drawing Area
- Layout Tabs
- Prompt History Window
- Keyboard
- Coordinate Input
- Point Filters
- Selecting Entities

- Tip of the Day
- Menu Bar
- Cursors
- Prompt Boxes
- Scroll Bars
- Shortcut Keys
- Vector Entities
- Direct Distance Entry
- Grips Editing

- Title Bar
- Toolbars
- UCS Icons
- Command Bar
- Status Bar
- Mouse and Tablet
- Keyboard Input
- Mouse Input
- Select Set Modes

Exploring the User Interface

When progeCAD starts, it presents itself like the image shown below. (Shown is the user interface for progeCAD 2009 Pro. If you use another version of progeCAD, you may see something different due to minor differences among user interfaces.)
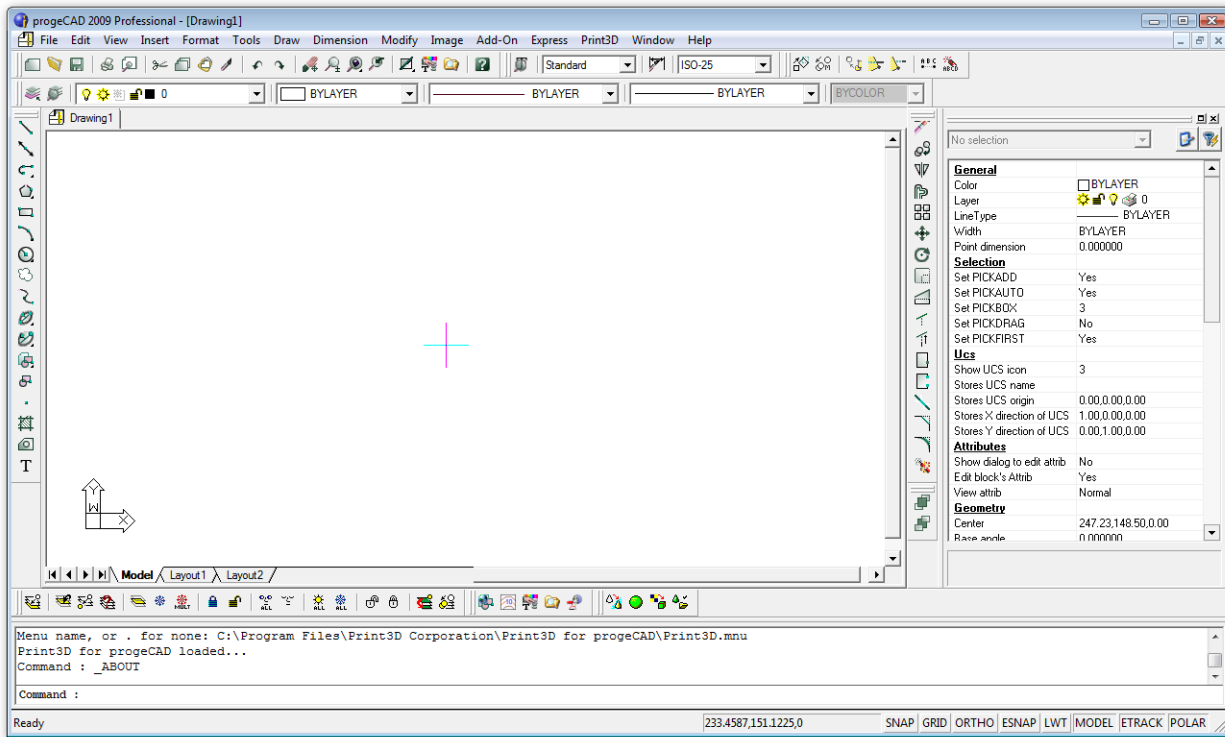
Tip of the Day

If you would like to see a handy tip on using progeCAD each time you start the software, then you may want to turn on Tip of the Day, as follows: Go to Help | Tip Of The Day, and then turn on the Show Tips on Startup option.

Title Bar

At the top of every Windows program is the *title bar*. It reports the name of the software (**progeCAD 2009 Professional** in this case) and the name of the current drawing: **[Drawing1]**, as in the figure below.
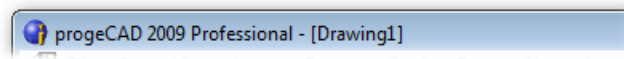
Sometimes the title bar shows additional information. When the drawing has not yet been saved, the ".dwg" file extension is missing from the drawing name. Once the drawing is saved for the first time, **.dwg** is added to the drawing name.

When drawings are opened in *read-only* mode, the phrase **(Read-Only)** is added to the file name. It reminds you that changes made to the drawing will not be saved to the same file.
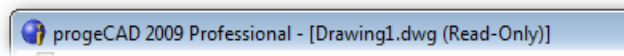
To save a read-only drawing, save it by another file name.

Control Menu

At the left end of the title bar is an icon ( ). Click the icon to see a hidden menu that controls the window. You can usually ignore this *control menu*. Sometimes, though, it is useful to know that **Alt+F4** is the keyboard shortcut that closes the current program.
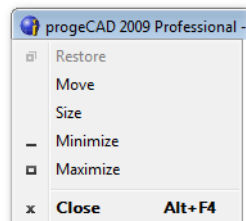
At the other end of the menu bar are three buttons that do some of the same things as this hidden menu. From left to right, they minimize (**_**) , restore (•), and close (**X**) the program window.

TIPS To maximize and restore the progeCAD window, double-click the title bar.
To quickly exit progeCAD (and save the drawing), click the **X** button or press **Alt+F4**. progeCAD automatically asks if you want to save the drawing; click **OK**. The drawing is saved and the program is closed.

Title Bar   Menu Bar

progeCAD 2009 Professional - [Drawing1]

File   Edit   View   Insert   Format   Tools   Draw   Dimension   Modify   Image   Add-On   Express   Print3D

Standard

BYLAYER   BYLAYER

Drawing1

Drawing Tabs

Properties Bar

Toolbars

Drawing Area

Cursor

UCS Icon

Layout Tabs

Command Bar

Model   Layout1   Layout2

Menu name, or . for none: C:\Program Files\Print3D Corporation\Print3D for prog
Print3D for progeCAD loaded...
Command : _ABOUT

Command :

Ready

Command Prompt

Status Bar

Window   Help

ISO-25

―――――― BYLAYER   BYCOLOR

No selection

General	
Color	☐BYLAYER
Layer	☼ ☞ ♀ ☁ 0
LineType	―――――― BYLAYER
Width	BYLAYER
Point dimension	0.000000
Selection	
Set PICKADD	Yes
Set PICKAUTO	Yes
Set PICKBOX	3
Set PICKDRAG	No
Set PICKFIRST	Yes
Ucs	
Show UCS icon	3
Stores UCS name	
Stores UCS origin	0.00,0.00,0.00
Stores X direction of UCS	1.00,0.00,0.00
Stores Y direction of UCS	0.00,1.00,0.00
Attributes	
Show dialog to edit attrib	No
Edit block's Attrib	Yes
View attrib	Normal
Geometry	
Center	247.23,148.50,0.00
Base angle	0.000000

Properties Palette

Scroll Bars

geCAD\Print3D.mnu

233.4587,151.1225,0   SNAP GRID ORTHO ESNAP LWT MODEL ETRACK POLAR

X, Y, Z Coordinates

Toggles

Menu Bar

Below the title bar is the *menu bar*. Its purpose is to sort progeCAD's many commands into logical groups. For example, almost all file-related commands, such as Open, Save, and Close, are found in the File menu. All commands related to dimensions are in the Dimensions menu.
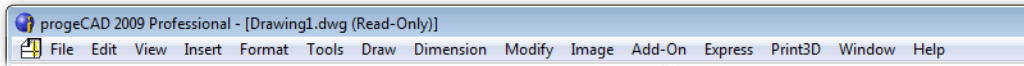
progeCAD's menus operate like those in other Windows programs. Select a menu item, and then select a command from the dropdown menu.

For example, to save the drawing, use the mouse and cursor to select the **File** item, and then choose **Save**.
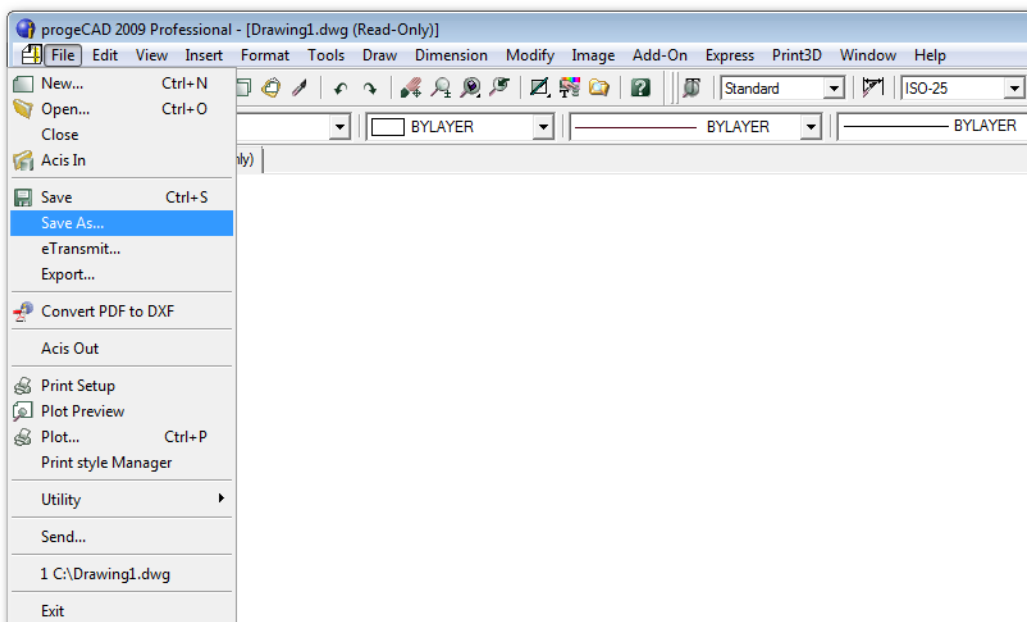
Menus use these notations:

File (underline) means that the menu item can be accessed from the keyboard, in place of using the mouse. To do so, hold down the **Alt** key, and then press the letter — **F** in this case.

Ctrl+N means the command can also be activated with a *keyboard shortcut*. In this case, hold down the **Ctrl** key, and then press the **N** key to activate the **New** command. Other keyboard shortcuts involve the **Alt**, **Shift**, and **F** (function) keys. You can customize the menu bar and its dropdown menus. From the **Tools** menu, select **Customize**, and then select the **Aliases** tab.

New... (ellipsis or three dots) means the command displays a dialog box. When there are no ellipses, look for prompts on the command line or in the prompt box.

———— (separator line) separates groups of commands within the dropdown menu.

✔ (check mark) means the option is turned on; no check mark means the option is turned off. You turn on and off the option by selecting the menu item.

‣   (arrowhead) means the menu item displays an additional menu, also called a "sub menu." In some cases, sub menus display further sub menus.

(You can customize aliases, the menu bar, and dropdown menus. See our *Tailoring progeCAD* e-book for details.)

Toolbars

While menus display commands as words, toolbars display commands as pictures on buttons called "icons." As with other Windows programs, you activate commands by clicking the buttons.
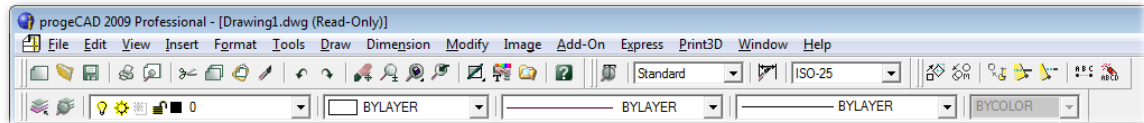
progeCAD provides a couple of assistants for decoding the meaning of icons. When you move the cursor over a button, look to the status line at the bottom of the window: it displays a one sentence description of the button's purpose.
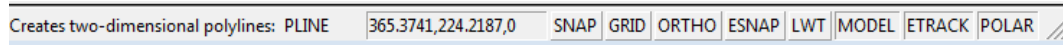
When you leave the cursor sitting over the toolbar button, a "tooltip" appears after a few seconds. The tooltip's yellow rectangle contains a word or two describing the command executed by pressing the button.
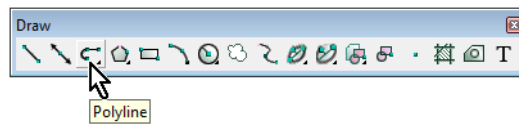
On high resolution monitors, the buttons might be too small to see. You can double their size by right-clicking any toolbar, and then select **Toolbars**. Select **Large Buttons**, and then click **OK**.
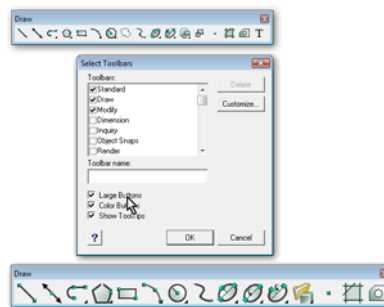
progeCAD displays five toolbars initially; other toolbars are accessed by right-clicking any toolbar, and then selecting its name from the list.

(You can customize any toolbar and its buttons. From the **Tools** menu, select **Customize**, and then select the **Toolbars** tab. See our *Tailoring progeCAD* e-book for details.)

The Entity Properties bar is a special toolbar that shows you the settings for (left to right) layers, colors, linetypes, lineweights, and plot styles.
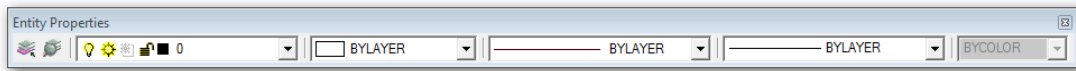
It works in three ways:

- No objects selected: the drawing's default settings are reported, as shown above.
- One or more objects selected:

  Reports settings for the selected object(s).

  Modifies the properties of the selected object(s).

Drawing Tabs

progeCAD can display one or more drawings at a time, each in its own window. To switch quickly between drawings, progeCAD sports a row of tabs between the toolbars and the drawing area. For every drawing that is currently open in progeCAD, a file name appears on a tab. Click a tab to switch to the drawing.
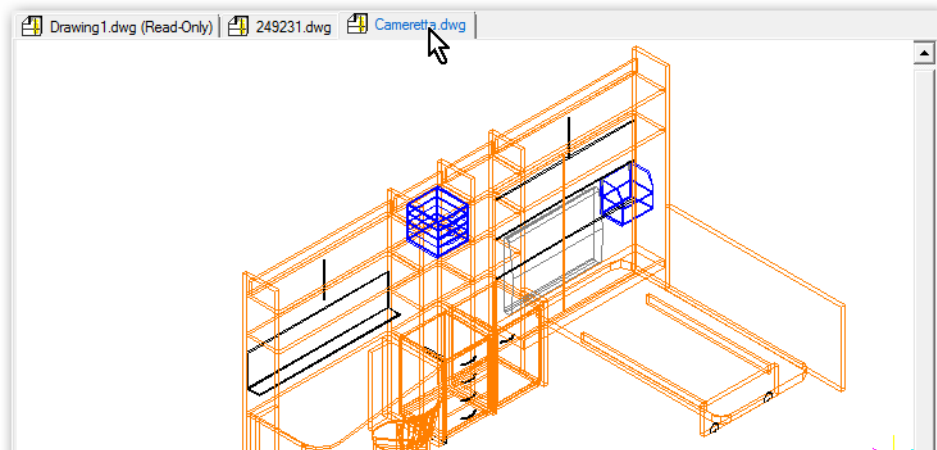
Alternatively, you can switch between drawings with the **Windows** menu, or by pressing **Ctrl+Tab**.

Right-click a drawing tab to see a shortcut menu of options:
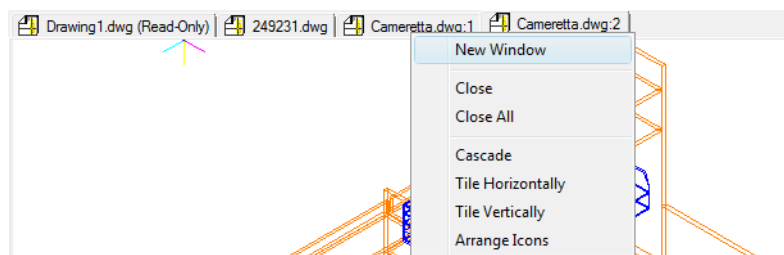
When you choose New Window, progeCAD creates a new window that contains a copy of the original drawing. Any changes made in one copy is reflected in the second.

Drawing Area

The drawing area is where you view, draft, and edit drawings.

When you start a new drawing with the **New** command, progeCAD opens a new, blank drawing. When you open existing drawings with the **Open** command, they appear in the drawing area.

You can change the color of the drawing area. Some drafters like it to be black, because it makes entity colors stand out. Other drafters (myself included) prefer white for the background color, because this color is the same as that of the paper upon which drawings will be printed.

You can change the background to any of 254 colors and shades of gray, as follows:

1. From the **Tools** menu, select **Options**, and then select the **Display** tab.
2. Click the **Color** button, found next to Graphics Screen Color.
3. Select a color in the Color dialog box.
4. Click **OK** twice to dismiss both dialog boxes.
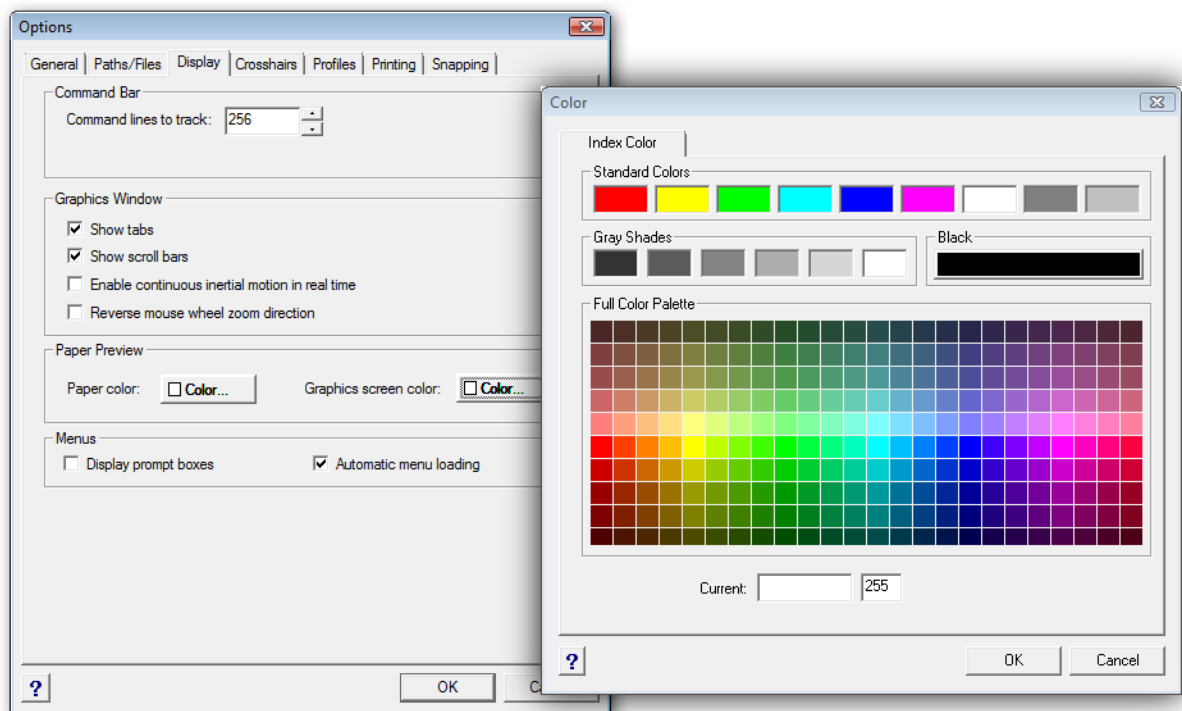
(This book illustrates progeCAD with the white drawing area.)

Cursors

The cursor shows you where "you" are in progeCAD. It also provides you with information: in different areas and in different situations, it changes its shape.

Outside of the drawing area, the cursor displays the familiar arrowhead found in other Windows applications.

When the cursor is paused over a toolbar button, a *tooltip* appears after a second or two. The tooltip explains the purpose of the button. If you do not see tooltips, you can turn them on: right-click any toolbar, and then select **Toolbars** from the shortcut menu; choose the **Show Tooltips** option.

On the edge of windows, toolbars, and stretchable dialog boxes, the stretch cursor appears, indicating you can change the size of the user interface element.

In the drawing area, the cursor changes in these ways:

Cursor	Meaning
	No command is active: progeCAD is ready to select an object. The small square is called the "pickbox": within it, progeCAD searches for objects. The size of the pickbox is changed with the **Pickbox** system variable.
	The colored cursor indicates the direction of the x (cyan) and y (pink) axes. In 3D space, the z axis is indicated by a yellow cursor line. The colors of the cursor can be changed with the **Option** command's **Cursor** option.
	Command and esnap (entity snap) are active: progeCAD is ready to select objects. The square is called the "aperture," and is displayed only when esnaps are turned on. progeCAD searches for geometric features within the aperture. The size of the aperture can be changed with the **Aperture** system variable.
	Entity snap found: When progeCAD finds a geometric feature, it displays a yellow icon called a "marker," which shows the location of the geometric feature; the yellow tooltip describes the feature (Endpoint, in this case); see full list below. The color and size of the marker can be changed with the **Options** command's **Snap Cursor** option.
	The "drumstick hitting the bassdrum" cursor is called "snap cursor decoration." It alerts you that two or more entity snaps are active. Other decorations are displayed for specific entity snaps; entire list described later. Decoration are off by default; to turn them on, use the **Options** command's **Show Snap Setting on Cursor** option.
	The magnifying glass icon is displayed in place of the cursor during the **Zoom** command.
	The hand icon is displayed in place of the cursor during the **Pan** command.
	The 3D spin icon is displayed in place of the cursor during the **RtRot** command. You will see these icons during other commands:
	RtRotX: Spin about x axis.
	RtRotY: Spin about y axis.
	RtRotZ: Spin about z axis.

Entity snaps employ a different icon for every mode. There are *decoration* icons that report the current entity snsap mode, and *marker* icons that report when progeCAD finds the matching geometric feature. Here is the list of entity snap icons and their meanings:

Esnap Mode	Cursor Decoration	Geometric Marker
Endpoint		
Nearest		
Perpendicular		
Quadrant		
Insertion		
Center		
Midpoint		
Point		
Tangent		
Deferred	*no icon*	
Intersection		
Plan View		
Parallel	*no icon*	

UCS Icon

The purpose of the *UCS icon* is to show you the drawing's orientation in 3D space. (UCS is short for "user coordinate system.") The icon is usually found in the lower left corner of the drawing area.

In 2D space, the x and y axes are shown; the z axis is looking straight down on the x,y-plane. The letter W means that the view is in the "world coordinate system" — basically, the default and any time a UCS is not invoked. (See figure below, at left.)

In 3D space, the z axis shows up. The colors of the axes mimic that of the cursor.

The UCS icon is pointless for 2D drafting, and can be turned off with the **UcsIcon** command's **Off** option. When in paper space (layout mode), the UCS is also pointless, because paper space is 2D-only. (Paper space is discussed in Chapter 11.)

The UCS icon changes to an icon that mimics a sheet of paper.

Layout Tabs

Layout tabs are located below the drawing area. They allow you to switch quickly between model space and named layouts.

Model space is where the drawing is created, where the 2D or 3D model is designed. *Layouts* are meant primarily for plotting: you arrange 2D views of the model inside viewports with drawing borders or title blocks.

While there is just one model tab, drawings can have many layouts. You would work with more than one layout when you have color and black-white plot layouts, for example, or when you have more than one sheet showing the drawing.

When you right-click a layout tab, a shortcut menu displays options for creating, renaming, and deleting tabs. (The Model tab cannot be renamed nor erased, and the first layout tab cannot be erased.)

Command Bar

The *command bar* is the area where you type commands; this is also the area where progeCAD most often reports information to you. Typically, you look here to see command options; this does not apply to dialog boxes, 'natch.
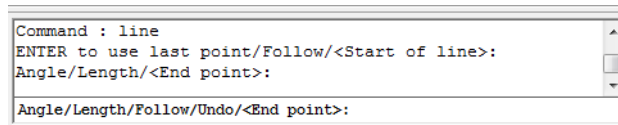
The command bar normally hangs off the bottom of the progeCAD window, but you can drag it away to make it a floating window. Once floating, you can resize the command bar to be wider, narrower, taller, and shorter.

You can double-click the status bar to toggle the display of the command bar. *Toggle* means to turn on and off.

Command Prompt

The command *prompt* is the word 'Command' followed by the colon (Command:). When you see the prompt by itself on the lowermost line of the command bar, you know that progeCAD is ready for you to enter a command. If it is not clear, press **Esc** to clear the prompt line.

TIP   Use the **CmdLnText** system variable (short for "command line text") to change the prompt character.

Press **F2** to see the *prompt history window*. It remembers the last several hundred lines of commands and prompts.

```
progeCAD Prompt History
Command : '_help
Command : ucs
?/3point/Delete/Entity/Origin/Previous/Restore/Save/View/X
/Y/Z/ZAxis/<World>: v
Auto saving open drawings...
C:\Users\ralphg\AppData\Local\Temp\Drawing1.SV$
C:\Users\ralphg\AppData\Local\Temp\Cameretta.SV$
Command : rtrot
>>ENTER, Right click or Esc to complete...
Cancel
Command :
Cancel
Command :
Opposite corner:
Command : line
ENTER to use last point/Follow/<Start of line>:
Angle/Length/<End point>:
Angle/Length/Follow/Undo/<End point>: '_CMDBAR
Floating/Lower/Upper/<Hide Command Bar>: T
Angle/Length/Follow/Undo/<End point>: '_CMDBAR
Floating/Lower/Upper/<Show Command Bar>: T

Command : |
```

Right-click the prompt history window to display a shortcut menu that lets you copy the text to the Clipboard, or to paste text into the window.

You can change the number of lines remembered by progeCAD: use the **Tools | Options | Display** command. The maximum and minimum values are 500 and 1 lines.

Prompt Boxes

Many of progeCAD commands involve *options*. The **Line** command, for example, has options for undoing (undrawing) the last segment, closing the lines as a polygon, and so on. Typically, you specify the option by entering its first letter, such as "U" for Undo and "C" for Close.

progeCAD provides an alternative to typing characters: *prompt boxes* are shortcut menus that display lists of option names, which you can select with the cursor.

The display of prompt boxes can be turned on and off through the **Config** command's Display tab: change the setting of **Display prompt boxes**.

Here is an example of the different prompt boxes that appear during the **Line** command:

(no prompt box)   Command: **line**
(no prompt box)   Start of line: *(Pick a point.)*
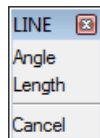
```
LINE
Angle
Length
Cancel
```
Angle/Length/<End point>: *(Pick another point, or select an option.)*
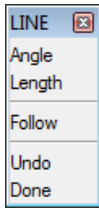
Angle/Length/Follow/Undo/<End point>: *(Pick another point, or select option.)*
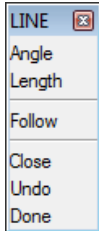
Angle/Length/Follow/Close/Undo/<End point>: *(Pick a point, or select option.)*

Prompt boxes disappear when commands end.

Some commands, such as **Circle** and **Move**, end on their own; for other commands, such **Line** and **Trim**, you need to press **Esc** or **Enter** to end them.

Scroll Bars

Scroll bars appear at the right and bottom edges of the drawing area. They are an alternative to using the **Pan** command to move the drawing view. By default, scroll bars are turned off, because that provides a bit more drawing area.

To turn on the scroll bars, use the **ScrollBar** command. Or, from the **View** menu, select **Scroll Bars** item.

Status Bar

The status bar reports information on the status of the drawing and the drafting process. From left to right, the bar provides the following info:

Creates two-dimensional polylines: PLINE   365.3741,224.2187,0   SNAP | GRID | ORTHO | ESNAP | LWT | MODEL | ETRACK | POLAR
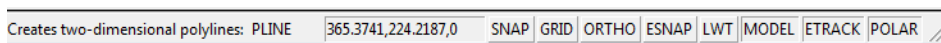
- **Help Text** — a sentence of help appears when selecting menu items and toolbar buttons; otherwise, the word "Ready" appears. You can right-click the status bar to change the data it displays. Double-click the status bar to toggle the display of the command bar.

- **X,y,z** — reports the coordinates of the cursor position in the drawing area; the z coordinate reports the current elevation setting. During drawing and editing commands, the display changes to show distance and angle:

Length: 1'-2" @ Angle: 0

Right-click to change the display of units.

- **SNAP** — toggles snap mode; right-click to access snap settings.
- **GRID** — toggles the display of the grid; right-click to access grid settings.

- **ORTHO** — toggles orthographic mode, which constrains cursor movement to right angles.
- **ESNAP** — toggles entity snap mode; right-click to access esnap settings.
- **LWT** — toggles the display of lineweights.
- **MODEL** — toggles model and paper space.
- **ETRACK** — toggles entity tracking mode.
- **POLAR** — toggles polar mode.
- **Drag corner** — drag with cursor to make the progeCAD window larger and smaller.

To change a toggle (snap, grid, ortho, esnap, model, and tablet), click its "button." When on, the word (SNAP, GRID, etc) looks indented; when off, outdented.

Changing the Status Bar

The display of the status bar can be turned on and off with the **StatBar** command. You might want to turn it off to gain a fraction of an inch more drawing area, but the status bar is far too useful to keep off. It provides an overview at the state of the drawing, and provides instant access to frequently-used toggle commands.
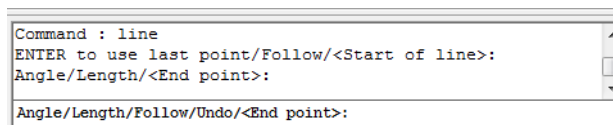
Right-clicking different areas of the status bar provides a variety of shortcut menus.

User Input

progeCAD accepts input from the keyboard, mouse, and digitizing tablet. It also supports other input devices, provided drivers are available that are recognized by progeCAD.

Keyboard

It is possible to run progeCAD almost entirely by mouse, the keyboard is still needed for inputting some coordinates, entering command options, and naming files. Indeed, there are some — such as me — who prefer the keyboard over the mouse. Commands and options are entered by typing their names in the command bar.

```
Command : line
ENTER to use last point/Follow/<Start of line>:
Angle/Length/<End point>:

Angle/Length/Follow/Undo/<End point>:
```

(Right-click the command bar to display a shortcut menu that lets you copy all text to the Clipboard, paste text, and access the Options dialog box.)

Spacebar and Enter

Pressing the *spacebar* performs several functions, whose meaning depends on *when* you press it: end input, exit commands, or repeat commands. (Pressing the spacebar is equivalent to pressing the **Enter** key.)

The spacebar tells progeCAD that you are done entering data, such as the name of a command, an option, or coordinates:

    Command: **line** *(Press spacebar or **Enter**.)*
    Start of line: *(Type coordinates, and then press spacebar or **Enter**.)*

Press the spacebar to tell progeCAD you want the command to end:

    Angle/Length/Follow/Undo/<End point>: *(Press spacebar or **Enter**.)*
    Command:

At the **:** prompt, the spacebar tells progeCAD to repeat the previous command:

    Command: *(Press spacebar or **Enter**.)*
    LINE Start of line:

Esc

Pressing the **Esc** key means "cancel." You press it to get out of commands and to cancel grips.

Sometimes you have to press **Esc** two or three times to get out of deeply nested commands, like **PEdit**. It is also the equivalent of clicking the **Cancel** button in dialog boxes.

Cursor Keys

The keys found on the keypad control *panning*. Panning moves the drawing inside the window; it is most useful when zoomed in close. Press the cursor keys (up, down, left, right) to move the drawing a small amount.

To pan by an entire screenfull, hold down the **Shift** key while pressing the cursor keys.

Shortcut Keys

Shortcut keys let you carry out commands by pressing assigned keys on the keyboard. Some users find this is much faster than selecting commands from menus, or typing entire command words at the keyboard. Out-of-the-box, progeCAD defines the shortcut keystrokes shown by the table below.

Shortcut	Meaning	Command Executed
Function Keys		
F1[a][w]	Displays on-line help.	HELP
F2[a]	Displays Prompt History window.	PMTHIST
F3[a]	Toggles object snap mode.	ESNAP T
F4[a]	Toggles tablet mode.	TABLET T
F5[a]	Switches to next isoplane.	ISOPLANE
F6[a]	Toggles coordinate display.	COORDINATE T
F7[a]	Toggles display of the grid.	GRID T
F8[a]	Toggles ortho mode.	ORTHOGONAL T
F9[a]	Toggles snap mode.	SNAP T
F10	Toggles status bar.	STATBAR T
Control Keys		
Ctrl+A[a][w]	Selects all objects in drawing.	SELGRIPS ALL
Ctrl+C[a][w]	Copies selected objects to Clipboard.	COPYCLIP
Ctrl+D[a]	Toggles coordinate display.	COORDINATE T
Ctrl+E[a]	Switches to next isoplane.	ISOPLANE
Ctrl+H[a]	Backspaces the command line.	…
Ctrl+K	Displays the previous command.	…
Ctrl+L	Displays the next command.	…
Ctrl+M[a]	Same as pressing **Enter**.	…
Ctrl+N[a][w]	Starts a new drawing.	NEW
Ctrl+O[a][w]	Opens a drawing file.	OPEN
Ctrl+P[a][w]	Prints the drawing.	PRINT
Ctrl+S[a][w]	Saves the drawing.	QSAVE
Ctrl+T[a]	Toggles tablet mode.	TABLET T
Ctrl+V[a][w]	Pastes from Clipboard into drawing.	PASTECLIP
Ctrl+X[a][w]	Cuts selected objects to Clipboard.	CUTCLIP
Ctrl+Y[a][w]	Redoes the last undo.	REDO
Ctrl+Z[a][w]	Undoes the last command.	U
Other Keys		
Del[a][w]	Deletes selected objects.	DELETE
Left Arrow	Pans left.	PAN L
Right Arrow	Pans right.	PAN R
Down Arrow	Pans down.	PAN D
Up Arrow	Pans up.	PAN U
Shift+Left Arrow	Pans left a pagefull.	PAN PGL
Shift+Right Arrow	Pans right a pagefull.	PAN PGR
Shift+Down Arrow	Pans down a pagefull.	PAN PGD
Shift+Up Arrow	Pans up a pagefull.	PAN PGU
Page Down	Pans down a pagefull.	PAN PGD
Page Up	Pans up a pagefull.	PAN PGU

[a] Shortcut key is the same as in AutoCAD.

[w] Shortcut key is standard among Windows software.

Mouse

Like other Windows software, progeCAD supports the multi-button mouse as an input device. It does not, however, allow you to change the meaning of the buttons. The functions of mouse buttons vary, depending on the current circumstance. Here is an overview of their actions.

First (Left) Button

One click of the first (left) button always selects something — whether a menu, a toolbar button, or an entity in the drawing.

You can click dfferent parts of the status bar for different effects (see the section on the status bar, earlier.)

Double-clicking the left button performs additional tasks that depend on the context. (Double-clicking means pressing the first button twice quickly.) For instance, when you double-click:

- A single-line text entity, progeCAD displays the edit text dialog box.
- A multiline text, progeCAD displays the mline text dialog box.
- A dimension, progeCAD displays the dimension setting dialog box.
- A hatch pattern, progeCAD display the boundary hatch dialog box.
- Any other entity, progeCAD displays the Properties palette box with the properties of the entity.
- progeCAD's title bar, the window changes between maximized and windowed.

> **TIPS**   Through the mouse driver, I like to assign double-click to the middle button. That saves wear and tear on my hand, because I press the middle button once to get the double-click effect.
>
> The **MButtonPan** system variable toggles behavior of the middle mouse button.
> | | |
> |---|---|
> | **0** | Operates as defined by the *.mnu* file. |
> | **1** | Hold and drag to pan the drawing. |

When you hold down the **Ctrl** key and press the first button, progeCAD goes into real-time 3D motion (same as executing the **RtRot** command).

Second (Right) Button

One click on the second (right) button repeats the last command.

Right-click a toolbar, the status bar, the command bar, or a selected entity brings up a context-sensitive shortcut menu.

Hold down the **Shift** key, and press the right button to display a shortcut menu of entity snap modes.
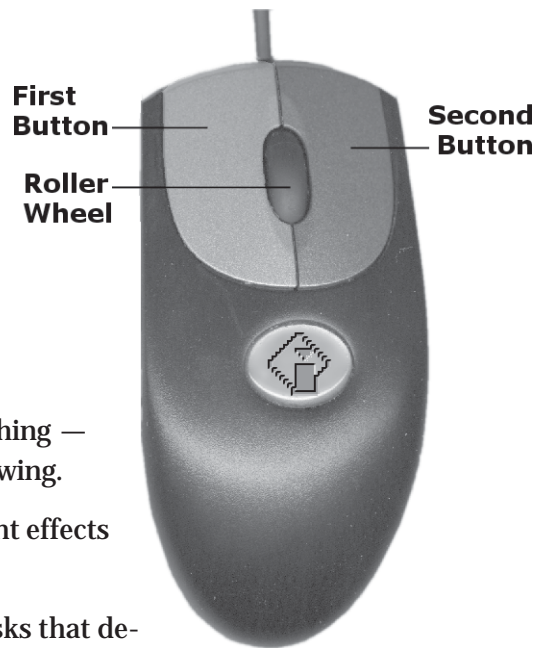
Hold down the **Ctrl** key, and hold down the right button for real-time rotation around the z axis (**RtRotZ**).

Roller Wheel

Rolling the roller wheel on the mouse (if it has one) pans the drawing up and down. To pan left or right and to zoom the drawing:

- Hold down the **Shift+Ctrl** keys together to pan left and right.
- Hold down the **Ctrl** *or* **Shift** keys to zoom the drawing in and out.

You can reverse the direction of the zoom relative to the roller wheel through the **Options** command's **Reverse mouse wheel zoom direction** option (found in the Display tab).

Tablet

progeCAD supports digitizing tablets of any size, and digitizer pucks with up to 16 buttons. Digitizing tablets consist of two parts: the *tablet* and a mouse-like *puck* or *stylus.* The tablet determines the precise location of the puck.

Illustrated at right is a typical digitizing tablet with 16-button puck. As an alternative, a pencil-like stylus can replace the puck.

Tablets were the preferred method of interacting with CAD software in the 1980s and into the 1990s. When Windows became popular in the mid-1990s, the mouse took over, because it was much, much cheaper (less than a hundred dollars versus several hundred or more), took much, much less desk space (a quarter-square foot versus a square foot or more), and was much, much easier to use (few options versus many options).

Tablets are still in use today, because they have features that mice lack:

- Pucks typically have 4, 12, or 16 buttons, whereas mice have fewer buttons.
- Tablets can select hundreds of commands in plain view on overlays; mice cannot.
- Tablets can trace paper drawings into CAD, because they employ absolute coordinates; mice cannot, because they use relative coordinates.

Using Tablets with progeCAD

Before you can use a digitizer with progeCAD, you need to implement the following steps:

1. Attach the digitizing tablet to your computer.

2. Configure the tablet with the WinTab driver.

3. Start progeCAD, and print a copy of the *Digitizer progeCAD.dwg* to fit your tablet's surface area. (See the figure of the tablet overlay on the next page.) The overlay drawing is provided with progeCAD in the *progeCAD 2009* folder.

4. Align the overlay with progeCAD's **Tablet** command. See below.

5. For additional commands specific to tablets, load the *tablet.lsp* file, found in one of folders included with the progeCAD installation. See below.

Commands Specific to Tablets

The **Tablet** command provides four options:

- **Configure** adapts the overlay to the tablet.
- **Calibrate** maps the paper drawing to the electronic drawing.
- **ON** uses the tablet for drafting.
- **OFF** uses the tablet for selecting commands from the overlay.

The *tablet.lsp* routine defines additional commands:

ButM (Button_Modify) changes the commands associated with the buttons on the puck. The first (pick) and second (Enter) buttons cannot be changed.

TabM (Tablet_Modify) changes the commands associated with cells on the overlay.

TabI (Tablet_Import) imports AutoCAD-style tablet menu files (.*mnu* but not .*cui*) into progeCAD. Note that the tablet overlay provided with AutoCAD is a 25 x 25 grid, while progeCAD's tablet menu is 26 x 26.

TabE (Tablet_Export) exports tablet and button settings to a file for use by other computers.
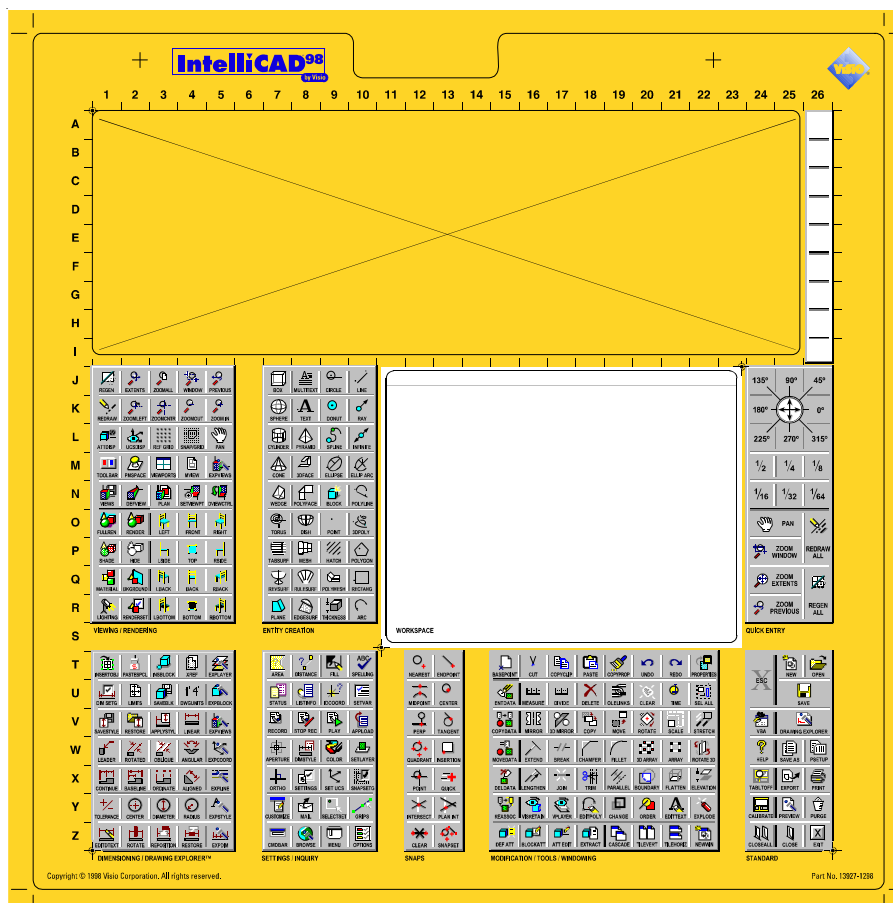
Coordinate Input

Now that you have a familiarity with progeCAD's user interface and how to control the software, the next items to understand are *commands* and *methods of input*.

To draft drawings, you use commands that create and modify entities. Commands also perform utility functions, such as saving and plotting drawings, and extracting information from drawings. You learn how to use progeCAD's commands during the tutorial sections of this book.

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

Vector Entities

All CAD software works with *vector* entities. "Vector" means that entities are defined by their geometric features. You never draw the entities themselves; instead, you specify their features, and then progeCAD draws the entities.

For instance, instead of drawing a line, you specify the two endpoints of the line:

```
Command: line
Start of line: (Pick the first end point.)
Angle/Length/<End point>:  (Pick the second end point.)
```

progeCAD then fills in the line segment between the two points.
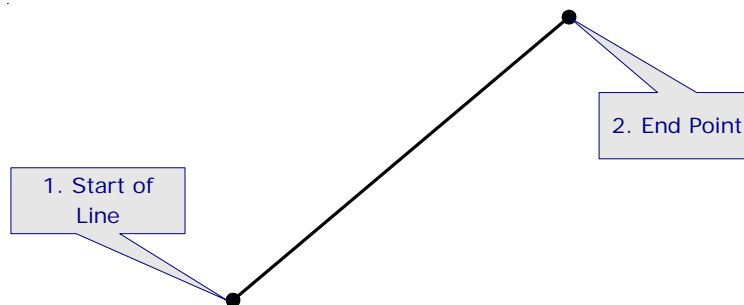
The same technique applies to other entities: progeCAD draws them after you specify their geometric features. Here are a few more examples:

Circles — indicate the location of the center point and the radius; progeCAD draws the circle.

```
Command  : circle
2Point/3Point/RADTT/TTT/Arc/Multiple/<Center of circle>: (Pick the center point.)
Diameter/<Radius>: (Specify the radius.)
```
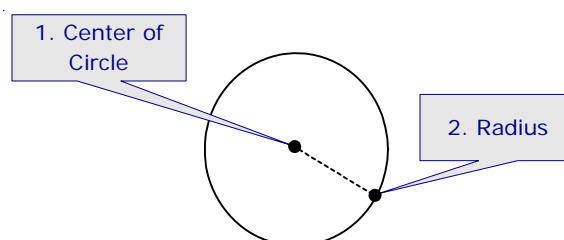
Arcs — indicate two endpoints and another point on the arc.

Command: **arc**
Center/<Start of arc>:
Angle/Center/Direction/End/Radius/<Second point>:
End point:

2. Another
Point

1. Start of Arc

3. End Point
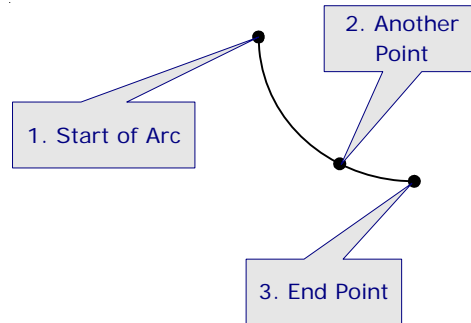
Ellipses — indicate the location and length of two axes.

1. First End of
Ellipse Axis

2. Second End
of Axis

3. Other
Axis
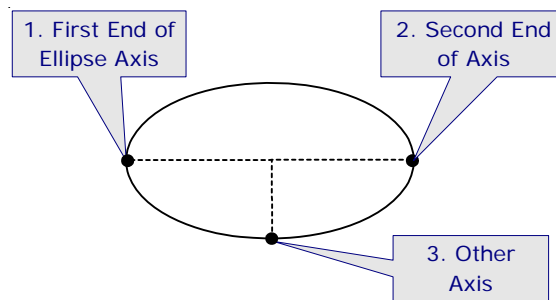
Text — indicate the starting point of the text, and its height and angle.

2. Height of
Text

1. Start Point

Text.

3. Rotation
Angle of Text
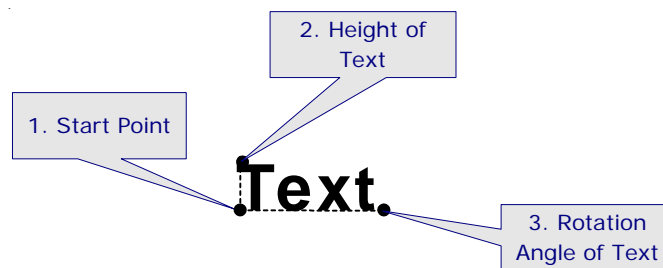
Dimension — indicate the location of two extension lines, and the dimension line and text.

3. Orientation of
Dimension Line
and Location of Text

1"

1. Origin of First
Extension Line

2. Origin of Second
Extension Line
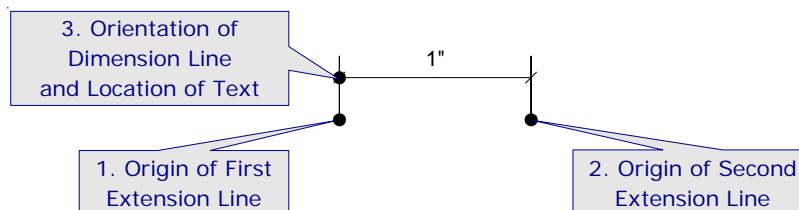
While you can specify line segments by their two endpoints, this may not be the most convenient method. For example, in drawing lines that represent walls, you often know their length, but not where their end points are located.

progeCAD is flexible enough to provide multiple methods of defining entities. To draw walls, for instance, you can specify the start point, the *length*, and the *angle* of lines. Again, you don't draw the line itself, but specify its parameters.
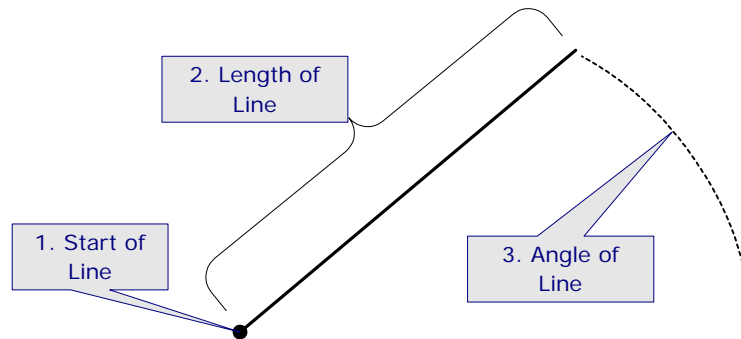
(The angle is measured from the x axis.)

The same is true for nearly all entities created by progeCAD: often, there are two or more ways to draw each of them. The arc, for example, is the most flexible: there are more than a dozen different ways to draw it.

Specifying Coordinates

To draw entities, you need to specify the coordinates (locations) of geometric features. progeCAD provides two methods for specifying geometric features: the keyboard, and the mouse (cursor). The keyboard is used to enter specific numbers, distances, and angles; the mouse is used to show locations in the drawing.

Let's take the **Line** command as an example.

Keyboard Input

The keyboard enters the names of commands and options, as well as to specify exact x, y coordinates, distances, angles, and amounts.

```
Command: line
ENTER to use last point/Follow/<Start of line>: (Enter coordinate, or enter an option.)
```

At the "ENTER to use last point/Follow/<Start of line>" prompt you can do any of the following at the keyboard:

- Press **Enter** to continue the line from the last point entered in the drawing.
```
ENTER to use last point/Follow/<Start of line>: (Press Enter.)
```

- Press **F** and then **Enter** to specify the Follow option.

    ENTER to use last point/Follow/<Start of line>: **f** *(and then press **Enter**.)*

- Type x, y coordinates, such as **1.2,3.4**, and then press **Enter**.

    ENTER to use last point/Follow/<Start of line>: **1.2,3.4** *(and then press **Enter**.)*

    Angle/Length/<End point>: *(Enter the next set of coordinates, or enter an option.)*

Specifying Coordinates

Although entering x, y coordinates is very common among CAD users, it is not the only way; progeCAD supports a variety methods of inputting coordinates.

- **x,y** — two-dimensional coordinates are used for 2D drafting. A comma (,) separates the x and y values.

    <Start of line>: **1,2**

    You can specify coordinates in imperial or metric units.

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

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

    <Start of line>: **1,2,3**

- **d<a** — polar coordinates specify the *distance* and *angle* from the current point. Think of being at the North Pole, and finding any point on the earth by a distance (from the North Pole) and an angle (around the North Pole).

    <Start of line>: **23<45**

    The angle bracket (<) prefix indicates the angle. You can specify the angle in degrees, radians, grads, or surveyor's units.

- **x,y<a** — cylindrical coordinates combine 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.

    <Start of line>: **1,2<45**

- **d<a<A** — spherical coordinates, the 3D version of polar coordinates. These 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 left-and-right around the equator (x, y plane), while the **A** measures the angle up-and-down along the International Data Line (y, z plane).

    <Start of line>: **1<45<90**

Remember that commas (,) separate distances, while brackets (<) signify angles. There are additional symbols that modify coordinates:

- **Negative distances and angles** — dash (-). Negative angles are measured clockwise from the x axis, and are indicated by the dash prefix.

    <Start of line>: **-1,-2<-45**

    (Positive angles are measured counterclockwise from the x axis, although the direction can be reversed with the **Units** command, as can the location from which they are measured.)

- **Relative coordinates** — at (@). Most coordinates are relative to the origin of the drawing (0,0) and are called "absolute coordinates." By placing the @ symbol in front of the coordinate, you measure distances and angles relative to the last point picked in the drawing.

  > <Start of line>: **@1<45**
  > End point: **@7,5**

- **UCS in WCS coordinates** — asterisk (*). When used in a user-defined coordinate system, the * prefix forces coordinates to be interpreted relative to the world coordinate system.

  > <Start of line>: ***1,2,3**

Point Filters

Finally, you can enter partial coordinates by using *point filters*. "Point" refers to the dot (.) that you first enter to alert progeCAD that you are entering a partial coordinate. "Filter" means that the coordinates are being provided partially: first x, y, z, or some combination thereof, and then the "missing" coordinate(s). The table below lists all possible combination of point filters:

Point Filter	Specify…	…and then specify
.x	Select x of	Still need y, z
.y	Select y of	Still need x,z
.z	Select z of	Still need x,y
.xy	Select x,y of	Still need z
.xz	Select x,z of	Still need y
.yz	Select x,y of	Still need x

Well, not finally. Coordinate input can also be modified by entity snaps, as described later in this chapter.

Direct Distance Entry

progeCAD has a "hidden" form of coordinate entry known as "direct distance entry." Here you move the cursor in a specific direction, and then type in the distance. This is an interactive form of relative coordinates (@d<a).

Here's how to use direct distance entry with the **Line** command:

> Command: **line**
> ENTER to use last point/Follow/<Start of line>: **1,2**
> Angle/Length/<End point>: *(Move mouse/cursor in any direction, and then enter a distance.)* **5**

While you can move the cursor in any direction to indicate the angle, it's more accurate to use ortho mode or an esnap mode to assist you in specifying the angle.

Units of Measurement

progeCAD accepts distance input in a variety of formats, including decimal (metric) and feet-inches. Angles can be input in degrees, radians, grad, and surveyor's units. Internally, however, progeCAD converts them to unitless scientific notation.

Note that you can enter distances and angles in any units format, even when it is different from that displayed by the status bar. You can change the display of units at any time by right-clicking the coordinate display on the status line.
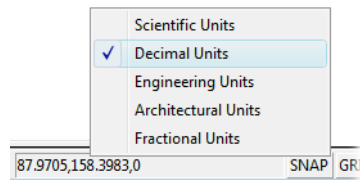
Mouse Input

You can use the cursor to show progeCAD locations in drawings; the mouse (or digitizer) controls the cursor. The problem with using the mouse is that it is not particularly accurate by itself. You would find it difficult to draw, for example, a 31.01"-diameter circle using just the mouse.

For this reason, progeCAD provides several aids for more accurately drawing with the mouse and cursor.

Status Bar shows the distance (**Length:**) and angle (**@ Angle:**) from the last pick point. As you draw, keep an eye on the status bar to see the size of the entity. (See figure below.) The drawback is that watching the status bar is too slow for production use.
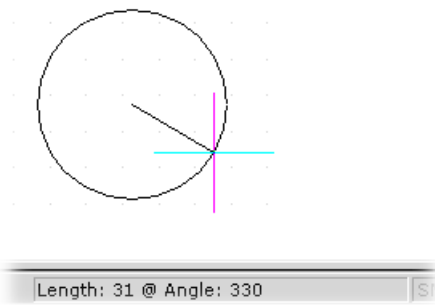
Snap specifies the cursor resolution. For instance, when you set the snap distance to 1", then mouse movements in the drawing area are accurate to the nearest one inch. (The snap does not affect the cursor outside of the drawing area.) Use the **Snap** command to specify the snap distance.

The snap distance is invisible, however, so many users also turn on the grid to match the snap distance. (In the figure above, the grid is just visible as an array of tiny dots.)

Entity Snap causes the cursor to snap to geometric features of entities, such as their end, mid, and tangent points. This is useful when you want to draw from one geometric feature to

another, such as drawing a line from the precise center of one circle to the precise tangency of another, as illustrated below.
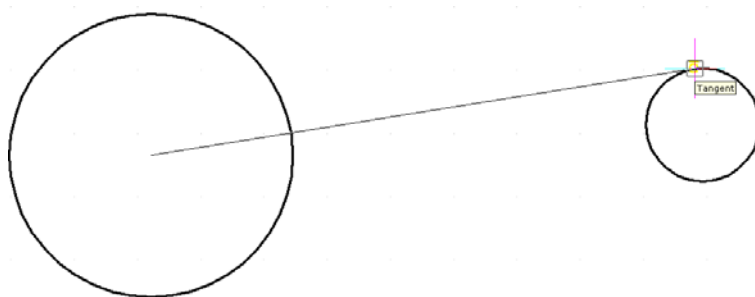
Use the **Settings | Entity Snap | Entity Snap Settings** command to toggle entity snap modes. (Entity snaps are called "esnaps" for short.)

To further help you, progeCAD displays markers and tooltips that identify geometric features. All of progeCAD's entity snap features are listed later in this chapter.

Selecting Entities

When it comes to editing drawings, you need a way to select the entities to be edited. *Selection* tells progeCAD which group of entities you need to modify. In most cases, you must use the mouse to select entities; in a few cases, you 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 can use the method you find easier.

Grips Editing

Here's how to perform grips editing: With no command active, select one or more entities by picking them with the cursor. (To select all entities in a drawing, press **Ctrl+A** on the keyboard.)

Notice that the entities change to dashed lines and that green squares appear; this is how progeCAD responds, letting you know which entity you picked.
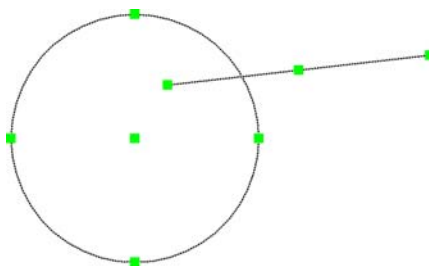
The green squares are called "grips." Move the cursor over a grip, and then drag the grip to relocate it. This action changes (edits) the entity.

Depending on the entity and which grip you move, you either change the entity's size (stretch) or move the entity's position. Check the command line for a description of the grip action. Examples include:

New end of line: *(Changes length and angle of the line.)*
New location for line: *(Moves the line.)*
New circle radius: *(Changes the size of the circle.)*
New basepoint for text: *(Moves the text.)*

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

To see a list of additional editing commands, press the mouse's right button. The resulting shortcut menu is illustrated at left, and lists the names of dozens of editing commands that operate on selected entities. For example, selecting **Print** from the menu plots only the selected entities.

To change the size and color of grips, use the **Settings | Drawing Settings | Display | Grips** command.

Repeat last command	
Redo	Ctrl+Y
Properties	Ctrl+1
Cut	Ctrl+X
Copy	Ctrl+C
Copy with base point	Ctrl+Shift+C
Paste	Ctrl+V
Paste as block	
Delete	
Copy	
Move	
Rotate	
Scale	
Group	
Explode group	
Draw order	
Print	
Open calculator	
Hyperlink	
Open link	

Multiple Entity Selection

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.

After a while, picking entities one-by-one becomes tedious, and so progeCAD provides ways to pick groups of entities. One method is the *selection rectangle*. It selects entities within an area specified by a rectangle, which is created like this:

1. Pick a blank point in the drawing (away from any entities).
2. Form a rectangle by moving the cursor to the left or to the right.

   • **Moving the cursor right** forms a *windowed* selection. progeCAD selects all entities that fall entirely within the selection rectangle. Entities crossing the rectangle and outside the rectangle are ignored.

1. Pick a point away from any entities.

2. Move cursor to the right.

Window Selection icon
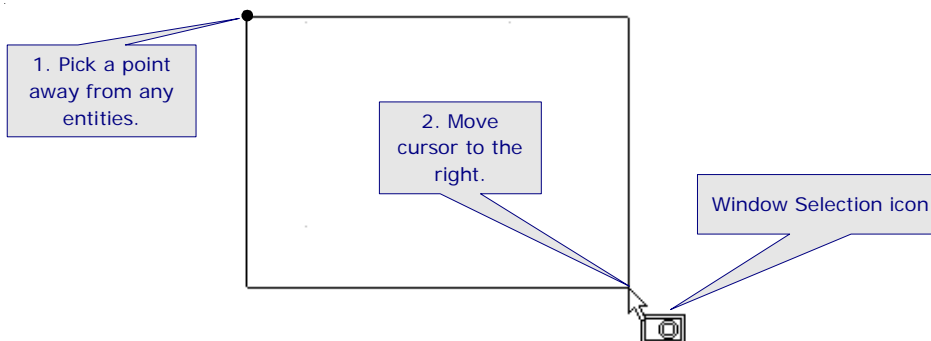
Notice that progeCAD displays an iconic cursor to remind you of the type of selection rectangle you are creating. For windowed selections, the cursor shows a circle entirely within a rectangle.

• **Moving the cursor to the left** forms a *crossing* selection. progeCAD selects all entities within and crossing the selection rectangle.

Crossing Selection icon

2. Move cursor to the right.

1. Pick a point away from any entities.
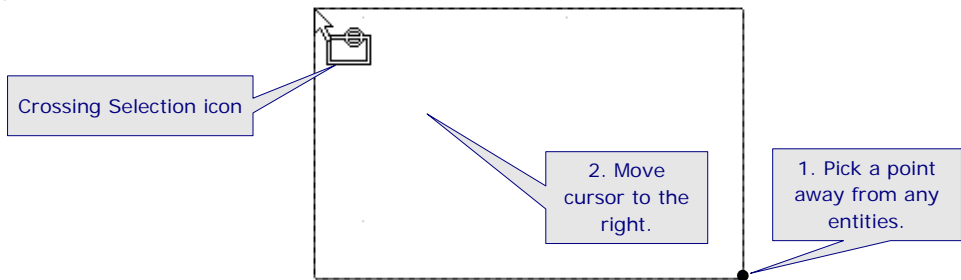
For crossing selections, the cursor shows a circle crossing the rectangle.

Selection Set Modes

But there are more selection modes! Selection areas need not be rectangles; they can also be circles, polygons, and fences. Typically, editing commands asks you to select objects, as follows:

> Command: **copy**
> Select entities to copy: **cp**

At the "Select entities" prompt, you can pick entities one at a time with the mouse, or enter one of the following options. You enter the highlighted letters only, or pick an option from the prompt box.
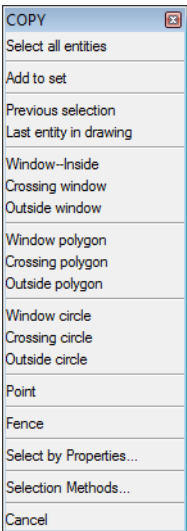
Select	Comments
Add to set	Adds one or more entities to the selection set (can also press **+**).
Crossing **c**ircle	Selects entities contained within or crossing the boundary of a circular selection area.
Crossing **p**olygon	Selects entities contained within or crossing the boundary of a polygon selection area.
Crossing window	Selects entities contained within or crossing the boundary of a rectangular selection area.
Fence	Selects entities crossing a line or line segments.
Last entity in drawing	Selects the entity most recently added to the drawing.
Outside **c**ircle	Selects entities falling completely outside a circular selection area.
Outside **p**olygon	Selects entities falling completely outside a polygon selection area.
Outside **w**indow	Selects entities falling completely outside a rectangular selection window.
Previous selection	Selects entities included in the previous selection set.
Select **all** entities	Selects all entities in the current drawing (**Ctrl+A**).
Subtract from set	Removes one or more entities from the selection set (**-**).
Window inside	Selects entities contained entirely within a rectangular selection window.
Window **c**ircle	Selects entities contained entirely within a circular selection window.
Window **p**olygon	Selects entities contained entirely within a polygon selection window.

COPY
Select all entities
Add to set
Previous selection
Last entity in drawing
Window--Inside
Crossing window
Outside window
Window polygon
Crossing polygon
Outside polygon
Window circle
Crossing circle
Outside circle
Point
Fence
Select by Properties...
Selection Methods...
Cancel

The list above contains selection modes lthat are best for selecting entities based their location in the drawing.

A second method is to select them by their *properties*. Notice the **Select by Properties** option in the prompt box above. The options are:

> **Color** — selects all entities of the specified color.
>
> **Layer** — selects all entities residing on the specified layer.
>
> **Linetype** — selects all entities of the specified linetype.
>
> **Thickness** — selects all entities of the specified thickness.
>
> **Width** — selects all entities of the specified width.
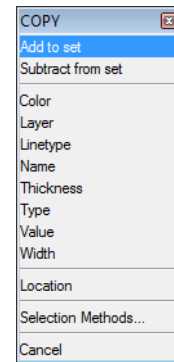
Of the other options — **Name**, **Type**, and **Value** — their purpose is not documented, but I suspect they allow programmers to select entities based on their internal name, entity type, and property value.

Add to set and **Subtract from set** allow you to add more entities based on their properties.

The **Location** option returns you to the previous cursor menu. **Selection Methods** displays the Drawing Settings dialog box. **Cancel** cancels the editing command in progress.

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**
        ENTER to use last point/Follow/<Start of line>: **.xy**
        Select x,y of: **mid**
        of *(Pick an entity with the mouse/cursor.)*
        Still need z: **3**
        Angle/Length/<End point>: **@1<-45**

123+ Tips

the 123+ tips listed in this chapter are included in the *icad.tip* and *icad.chm* files that accompany progeCAD. Some have been updated or edited for clarity.

User Interface

1. You can right-click different areas on the screen to bring up different cursor menus. Try doing this on the status bar at the bottom of the window.

Command Help

2. You can press **F1** at any time to display progeCAD Help. If you are using a command, pressing F1 displays help for that command.

progeCAD provides:

- Online user's guide.
- Online Help, which gives details about commands, tools, aliases, and system variables.
- Online developer references for programmers.

3. You can learn about any command on menus by highlighting the item, and then reading the text in the status bar at the bottom of the main window.

Screen

4. You can change the color of your screen background: from the **Tools** menu, choose **Options**, and then click the **Display** tab. Under **Graphics Screen Color**, click **Color**, and then choose the background color you want.

5. You can maximize the drawing area on the screen by hiding items you do not need at the moment: toolbars, command bar, scroll bars, and status bar. To hide or show elements of the user interface, select or clear the item that you want to hide or show in the **View** menu's **Display** submenu.
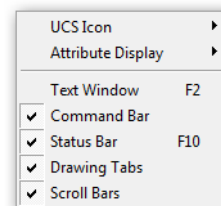
6. You can display a reference grid, and snap to regularly spaced points on the screen. To see these options, go to the **Tools** menu, choose **Drawing Settings**, and then click the **Coordinate Input** tab. Choose the **Snap and Grid** option.
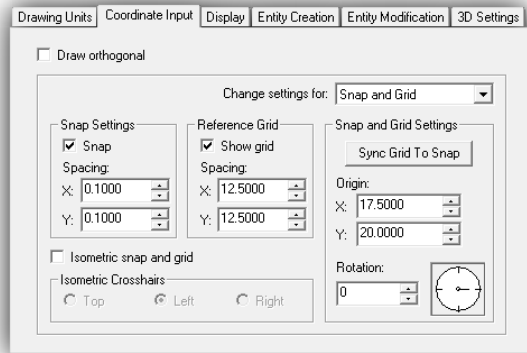
7. The drawing crosshairs use a different color for each axis. To select the colors you want, go to the **Tools** menu and choose **Options**, then click the **Crosshairs** tab.
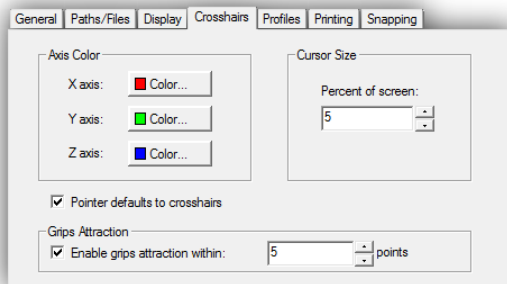
8. You can show a cursor menu of commands to modify specific entities. Select an entity, and then right-click to see the cursor menu.

Command Bar

9. If you are accustomed to using the command line in other CAD packages, such as AutoCAD, you can type the same commands and use the same options in progeCAD — even when you do not see them on the screen. Simply type the commands, and then watch the status bar.

When the command bar is not displayed, you can see each step of an active command in the status bar.

10. If you prefer to hide the command bar for more screen space, double-click the status bar.

11. You can specify any word or symbol for that progeCAD to indicate it is ready for your commands, such as 'Ready' or 'OK'. To change the text appears in the command bar, enter **CmdLnText**, and then enter the new word or symbol.

12. You can dock the command bar at the top or bottom of the progeCAD window or float it anywhere you want.

13. You can change the number of lines displayed by the docked command bar by dragging the separator line.
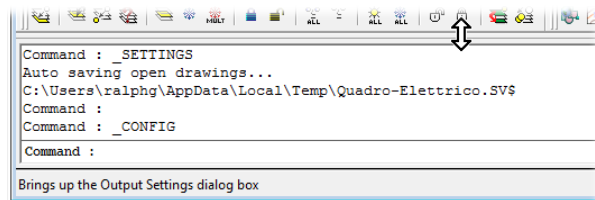
You can resize the floating command bar: drag it to the middle of the window, change the size, and position it where you want. To prevent the command bar from docking, press and hold the **Ctrl** key as you move it.

14. You can quickly hide and show the command bar by double-clicking in the text area on the left side of the status bar.

15. You can display entity information in the Prompt History window by typing **List**. Follow the prompts to choose how you want information displayed.

16. progeCAD automatically tracks your command history. The default number of recorded lines is 256, but you can change the number to between 1 and 400. From the **Tools** menu, choose **Options**, then click the **Display** tab.

Enter the number of lines you want to track next to **Command lines to track**.

17. Many commands can be "nested." To start a second command while one is in progress, type an apostrophe ('), and then enter the second command. While the nested command is active, an angle bracket (>) is displayed in the command prompt.

Prompt Box

18. After you select a command, a prompt box displays options. You can use the mouse to select command options from the prompt boxes.

19. You can turn the prompt boxes on and off. From the **Tools** menu, choose **Options**, click the **Display** tab, and then change the check box fo**r Display Prompt Boxes**.

20. You specify command options by typing the capital letter(s) shown in the command bar. For example, to choose the **Length** option in the **Line** command, simply type **L** at that prompt.

21. To change where prompt boxes appear on the screen, move one to a different location. Future prompt boxes will display in the new position.

Menus

22. You can set progeCAD to show the tools and menu items appropriate for your experience level. From the **Tools** menu, choose **Options**, click the **General** tab, and choose **Beginner**, **Intermediate**, or **Advanced** from the **Experience Level** droplist.
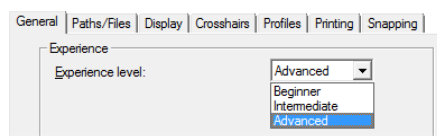
23. On menus, a check mark indicates that the setting is in use.

24. You can save your arrangements of menus (export) and you can read menus from AutoCAD (import). To export and import menus, go to the **Tools** menu, choose **Customize**, and then click the **Menus** tab.

25. You can replace all your menus with new sets or append a few new ones to existing menus. Click the **Append To Current Menu** check box, as appropriate.

Dialog Boxes

26. You can close dialog boxes by pressing the **Esc** key.

Toolbars

27. You can move the toolbars and the command bar to any place on the screen by dragging them away from the edge of the window.

28. You can make the tool icons larger to help you recognize them more easily. To change to large toolbars, go to the **View** menu and choose **Toolbars**, and then select the **Large Buttons** check box.

29. You can remove from your toolbars tools that you rarely use. Hold down the **Shift** key and drag the tool away from the toolbar. For more toolbar customization options or to replace tools you have removed, go to the **Tools** menu and then choose **Customize**.

30. Several toolbars include *flyout* toolbars with related commands and options. Flyout toolbars are indicated by a small, black triangle in the lower right corner of the tool.

31. You can display as many or as few of the toolbars as you want. From the **View** menu, choose **Toolbars**, and then click the check box for the toolbars that you want to display.
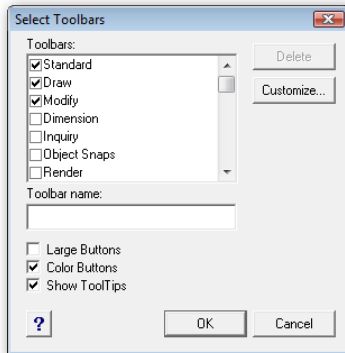
32. To hide a toolbar, drag it away from the edge of the window, and then click the **X** button.

33. You can display a cursor (shortcut) menu of available toolbars by right-clicking any toolbar.

Status Bar

34. You can change the units display method by right-clicking the coordinates display in the status bar. Choose a different set of units from the cursor menu.

Dynamic Pan and Zoom

35. progeCAD provides dynamic pan and zoom using the mouse:

- To pan dynamically, press **Ctrl+Shift** and hold down the *right* mouse button.
- To zoom dynamically, press **Ctrl+Shift** and hold down the *left* mouse button.

To stop panning or zooming, release the mouse button.

36. You can pan horizontally and vertically using the scroll bars at the side and bottom of the drawing window. The size of the scroll boxes are proportional to the view size.

Drafting and Drawing

Creating Drawings

37. You can use any drawing you want as the basis for future drawings; these are called "templates." Template can include borders, title blocks, and your preferred settings.

38. To use the New Drawing wizard, go to the **File** menu and select **New**.
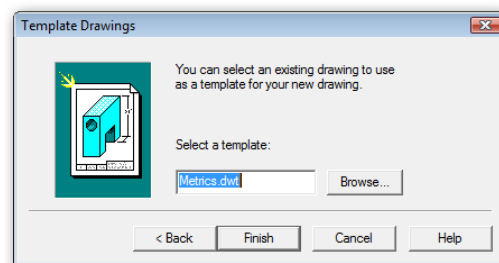
To open a new drawing *without* the assistance of the wizard, from the **Standard** toolbar, select **New**.

39. You can open multiple windows to view your drawing in different ways at the same time. Right-click a drawing tab, and then choose **New Window**.

40. You can include progeCAD drawings in the documents of other programs. Simply drag and drop the progeCAD drawing onto the document. To edit the drawing in-place, double-click it.

41. Similarly, you can drag and drop documents from other programs into progeCAD drawings. For example, you could add a Word document or a Visio diagram.

Information from the Drawing

42. You can find distances, areas, and exact locations in your drawings using the tools on the **Inquiry** toolbar.

43. You can create drawings in any way convenient for you in model space (the default), and then arrange it for printing in paper space.

Blocks

44. You can create blocks, which are combinations of entities that you use repeatedly.

To create a block, draw something you want to reuse. Go to the **Draw** menu, choose **Block**, and then choose **Make**.

To use the block later, go to the **Insert** menu and choose **Insert Block**, or select the block in the progeCAD Explorer.

45. With the progeCAD Explorer, you can view blocks before inserting them.

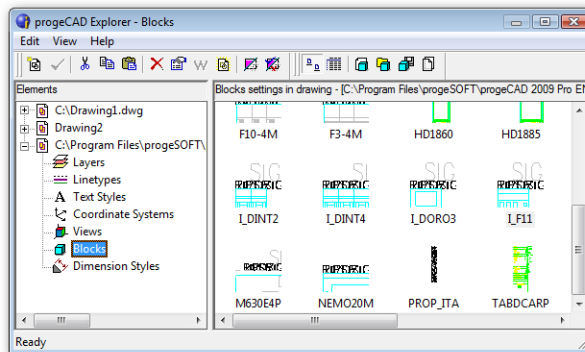46. You can associate text (called "attributes") with your blocks when you define them. Each time you insert the block, you enter the appropriate information for that instance, such as a part number, price, or supplier.

47. The array of blocks inserted with the **DdInsert** command's **Multiple Blocks** option is itself a block.

48. You can insert blocks and hatches in exploded form so that they are placed as pieces rather than as a single unit. To do this, precede the name of the block or hatch pattern with an asterisk (*).

49. You can use block attribute information in an external drawing database. To export data from the attributes in your drawings, go to the **Tools** menu and choose **Extract Attributes**.
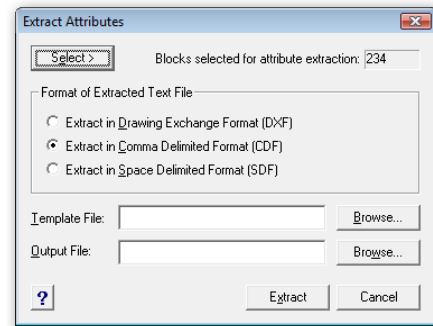
50. You cannot explode blocks inserted with the **MInsert** and **Xref** commands.

progeCAD Explorer

51. You can see and work with the layers, linetypes, styles, and other elements of external references (xrefs) in the progeCAD Explorer.

52. You can save a block to a separate drawing file using the progeCAD Explorer. Select the block to save in its own drawing, and then click the **Save Block** tool.

53. In progeCAD Explorer, you can double-click the name of an item to make it current. For example, to set a layer as the current layer, double-click the name of the layer.

54. You can copy anything in the progeCAD Explorer from one drawing to another: layers, linetypes, saved views, blocks, text styles, dimension styles, and user coordinate systems.

55. You can use the progeCAD Explorer to easily change your layers, linetypes, styles, coordinate systems, views, blocks, and dimension styles. From the **Tools** menu, choose **progeCAD Explorer**. Right-click the item you want to edit, and then choose **Properties**.

You can alter layer, color, linetype, and linetype scale.

56. To make information from external references available in the progeCAD Explorer, reload the drawing using the External Reference (**XRef**) command.

57. You can rename elements from within the progeCAD Explorer.

58. You can constrain lines and other entities to draw only vertically and horizontally. For just a few lines, hold the **Shift** key down while you draw. To work this way continually, click the **ORTHO** button on the status bar.

59. You can place point entities or blocks at regular intervals along an entity using the **Divide** and **Measure** tools:

> Divide marks the entity in a specified number of segments.
>
> Measure marks off segments of specified length. Neither command alters the original entity.

60. You can draw quickly and precisely by snapping to points on existing entities, such as the end of a line or center of a circle. To set entity snaps and see their current states, use the **Entity Snaps** toolbar.

61. *Polylines* are sets of line and arc segments handled as a single entity. Polylines can have qualities that ordinary lines cannot, such as width. When you move a vertex of a polyline, both adjacent segments move with it.
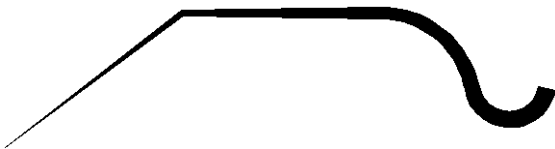
62. When drawing lines, arcs, and polylines, you can use the **Follow** option to continue drawing from the last point and in the same direction.

63. A *donut* is a planar, circular entity that has an inside and outside diameter, like a flat washer. You can draw donuts by using either the two-point, three-point, or tangent-radius method.
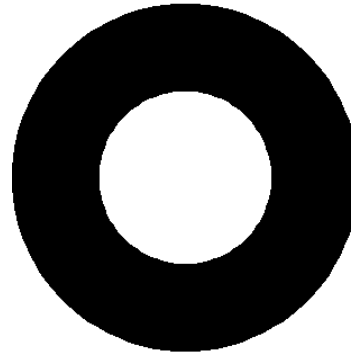
64. When you create planar entities using the **Solid** command (aka Plane), you can constrain the planes to squares, rectangles, and equilateral triangles.

65. You can use the **Rectangle** command to draw squares and rectangles at a specific angle. You can also adjust the width of them.

66. Your drawing may contain blocks that you no longer need. Use the **Purge** command to clean out unused blocks.

67. You can use the **Center** entity snap to snap to the center of any two-dimensional closed polyline, including rectangles and polygons.

68. You can bisect lines, arcs, and polylines using the Ray command. Start the **Ray** command, select **Bisect**, and then select **Entity**.

69. You can create regular polygons such as hexagons and pentagons using the **Polygon** command. All polygons are created as closed polylines.

70. Polylines have a direction. To reverse the direction of a polyline, first select it, and then from the **Modify** menu, choose **Edit Polyline**, and then **Reverse Direction**.

71. You can snap lines perpendicular or tangent to existing entities. Simply select a point on the entity, and progeCAD adjusts the point so that the line you are drawing is perpendicular or tangent to the entity.

72. A drawing can contain a reference to another drawing, called an "external reference," or *xref*. The display of the referenced drawing updates each time the reference is reloaded using the **XRef** command.

73. When drawing lines, you can specify the angle or length first, and then adjust the other parameter dynamically.

74. You can use the **PEdit** command to specify whether linetypes are broken or continuous across vertices. Select the entity, and from the **Modify** menu, choose **Edit Polyline**, and then choose **Linetype Mode**. From the prompt box, choose **Continuous On** or **Continuous Off**.

75. You can join colinear lines and coplanar arcs with the same center and radius using the **Join** command.

76. If you are drawing in isometric planes and the grid is on, the x axis grid spacing is automatically adjusted to isometrically match the y axis spacing.

77. To draw isometrically, from the **Tools** menu, choose **Drawing Settings**. Click the **Coordinate Input** tab, and in the **Change Settings For** list, click **Isometric Snap and Grid**.

78. When using the **Polygon** command, you can change the line width of the polygon you are drawing.

79. To close an arc, use the **Arc** option of the **Circle** command.

80. You can set points to be displayed in any of twenty different ways. From the **Tools** menu, choose **Drawing Settings**, and then click the **Entity Creation** tab. Change the settings for points by choosing one of the displayed options.
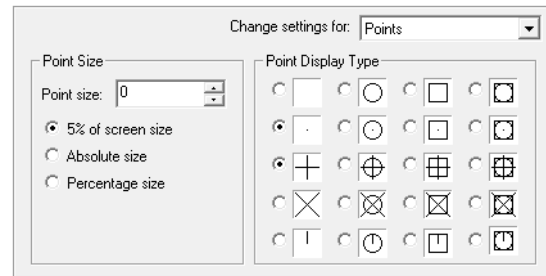
81. You can make a layer current by selecting an entity on that layer. From the **Tools** menu, choose **Set Layer By Entity**. Select an entity on the layer you want to make current.

82. The length of the last entity you specified or drew before starting the **Lengthen** (aka EditLen) command is displayed by the prompt.

83. To create a planar face with the **Face** command, position all points on the same plane.

84. When **Fill** is turned off, the program displays and plots filled drawing entities faster on slower computers.

85. For more accurate tracking during the **Sketch** (aka Freehand) command, specify shorter segments. Doing this results in a smoother freehand entity, but creates larger drawing sizes.

86. Displaying objects as icons with the **InsertObj** command is faster than displaying the objects themselves.

87. Right-clicking an object opens a menu of commands you can use on the object.

88. To create a polygon mesh with the **RuleSurf** command, select each entity at the same end. To create a self-intersecting polygon mesh, select each entity at opposite ends.

89. Entities that you preselect with the **Select** command are placed into a *previous* selection set. When you perform many progeCAD com-

mands (such as Delete or Change), you can choose Previous selection set.

90. The current layer name is displayed in the Properties toolbar.

91. To continue text under the previous line, start the **Text** command and then press **Enter** until it displays the 'Text:' prompt.

92. progeCAD provides three visual indicators to show that an entity snap is turned on:

> When any entity snap is turned on, ESNAP appears in black on the status line.
>
> When the Entity Snaps toolbar is displayed, the entity snap tool that is on is displayed as recessed.
>
> When drawing or editing, the cursor displays icons indicating the current entity snap.

93. The **Flatten** command allows you to move sets of entities, such as three-dimensional walls, to a new elevation. When you are in plan view, this command has no visible effect.

94. When the grid spacing is **0**, it is not turned off. Rather, the grid is set to the same spacing as the snap spacing.

95. Use the **Style** option of the **Snap** command to turn on isometric drawing mode. In addition, turn on the grid to help you visualize the isometric grid.

Draw isometric circles with the **Isocircle** option of the **Ellipse** command.

96. When the system variable **LimCheck** is turned on, you are restricted from drawing entities outside the rectangular area you define when you use the **Limits** command:

> To draw entities outside the drawing limits, set the **LimCheck** variable to **0**.
>
> To restrict drawing to within the drawing limits, set the **LimCheck** variable to **1**.

97. The **Line** command's **Follow** option uses the angle stored in the **LastAngle** system variable.

98. Choosing the **Batch** option of the **Purge** command purges all unused elements without asking you to confirm each element.

99. There are two ways to chamfer entities: you can specify the distances from the intersection point of each entity, or you can specify the distance on the first entity and an angle for the chamfer line.

The distance you specify for the **Chamfer** command's first chamfer distance is saved in the **ChamferA** system variable. The distance you specify for the second chamfer distance is saved in the **ChamferB** system variable.

To draw rectangles with the **Rectangle** command with chamfered corners, set the chamfer distances with the Chamfer option.
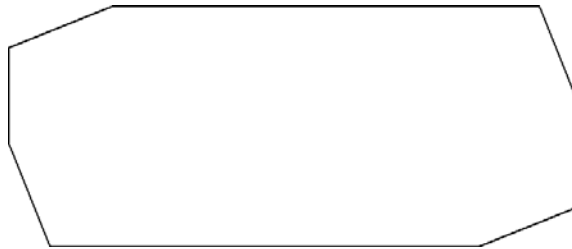
100. The distance you specify for the **Rectangle** command's fillet distance is used for all subsequent rectangles you draw. To change the fillet distance, choose **Fillet** again and enter a new value.

101. progeCAD supports the standard Windows ANSI character set. To enter an extended character, press **Alt** plus the numeric equivalent (be sure that **Num Lock** is turned on) on the keyboard's numeric keypad. For a list of the ANSI characters supported and their numeric equivalents, refer to the Character Map utility included with Windows.

102. You cannot use the **View** command's **Save** and **Restore** options as a transparent

command when you are in paper space, during **Viewpoint** or **DView**, or when zooming or panning the display.

103. You can apply the qualities of one entity, including color, linetype, and thickness, directly to other entities. From the **Standard** toolbar, choose the **Copy Entity Formatting** tool.

104. If you press **Enter** to select all entities in the drawing as boundaries during the **Trim** command, the program immediately continues to the next prompt.

105. You can simultaneously modify the properties of any number of selected entities. From the **Modify** menu, choose **Properties**, and then make the changes that you want.

106. You cannot edit the length of a polyline that has been splined or fitted.

107. To edit the attributes of more than one block at a time, use the **Tools | Edit Attribute** command.

108. You can use the **Explode** command to alter various properties of entities as they are broken into their component pieces.

109. You can use the **Copy** and **Move** commands between drawings. When prompted for a displacement point, specify a point in the destination drawing.

110. You can fillet or chamfer selected vertices of a polyline. Select two points on the polyline, and all vertices between the selected points will be modified. The operation goes in the direction of the polyline, so on closed polylines the order of point selection determines which vertices are modified.

111. You can use the system variable **MirrText** to specify whether you want text mirrored during the **Mirror** command:

> To change the text direction, set the **MirrText** variable to **1**.

> To retain the text direction, set the **MirrText** variable to **0**.

112. The **Fillet** command fillets closed polylines by two different methods, depending on how the polylines were closed:

> If you used the **Close** option of the **Polyline** command, then all vertices are filleted.

> If you specified the endpoint of the last segment, the last vertex is not filleted, even if you used entity snap to place the endpoint.

113. You can change entities to make them the same length. From the **Modify** menu, choose **Edit Length**. From the prompt box, choose **Total**. Enter the length you want, and then select the entities you want to lengthen.

114. You can alter the way the Change command affects rays and infinite lines. First select the entities to change, and then use the **Change** command. Choose the **Entities** option to select **No Change**, **Change Angles**, or **Change Definition Points**.

115. You can fillet a polyline, even if the intersection lies on a curved segment of the polyline.

116. You can change the hatching style for nested regions to hatch either the outer and inner regions, only the outer region, or all regions. To set this variable, from the **Tools** menu, choose **Drawing Settings**. Click the **Entity Creation** tab, and in the **Change Settings For** list, click **Hatches**.

117. You can erase the ends of sketches you are drawing using **Sketch**. In the prompt box, select **Delete On** and move the cursor to erase. To continue drawing, go back to the prompt box and select **Delete Off**.

118. You can straighten polyline segments. From the **Modify** menu, choose **Edit**

Polyline. Select the entity, and then from the prompt box, choose **Edit Vertices**. To select the vertex, choose **Next Vertex** or **Previous Vertex**, and then choose **Straighten**.

119. You can also change the included angle of individual polyline segments. From the **Modify** menu, choose **Edit Polyline**. Select the entity, and then from the prompt box, choose **Edit Vertices**. Choose **Next Vertex** or **Previous Vertex** to select the vertex, and then choose **Angle**.

120. You can explode traces, planes, and 3D faces into their component lines. From the **Modify** menu, choose **Explode**, and then select the entities to explode.

121. When editing a polyline, you can taper the width of the polyline continuously from start to end. All vertex widths are automatically calculated. From the **Modify** menu, choose **Edit Polyline**, and then choose **Taper**.

122. You can easily convert arcs into circles. From the **Draw** menu, choose **Circle**. From the prompt box, choose **Turn Arc Into Circle**.

Selection Sets

123. You can create a selection set and add entities to it by their properties, such as layer or color. Type **Select**, and from the prompt box, choose **Select By Properties**.

124. After you create a selection set for one command, you can easily recall that same selection set for the next command you use by choosing **Previous Selection** or by typing **P** at the 'Select Entities' prompt.

125. You can adjust the size of the cursor used for selecting entities. To set the size, go to the **Tools** menu and choose **Drawing Settings**. Click the **Coordinate Input** tab, and in the **Change Settings For** list, choose **Entity Selection**.

126. You can select several entities and change their properties simultaneously. Select the entities, and then right-click to see the cursor menu. Select **Properties**, and make the global changes.

UCS and WCS

127. The World Coordinate System (WCS) is the default and the foundation on which user coordinate systems are built. You can define your own coordinate systems to make specifying points and reading entity information easier: from the **Tools** menu, choose **User Coordinate Systems**, or create a new UCS in the progeCAD Explorer. You return to the WCS at any time by entering **UCS** in the command bar and choosing the **World** option.

128. Defining a user coordinate system (UCS) can involve anything from simply shifting the origin (0,0,0) to reorienting the axis so you can easily draw in a different plane. You can create as many user coordinate systems as you want, and then save and recall them as you need them.

129. Coordinates displayed in the status bar and through the **List** and **Distance** commands are relative to the current coordinate system, either a user coordinate system (UCS) or the World Coordinate System (WCS).

130. You can use the **Plan** command to change to the plan view for any user coordinate system or the World Coordinate System. From the **View** menu, choose **Plan View**.

Dimension Settings

131. There are hundreds of settings that affect how dimensions are created and displayed. To see how they work and to customize them, go to the **Tools** menu and choose **Dimension Settings**.

Customizing and Programming

132. You can open a drawing, run a script, or load a LISP program by dragging and dropping the file on progeCAD.

Command Aliases

133. You can make aliases for any of the progeCAD commands. For example, you may prefer "guideline" over "infline." To make an alias, go to the **Tools** menu, select **Customize**, and then click the **Aliases** tab.

134. You can assign commands to any key on your keyboard or change existing key assignments. You can also use key combinations with **Shift**, **Alt**, and **Ctrl**. From the **Tools** menu, choose **Customize**, and then click the **Keyboard** tab.

Recording Scripts

135. You can record command bar entries and mouse coordinate selections in a script file, which you can play back later. Doing this is handy for avoiding repetitive work. Scripts are saved in files with the *.scr* extension. From the **Tools** menu, choose **Record Script**.

When you have finished, choose **Stop Recording**.

Customizing Menus

136. You can customize your menus, toolbars, keyboard shortcuts, and aliases. From the **Tools** menu, choose **Customize**, and then make the changes you want.

To set them back to their original state, click **Reset**.

LISP

137. LISP is an interpreted programming language invented by John McCarthy in 1958 for work in artificial intelligence. You can write your own LISP programs for progeCAD to automate frequent or complex operations. See the online *progeCAD Developer Reference* for complete information.

You can enter a LISP expression in response to a request from a LISP function for input. For example:

Enter circumference: **(* 2 3.14 5)**

138. The **Multiple** command is not valid in LISP. See the *Developer Reference* section of Help for more information.

139. Any LISP function placed in the *icad.lsp* file is automatically loaded each time you start progeCAD.

140. You can email a drawing to someone else while working in progeCAD. From the **File** menu, choose **Send Mail**. Your current drawing is sent as an attachment.

You cannot email drawing files using this command if your system is configured with a Browser-based email system, such as any version of Microsoft's Internet Explorer older than version 4.0. However, Internet Explorer version 4.0 and higher includes Microsoft's Exchange, making it compatible with the **File | Send Mail** command.

Alternatively, use the **eTransmit** command to package the drawing with all support files.

141. progeCAD automatically saves your work at a time interval you specify. From the **Tools** menu, choose **Options**, click the **General** tab, and then set the **Autosave** time.
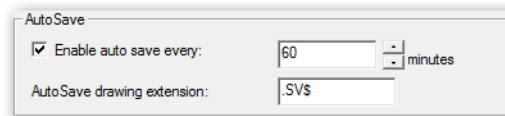
progeCAD for AutoCAD Users

autoCAD is popular, but it also continues to become more expensive. Over time, its price has quadrupled from the original $1,000 price tag to today's $4,000. And so we have a software package, progeCAD, that provides some 90% of AutoCAD's drafting capabilities for 15% of the cost. (For many 2D drafters, the missing 10% are inconsequential.)

This chapter describes the similarities and the differences between progeCAD and AutoCAD. Terminology between the two is virtually identical; the only significant difference is progeCAD's use of the words "esnaps" or "entity snaps" in place of AutoCAD's "osnaps" and "object snaps."

In This Chapter

- User Interface Comparison
- Other Elements, and Right-click Menus
- Command Name Differences
- Commands Enhanced in progeCAD
- Reading and Writing DWG Files
- Customizing and Programming progeCAD
- Menu Bars, Toolbars, and Status Bars
- Commands and System Variables
- Additional Commands & System Variables
- AutoCAD Commands Missing from progeCAD
- progeCAD-AutoCAD Dictionary
- Folder Structure

AutoCAD 2009 (Legacy Workspace)
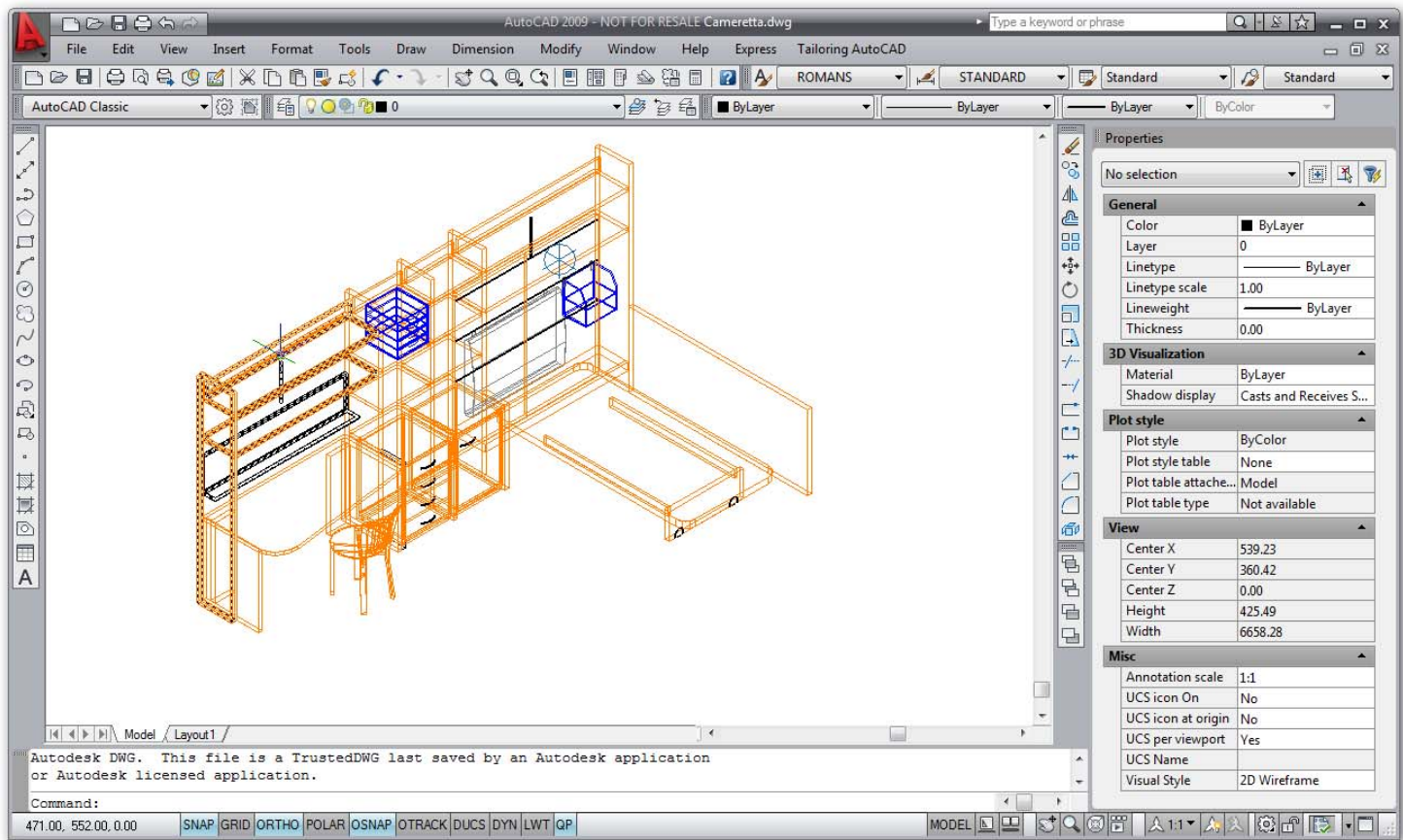
AutoCAD and progeCAD Share Similarities

Many aspects of AutoCAD and progeCAD are the same. The list includes commands, system variables, the user interface, status bar, toolbars, menus, fonts, linetypes, hatch patterns, toolbar and menu macros, scripts, LISP and VBA programming, *.dwg*, *.dxf*, and *.dwf* file formats, 3D and ACIS, paper space and layouts, and UCSs.

For example, commands and system variables are entered identically. To draw a line in progeCAD, you enter the **Line** command and press **Enter** — or enter the **l** alias, or click the Line button on the Draw 2D toolbar — just as in AutoCAD.

Some aspects differ, and that's what this chapter concentrates on.

part i: introducing progeCAD 6

progeCAD 2009
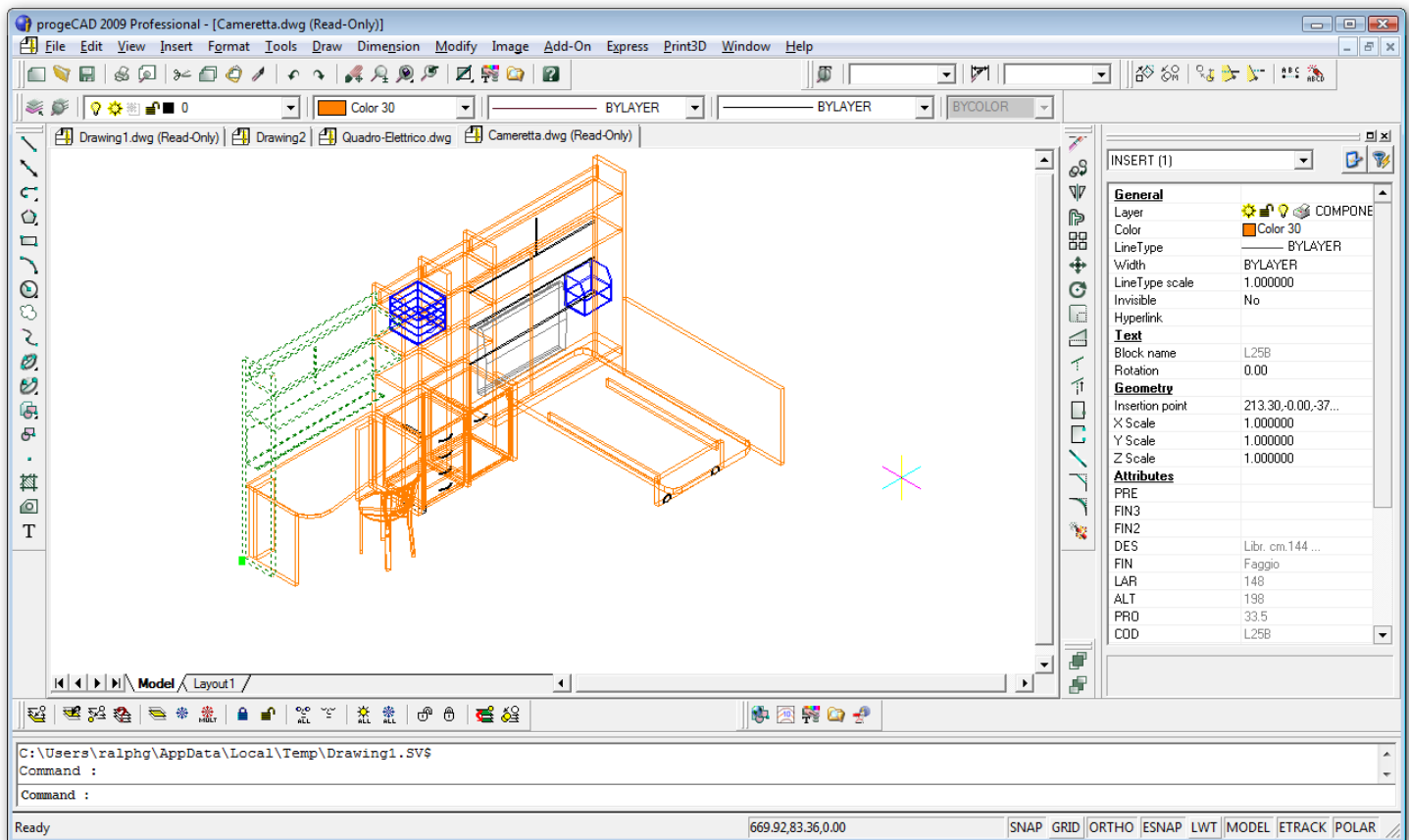
User Interface Comparison

The user interfaces of both CAD programs are similar, and recent changes in progeCAD make it look and act even more similarly to AutoCAD.

Comparison of Menu Bars

AutoCAD and progeCAD's menu bars are similar, but not identical — as illustrated below.

Menu bar from AutoCAD 2009:

Menu bar from progeCAD 2009:

The menu bars, menus, and macros of both programs can be modified (customized). Thus, it is possible to create a progeCAD menu that looks identical to that of AutoCAD.

Comparison of Toolbars

The buttons on AutoCAD and progeCAD's toolbars use icons that differ from each other. In the figure below, compare the difference in **New** icon (top row, far left) between the two.

Some of the toolbars in AutoCAD 2009:

Some of the toolbars in progeCAD 2009:
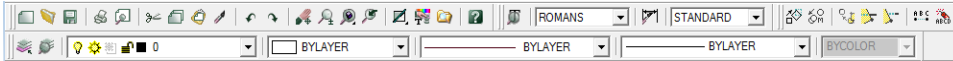
The two programs sometimes place buttons in different toolbar groupings. The toolbars, icons, and related macros of both programs can be customized. As with menus, it would be possible to create exactly AutoCAD-like toolbars for progeCAD.

Comparison of Status Bars

AutoCAD and progeCAD's status bars show similar information, although AutoCAD's has many more buttons. progeCAD displays help text together with the other information fields; in contrast, AutoCAD replaces the entire status bar with help text.

Status bar for AutoCAD 2009:
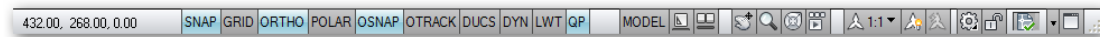
Status bar for progeCAD 2009:
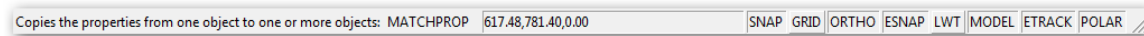
Both use a single-click to change toggles (progeCAD no longer requires a double-click). AutoCAD's buttons turn blue when turned on; progeCAD's look depressed.

progeCAD does not have AutoCAD's "tray" found at the far right end of the status bar, but it has a greater number of useful options for establishing the information displayed by the bar.
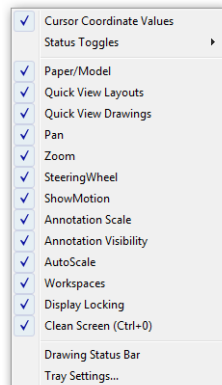
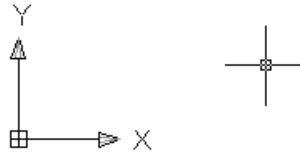*AutoCAD 2009 status bar options:* *progeCAD 2009 status bar options:*
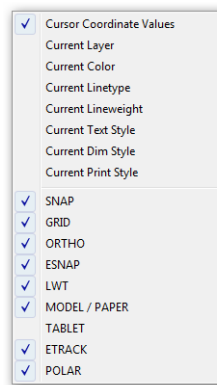
Both programs can customize the status bar with the Diesel macro language.

Comparison of Other User Interface Elements

AutoCAD and progeCAD share similar user interface elements with differences:

- **Cursor** — the size of the cursor can be changed in both programs. Both can display a different color for each axes, something AutoCAD cloned from progeCAD. The cursors of both programs show the pickbox (entity selection area) and the aperture (object snap search area).
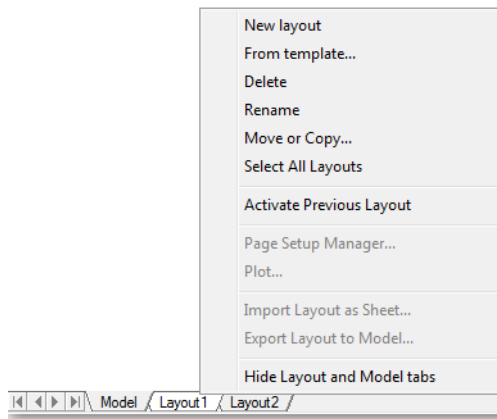

*AutoCAD 2009's cursor and UCS icon:*

progeCAD 2009's cursor and UCS icon:

- **UCS** — both programs optionally display the UCS (user-defined coordinate system) icon in the left hand corner. AutoCAD's can be modified for style, size, and thickness; progeCAD's can be modified for the color of each axis.

- **Scroll Bars** — both programs optionally display scroll bars for panning the drawing view.

- **Layout Tabs** — both programs have Model and Layout tabs at the bottom of the drawing area. AutoCAD's tabs have more options than do progeCAD's; right-click a tab to see either shortcut menu.

AutoCAD 2005 layout options: *progeCAD 2009 layout tab options:*
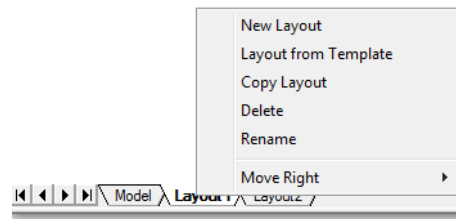
## Comparison of Right-click Menus

AutoCAD and progeCAD always show cursor (shortcut) menus when you press the right mouse button, but sometimes they show different content from each other. It's not possible to show all of the menus here, so I'll illustrates the differences with just a few examples.

Hold down the **Shift** key and then right-click. The resulting shortcut menu lists entity (object) snap settings. The menus are similar, but different:
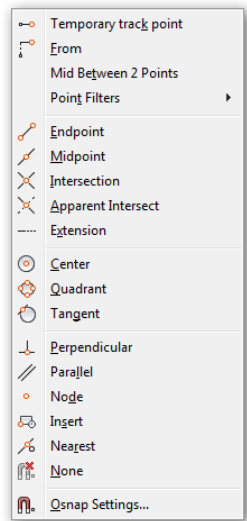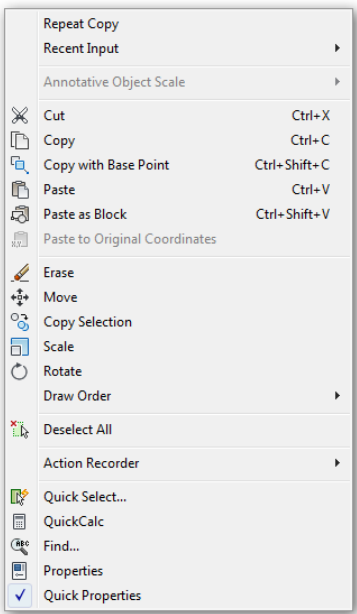
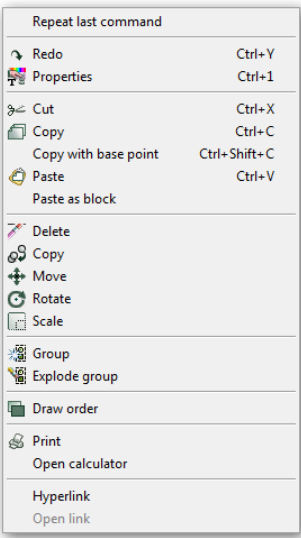*AutoCAD 2009 shortcut menu:*   *progeCAD 2009 shortcut menu:*
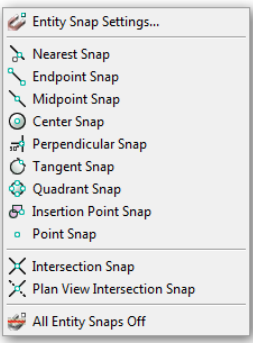
Similarly, right-clicking a selected entity shows shortcut menus with some editing commands the same, and some different:

AutoCAD 2009 shortcut menu: *progeCAD 2009 shortcut menu:*

Commands and System Variables

This section describes some of the differences of commands and system variables between AutoCAD and progeCAD. Because both programs are constantly updated, these lists are not exhaustive.

Command Name Differences

Some progeCAD commands have names that differ from those in AutoCAD. Square brackets indicate equivalent keystrokes, such as [Ctrl] and [F6].

progeCAD	AutoCAD	Action in progeCAD differing from AutoCAD
Cal	Cal	Displays Windows calculator.
Coordinate	[Ctrl]+[D] *or* [F6]	Changes display of coordinates on status line.
Customize	Toolbar	Displays Customize dialog box.
ESnap	-Osnap	Sets entity snaps at the command bar.
ExpBlocks	AdCenter *or* Insert	Displays Explorer - Blocks.
ExpDimstyles	AdCenter *or* DimStyle	Displays Explorer - Dimension Styles.
ExpFonts	AdCenter *or* Style	Displays Explorer - Styles.
ExpLayers	AdCenter *or* Layer	Displays Explorer - Layers.
Explorer	AdCenter	Displays Explorer.
ExpLTypes	AdCenter *or* Linetype	Displays Explorer - Linetypes.
ExpUcs	DdUcs	Displays Explorer - UCS.
ExpViews	View	Displays Explorer - Views.
NewWiz	New	Runs New Drawing wizard.
RtRot	3dOrbit	Rotates view of entities.
RtRotX	3dOrbit	Rotates view of entities about x-axis.
RtRotY	3dOrbit	Rotates view of entities about y-axis.
RtRotZ	3dOrbit	Rotates view of entities about z-axis.
SetESnap	OSnap	Displays Drawing Settings dialog box.
StopScript	[Backspace]	Stops running the script.

Additional progeCAD Commands

progeCAD includes additional commands not found in AutoCAD. Some of these include:

Command	Action in progeCAD
CmdBar	Positions the command window.
CopyEData	Copies extended entity data from one entity to another.
DelEData	Deletes extended entity data from an entity.
EditEData	Edits extended entity data found in an entity.
Flatten	Sets thickness to zero at user- specified elevation.
MoveEData	Moves extended entity data from one entity to another.
QPrint	Quickly prints the current viewport (window) with no options.
ReassocApp	Re-associates extended entity data with an application.
RecScript	Starts the Script Recorder.
TipOfDay	Displays Tip of the Day dialog box.
SaveAll	Saves all currently open drawings.
Scrollbar	Toggles the scrollbars.
SelGrips	Select objects to display grips.

StatBar	Toggles the status bar.
StopScript	Stops recording the script.
VmlOut	Exports drawings in VML format.

Additional progeCAD System variables

progeCAD includes additional system variables not found in AutoCAD. Some of these include:

Command	Action in progeCAD
AutoMenuLoad	Determines whether default menu or menu associated with drawing is loaded.
BaseFile	Name of default prototype drawing used to create new drawings.
CmdLnText	Specifies the command bar prompt.
ColorX	Color of x axis in the cursor and UCS icon.
ColorY	Color of y axis in the cursor and UCS icon.
ColorZ	Color of z axis in the cursor and UCS icon.
ExplStBlk	Column width of block settings in Explorer.
ExplStLay	Column width of layer settings in Explorer.
ExplStLt	Column width of linetype settings in Explorer..
ExplStTs	Column width of text styles settings in Explorer.
ExplSstUcs	Column width of UCS settings in Explorer.
ExplStVw	Column width of named view settings in Explorer.
IcadVer	Reports the progeCAD version number.
LastCmdPt	Stores the last point entered relative to the current UCS.
LastPrompt	Stores the last prompt string displayed by command bar.
LtSclWblk	Specifies whether global linetype scale is applied to inserted blocks.
MacroRec	Toggles macro recorder.
MacroTrace	Toggles macro tracer.
MakeBak	Renames drawing as *.bak* with each save.
NFileList	Number of most recently used files in File menu.
ProgBar	Toggles display of the progress bar.
PromptMenu	Toggles display of the prompt menu.
SaveRoundtrip	Determines whether unknown entities are saved R14 format.
ScrlHist	Specifies number of command lines to track in command bar.
SrchPath	Specifies the search path for support files.
TbColor	Switches toolbar color between color and monochrome.
TbSize	Switches toolbar size between small and large.
Tipofday	Toggles Tip of the Day dialog box.
ViewAspect	Aspect ratio of the view.
WndBtExp	Specifies width of the list pane in Explorer.
WndLCmd	Toggles command bar visibility.
WndLMain	Toggles display of the main window.
WndLMdiW	Controls display of MDI child window.
WndLStat	Toggles display of the status bar.
WndLtExp	Toggles display of the Explorer window.
WndLText	Toggles display of the prompt history window.
WndPCmd	Screen coordinates of the command bar when floating.
WndPMain	Screen coordinates of drawing window position.
WndPPmenu	Screen coordinates of the prompt menu.
WndPtExp	Screen coordinates for Explorer window.
WndPText	Screen coordinates for prompt history window.
WndSCmd	Size of command bar, when floating.

```
WndSMain          Size of the drawing window.
WndStExp          Size of the Explorer window.
WndSText          Size of the prompt history window.

ZoomPercent       Incremental zoom percentage.
```

Commands Enhanced in progeCAD

Some of progeCAD's commands provide more options than in AutoCAD. Here is a selection of them.

AutoCAD's **MSlide** command saves the image in Autodesk's proprietary SLD format. In progeCAD, the command also saves the image in Microsoft's WMF (windows meta file) or EMF (enhanced meta file) formats.

AutoCAD's **Offset** command copies to one side at a time. In progeCAD, the command's **Bothsides** option copies to both sides at once.

AutoCAD's **Solid** command draws solid quadrilaterals and triangles. In progeCAD, the command's **Rectangle** option draws solid-filled rectangles; **Plane Square** draws solid-filled squares; and **Plane Triangle** draws solid-filled equilateral triangles.

progeCAD's **Circle** command's **Arc** option converts arcs to circles. AutoCAD does this through its **Join** command.

In progeCAD, the **Line** command's **Angle** option specifies the angle, followed by a length; similarly, the **Length** option specifies the length, followed by an angle. AutoCAD can do this only through direct distance entry.

progeCAD's entity (object) snaps can be entered directly at the command prompt. AutoCAD's requires that its -**OSnap** command be employed.

progeCAD's **Select** command has additional selection options not found in AutoCAD:

Select		Meaning
CC	Crossing Circle	All entities within and crossing a circular area.
O	Outside Window	Selects all entities outside a rectangular window.
OC	Outside Circle	Selects all entities outside a circular area.
OP	Outside Polygon	Selects all entities outside a polygonal area.
PRO	Properties	Selects all entities with specific properties.
WC	Window Circle	Selects all entities within a circular area.

As well, the + and - keys can be used to add and remove objects from selection sets.

AutoCAD Commands Missing from progeCAD

Each release of AutoCAD adds more commands, some of which progeCAD also adds, but some of these new ones are not added. In addition, there are two versions of progeCAD: Standard and Professional; the Professional edition has commands not found in the Standard version.

progeCAD reads and writes drawings created by AutoCAD through to 2009. Because progeCAD does not support everything in AutoCAD, some objects are difficult to handle. Tables, for instance, are displayed but as anonymous blocks and cannot be edited in progeCAD.

The addition of ObjectARx to AutoCAD Release 13 enabled Autodesk and third-party developers to create "custom objects" that can be created and edited only when the associated ObjectARx programming code is present. When the code is missing, then a *proxy* object is displayed instead. Autodesk provides *object enablers* that provide sufficient code to edit custom objects. progeCAD converts proxy entities into blocks using their exclusive proxy conversion technology built into the software.

progeCAD-AutoCAD Dictionary

This brief progeCAD-AutoCAD dictionary explains some progeCAD jargon:

progeCAD Term	AutoCAD Equivalent
command bar	command prompt area
edit length	lengthen
entity	object
entity snap	object snap
entity snap precision	aperture
edata	xdata
fixed attribute	constant attribute
follow	continue
freehand	sketch
hidden attribute	invisible attribute
infinite line	XLine
insert	draw
orthogonal	ortho
parallel	offset
plane	solid (2D)
point snap	node snap
predefined attribute	preset attribute
prompt box	context menu
prompt history window	text screen
reference grid	grid
snapshot	slide
validate attribute	verify attribute

Customizing and Programming progeCAD

progeCAD works with most of the support files used by AutoCAD, including *.lin*, *.pat*, *.mnu*, *.fmp*, *.unt*, *.dcl*, *.lsp*, *.shx*, and *.sld* files.

progeCAD mimics AutoCAD in writing menu and toolbar macros, scripts, DCL (dialog control language), LISP routines, i-drop, and VBA applications (visual basic for applications).

Unlike present-day AutoCAD, progeCAD can work with programs that use the SDS API (solutions development system applications programming interface), called "ADS" in AutoCAD until it was removed with Release 13.

AutoCAD's rarely-used side-screen menu is not included in progeCAD; prompt menus replace some of the functions provided by the side-screen menu.

progeCAD also works with Diesel.

Introducing the Boardwalk

to help you learn progeCAD, this tutorial walks you through a fictional condominium design project called "The Boardwalk on 18th Avenue." For this project, you are to produce the floor plans for one-bedroom condo units.

The design project goes through the following stages:

1. Draw the *exterior walls*; these enclose the unit.
2. Define the *interior spaces*; these include the living room, bedroom, and closets.
3. Determine the *access points and views*, such as doors and windows.
4. Specify the *fixed amenities*, such as bathroom fixtures, kitchen arrangement, and fireplace. Also specify the *non-fixed amenities*, such as living room seating, dining table, and bed.
5. Label important areas with text.
6. Add border and title block.
7. Apply dimensions to show sizes.
8. Create a printout of the floor plan drawing.

The following chapters describe how to work on the drawings for each stage using progeCAD.

About The Condominium Project

The Boardwalk on 18th Avenue is a low-rent, one-story condominium development consisting entirely of one-bedroom apartments. (A *condominium* is an apartment or flat that is purchased, and not rented or leased.) The development takes up half a city block, and so is surrounded on three sides by streets.

Development on the site is limited by the city's bylaws, which define the *building lines* and parking requirements. No part of the structure can be beyond the building lines; a pair of units are positioned in the illustration below. (P/L indicates the property lines.)
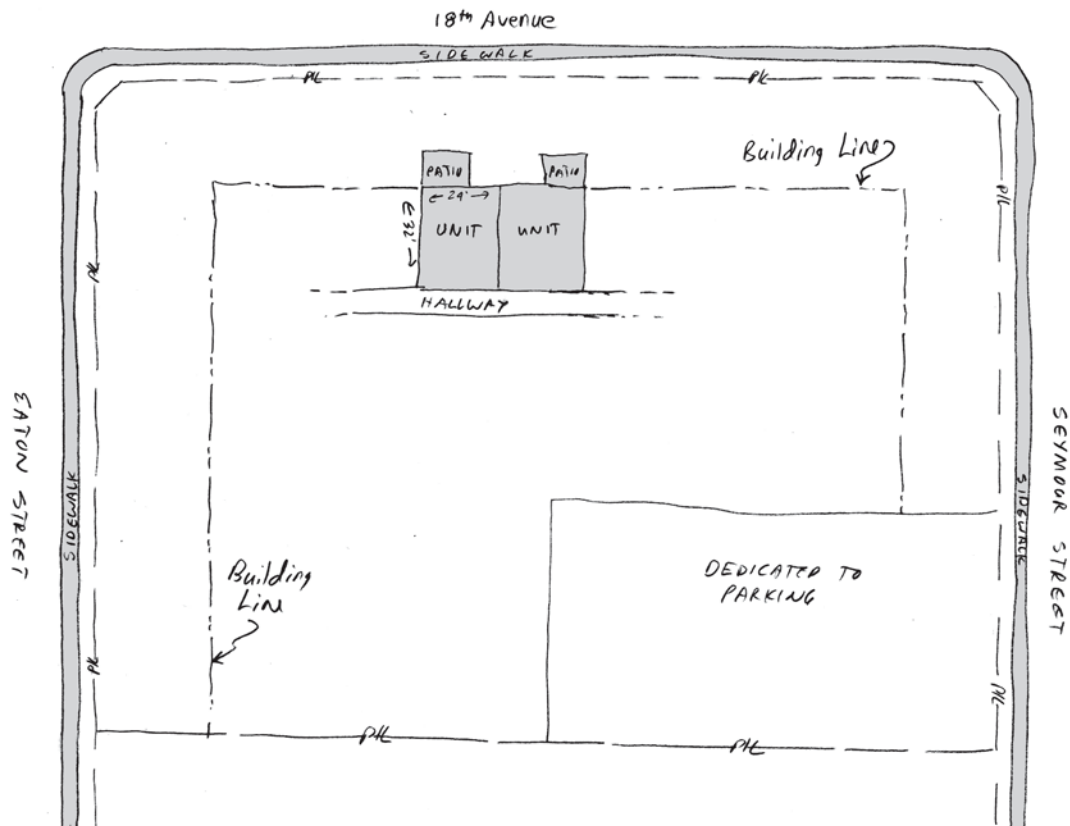
Each unit is the same: 32 feet deep by 24 feet wide (768 square feet). Inside the unit, the layout is *open plan*, meaning the kitchen, dining room, and living room are open to one other, with no full walls between.

Each unit consists of the following items:

- Kitchen.
- Dining room.
- Living room with fireplace.
- Three-piece bathroom with sink, toilet, and bathtub.
- Bedroom with closet and ensuite bathroom. (An ensuite has a sink and toilet, and is accessed only from the bedroom.)
- Hallway with coat closet and laundry closet (stores the hot water tank).

Residents access their units from an outside door or from the central hallway. Outside the living room is a 10' x 14' concrete slab patio.

A rough sketch of the layout of a unit is illustrated below.
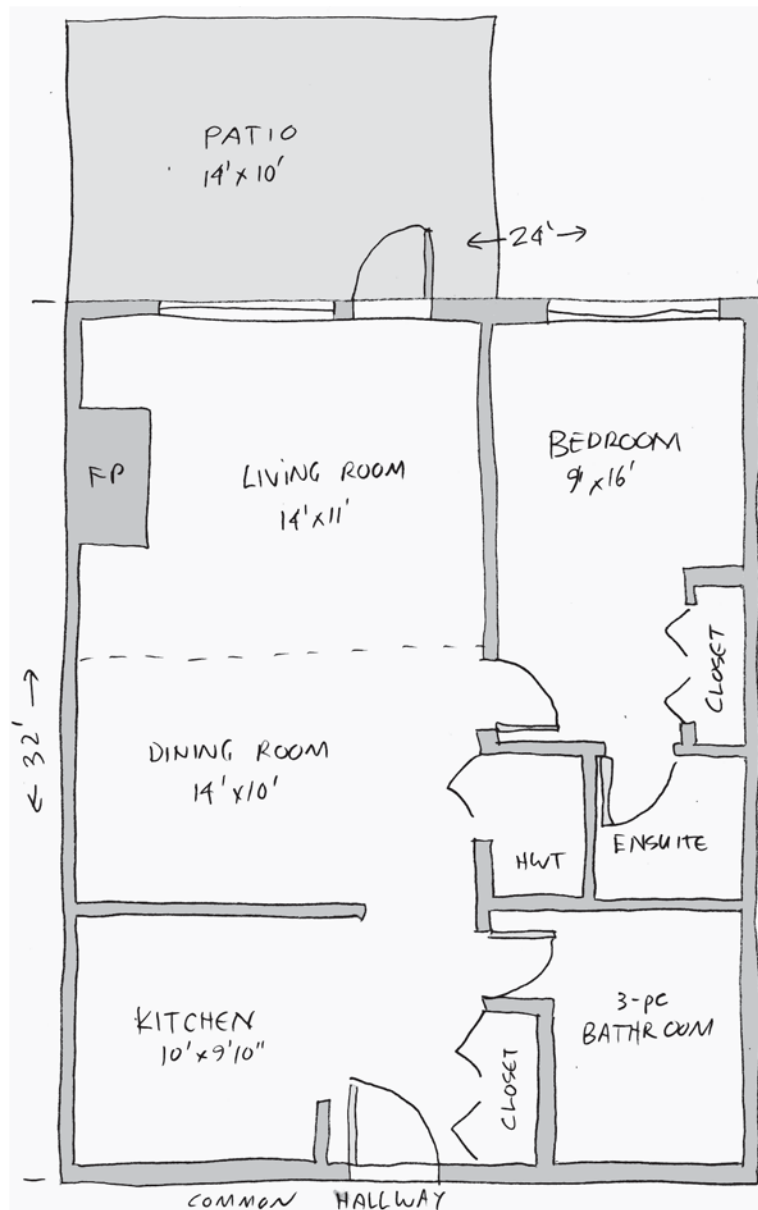
Every other unit is a mirror image of this one, so that similar facilities adjoin each other: bathrooms with their plumbing needs are adjacent; (quieter) bedrooms have common walls, as do the (nosier) living rooms and kitchens.

As the drafter for this project, your task is to draw the 2D floor plan of one unit. You use layers to segregate information, and blocks to place common objects, such as doors and chairs. You must place dimensions to show the size of important parts of the drawing, and add text to explain functions. You plot the drawings to scale so that your client can review them.

The completed progeCAD drawing should look something like this:
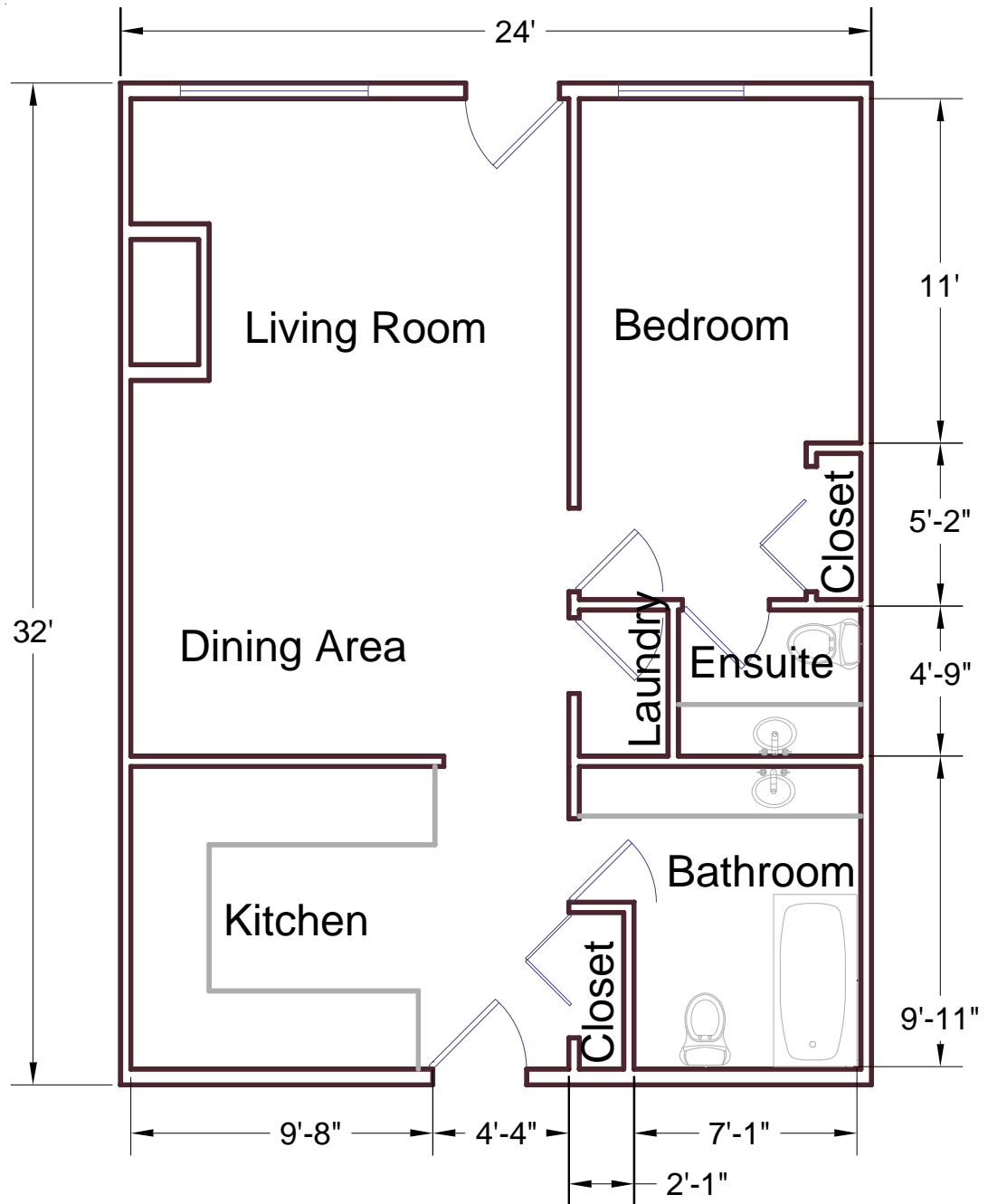
Setting Up New Drawings

before drawing even one line, you must first prepare new drawing files with suitable settings. For this tutorial, you are creating a medium-size architectural drawing, and so in this chapter you prepare the new drawing file for architectural drafting.

To help you prepare the drawing, you use two sets of tools:

- New Drawing Wizard sets up some parameters.
- Additional commands set up the remaining parameters.

In This Chapter

- New Drawing Wizard
- Setting Limits
- Toggling Ortho Mode
- Calculating Scale Factors
- Naming (and Saving) Drawings
- Specifying Snap & Grid Spacing
- Selecting Entity Snap Modes
- Alternative Method: Settings Dialog Boxes

New Drawing Wizard

To set up new drawings, you can use the New Drawing wizard, as follows:

1. Start progeCAD. Notice that it opens with a blank, new drawing.

2. To set up the drawing for architectural drafting with the New Drawing wizard. From the **File** menu, select **New**.
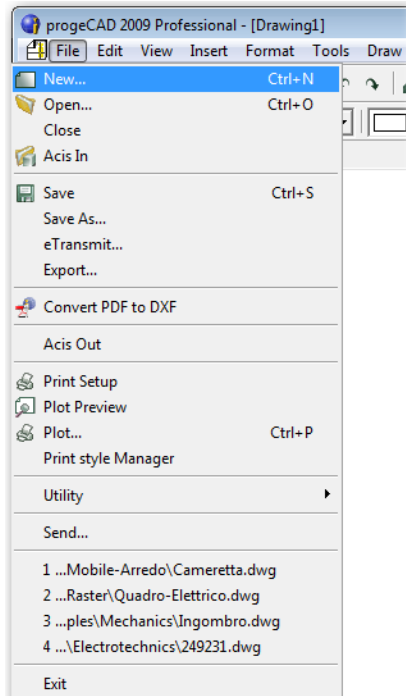
(Alternatively, you can enter **NewWiz** at the command prompt.)

3. *Wizards* are multi-page dialog boxes that take you through a complex process step by step. The New Drawing Wizard takes you through the steps to specify settings for new drawings. It presents a variety of options.
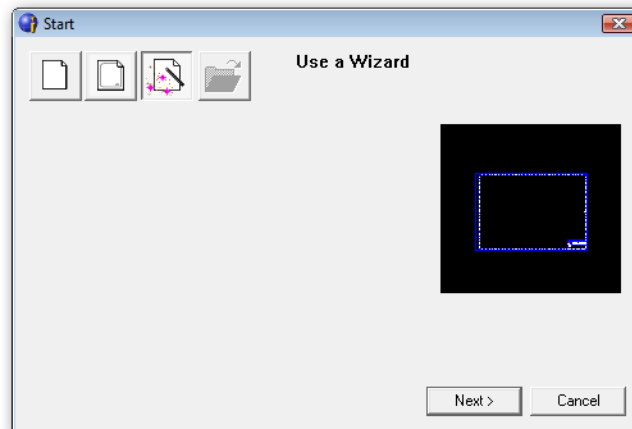
The wizard does not, unfortunately, provide all the settings needed for this tutorial, so you set additional options later.

In the first page asks about the basis for the new drawing. You have two choices:
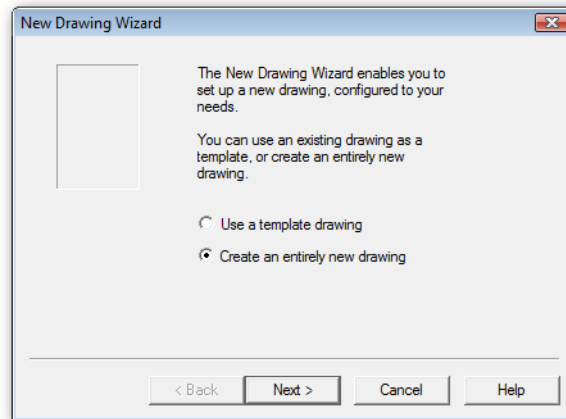
○ **Use a template drawing** — the new drawing is based on an existing drawing known as a "template." progeCAD copies all the settings from the template file. This option is a good pick when your office has already created *.dwt* template files.

(If you were to choose this option, the wizard's next page would ask you to select the name of the template file.)

⊙ **Create an entirely new drawing** — the new drawing is based on *icad.dwt*, a blank drawing that contains progeCAD default settings.

Select the "entirely new drawing" option, and then click **Next** twice in a row.

4. progeCAD lets you work with five types of linear measurement units. "Linear" refers to distances, as opposed to angles.
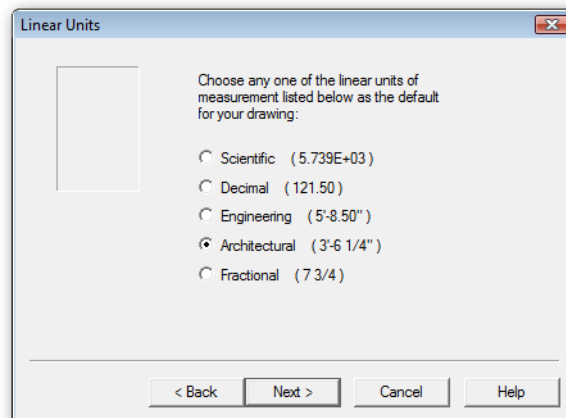
○ **Scientific** – displays scientific notation, such as 5.739E+02.

○ **Decimal** – displays unitless decimals, such as 573.9. This is also used for drawings created in metric units.

○ **Engineering** – displays feet and decimal inches, such as 5'-7.39".

⊙ **Architectural** – displays feet and fractional inches, such as 5'-7 3/4".

○ **Fractional** – displays fractional inches (no feet), such as 67 1/4".

Select **Architectural** units, and then click **Next**.

5. progeCAD lets you work with five kinds of angular measurement units.
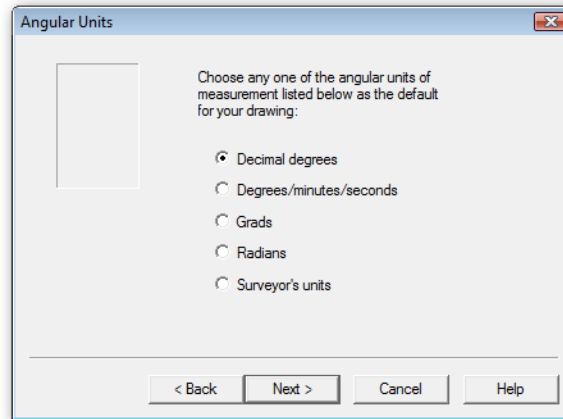
○ **Decimal degrees** – displays degrees and decimals, such as 5.739°.

○ **Degrees/Minutes/Seconds** – displays degrees with minutes, seconds, and decimals of seconds, such as 5° 73' 29.33".

○ **Grads** – displays grad and decimals, such as 234.55g. There are 400 grads in a circle; grads are used in Germany.

○ **Radians** – displays radians and decimals, such as 57.34r. There are 2*pi radians (approximately 6.282) in a circle; radians are used by scientists and programmers.

○ **Surveyor's Units** – displays angles relative to North, such as 67 1/4".

Select **Decimal degrees**, and then click **Next**.

6. The next step is to decide whether you want to used *named style tables*.
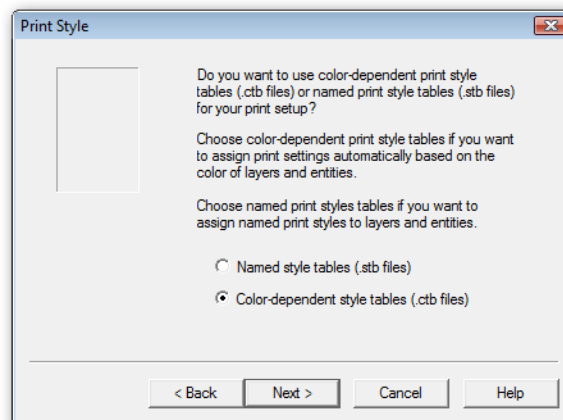
These allow you superb control over the plotted look of every entity in the drawing.

For our drawings, however, the older, color-dependent style tables are just fine.

○ **Named style tables (.stb files)** – controls the look of every entity, such as thickness, grayscale, line ends, and more; applies named styles to entities and layers.

◉ **Color-dependent style tables (.ctb files)** — only the color of entities can be used to affect the plotted output.

Both files — *.stb* (short for "style table") and *.ctb* ("color table") — store settings; both are compatible with AutoCAD.

Select **Color-dependent style tables**, and then click **Next**.

7. The final page of the wizard calls itself "entity creation," but deals with more than just that.
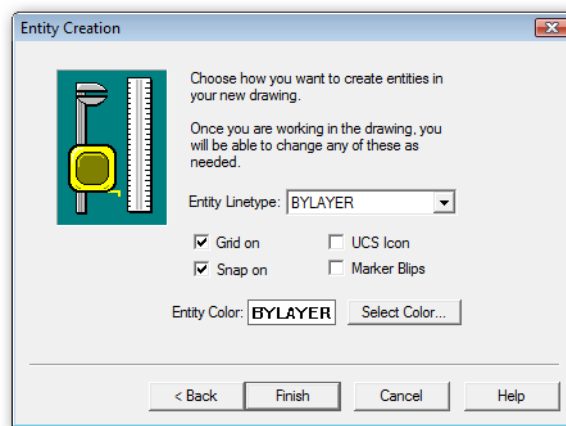
The meaning of the options are:

Entity Linetype specifies how entities are assigned linetypes. This is an odd question to ask when defining a drawing globally, in my opinion. In any case, always select **ByLayer** from the options in the droplist:

- **ByLayer** means that layers define the linetype for all entities residing on that layer.
- **ByBlock** means that blocks define the linetype for all entities in the block.
- **Continuous** means that entities are defined on a by-entity basis. In this case, the default is continuous (unbroken) lines.

☑ **Grid on** displays the grid, a matrix of dots that help you visualize relative sizes in drawings. Turn this option **on**.

☑ **Snap on** turns on the cursor increment, the distance the cursor jumps. Instead of moving freely, the cursor can be made to move in increments of one inch, a foot, a quarter-inch, whatever. Turn this option **on**.

☐ **UCS Icon** displays the UCS (user-defined coordinate system) in the drawing. This is useful for determining your 3D position in space, but is pointless for 2D drawings. Turn this option **off**.

Alternative Method: Settings Dialog Box

For setting up new drawings, there is an alternative method to the New Drawing wizard or using commands. You can use the Settings dialog box to adjust many settings. From the **Tools menu**, select **Drawing Settings**. (Alternatively, enter the **Settings** command.)

Drawing Units Tab

The Drawing Units tab specifies settings for linear and angular units, and user-defined coordinate systems. Select categories from the **Change Settings For** droplist.
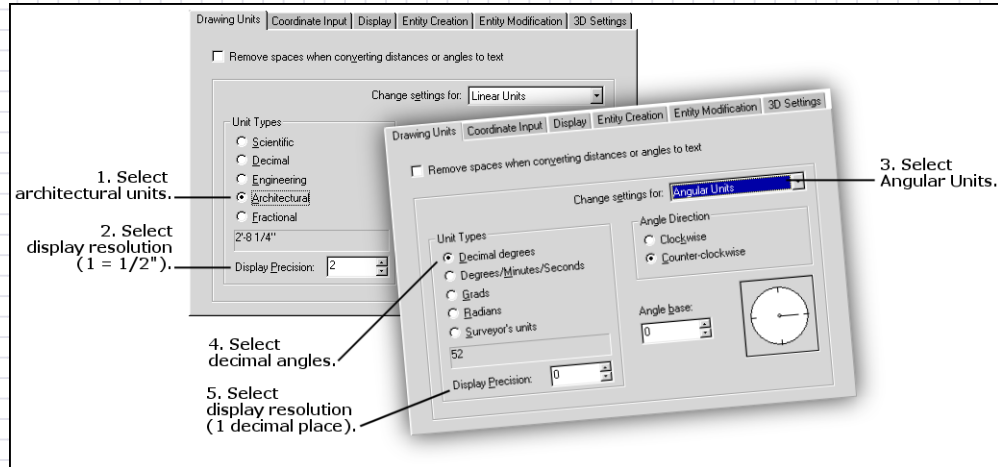
Coordinate Input Tab

The Coordinate Input tab specifies settings for snap and grid, limits, entity snaps, and entity selection.
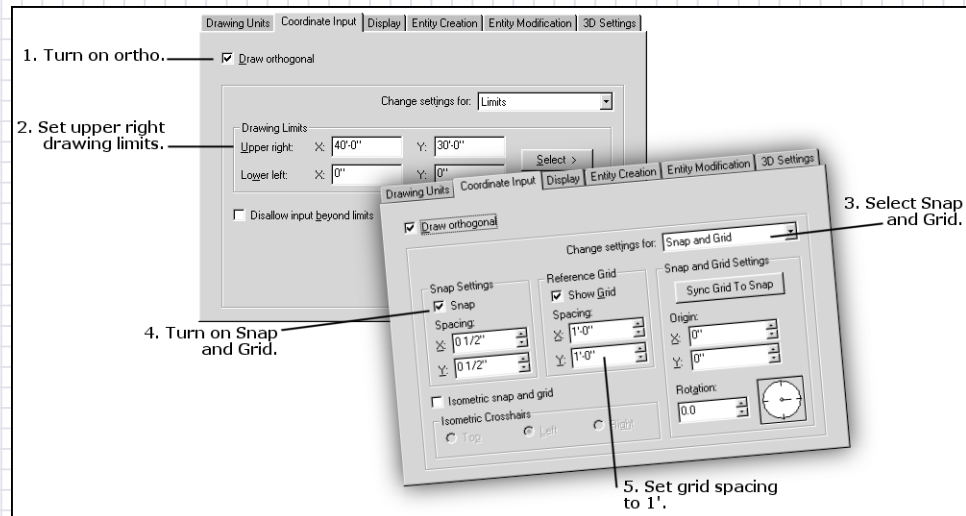
☐ **Marker Blips** displays small + marks each time you click a point in the drawing. These blips can be helpful to determining where you clicked, but after a while clutter the drawing with otherwise-useless markers. (You clean the blips off drawings with the Redraw command.) Turn this option **off**.

Entity Color specifies the default color for entities in the drawing. As with linetypes, I find this an odd question; the best option is **ByLayer**. If you want, however, you can click the **Select Color** button and pick a default color. You have these two options:

- **ByLayer** means that layers define the color for all entities residing on that layer.
- **ByBlock** means that blocks define the color for all entities in the block.

8. Click **Finish**. progeCAD applies the settings you selected to the drawing.
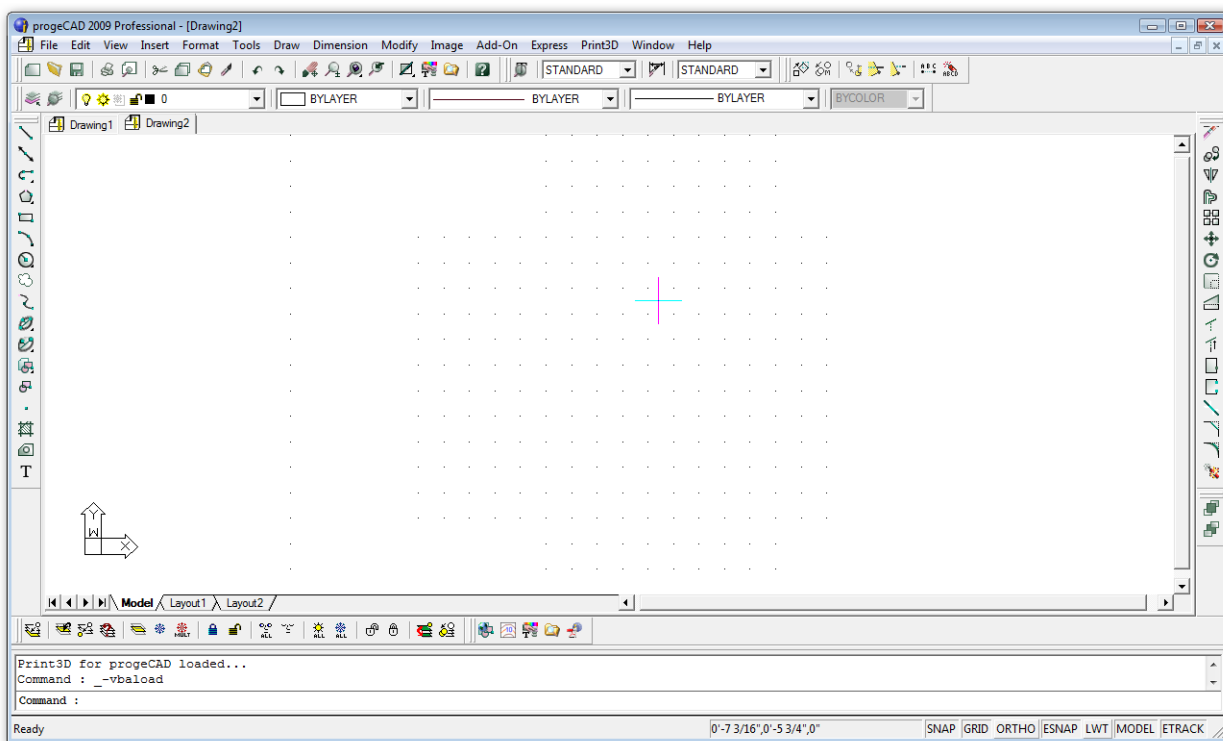
Notice that the grid is turned on and that the UCS icon is missing. On the status line, the SNAP and GRID settings are in black, meaning they are turned on. (If you cannot see the grid, use the **Zoom Extents** command, as follows:

```
: zoom
In/Out/All/Center/Extents/Left/Previous/Right/Window/Dynamic/<Scale (nX/nXP)>: e
```

There are more settings that every new drawing needs that are missing from the wizard. They include a scale factor, limits, ortho, snap and grid spacing, and the file name for the drawing. We'll tackle them after we save and name this drawing.

part ii: the tutorial projects

Naming (and Saving) Drawings

At this point, the drawing is devoid of entities, but still contains data. It is crucial that you save your work so that you don't lose fruits of your labor when Windows crashes your computer. In addition to you manually saving your work, progeCAD automatically saves your drawing every 60 minutes.

The first time you save a drawing, progeCAD asks you to give it a name. After that, the drawing is saved by the same name until you chose to change it.

Use the **Save** command to save drawings and give them a name.

1. From the **File** menu, select **Save**.

   (Alternatively, enter the **Save** command at command prompt.)

2. The Save Drawing As dialog box presents numerous options:
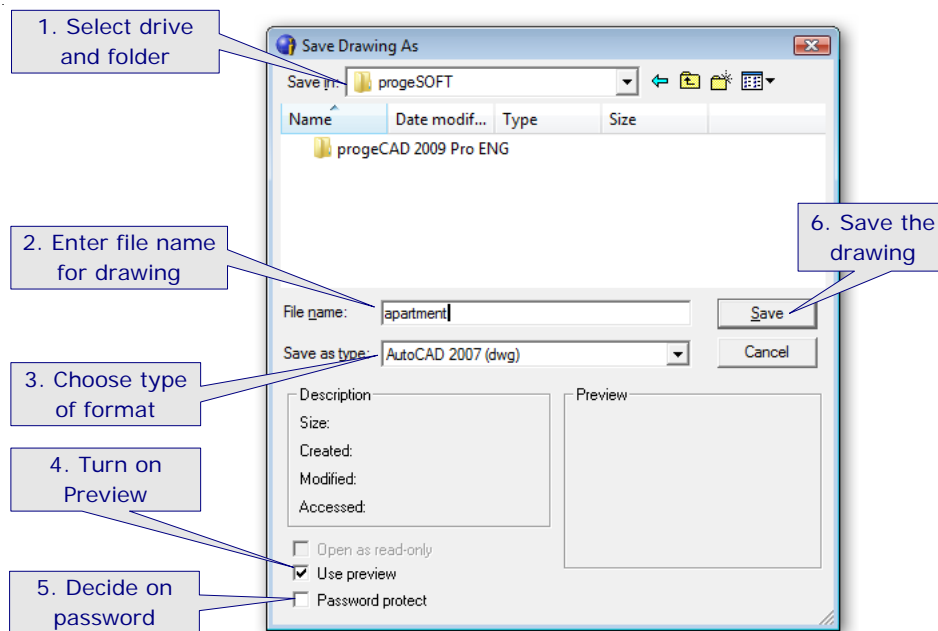
Save in specifies the name of the drive and folder in which drawings are to be saved. You can also access *networked drives*, which are drives on computers connected to yours through a network. Click the **Save In** droplist, and then select a drive and folder for this drawing.

File name identifies drawings by names of up to 255 characters. Firms often have standards for naming drawings logically; for this tutorial, use the generic name "apartment."

Save as type specifies the format in which to save the drawing. progeCAD supports the following AutoCAD-compatible formats:

- DWG (standard drawings).

- DXF binary.
- DXF ASCII.
- DWT (template drawings).
- Formats compatible with older releases of AutoCAD and progeCAD.
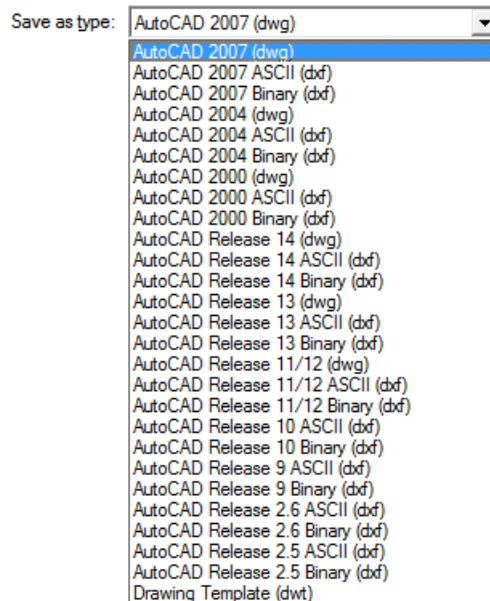
Unless you understand the reason for using another format, select "AutoCAD 2005 2006 Binary (dwg)."

☑ **Use Preview** creates a preview image of the drawing. This is useful when browsing through drawings, in deciding which one to load. Turn this option **on**.

☐ **Password Protect** prompts you to specify a password. Later, when opening the drawing again, you are asked for the password before it will be displayed. If you forget the password, you lose access to the drawing. Turn this option **off**.

3. Click the **Save** button to save the drawing, giving it the new name. You should see the new file name on progeCAD's title bar.

TIP progeCAD automatically backs up (saves) your drawings every 60 minutes. This isn't frequent enough, because you can do — and lose — a lot of work in an hour. To change the frequency of automatic backups, follow these steps:
   1. From the **Tools** menu, select **Options**.
   2. In the Options dialog box, select the **General** tab.
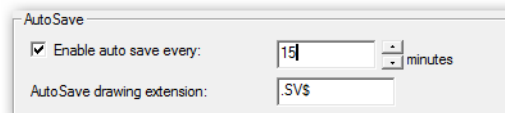
3. In the AutoSave section, change the **minutes** setting to 10 or 15.
   4. Click **OK**.

Setting Limits

Limits define the extents of the drawing: how big is the drawing horizontally and vertically? In practice, limits don't matter, because progeCAD can create drawings larger than our solar system. (Printing a drawing of that size might be a problem.)

Limits are useful for a different reason: they define the limits of the grid and the zoom-all display. This provides you with an idea of the working area.

For the tutorial drawing, the limits are the size of the condominium apartment, plus some extra space around the edges. The apartment is 32' x 24', so make the limits 40' x 30'. Limits are set with the **Limits** command.

1. From the **Tools me**nu, select **Drawing Limits**.

   (Alternatively, enter the **Limits** command.)

2. In the command bar, progeCAD asks you for the coordinates of the lower left corner. They are at 0",0" so just press **Enter** to accept the default values:

   ```
   Command: limits
   Limits are off:  ON/<Lower left corner> <0",0">:  (Press Enter.)
   ```


40',30'

Limits of the Drawing

0',0'
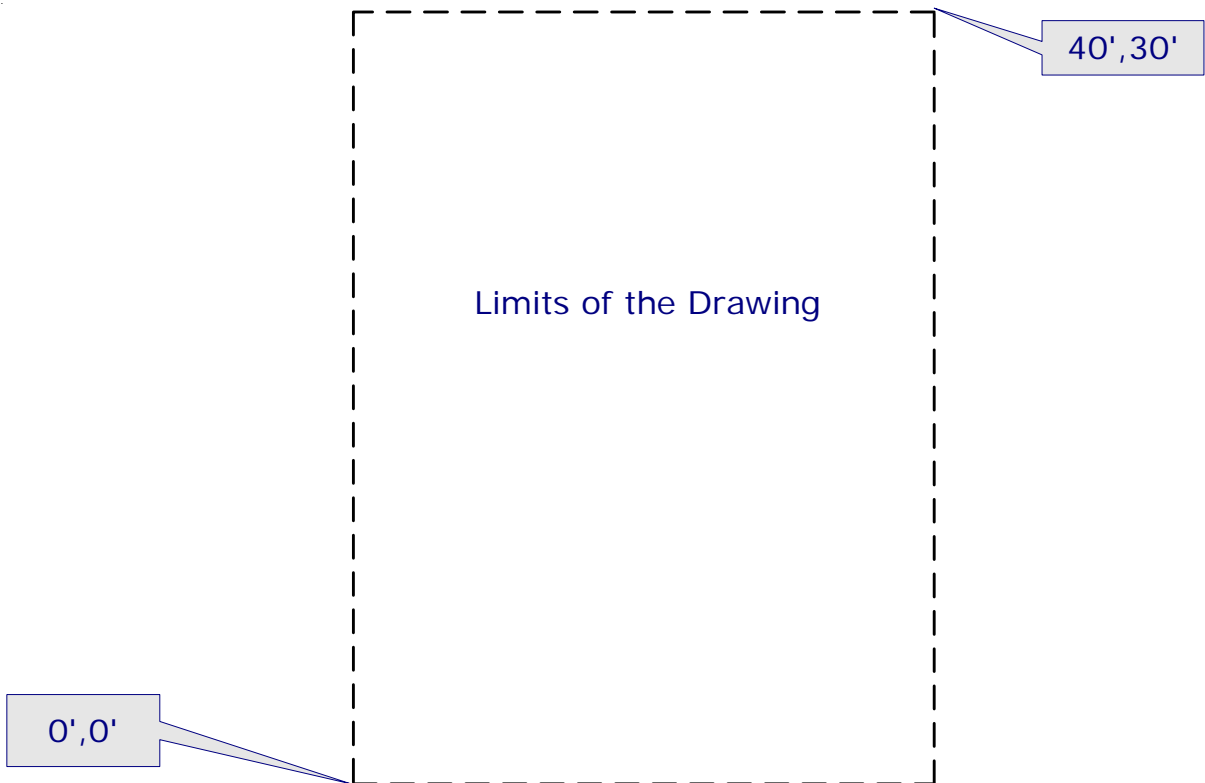
3. The upper right corner uses the coordinates specified earlier. Remember to include the apostrophe (') after each dimension, otherwise progeCAD thinks you are entering inches.

> Upper right corner <0'-12",0'-9">: **40',30'**

4. Again, use the **Zoom Extents** command to "see" the limits of the drawing. progeCAD most likely complains, "The grid spacing is too small to display in this view." You fix the problem in the next section.

Specifying Snap and Grid Spacing

The New Drawing wizard had preset the spacing between snap points and grid dots to 1/2". When you zoom out to the full extents of the drawing, progeCAD displays some 350,000 grid dots. That many is too many, making the screen look black.

When the grid becomes too dense, progeCAD refuses to display it; instead it displays a warning message in the command bar: "The grid spacing is too small to display in this view." There are two ways to deal with the warning message: ignore it, or change the grid spacing to larger values.

With the **Grid** command, change the grid spacing to 1', as follows.

1. From the **Tools me**nu, select **Reference Grid**.

   (Alternatively, enter the **Grid** command, or else right-click GRID on the status bar, and then select **Settings**.)

2. In the command bar, enter the spacing:

   > Grid is on: OFF/Snap/Aspect/<Grid spacing (x and y = 0 1/2")>: **1'**

   Notice that progeCAD displays the grid dot inside of the limits. Each dot is one foot from its neighbor.

3. The *snap spacing* determines how far the cursor moves. You can think of it as cursor resolution. When you draw the floor plan of this apartment, you will probably draw entities to the nearest 1/2" or so. By setting snap to 1/2", you make it easier to pick points with the cursor.

   Snap spacing is related to grid spacing. In fact, the two can share the same value. In practice, I find that the grid should have a larger value (like 1 foot or 1 meter), and the snap a smaller on, like 1/2" or 1 mm.

   The snap spacing had been set to 1/2" by the New Drawing Wizard. If you wish to confirm this, use the **Snap** command.

GRID COMMAND

Controls the grid display.
Command: **grid**
Alias: **g**
Menu: **Tools | Reference Grid**
Toolbar: **Tools | Reference Grid**
Shortcut: **F7**
Status Bar: **GRID**
Options: **On** or **Off**
          Snap
          Aspect
          Grid spacing

ALTERNATIVE

DdRModes specifies grid settings via dialog box.

SNAP COMMAND

Controls the snap increment.
Command: **snap**
Alias: **sn**
Menu: **Tools | Snap**
Toolbar: **Settings | Snap**
Shortcut: **F9**
Status Bar: **SNAP**
Options: **On** or **Off**
          Rotate
          Style
          Aspect
          Snap spacing

ALTERNATIVE

DdRModes specifies snap settings via dialog box.

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part ii: the tutorial projects

Toggling Ortho Mode

The drawing of the apartment is *rectilinear*, which means it consists mostly of straight lines drawn at right angles. progeCAD helps you draw straight lines at right angles with *ortho* mode, which is short for "orthographic" — another way of saying rectilinear.

To turn on ortho mode, use the **Ortho** command.

1. From the **Tools** menu, select **Draw Orthogonal**. Notice that the ORTHO button turns on the status line, and that progeCAD confirms the change in the command box.

> ORTHOMODE is currently off: ON/Toggle/<Off>: _T

Ortho mode is called a "toggle," because it only has two settings: on and off — just like a light switch. Because it is common to toggle ortho mode while drafting, there are several ways to turn it on and off:

- Press function key **F8**.
- Click **ORTHO** on the status bar.
- Hold down the **Shift** key.
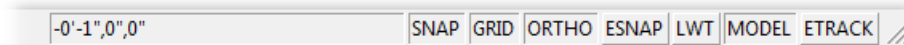
You can use the shortcuts to toggle ortho mode at any time, including during any command.

Selecting Entity Snap Modes

Entity snaps are among the most powerful tools in the progeCAD toolbox. Esnaps snap the cursor to geometric features, such as to the middle or ends of lines and arcs. They can snap to the center of circles and arcs, to the intersection of two entities, and to many more features.

Like ortho mode, you sometimes want entity snaps on, and sometimes off. You can toggle ensaps during commands by using these shortcuts:

- Press function key **F3**.
- Click **ESNAP** on the status bar.

For turning on individual entity snap modes, I find the dialog box easier than the command line.

ORTHO COMMAND

Controls the snap increment.
Command: **ortho**
Alias: **sn**
Menu: Tools | Draw Orthogonal
Toolbar: Settings | Draw Orthogonal
Shortcut: **F8**
Status Bar: **ORTHO**
Options: **On** or **Off**

ALTERNATIVE

DdRModes specifies the ortho setting via dialog box.

ESNAP COMMAND

Specifies entity snap modes.
Command: **esnap**
Aliases: **-os**, **osnap**, **-osnap**
AutoCAD: **-osnap**
Menu: **Tools | Entity Snap**
Toolbar: **Entity Snaps**
Shortcut: **F3** (toggle)
Status Bar: **ESNAP**
Mouse: **Shift**+right click
Options: **On**, **Off NEArest**, **ENDpoint**, **MIDpoint**, **CENter**, **PERpendicular**, **TANgent**, **QUAdrant**, **INSertion**, **POInt**, **INTersection**, **PLAnview**

ALTERNATIVE

DdRModes specifies the esnap setting via dialog box.

---

1. From the **Tools me**nu, select **Entity Snap**, and then select **Entity Snap Settings**.
    (Alternatively, you can enter the **ESnap** command, or else right-click ESNAP on the status bar, and then select **Settings**.)

2. In the Settings dialog box, click **Clear** to turn off all esnap modes. ("Esnap" is short for entity snaps.)
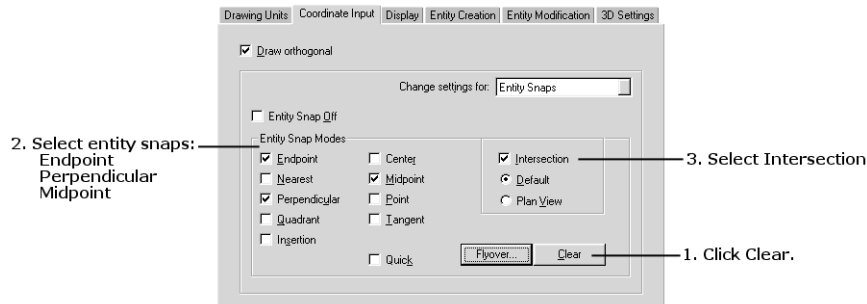
3. Now select the entity snaps that work best with rectilinear drawings, like our apartment floor plan:
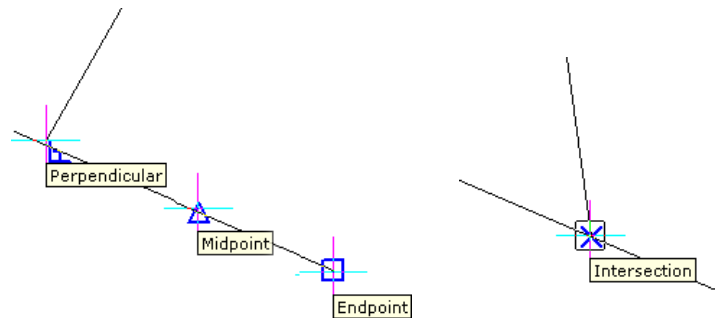
> ☑ **Endpoint** — snaps to the ends of lines, polylines, and arcs.
>
> ☑ **Perpendicular** — snaps at right angles to lines and circles.
>
> ☑ **Midpoint** — snaps to the midpoint of lines, polyline segments, and arcs.
>
> ☑ **Intersection** — snaps to the intersection of two lines, circles, arcs, other entities, and combinations thereof.

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

Calculating Scale Factors

One issue with CAD is that there are two kinds of *scale factors*. Normally, you draw everything "full size" in progeCAD. If a wall is 24 feet long, you draw it 24'-long in the drawing.

When it comes time to plot, the drawing must be scaled down to fit the paper. That 24-foot wall might be plotted just 8 inches long. The scale of the drawing is 24'=8", or 36:1. (That's 24' x 12 inches/foot = 8", or 288:8 = 36:1.) When plotted, everything in the drawing is 36 times smaller.

There is a problem, however. Some aspects of the CAD drawing cannot be 36x smaller. These include text, linetypes, hatch patterns, dimension text, and arrowheads. These items should be "full size" when plotted. That means they must be 36x larger when drawn in CAD, in order to turn out 1x on the printed page.

Text, for instance, is typically 1/4" tall in paper drawings. That means it should be 9" tall in the CAD drawing (1/4" x 36, the scale factor).

To set the scale factor in progeCAD, use the commands and system variables for each type of scale-sensitive entity, as illustrated by the table below.

Entity	Command or System Variable
Linetypes	**LtScale** sets the linetype scale factor.
Hatch Patterns	**HpScale** sets the hatch scale factor.
Dimensions	**DimScale** sets the dimension scale factor.
Text	**Style** or **Text** set the height in plotted inches.

For this drawing, set the scale factors, as follows:

1. Use the **LtScale** system variable to preset the scale factor for linetypes:

    : **ltscale**
    New current value for LTSCALE (greater than zero) <1.0>: **36**

2. Use the **HpScale** system variable to preset the scale factor for dimensions:

    : **hpscale**
    New current value for HPSCALE (greater than zero) <0'-1">: **36**

3. Use the **DimScale** system variable to set the scale factor for dimensions:

    : **dimscale**
    New current value for DIMSCALE (0 or greater) <1.0>: **36**

Text aren't scaled until later in this tutorial when you use the **Style** and **MText** commands

Save As Template Drawing

You start to see the benefit of template drawings, where all settings are already established. By saving the current drawing as a template, you can later reuse it as the start for similar drawings.

Templates are saved with the **SaveAs** command.

1. From the **File** menu, select **Save As**.
2. In the Save Drawing As dialog box, change the extension of the file name from .dw*g* to .*dwt*.

    File name: **apartment.dwt**

3. Click **Save**, and the drawing is saved as a template. The next time, you can use the **Open** command's "Drawing Template (dwt)" file type to access it.

4. There is a problem: notice the title bar reads "apartment.dwt". We want to work with a drawing, not a template.

    Use the **SaveAs** command again, saving the drawing as "apartment.dwg."

Drawing Exterior Walls

the *exterior walls* enclose the unit. Everything inside the apartment must fit between four walls. Two of the exterior walls (top and bottom) are 6" thick to provide extra insulation; the other two walls (left and side) are 4" thick, because they adjoin other units. In this chapter, you draw the exterior walls with the **Line** and other commands, and then save your work.
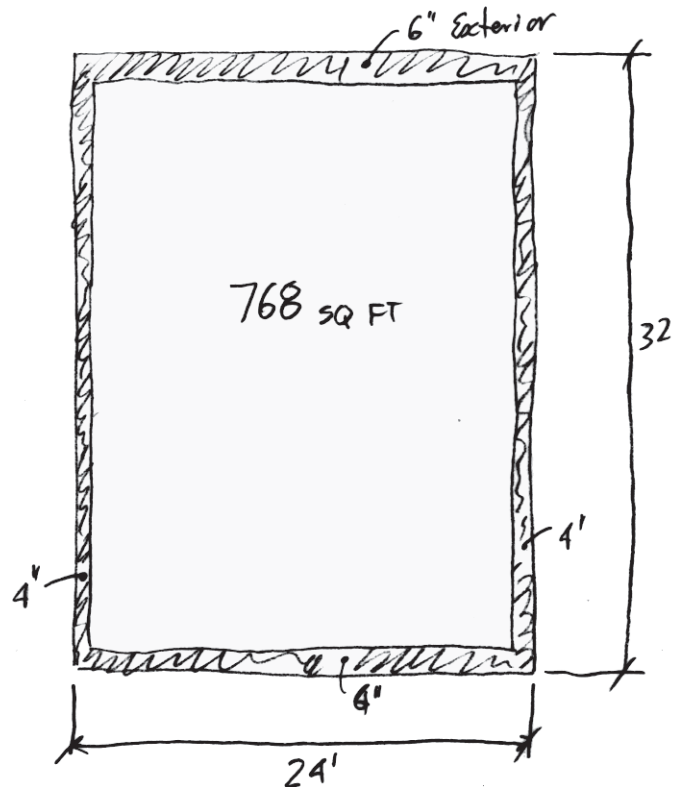
In This Chapter

- **Line** command.
- **Offset** command.
- **Trim** command.

Drawing Lines

The **Line** command draws line segments. You use it to draw the outline of the apartment's four exterior walls. The location of the lines are specified with absolute x, y coordinates (Cartesian coordinates); in later tutorials, you learn about other methods for inputting coordinates.

To use the **Line** command, you need to know the starting and ending x,y coordinates of each line segment. From the sketch of the exterior walls on the previous page, we can see that two walls are 24' long, and that two are 32'. Starting the apartment in the lower left corner at 0,0 (aka *origin*), I've noted the coordinates of each segment's endpoints in the figure below.

1. If necessary, use the Open command to open the *apartment.dwg* saved in the previous chapter.

2. Start the **Line** command.
   Command: **line**

3. Specify where the first line segment should be drawn, starting at the origin (0,0)...
   Start of line: **0,0**

4. ...and where the line segment ends, 24' to the right.
   Angle/Length/<End point>: **24′,0**

5. The second segment is 32' tall:
   Angle/Length/Follow/Undo/<End point>: **24',32'**
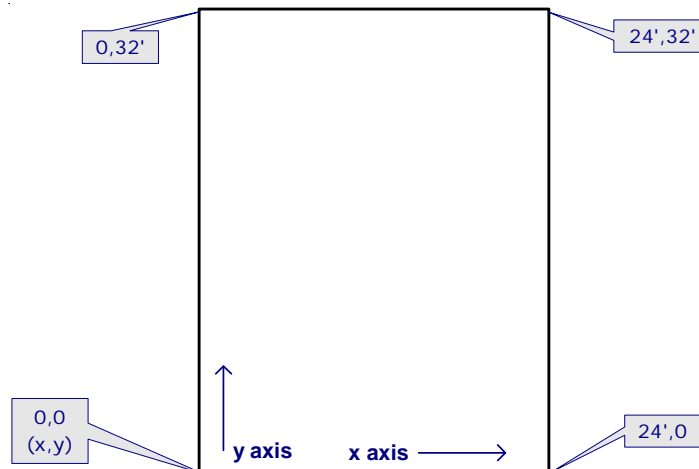
6. Draw the next segment is 24' to the left.
   Angle/Length/Follow/Close/Undo/<End point>: **0,32'**

7. To close the rectangle, you could enter the final coordinates (0,0), or you could use the **Close** option to have progeCAD draw the final segment for you.

> Angle/Length/Follow/Close/Undo/<End point>: **c**

At this point, the drawing consists of a rectangle. If you do not see it all, use the **Zoom Extents** command.

Offsetting Lines

The exterior walls are 4" and 6" thick. To show the thickness, you draw a second set of lines inside the large rectangle. You could use the Line command again, but there is an easier method. The **Offset** command makes copies by offsetting the original lines.

To use the Offset command, you need to know the following:

- The entities to be copied.
- The offset distance.
- The side(s) on which to create the copy.

1. Start the **Offset** command.

> Command: **offset**

2. Specify the offset distance of 4".

> Parallel: ENTER for Through point/<Distance>: **4**

3. Select the entity to copy.

> Select entity: *(Pick one of the vertical lines.)*

4. Pick a point inside the rectangle. progeCAD will make the copy through that point.

> Both sides/<Side for parallel copy>: *(Pick inside the rectangle.)*

5. The command repeats itself so that you can create parallel copies of additional entities. Pick the other vertical line.

> Select entity: *(Pick other vertical line.)*
> Both sides/<Side for parallel copy>: *(Pick inside the rectangle.)*

6. Press **Enter** to exit the command.

> Select entity: *(Press **Enter**.)*

At this point, your drawing consists of a rectangle with two extra vertical lines, as illustrated below.
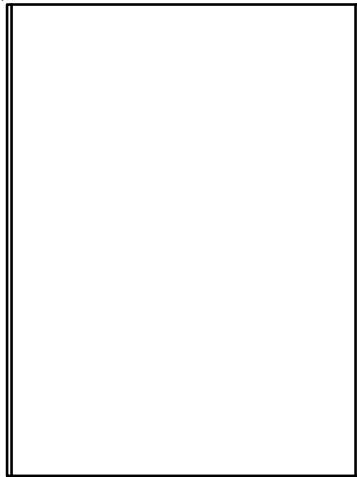
Repeat the **Offset** command to add the 6" offset lines.

> Command: **offset**
> Parallel: Through point/<Distance> <0'-4">: **6"**
> Select entity: *(Pick one of the horizontal lines.)*
> Both sides/<Side for parallel copy>: *(Pick inside the rectangle.)*
> Select entity: *(Pick the other horizontal line.)*
> Both sides/<Side for parallel copy>: *(Pick inside the rectangle.)*
> Select entity: *(Press **Enter** to end the command.)*

The result should look like the figure illustrated below.
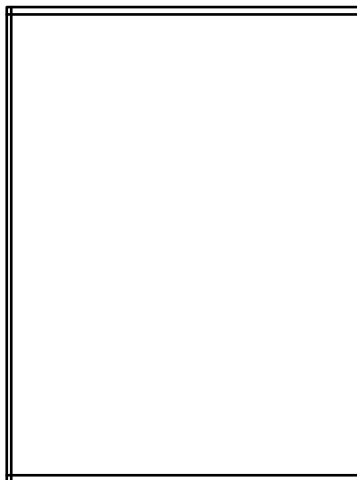
Trimming Lines

Creating the parallel copies resulted in messy intersections where the lines overlap. You use the **Trim** command to clean up intersections.

Trim works by shortening entities back to a *cutting entity*. Because the four inner lines overlap each other, you can use them simultaneously as cutting entities *and* the entities to be trimmed.

To use the **Trim** command, you need to know:

- The cutting entities.
- The entities to be trimmed.
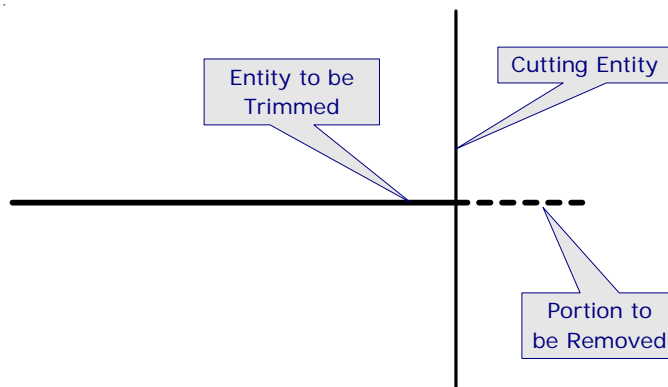
1. Start the **Trim** command.

   Command: **trim**

2. Select all the entities in the drawing by pressing **Enter**.

   Select cutting entities for trim <ENTER to select all>: *(Press **Enter**.)*
   Using all entities as trim boundary.

3. Pick a line to trim. This can be tricky: *pick the portion of the line you want removed*; do not pick the portion you want to retain.

   Edge mode/Fence/Projection/<Select entity to trim>: *(Pick the end of a line.)*
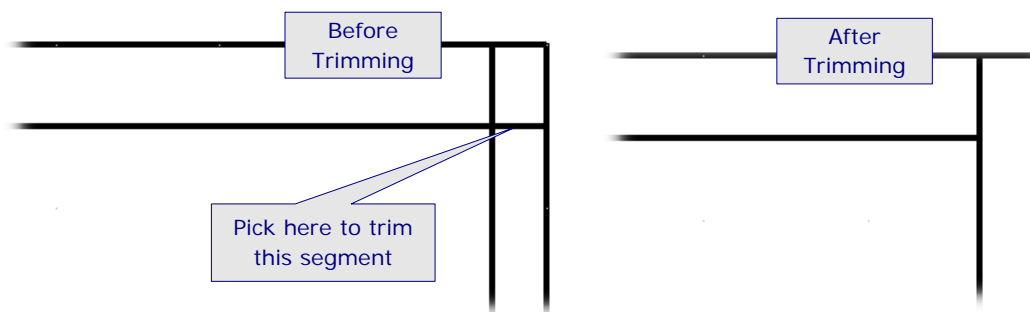
4. Repeat seven more times to trim the ends of all four inner lines.

        Edge mode/Fence/Projection/Undo/<Select entity to trim>: *(Pick another segment.)*

5. When done, press **Enter** to end the command.

        Edge mode/Fence/Projection/Undo/<Select entity to trim>: *(Press **Enter**.)*

At this point, your drawing should look like the double-line rectangle below.

Be sure so save your drawing! Use the **QSave** command, or press **Ctrl+S**.

Defining Interior Spaces

the interior spaces contain the living room, kitchen, a single bedroom, the closets, and so on. They are constrained by the exterior walls, and each other: give one area more space and it takes away space from other areas. In general, the kitchen and bathroom receive less space, while major living areas — dining, living, and bedroom — receive more space.

PATIO

LIVING ROOM   BED ROOM

DINING ROOM

4" walls

KITCHEN   BATHROOM

HALLWAY
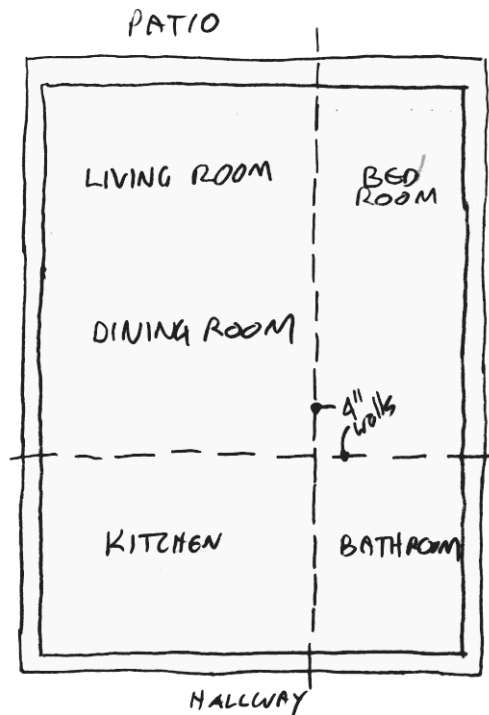
In This Chapter

- **ExpLayers** command
- **Move** command
- **Erase** command
- **Lenthen** command
- **Redraw** command
- **Fillet** command

For this project, each unit is subdivided into four major areas:

- Living and dining room.
- Kitchen; the kitchen shares space with the entrance hall.
- Bathroom; the bathroom shares space with the coat closet.
- Bedroom; the bedroom is further subdivided into three spaces:

    Bedroom closet.

    Ensuite bathroom.

    Hotwater tank closet.

All interior walls are 4" thick.

Creating Layers

Once drawings consist of more than just a few lines, it becomes important to segregate information. For instance, external walls are different from internal walls, and so they should be segregated. Similarly, the content of rooms (furniture, appliances) should be segregated from the content of walls (doors, window). And so on.

There are two ways to segregate information in CAD drawings: through *layers* (discussed in this chapter) or through *externally-referenced drawings*. (Externally-referenced drawings keep entire drawings separate from each other, yet display them as one drawing. Here we concern ourselves with layers only.)

About Layers

Layers are like transparent sheets of paper; on each sheet you draw different parts of the drawing, such as walls, electrical, and furnishings. You overlay the sheets to see the entire drawing. For instance, an electrician is interested in the walls and electrical, so he doesn't need the furnishings layer. That layer can be turned off for him.

Entities are assigned to layers. (If you don't assign layers, then all entities end up on a single layer, named "0").

Layers have properties that determine the way that entities appear. The properties include color, linetype, and visibility. This aspect of layers is powerful, because it means that all entities of a layer can have their properties changed at once. For instance

- Change a layer's color from black to red: all entities assigned to that layer also change from black to red.
- Change the layer's linetype from continuous to hidden: the entities' linetype change to hidden.
- Turn off a layer, and its entities disappear from the screen; turn on the layer, and the entities reappear.

There is no limit to the number of entities that can be assigned to a layer. There is no limit to the number of layers that can be created in a drawing.

Layer Standards

It is common for architectural and engineering firms to establish *layer standards.* These specify the names and properties of layers, and other aspects of drawings.

Standards are usually specific to disciplines; for example, the layers used for an architectural drawing are not suitable for mechanical drawings.

Standards are sometimes dictated by clients or by industry associations. Examples include the US Coast Guard and the Architectural Institute of America.

TIPS It is good practice to create template drawings containing standard layer names.

To copy layers from one drawing to another, use the Explorer window's **Copy** and **Paste** commands.

Properties of Layers

Layers can have the following properties:

Name — any combination of letters, numbers, and spaces up to 255 characters long. The name cannot contain these symbols / ? : > " < | and \ . You can use descriptive names ('Walls', 'Electrical', 'Furnishings') and prescriptive names ('A-WALLS', 'E-LIGHTS').

Current (Set) — only one layer can be current at a time. This is the *working* layer; when new entities are drawn, they are assigned to the current layer.

Color — layers can be assigned any of progeCAD's colors. The default color is black.

Linetype — layers can be assigned any linetype loaded into the drawing. The default linetype is Continuous.

Lineweight — layers can be assigned any of progeCAD's 24 lineweights, from 0 mm to 2.11mm. The default lineweight is called "Default," and is determined by the setting in the **Drawing Settings | Display | Lineweights** dialog box.

On and **Off** — layers can be on (visible and editable) or off (not visible). The default is on.

Plot — layers can be plotted or not plotted. The default is to be plotted.

Freeze and **Thaw** — layers can be frozen (not visible and ignored) or thawed (visible). The default is thawed.

Lock and **Unlock** — layers can be locked (visible but not editable) or unlocked (visible and editable). The default is unlocked.

(Plot Styles names can be specified only when the drawing uses named plot styles; otherwise they are grayed out.)

Frozen Layers

The difference between *off* and *freeze* is subtle. When a layer is turned off, its entities are no longer visible, but they are taken into account when the extents of the drawing are calculated, such as during the **Regen**, **Zoom**, and **Plot** commands. As well, you can continue to draw on layers turned off, but the entities remain invisible until the layers are turned on.

When a layer is frozen, the extents of its entities are ignored, and you cannot draw on it.

In addition, frozen layers occur in two different ways: in *current* viewports and in and *new* ones. (These are the viewports found in paperspace or layout mode.) By freezing layers in individual viewports, you create different kinds of views of drawings. The "New Viewports" option determines whether layers are frozen when new viewports are created.

ByLayer and ByBlock

You may have seen colors or linetypes called "BYLAYER." This special name means that entities take on the color, linetype, and other properties specified by their layer — sort of like an automatic color and linetype.

Similarly, "BYBLOCK" colors and linetypes mean that entities take on the color and other properties specified by the block of which it is a part.

Local Overrides

I have described the preferred method of controlling the properties of entities: by layers. This can be overridden, a practice that some CAD instructors frown upon. For instance, you can assign the color green to an entity whose layer is red. This is called a "local override."

Local overrides can be applied with the **Color** and **Linetype** commands, or with the **Entity Properties** toolbar: select the entity, and then change its color or linetype.

Layer Command

You create layers with the **Layer** commands. (Alternatively, you can also use the ExpLayers command, but Layer is more like AutocAD's.)

If you had problems completing the previous tutorial, the exterior walls drawing is available as *1ExteriorWalls.dwg*.

In this tutorial, you create layers to segregate details. As I noted earlier, layers can have *any* name — so what names should you use? Spend a moment thinking about the content of the apartment design.

LAYER COMMAND

Creates and controls layers.
Command: **layer**
Aliases: **la**, **ddlmodes**
AutoCAD: **layer**
Menu: **Settings | Explore Layers**
Toolbar: **Entity Properties | Explore Layers**
Options: **Layer name**
      New
      Make
      Current (Set)
      Color
      Linetype
      Lineweight
      On and **Off**
      Plot
      Plot Style
      Freeze and **Thaw**
      Lock and **Unlock**

ALTERNATIVES

ExpLayer makes and controls layers at the Explorer.

SetLayer sets current layer from selected entity.

Exploring the Explorer Window

progeCAD's Explorer provides a detailed view into the elements that make up drawings — just about everything except for entities and system variables. Explorer lists the names of open drawings, the states of layers, linetypes, text styles, UCSs (named coordinate systems), named views, user-defined blocks, and dimension styles. (AutoCAD's DesignCenter is similar to the Explorer.)

Explorer is used to inspect the status of elements, to create new ones, and to change their properties. Perhaps the most powerful feature of Explorer is the ability to copy elements among drawings.
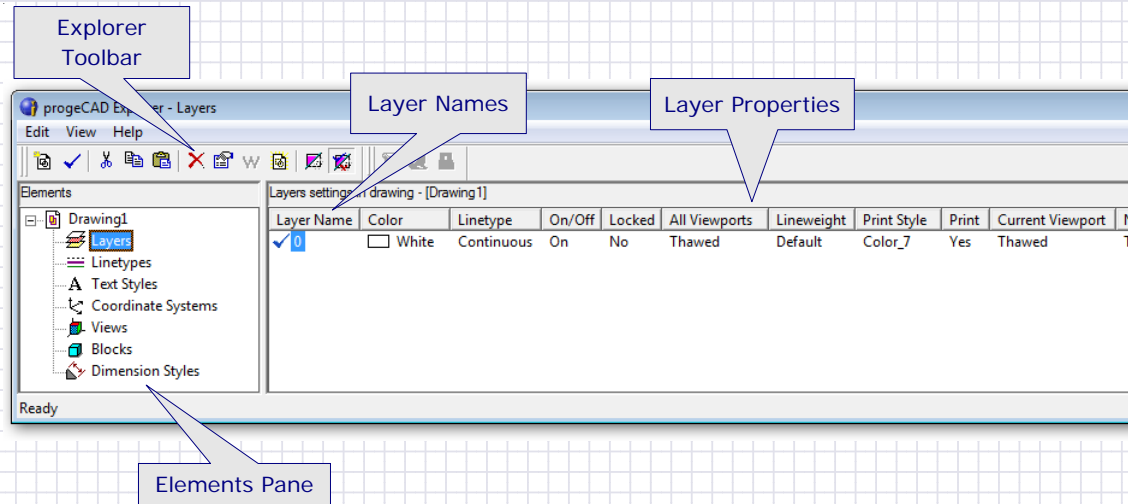
The window is opened by the **Explorer** command. Related commands open the window with the specified element showing:

- **ExpBlocks** — blocks stored in drawings.
- **ExpDimstyles** — dimension styles defined in drawings.
- **ExpFonts** — text styles names.
- **ExpLayers** — layers and settings.
- **ExpLtypes** — linetypes loaded into drawings.
- **ExpUcs** — user-defined coordinate systems.
- **ExpViews** — named views stored in drawings.

Toolbars

You can manipulate elements in a variety of ways, as indicated by the toolbar:

New Item — creates a new element with a generic name and default properties.

Current — makes the selected element current, such as the working layer or view.

Cut — copies selected elements to the Clipboard, and then erases them from the drawing.

Copy — copies selected elements to the Clipboard.

Paste — pastes elements from the Clipboard into the drawing, and adds them to the Explorer.

Delete — removes the element from the drawing; certain elements cannot be deleted. progeCAD warns you before removing the element.
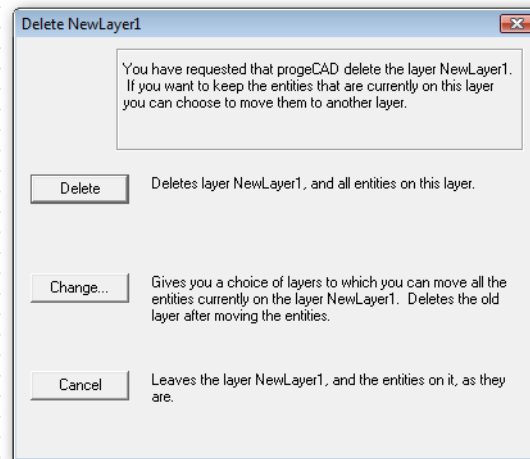
Properties — displays a non-modal dialog box showing property settings. "Non-modal" means that the dialog box can stay on-screen, and does not need to be dismissed in order to continue editing. Click the **x** to dismiss the dialog box.
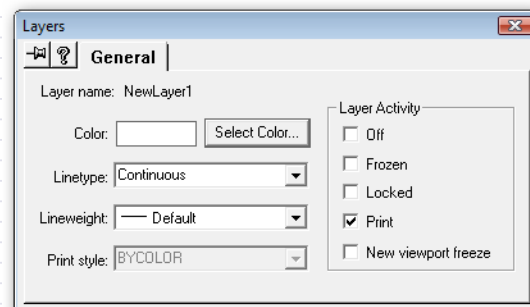
World (Coordinate Systems only) — returns a UCS to the WCS view.

Purge — removes unused elements from the drawing; certain elements cannot be purged.

Regen — forces the drawing to update to reflect changes made in the Explorer window.

On/Off Regen — toggles automatic regeneration.

Layer elements have three additional toolbar buttons:

Layer on/off — turns selected layer(s) on and off.

Freeze/thaw — freezes and thaws selected layer(s).

Lock/unlock — locks and unlocks selected layer(s).

The block elements have six toolbar buttons of their own:

Images — shows blocks as preview images.

Details — lists blocks by name and details, such as number of insertions and insertion points.

Insert — inserts the selected block in the drawing.

Insert external file blocks — displays Insert Block dialog box to insert any drawing as a block.

Save blocks — runs the **WBlock** command to export blocks as .*dwg* files.

Attach drawing — runs the **XRef** command to attach drawings as external references.

Curiously, the **Rename** option is not on the toolbar, perhaps because elements can be renamed directly. Click on a name twice, and then edit it.

Shortcut Keystrokes

Within the Explorer window, you can use the following shortcut keystrokes:

> **Ctrl+A** — selects all elements
>
> **Ctrl+C** — copies selected elements.
>
> **Ctrl+V** — pastes elements.
>
> **Ctrl+X** — cuts selected elements.
>
> **Del** — deletes selected elements.
>
> **F1** — displays help window.

Mouse Actions

To select an element, click on it.

To select a range of elements, hold down the **Shift** key, and then select the first and last elements in the range.

To select random elements, hold down the **Ctrl** key, and then select each element.

You can right-click elements to display a shortcut menu of options.
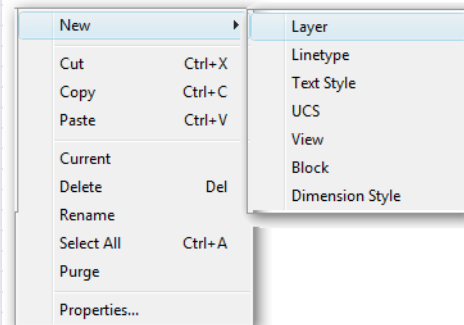
The Explorer cannot be customized.

It consists of these elements:

    - walls
    - doors and windows
    - fixed assets
    - nonfixed assets

A "real" drawing would also have plumbing, electrical and communications, alarm system, heating and ventilating, and many other details — and hence many more layers.

For this tutorial, use the above phrases as the layer names. To create layers, follow these steps:

1. Start the **Layer** command. Notice the Layer Properties Manager dialog box.
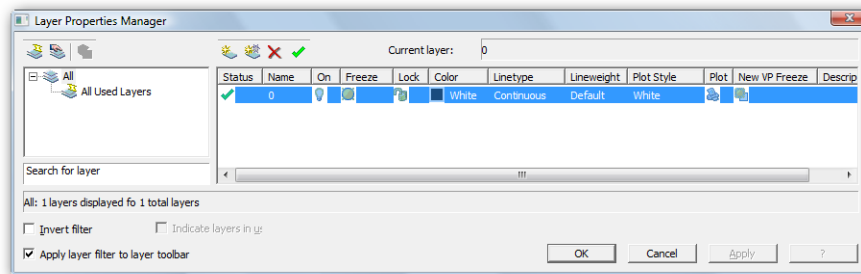
2. In the toolbar, click the **New Layer** button. Notice that progeCAD creates a new layer with the same properties as the current layer: white color, continuous linetype, and so on.

3. progeCAD names the new layer "Layer1." Change its name to "Walls".

4. Add the names of other layers, which you use later:

        Doors and Windows
        Assets-Fixed
        Assets-Nonfixed

There is a reason for using names with reversed words — such as "Assets - Fixed", instead of "Fixed Assets." progeCAD lists layer names in alphabetical order, and not in the order in which layers are created. By reversing the parts of the names, all Asset layers are listed together.

part ii: the tutorial projects

5. Now select **Walls**, and then click the **Current** ✅ button. This makes the layer current (i.e., the working layer), because the next lines you draw will be the interior walls.

   When done, the dialog box should look like the figure below.
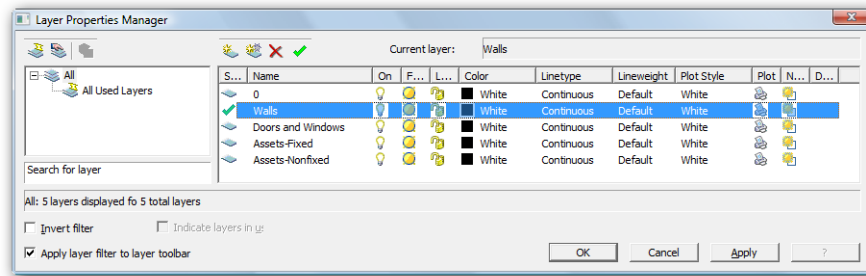
6. Close the dialog box by clicking the **OK** button.
7. On the Entity Properties toolbar, notice that the Layer Control displays "Walls."

Assigning Entities to Different Layers

Currently, all the lines, which represent exterior walls, are on layer 0. They really should be on layer 'Walls'. Follow these steps to assign entities to different layers.

1. Press **Ctrl+A** to select all entities in the drawing. Notice that they are highlighted by dashed lines and green grips.
2. From the Layer Control, select "Walls."
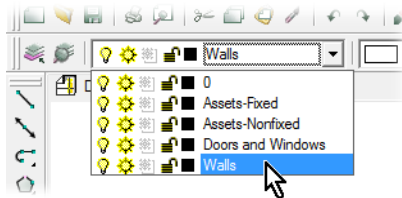
And you're done. Ja, it's that easy.

Adding Interior Lines

With the layers set up, you can now draw the interior walls. The technique is the same as for the exterior walls: draw one side of the wall with the **Line** command, and then draw the other side with the **Offset** command. Finally, clean up intersections with the **Trim** command.

You can make the rooms any size you like to fit the exterior walls. Or, you can follow the dimensions illustrated below (width' x depth').

Living Room
14'x21'
(including dining area)

Bedroom
9'x16'
(including two closets and ensuite bathroom)

Bathroom
9'x9'8"

Kitchen
10'x9'8"

Entrance
4'x9'8"
(including closet)
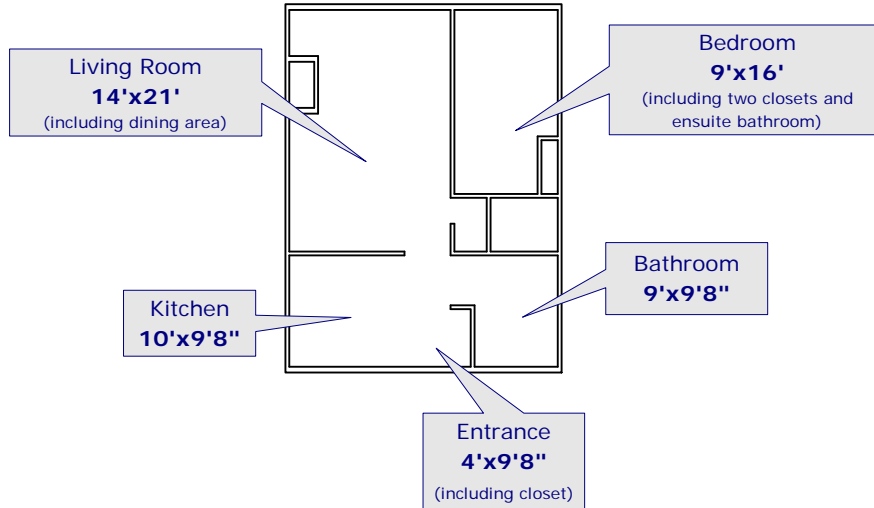
If you were to add up the dimensions, don't forget the wall thicknesses; otherwise, it may seem that the totals are incorrect. Across the top, for example, the living room + bedroom are 14' + 9' = 23' wide. Yet the apartment unit is 24' wide. Where did the missing foot go? The answer is in the thickness of the walls: the two outside walls and one inner wall are each 4" thick. That's 3 x 4" = 12" = 1'.

Here are some design hints:

- Interior walls are 4" thick.
- You don't need to create openings for doors or windows, because those will be indicated later when you insert the door and window symbols.
- The fireplace is roughly 4' x 2'.
- Read the alternate strategies described later in this chapter.

Starting Offset Lines

At this stage, the most difficult drafting task is starting a line at some point along an existing line. For instance, the wall that separates the kitchen from the dining/living room starts at 9'8" along the y axis.

How do you determine where this line starts?

Kitchen
10'x9'8"

9' 8"

0,0
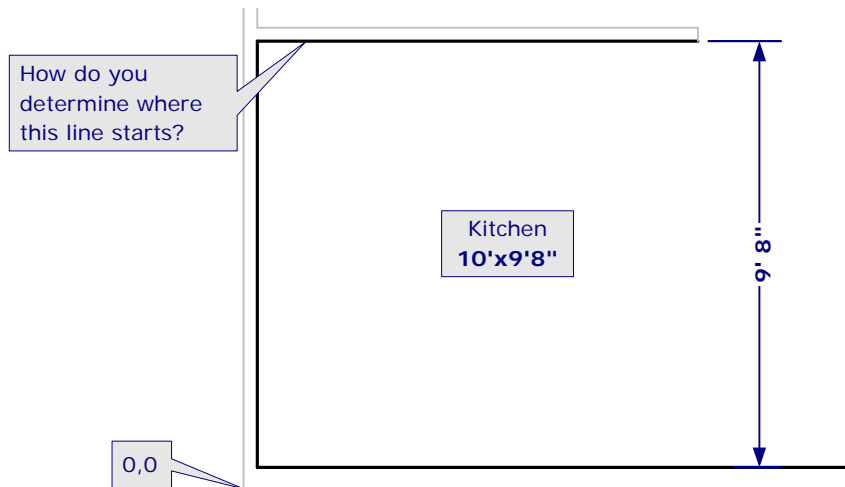
Drawing lines that start at the end of existing lines is pretty easy: use the ENDpoint entity snap. But how do you draw a line that starts 9'8" from the end of another line? That's not quite as easy. I'll show you several ways to do it; try each one to see which works best for you.

Brute Force Method

The *brute force* method draws a construction line to specify the distance, draws the wanted line, and then erases the construction line.

10'

3. "Wall" line is 10' long.

2. "Construction" line is 9'8" long.

9' 8"

1. Use INTersection entity snap at this corner.

4. Erase construction line.
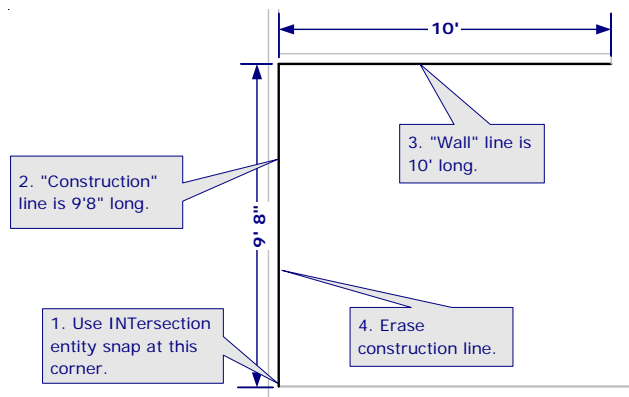
1. Ensure ortho mode is turned on. If necessary, double-click ORTHO on the status bar so that the word is shown in black, not gray.

2. Use the **Line** command to draw the construction line, followed by the wall line, as follows:

> Command: **line**
> Start of line: **int**
> Snap to intersection of: *(Pick intersection of lines at 1, as illustrated above.)*
> Angle/Length/<End point>: *(Move cursor upwards in the y direction)* **9'8"**
> Angle/Length/Follow/Undo/<End point>: *(Move cursor to the right in the x direction)* **10'**
> Angle/Length/Follow/Close/Undo/<End point>: *(Press **Enter** to end the command.)*

3. Use the **Erase** command to remove the construction line, as follows:

> Command: **erase**
> Select entities to delete: *(Pick the construction line.)*
> Entities in set: 1
> Select entities to delete: *(Press **Enter** to end the command.)*

4. The line is erased. If necessary, use the **Redraw** command to clean up the display.

> Command: **r**

TIP The **Line** command doesn't always require a pair of coordinates. Above, you entered a single value, 9'8", for the coordinate. How does progeCAD know if you mean x or y?
   When you turned on ortho mode, progeCAD restricted cursor movement in the x (horizontal) and y (vertical) directions.
   When you move the mouse (cursor) up, progeCAD understood you meant the y direction.
   And when you entered the distance of 9'8", progeCAD understood that to mean y=9'8".

Construction Line Method

The *construction line* method creates a layer for construction lines, and then uses the **InfLine** command to place a grid of construction lines. Following that, you switch back to the interior-walls layer, and trace over the construction lines with the **Line** command.

Calculated Coordinates Method

The *calculated coordinates* method works out the x,y-coordinate of the line's start point.

x = 0'4"
y = 0'4" + 9'8"
    = 10'0"

1. Determine starting coordinates of "wall" line.

2. "Wall" line is 10' long.

y = 9' 8"

x = 0'4"
y = 0'4"
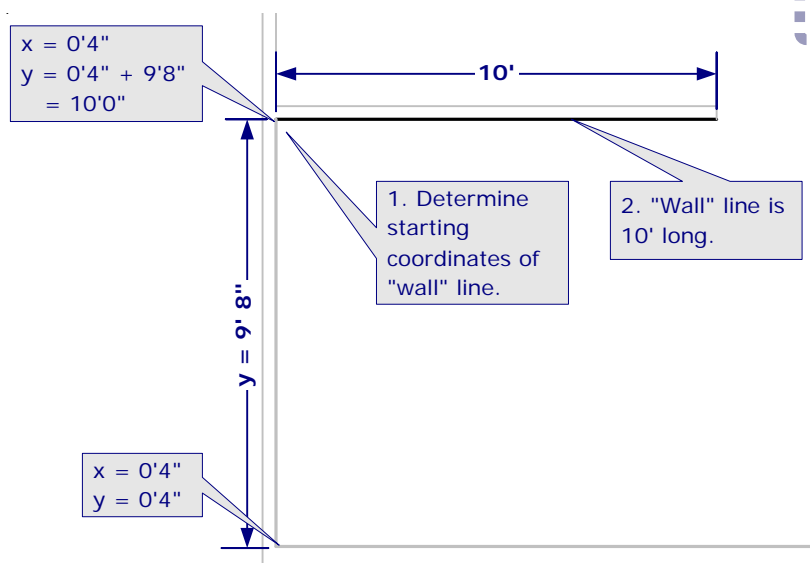
ERASE COMMAND

Deletes selected entities.

Command: **erase**

Aliases: **e, delete**

Keyboard: **Del**

Menu: **Edit | Delete**

ALTERNATIVES

CutClip copies selected entities to the Clipboard, and then erases them from the drawing.

Oops returns the last erased entity(ies).

U undoes this command.

REDRAW COMMAND

Cleans up the current viewport.

Command: **redraw**

Alias: **r**

Menu: **View | Redraw**

Toolbar: **View | Redraw**

ALTERNATIVES

RedrawAll cleans up all viewports.

Regen regenerates the view from the drawing database.

The figure above illustrates the math. The x coordinate stays the same, but the y coordinate changes, as follows:

```
y    = 0'-4" + 9' 8"
     = 10' 0"
```

(Having to add up feet and inches just about makes you pine for the metric system, eh?) You now draw the line using the coordinates you worked out:

Command: **line**
Start of line: **4",10'**
Angle/Length/<End point>: *(Move cursor to the right in the x direction)* **10'**
Angle/Length/Follow/Undo/<End point>: *(Press **Enter** to end the command.)*

Moved Entity Method

The *moved entity* method draws the line, and then uses the **Move** command to move it by 9'8".

2. … and then move by 9'8" to here.

1. Draw "wall" line here…

Point #1

9' 8"

10'
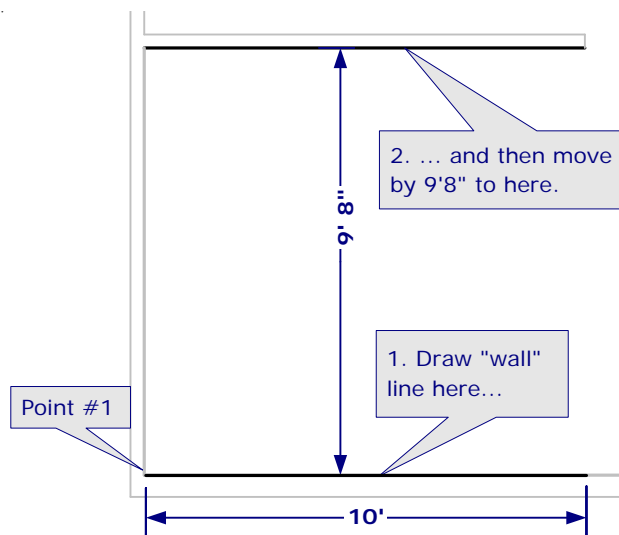
1. **Draw the 10'-long line over the existing line, as illustrated above:**

    Command: **line**
    Start of line: **int**
    Snap to endpoint of: *(Pick intersection at 1.)*
    Angle/Length/<End point>: *(Move cursor to the right in the x direction)* **10'**
    Angle/Length/Follow/Undo/<End point>: *(Press **Enter** to end the command.)*

2. **With the Move command, move the line up by 9'8", as follows:**

    Command: **m**
    Select entities to move: **l**
    Entities in set: 1
    Select entities to move: *(Press **Enter** to end entity selection.)*

MOVE COMMAND

Moves entities linearly.
Command: **move**
Alias: **m**
Menu: **Modify | Move**
Toolbar: **Modify |Move**
Options: **Vector**
         Base point
         Displacement

ALTERNATIVES

Grips editing moves entities.
Stretch move entities within the selection window.
Rotate moves entities through an angle.

Vector/<Base point>: **end**

Snap to endpoint of: *(Pick end of line at 1.)*

Displacement point: **0,9'8"**

TIP   Notice the use of "l" in the 'Select entities to move' prompt, above. It is short for "last" and tells progeCAD to select the last-created (or most-recently created) entity in the drawing.

> There are numerous selection options, but the few that I use the most include:
>
> **a**   Selects all entities in the drawing
> **c**   Prompts me to create a selection rectangle that includes all entities crossing and within the rectangle.
> **l**   Selects the last-drawn entity.
> **p**   Selects the previously-selected entities.
> **w**   Prompts me to create a selection rectangle that includes all entities within the rectangle.

Offset-Lengthen Method

The offset method uses the **Offset** command to offset an existing line, and then uses the **Lengthen** command to correct its length.
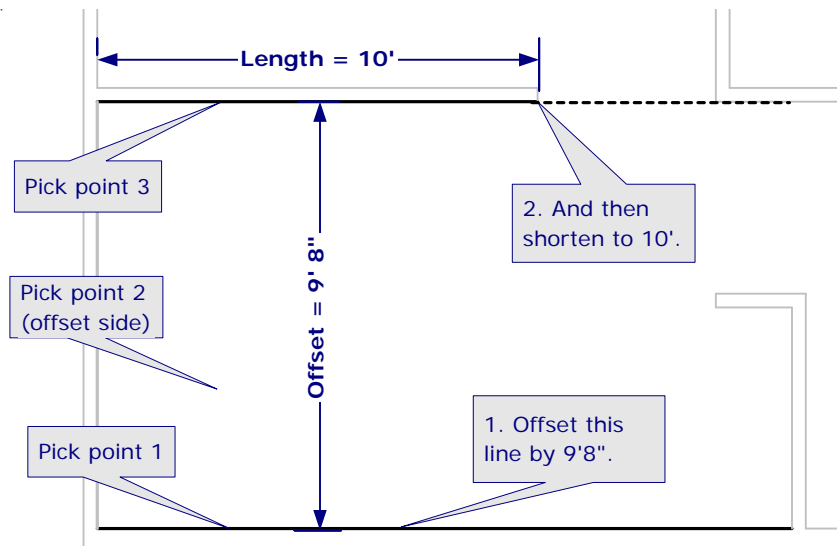
1. Use the **Offset** command to create an offset copy of an existing line, as follows:

    Command: **offset**

    Offset: Through point/<Distance> <0'-1">: **9'8"**

    Select entity: *(Pick line at 1.)*

    Both sides/<Side for parallel copy>: *(Pick above line, at 2.)*

    Select entity: *Press **Enter** to end the command.)*

2. Use **Lengthen** command to shorten the line to ten feet, as follows:

    Command: **lengthen**

    Edit length: DYnamic/Increment/Percent/Total/<Select entity to list length>: **t**

    Angle/<Enter total length (0'-1")>: **10'**

    Mode/<Select entity to change>: *(Pick line at 3.)*

    Mode/Undo/<Select entity to change>: *(Press **Enter** to end the command.)*

> **TIPS** The **Lengthen** command applies only to open entities, such as lines, arcs, and polylines.
>
> Your pick point at the 'Select entity to change' prompt determines from which end the entity is lengthened or shortened. The endpoint closest to the pick point is the end that changes; the other end point remains in place.

The "t" option specifies the *total*, new length of the line. The command's options include:

Lengthen	Meaning
DYnamic	Interactively changes the length as you move the cursor.
Increment	Adds to the entity's length by this amount; a negative value subtracts from the length.
Percent	Changes length by a percentage: Less than 100 = entity is shorter / 100 = no change / More than 100 = entity is longer
Total	Specifies the absolute length, longer or shorter.

Alternate Trim Strategy

The different approaches described above show the flexibility of progeCAD; it usually provides more than one way to perform operations.

Earlier, I suggested you draw the interior walls the same way as the exterior walls: place lines, make offsets, and trim the excess. Here is an alternative method to trimming.

The obvious way to trim intersections is with the **Trim** command to cut off unwanted portions of lines. Less obvious is the **Fillet** command; in fact, Fillet is preferred over Trim by most CAD operators.

1. Start the **Fillet** command, and then enter "s" for the **Settings** option, as follows:

   Command: **fillet**
   Fillet (radius=7'-10"): Settings/Polyline/<Select first entity>: **s**

2. progeCAD displays the Drawing Settings dialog box opened to the Entity Modification | Chamfer/Fillet tab. Here you set the fillet radius.

LENGTHEN COMMAND

Changes the length of open objects.

Command: **lengthen**

Aliases: **len, editlen**

AutoCAD: **lengthen**

Menu: **Modify | Edit Length**

Toolbar: **Modify |Edit Length**

Options: **DYnamic**
      Increment
      Percent
      Total
      List length

ALTERNATIVES

Stretch lengthens or shortens the portion of open and closed entities outside the selection window.

Extend lengthens open entities to a boundary.

Trim shortens open entities to a bdounary.

FILLET COMMAND

Rounds off intersections.

Command: **fillet**

Aliases: **f, fi**

Menu: **Modify | Fillet**

Toolbar: **Modify |Fillet**

Options: **Settings**
      Polyline

ALTERNATIVES

Chamfer angles off intersections.

Trim shortens open entities to a boundary.
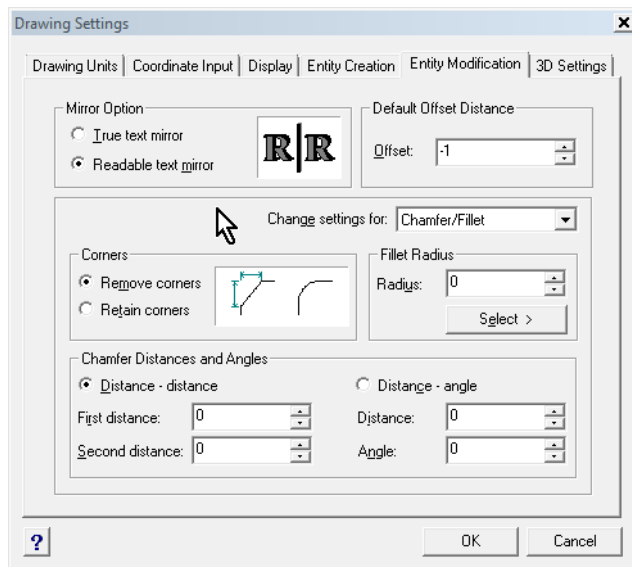
To trim lines with fillets:

a. Set the radius to 0". In the Fillet Radius area, change **Radius** to **0**,

b. Leave the **Corners** option set to "Remove Corners."

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

3. progeCAD returns to prompting you in the command bar. Notice that the radius is now set to 0". Pick the first line:

```
Fillet (radius=0"):  Settings/Polyline/<Select first entity>:  (Select a line.)
```

Important: Pick the part of the line you wish to keep, as illustrated below.

1. Pick one line.

2. And pick an intersecting line.
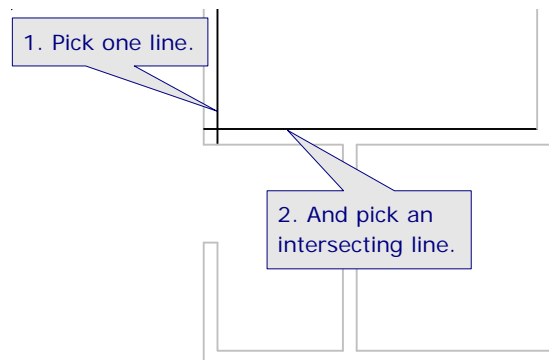
4. And then select the second line:

```
Select second entity:  (Select an intersecting line.)
```

progeCAD appears to trim the intersection by cutting back both lines to where they intersect.

Lines trimmed by Fillet command.
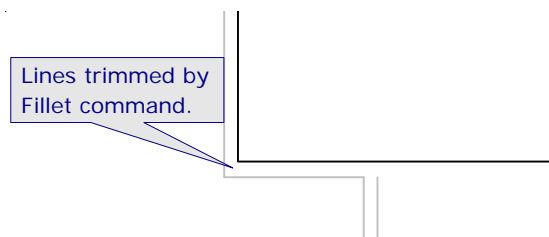
At this point, your drawing should look similar to the floorplan shown below.
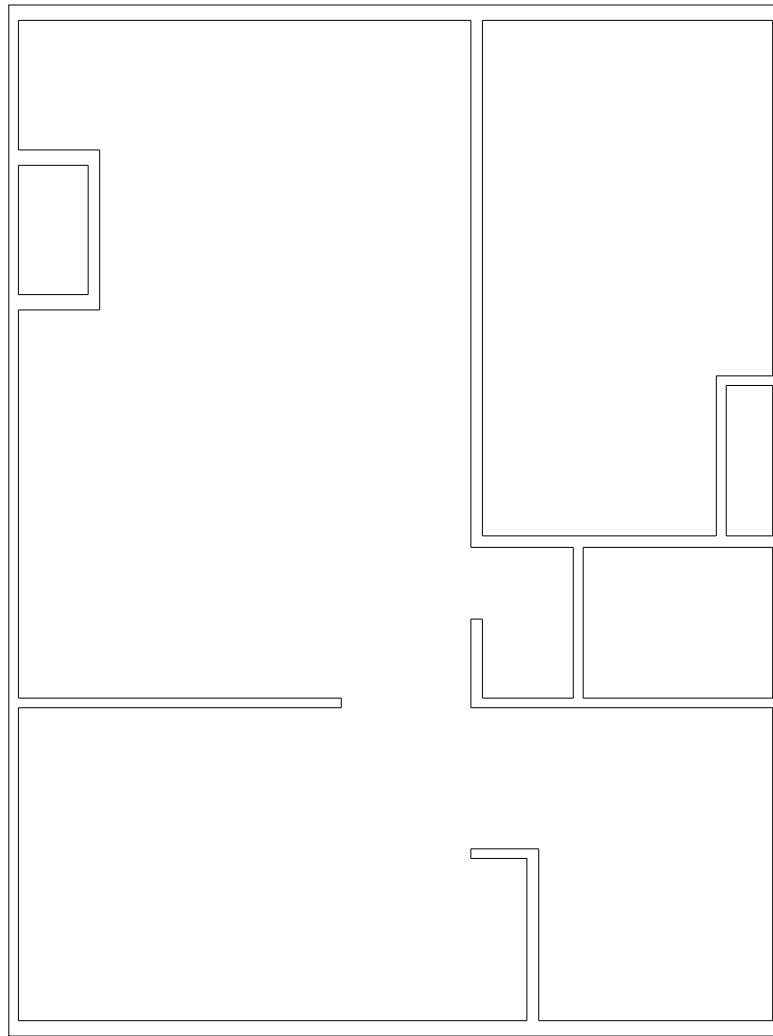
(If you had problems completing the previous tutorial, the apartment drawing is available as *2InteriorWalls.dwg*.)

Printing Drawings

at this point, your drawing should be looking good on your screen, but it would be great to see it on paper. Let's print the drawing now. Drawings are plotted with the **Print** or **QPrint** commands.

(If you had problems completing the previous tutorial, the apartment drawing is available as *2InteriorWalls.dwg.*)

Plotting vs. Printing

In the past, plotting and printing had different meanings:

- *Plotting* meant sending drawings to vector devices such as pen plotters. Ten to twenty years ago, plotting produced much higher quality output than did printing, because raster devices created cruder plots than did vector devices.

- *Printing* used to mean outputting drawings to raster devices such as dot-matrix printers. In the last ten years, raster technology has been refined to the point that nearly all CAD drawings are output on raster devices — either ink jet printers or laser printers. The quality is as good as the pen plotters of yore, but the speed is much faster.

In This Chapter

- **QPrint** command
- **Preview** command
- **Print** command
- **Export** command
- Headers and footers
- **Mail** command

Common today is *electronic plotting*, where drawings are saved in file formats suitable for sending by email or displaying at Web sites. The most common "eplot" formats are Acrobat PDF and DWF, both of which can be created by progeCAD.

Because they now mean the same thing, I use the words "plot" and "print" interchangeably In this chapter.

Plotting, Basically (Print)

To control the plotting process, you use the **Print** command. It displays the Print dialog box with its many options. For most plots, you do not need to worry about most options, and so this part of the chapter describes only the basic options.

(A complete description of the Print dialog box comes later in this chapter under the "Advanced Options" heading.)

1. To print a drawing, enter the **Print** command:
    Command: **print**

    (Alternatively, select **Print** from the **File** menu, or press **Ctrl+P**. The print button on the toolbar executes the QPrint command, as discussed later.)

    Notice that progeCAD displays the Print dialog box.
2. For now ensure that the most important options are set, as listed below. In the Device tab, choose the printer from the Printer droplist.

PRINT COMMAND

Displays dialog box to control plotting of drawings.
Command: **print**
Alias: **plot**
Shortcut: **Ctrl+P**
AutoCAD: **plot**
Menu: **File | Print**

ALTERNATIVES

QPrint prints the current drawing using the default printer settings.

Export saves the drawing in file formats that can be read and printed by other programs.
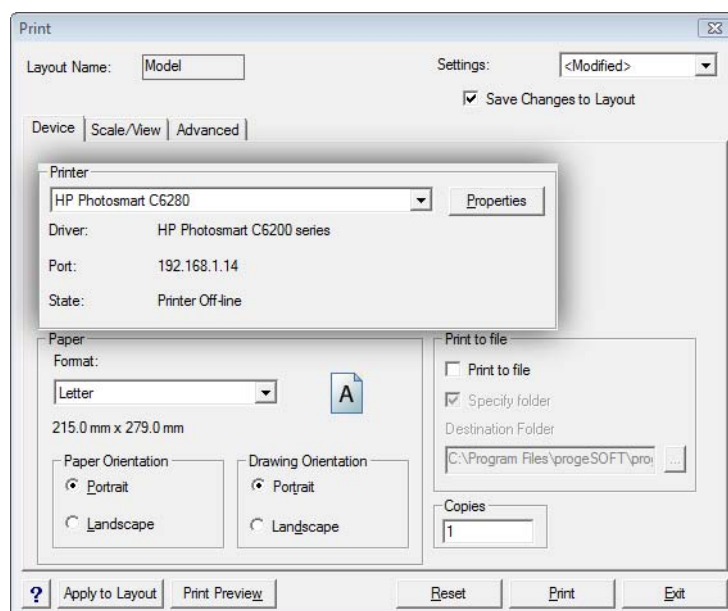
3. In the Scale/View tab, choose the following options:
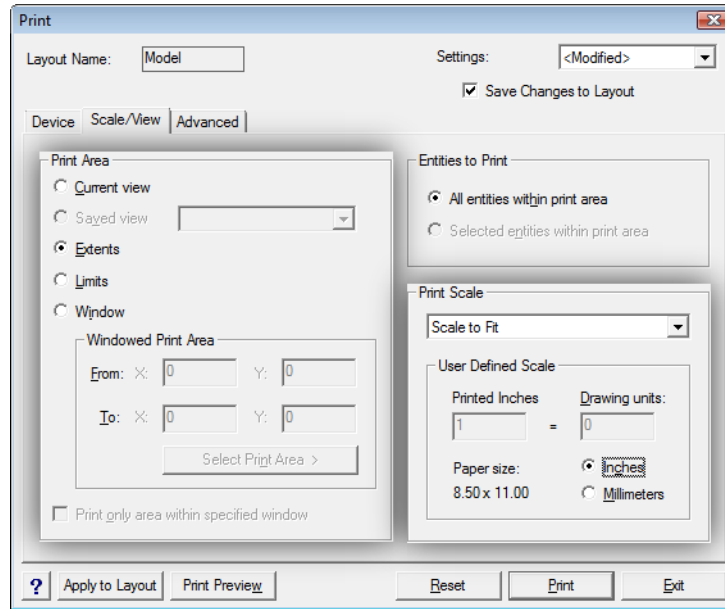
Print Area:	**Extents**
Print Scale:	**Scale to Fit**
Paper Size:	**Inches**

4. Before plotting, always, *always,* <u>*always*</u> perform a preview. It takes but a few seconds and ensures the plot will turn out as you expect. Trust me, plotting has so many options that there are bound to be one or more that are set incorrectly.

Preview lets you catch the mistakes before committing time, paper, and ink.

Click the **Print Preview** button. progeCAD immediately shows the floor plan as it will appear on the printed page.
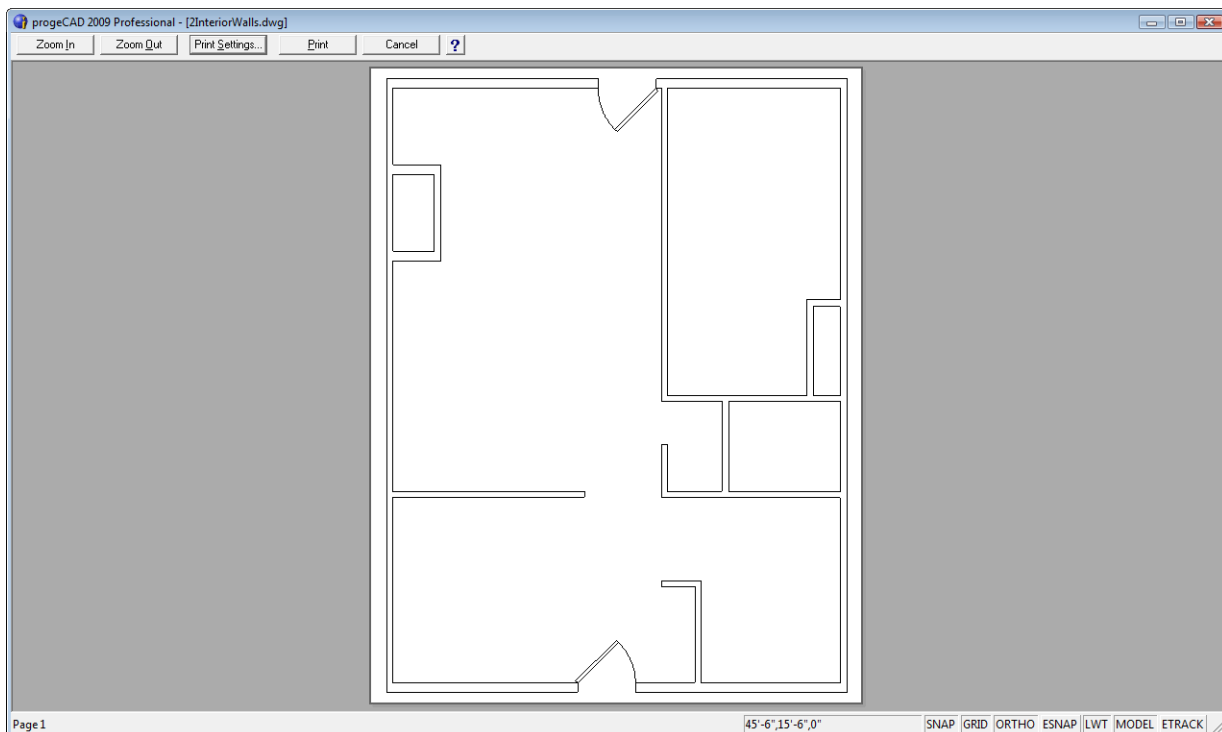
5. If it looks satisfactory to you, click **Print**. progeCAD sends the drawing to the printer. Wait a few moments for it to appear.

If you don't like the arrangement or size, click **Print Settings** to return to the Print dialog box.

Quick Prints (QPrint)

For quick plots, enter the **QPrint** command (short for "quick print"), or else click the **Print** button on the Standard tool bar.

> Command: **qprint**

progeCAD immediately sends the drawing to the printer or plotter.

The printing isn't accomplished any quicker than with the **Print** command; the difference is that you don't have access to the options: there is no Print dialog box and no print preview.

When you enter the **QPrint** command, progeCAD reads the settings of the Print dialog box and the default Windows system printer. Then it prints the current viewport according to those settings. (The very earliest releases of AutoCAD had a similar QPlot command, but it was removed with v2.0.)

You can get a similar effect with the **Preview** command: after progeCAD displays the preview window, click the **Print** button. progeCAD uses the stored print settings to control the plot. (This is identical to the operation in Word: click the printer icon on the toolbar, and Word sends the document to the printer using the default settings; select the Print item from the File menu, and Word displays the Print dialog box.)

> ### QPRINT COMMAND
>
> *Plots the current drawing using default values and printer.*
>
> Command: **qprint**
>
> Toolbar: **Standard | Print**
>
> ### ALTERNATIVE
>
> **Print** displays dialog boxes for controlling the plotting process.

TIPS To adjust settings before using **QPrint**:
1. Enter the **Print** command.
2. Make changes to the Print dialog box.
3. Click the **Apply** button to force progeCAD to remember the changes.
4. Click **Cancel** to exit the dialog box, and then use the **QPrint** command.

To preview the plot, use the **Preview** command, and then click the **Print** button.

To select a different printer or plotter:
1. Use the **PSetup** command to display Windows' Print Setup dialog box.
2. Make changes.
3. Click **OK**, and then use the **QPrint** command.

Preview Prints (Preview)

As noted above, the **Preview** command lets you see what the plot will look like before committing to printing it. It operates similarly to the print preview feature in word processors and other software.

1. From the **File** menu, select **Print Preview**.

   (Alternatively, click the **Print Preview** button on the toolbar, or else enter the **Preview** command.)

   Command: **preview**

2. progeCAD displays the drawing in the preview window. Along the top of the window are buttons for controlling the preview:
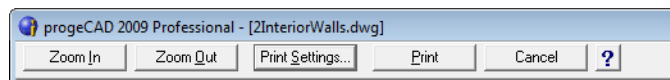
Zoom In enlarges the drawing. Alternatively, you can click the cursor on the drawing.

Zoom Out makes the drawing smaller, and see more of it. After the drawing is fully zoomed out, you can click the cursor twice more to make the drawing smaller again.

Print Settings closes the preview window, and then displays the **Print** dialog box.

Print closes the preview window, and then prints the drawing.

Cancel closes the preview window, and returns to the drawing editor.

? — displays on-line help.

TIP The **PSetup** command no longer exists. It has become the Device tab of the Printer dialog box, as described below.

Advanced Options

Let's return to the tabbed Print dialog box and examine the purpose of its options. First, the options common to all three tabs.

Layout Name

The name of the current layout tells you which layout will be plotted. MODEL means model space; LAYOUT means one of the layouts. You have to select a layout tab *before* starting the **Print** command. (Layouts are discussed later in Chapter 11.)
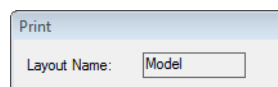
You can save different plot settings for each layout. That allows you to set up layouts for different printers, such as color and monochrome.

Settings allows you to copy settings from another layout.
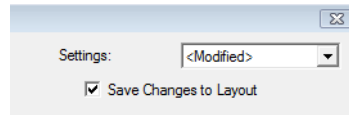
☑ **Save Changes to Layout** saves the settings to the layout on a layout-by-layout basis. Here "layout" also refers to the Model tab.

Common Buttons

The bottom of the dialog box contains a row of buttons common to all tabs:
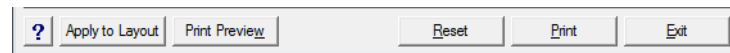
? displays online help.

Apply to Layout saves the options to the current layout; the next time you print this layout, the options are restored.

Print Preview switches to the preview window; see Preview command.

Reset changes all options to their default values.

Print prints the drawings.

Exit closes the dialog box.

Device Tab

The Device Tab selects the printer or electronic format for the plot.

Printer

The **Printer** droplist specifies the printer. The list includes:
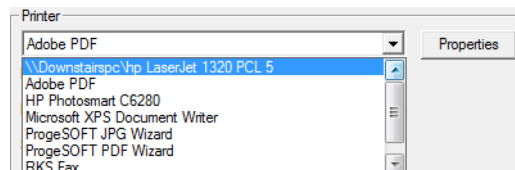
- Any printer set up for your Windows computer.
- Printers connected over networks.
- Non-printer drivers, such as for fax software and Adobe PDF.
- Non-printers specific to Windows Vista, such as XPS Document Writer.
- Additional file formats, such as JPG or DWF.

The **Properties** button displays the Properties dialog box specific to the printer. This is the same dialog box you see in any other program.

Paper

The **Format** droplist lists all the sizes of media supported by the selected printer. This means that the sizes listed vary, depending on which printer is chosen. The most common size is Letter (aka A-size).
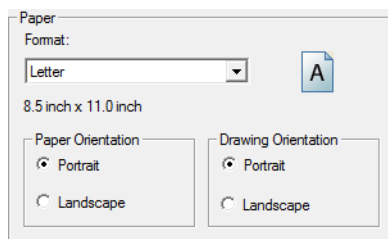
The **Paper Orientation** and **Drawing Orientation** radio buttons switch between Portrait (upright) and Landscape (sideways). This does not change the paper going through the printer; instead, this option rotates the drawing during printing, so that it fits the paper better.

Print to File

The **Print to File** check box determines whether the drawing is sent to the printer, or saved in the printer's data format in a file. (When you choose a format like JPG as the printer, then this option is turned on by default.
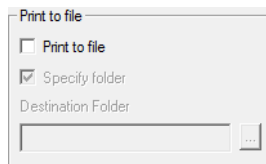
Copies

The Copies text entry box specifies the number of copies to print of the drawing.
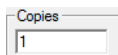
Scale/View Tab

The Scale/View tab controls how much of the drawing is plotted, and at what size.
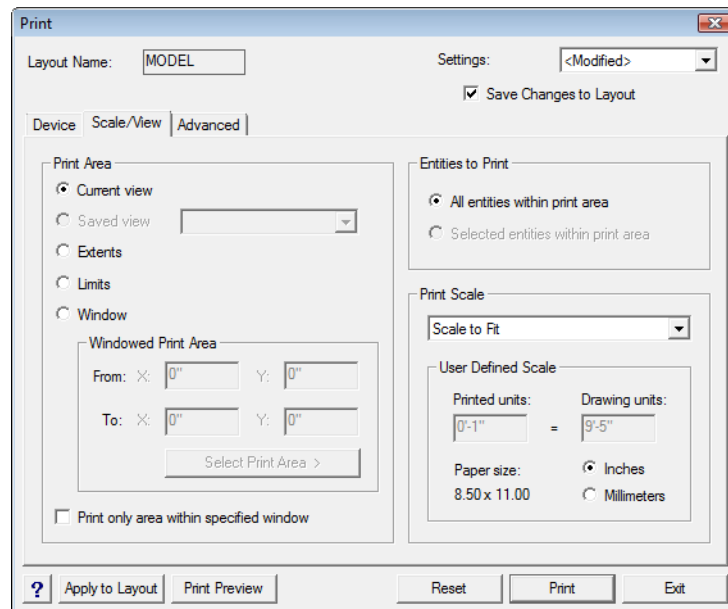
Print Area

The Print Area section tells how much of the drawing to plot. In most cases, you'll select the **Extents** option.
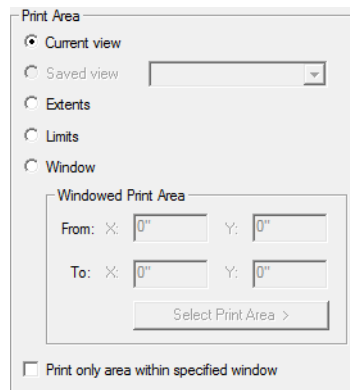
The options are:

○ **Current view** plots only as much of the drawing as is visible in the current viewport. That's exactly the view at the time you entered the **Plot** command. If the drawing is zoomed in, then only the zoomed-in portion is plotted.

○ **Saved view** plots named views defined by the **View** command. Select a view name from the droplist; when no views are defined, this option is not available. By defining views ahead of time, you can print selected areas of the drawing, such as a title block or a detail view.

- ⊙ **Extents** plots the entire drawing. This is the most commonly-used print area option, because it ensures everything is plotted. Everything, that is, except entities on frozen layers. Sometimes, drawings are printed really, really small, and you wonder why. Usually, it's because there are some entities way far, far away from the primary drawing. You can check by performing a **Zoom Extents** before printing.
- ○ **Limits** plots the portion of the drawing within the limits defined by the **Limits** command. The usefulness of this option is not clear to me, because limits are so rarely used.
- ○ **Window** plots a rectangular area. You enter either the x, y coordinates of the lower left and upper right corners of the area, or else click the **Select Print Area** button and then pick two points to define the rectangle. This option is useful for printing a portion of the drawing.

The **Print only area within specified window** check box prints just the entities within the rectangular area you specified. Why might there be entities outside the area? Because the rectangle you select usually does not have the same aspect ratio as the paper on which it is printed.

- ☑ **On** — only the entities (and parts of entities) within the selection rectangle are plotted.
- ☐ **Off** — entities outside the selection rectangle are plotted, if they fit the page.

In the figure below, the large white rectangle represents the paper, while the inner rectangle is the windowed plot area (designated by the first and second corner points).
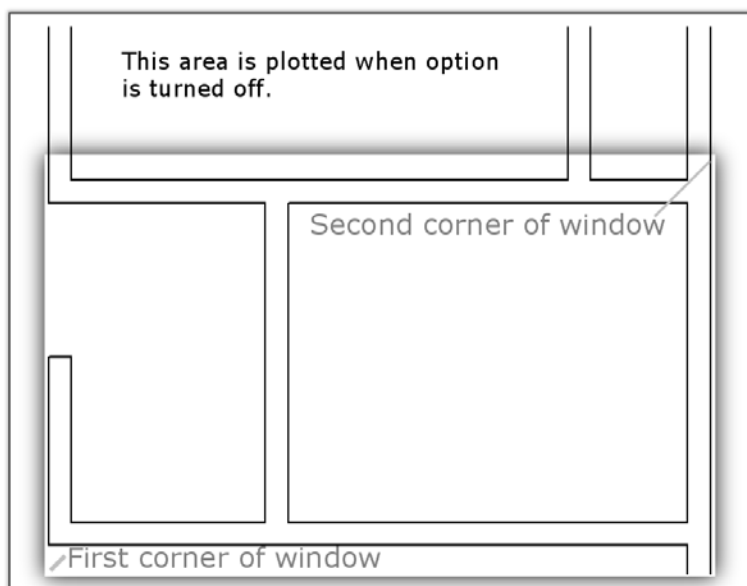
Entities to Print

The Entities to Print area allows you to print just selected entities. Problem is, you need to select the entities *before* you start the **Plot** command; then the second option becomes avail-

able. Otherwise, it is grayed out:
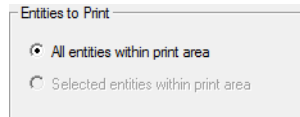
- ⊙ **All entities within the print area** prints every non-frozen entity.
- ○ **Selected entities within the print area** prints only the entities you selected prior to starting the Plot command.

In most cases, you'll select "All entities within the print area."

TIPS Both options have this disclaimer, "within the print area." This means that this option is subject to the option you selected under the Print Area section. For example, if you previously selected the **Window** option, then only the entities within the window are plotted.

Alternatively, you can select entities in the drawing, right-click, and then select **Print** from the cursor menu.

Print to Scale

The **Print Scale** area lets you size the drawing.
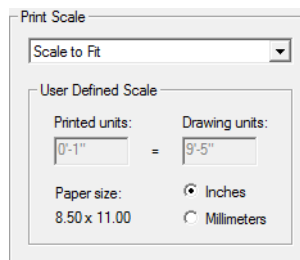
You have two choices: have progeCAD automatically scale the drawing to fit the paper, or else specify the scale factor that progeCAD must use. Click the droplist, and then select scale option:
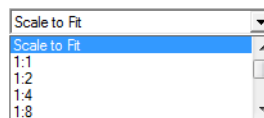
Scale to Fit automatically adjusts the scale to squeeze the entire drawing to the paper. No matter how big or small the drawing, progeCAD resizes it to fit the paper. progeCAD reports the scale under **Drawing units**, such as 9'5".

1:1, etc specifies the scale factor for the drawing. Many disciplines work with specific scales, such as 1:50 or 1"=250'.

Printed Units — the first number is the length on the paper (1"),

Drawing Units — the second number is the length in the drawing.

All drawings are plotted in inches or millimeters, depending on the units of the paper and the drawing:

- ⊙ **Inches** — uses inches (Imperial or British) units for the paper size. This allows you to plot metric drawings on imperial paper, as is common in Canada.
- ○ **Millimeters** — uses millimeters (metric) units for the paper size.

TIPS The **Paper size** is set by the printer driver, which is accessed through the **Device** tab **Properties** button.

The units of drawings are set with the **Units** command. Use decimal units for metric drawings.

Advanced Tab

Don't let the name of the Advanced tab intimidate you. It contains useful items, like plot centering and footers for labelling plots.
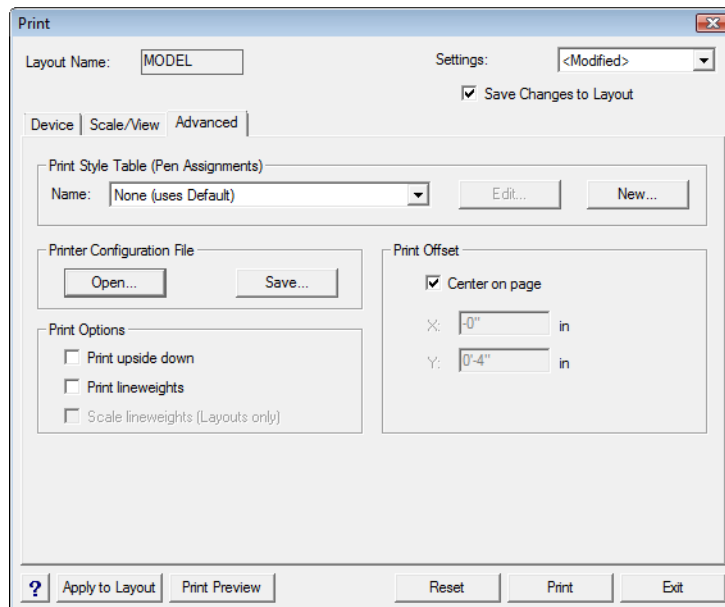
Print Style Table (Pen Assignments)

The Print Style Table section assigns plot properties to entities, such as width, screening, and lineweight. Don't worry if you see the "none-device (missing)" name. It means that a plot style has not yet been applied to the drawing.

Just as dimension styles affect the look of dimensions, plot styles affect the look of plotted drawings. They allow you to make the look of drawings seen on the screen independent of their look on paper. As noted earlier, there are two ways to assign plot styles: by colors and by entities.

Color-based styles map entity colors to plotter parameters. In the old days, back when the CAD universe was ruled by $20,000-pen plotters, this was the only way to do it: colors of entities were not meant to identify them in drawings, but to specify how they would be plotted.

Pen plotters typically had eight pens, usually black and of varying widths, so drafters would work with eight colors in their drawings. When it came time to plot, drafters specified which entity colors would be plotted by which pens. Weird? Yah.

Today's ink jet printers are capable of faking millions of colors and dozens of widths, so pen mapping just isn't needed anymore. But if you need to, here's how: the drafter might decide that all red entities should be plotted with the plotter's pen #1. Pen #1 could be red, but more often it would be black and a specific width. Once colors were mapped between entities and plotter pens, the settings can be saved to *.ctb* (**c**olor **tab**le) files for later reuse.

Today, drafters don't worry about matching pens; red entities are plotted red, and wide entities are plotted wide.

Entity-based styles map entities or layers to plotter parameters. When plot styles are enabled, the Explorer-Layer window displays an extra column so that you can assign plot styles. Settings are saved to *.stb* (**s**tyle **tab**le) files.

TIP   Drawings must be switched between color and entity plot styles. Drawings cannot have both styles at the same time. The plot style can be applied to new drawings only.

To switch between them, follow these steps:
1. From the **Tools** menu, choose **Options**.
2. In the Options dialog box, select the **Printing** tab.
3. In the Default Plot Style Behavior section, choose either of:
   - Use color-dependent print styles (*.ctb* files).
   - Use named print styles (*.stb* files).
4. From the droplists, choose a *.ctb* or *.stb* file.

About Plot Style Tables

When drafters want fine control over plots, they use *plot style tables*.

To see a sample plot style, select *icad.ctb* from the **Name** droplist, and then click **Edit**.
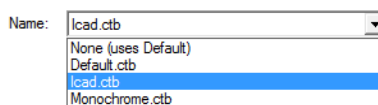
In the Print Style Table Editor dialog box, choose the **Properties** tab.
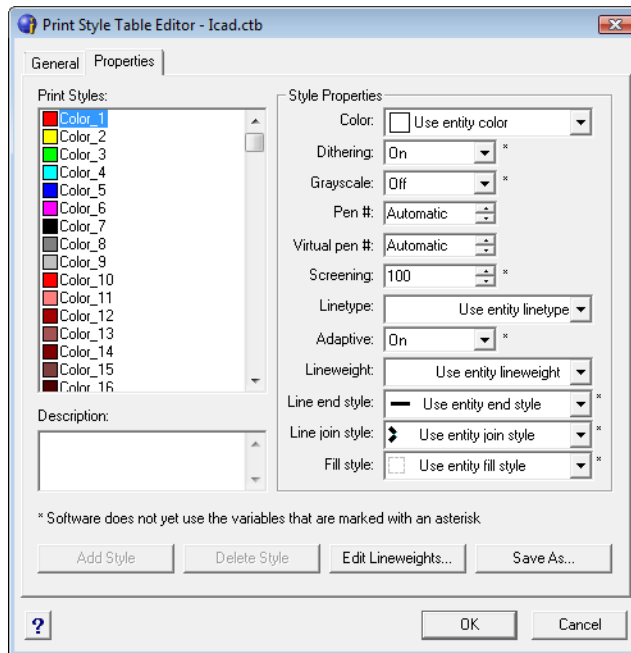
On the left are progeCAD's standard 255 colors, under **Print Styles**. On the right are the properties that can be assigned each color.

When a drawing is set up to use plot style tables, the dialog box looks very similar. The difference is that you see the names of plot styles, instead of the 255 entity colors. Each table can contain one or more styles, as illustrated below.
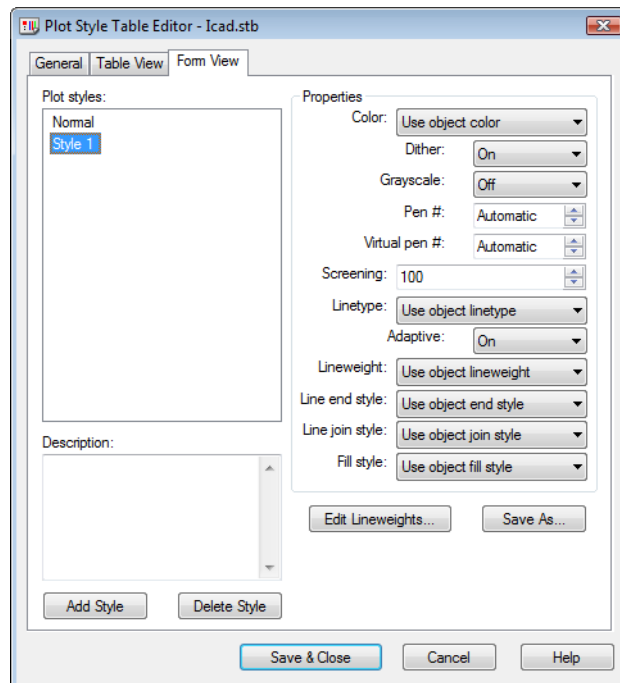
Color overrides the entity color with another color. To plot with the drawing's own colors, select "Use object color." The "True color" option is not yet supported.

Color: Use object color
- Use object color
- Red
- Yellow
- Green
- Cyan
- Blue
- Magenta
- Black
- Select Color...
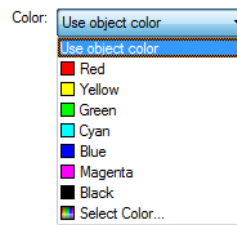
Pen # specifies which pen should plot the entity; range is 1 to 32. To let the printer determine the pen number, choose "Automatic."

Pen #: Automatic
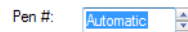
Virtual Pen # specifies a virtual pen number. Laser printers support the concepts of virtual pens, because they don't actually have pens. Range is 1 to 255.

Virtual pen #: Automatic

Screening specifies the amount of ink used for drawing objects. When set to a value less than 100%, the plotter uses many fine dots to print objects, instead of printing them in solid color.

Screening: 100

Linetype overrides the entity linetype with another linetype. For example, you may want to plot the entire drawing using dashed linetypes. To plot with the drawing's own linetypes, select "Use entity linetype."

Linetype: Use object linetype
- Use object linetype
- Solid
- Dashed
- Dotted
- Dash Dot
- Short Dash
- Medium Dash
- Long Dash
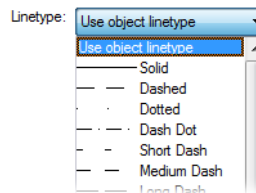
Adaptive determines whether linetypes are adapted to the length of entities. When this option is turned on, linetypes are forced to fit evenly between endpoints.

Adaptive: On
- Off
- On
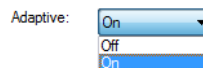
Lineweight overrides the entity line weight with another line weight. For example, you may want to plot the entire drawing using a single line weight. To plot with the drawing's own line weight, select "Use object lineweight."

Lineweight: Use object lineweight
- Use object lineweight
- 0.0000 mm
- 0.0500 mm
- 0.0900 mm
- 0.1000 mm
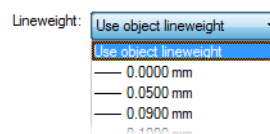
End Line Style specifies how the ends of lines (and other open objects) are plotted.
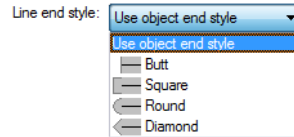
Line Join Style specifies how the corners of lines are plotted.
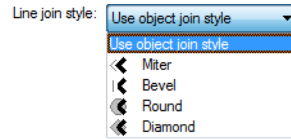
Fill Style specifies the fill pattern for objects. This options can be used in place of the Hatch command.
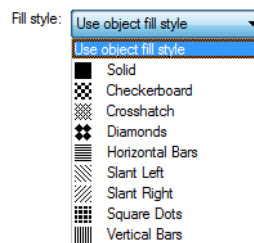
Configuration File

You can save plot settings to *.pcp* files. This is useful for sharing the settings with other progeCAD and AutoCAD users.
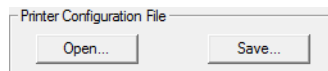
    Open prompts you to select a *.pcp* (plotter control parameters) file to open.

    Save saves the plotters settings in *.pcp* files.

Print Offset

progeCAD normally places the drawing in the lower left corner of the page. The Print Offset area allows you to change this position.
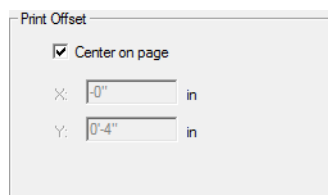
☐   **Center on page** centers the drawing on the page, when on. When off, the drawing is placed in the lower left corner. Frankly, I don't know why this option isn't turned on by default — centered plots look nicer.

X, Y allow you to specify an offset distance. When you plot drawings on preprinted sheets, you sometimes need to offset the drawing from the lower right corner of the page.

Finally, a few last options:
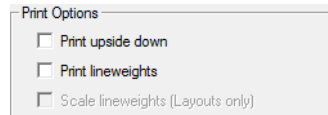
☐ **Prints upside-down** — turn on this option when you use preprinted paper and it is inserted the wrong way in the printer.

TIP   To produce mirrored plots, first mirror the entire drawing by pressing **Ctrl+A** followed by **Mirror**. Plot the drawing, and then use the **U** command to un-mirror the drawing.

☐ **Use Lineweight** — when on, tells progeCAD to send the printer wider lines, as defined by linetypes widths in the drawing. Generally, this option should be turned on.

☐ **Scale Lineweight** — changes the lineweight appropriate for the scale of the plot.

Header and Footer

The header and footer options are useful printing identifying text along the top (*header*) and bottom (*footer*) of plots. The header and footer information can be anything useful, such as the date of the plot or the file name. Simply type text in the text fields.

To specify headers and footers, use the Options command: From the **Tools** menu, select **Options**, and then choose the **Printing** tab.
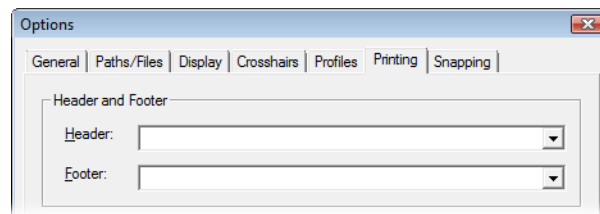
Header specifies the text to place along top of the plot.

Footer specifies the text to place along bottom of the plot.

progeCAD includes some preset text, such as today's date, the name and company of the registered user, and the word "Confidential." Click the droplist to access the preset text.

To see what the header and footer will look like in your plotted drawing, enter the **Print Preview** command.
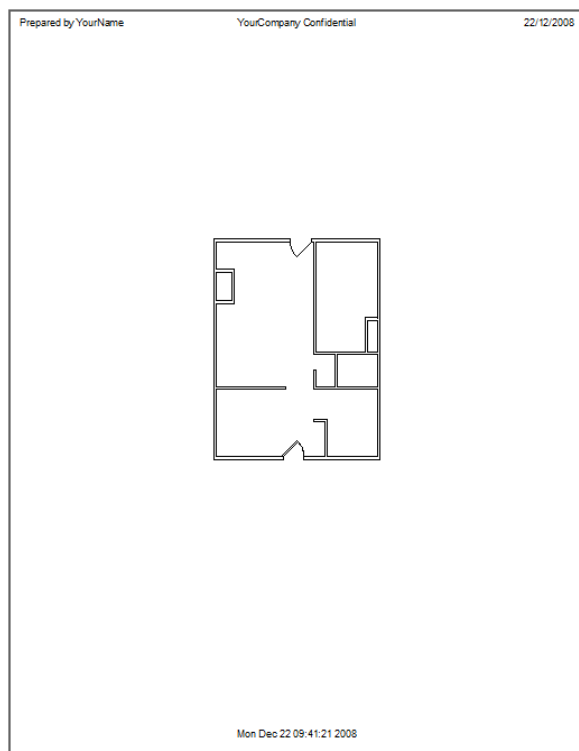
Electronic Plotting (Export)

Drawings can be plotted "electronically" with the **Plot** and **Export** commands. These saves drawing files in formats suitable for display in Web pages and delivery by email, such as DWF and PDF.

Export to DWF Format

DWF (design Web format) was developed by Autodesk in the late 1990s as a way to efficiently transmit drawings by email. The format strips out all unnecessary information and greatly compresses what remains to create replicas of drawings that are as much as 100x smaller.

For greater control over creating DWF files than provided by progeCAD, download the free DWF Writer software from progeSOFT's Web site at www.progesoft.com/?page=extras (scroll to the bottom of the page). Install it like a printer driver, and then select it with the **Plot** command. To access its options, click the **Properties** button, and then the **Advanced** button.

progeCAD can read and write DWF files. While AutoCAD can write DWF files, it cannot read them — ironically enough. To create DWF files of drawings, follow these steps:

1. From the **File** menu, select **Export**.

   (Alternatively, enter the **export** command.)

   > Command: **export**

2. In the Export Drawing As dialog box, click the **Save as type** droplist, and then choose one of the following DWF dialects:
   - **Design Web Format (.dwf)** — meant for 2D drawings.
   - **3D DWF (.dwf)** — meant for 3D drawings.
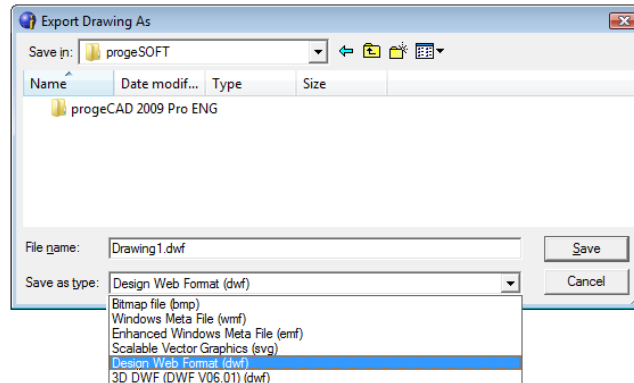
3. If necessary, use **Save In** to select a different folder and/or drive in which to save the file.

4. Click **Save**. Notice the dialog box. It lets you control the format of the DWF file.
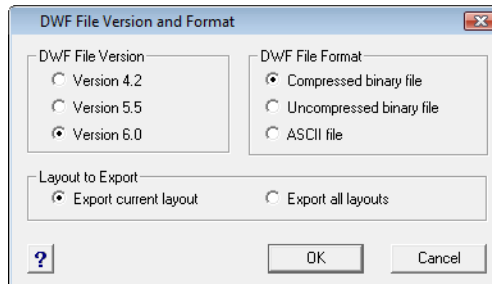
5. Choose the following options:
   - **DWF File Version** — Version 6. (Only use an older version if recipients cannot read the .dwf files you send them.)
   - **DWF File Format** — Compressed Binary File produces the smallest file format.
   - **Layout to Export** — choose Export Current Layout or Export All Layouts, depending on how much you want to include with the file. Note that "layout" refers to both model tab and to layout tabs.

6. Click **OK**.

7. In the command bar, progeCAD asks which entities to export. Enter **all** to export the entire drawing.

   > Select entities to export: **all**
   > Select entities to export: *(Press Enter to continue.)*

progeCAD exports the drawing; look for it in the folder specified by the Look In droplist. The files can be sent by email, added to a Web page, or viewed with viewing software. To view *.dwf* files, download the free DWF Viewer software from : www.autodesk.com/dwf. The package includes a stand-alone viewer and a plug-in for Internet Explorer.

TIP   To import DWF files, follow these steps:
  1. Enter the **Import** command.
  2. Select "Drawing Web Format (dwf)" as the file type.

Plot to PDF Format

PDF is short for "portable document format," a format created by Adobe for viewing documents. In recent years, it has also become popular for viewing and transmitting CAD drawings. To view *.pdf* files, you need the free Acrobat viewer software, available from www.adobe.com/acrobat.

To export drawings in PDF format, follow these steps:

1. Start the **Plot** command.
2. In the Print dialog box's Print tab, click the **Printer** droplist, and then choose "ProgeSoft PDF Wizard."
3. Change other options in the dialog box, as required.
4. Click **Print**. Notice the dialog box.
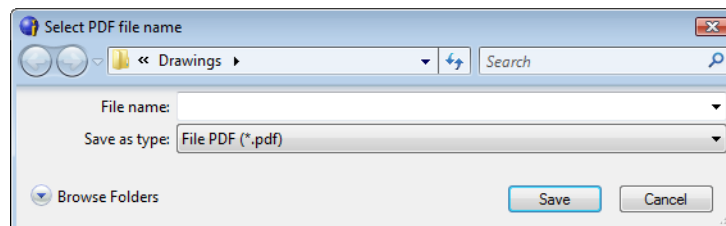
5. Enter a file name. If necessary, change the folder and drive in which to store the PDF file.
6. Click **Save**. progeCAD generates the PDF file, and then asks if you wish to see it.
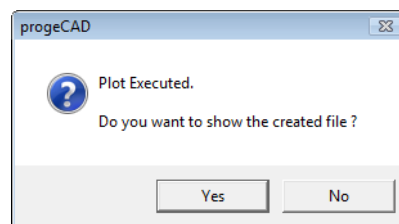
7. Click **Yes**, and the file is opened in Acrobat Reader.

Import PDF to DXF

progeCAD can read 2D and 3D PDF files, but in an indirect manner: the **Pdf2Cad** command converts the PDFs to DXF format. Vector data in PDF files are converted to vectors; images in the PDF files are attached as JPEGs. (DXF was invented by Autodesk in the mid 1980s as a way to exchange drawings; it is short for "drawing interchange format.")

To import PDF files, follow these steps:

1. From the **File** menu, choose **Convert PDF to DXF**.
2.   In the Pdf2CAD dialog box, enter the name of the PDF file; click **Browse** if you need help locating it.
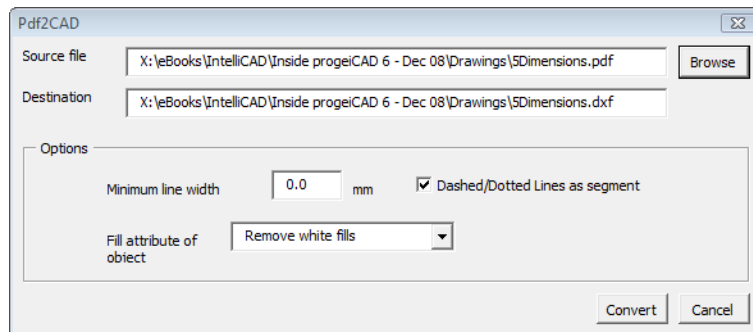
3. You can leave all of the options as they are. Here's what they mean:

    Minimum line width — specifies the lineweight of imported entities.

    Dashed/Dotted lines a segment — dashed lines are broken into segments.

    Fill attribute of object — specifies how to handle filled areas:

    - **Keep fills** keeps filled areas as they are.
    - **Outline all fills** removes all filled areas.
    - **Remove white fills** removes only fills that are colored white.

4. Click **Convert**.

    The PDF file is converted, and then attached to the drawing; non-vector entities are attached as a JPEG image.

Other Formats

progeCAD can export drawings in other electronic formats, all defined by Microsoft:

- BMP is a raster bitmap format.
- EMF is the enhanced metafile format, an improvement over WMF.
- WMF is the Windows metafile format, a kind of vector format.
- SVG is scalable vector graphics format.

SVG is the approved standard for displaying vector images (i.e., CAD drawings) on the Internet. Although it is an official standard, it isn't used much, although progeCAD can export it through the Export command. Once created, you can view the files with Internet Explorer, provided the appropriate plug-in is installed.

Learn more about SVG at www.w3.org/Graphics/SVG.

Sending Drawing Packages (eTransmit)

With the help of the eTransmit command, you can send a drawing and all of its related support files in a single package for transmission over the Internet or on a CD.

When sending drawing files to other users , it is can be necessary to attach dependent files, such as xrefs, images, and fonts. If these files are missing, then it becomes impossible for the recipient to view and edit the drawings correctly.

The eTransmit automatically includes dependent files in a transmission package, reducing errors down the road. If you wish, you can manually add other files to the package, such as documents, spreadsheets, and other drawings. You can also remove files from the package that eTransmit automatically added, such as those that might be restricted due to reasons of copyright or confidentiality.

TIP   If you want to send only a *.dwg* file, use the Send command from the File menu, as described below.

To create a group of files for transmission, follow these steps:

1. From the **File** menu, select **eTransmit**. (Alternatively, enter the eTransmit command.)
2. Notice that the Create Transmittal dialog box lists the name of the current drawing file (*5Dimensionsl.dwg*, in the figure below) and names of dependent files (*icad.fmp*).
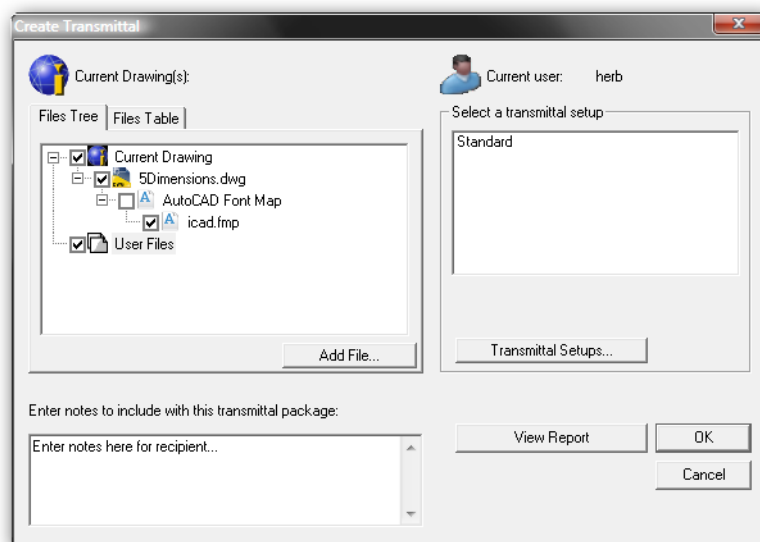
    The Create Transmission dialog box contains two two tabs that let you view and change the files to be added to the transmission package:

    Files Tree lists files in order of dependency. You can expand each drawing file to

view the related dependent files that progeCAD adds automatically to the transmis-sion package. To remove dependent files that you don't want to include in the trans-mission package, click the check boxes to remove the check mark.
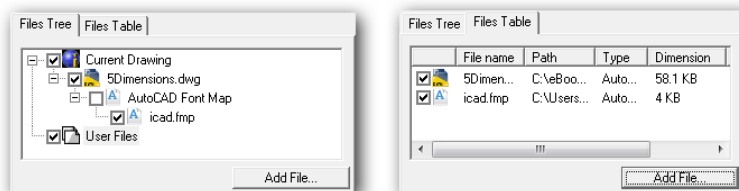
Files Table lists files in alphabetical order. This is a summary table that presents the names files, the paths of relative folders, and file details. You can select and deselect each file by clicking the check boxes.

3. To add files, click the **Add File** button. progeCAD displays the Add File to Transmit-tal dialog box. Choose a file, and then click **Open**. Repeat to add additional files.

4. Enter notes to include with this transmittal package. The notes are for the persons who receive the transmission package. (Later, eTransmit will generate a report file automatically that contains the list of the files present in the transmission package.) The contents of this report are displayed when you click the **View Report** button.
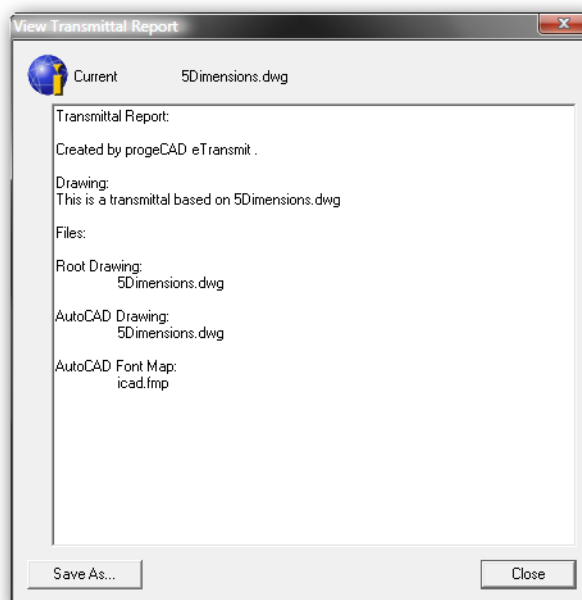
5. You can further fine-tune the transmittal with the Transmittal Setups options.

Transmittal Setups

Transmittal setups allow you to change settings of transmission packages.

If you need to send transmission packages several times within one project, it is better to save

it as a transmittal setups. The default setup is Standard. The Create Transmittal dialog box shows you a list of saved setups from which you can choose each time when you transmit a group of files.

To create new transmittal setups, follow these steps:

1. Click **Transmittal Setups**.

2. Notice the Transmittal Setups dialog box. It lists the names of saved setups, and lets you rename, modify, or delete them. (You cannot rename or delete the default setup, named "Standard.")
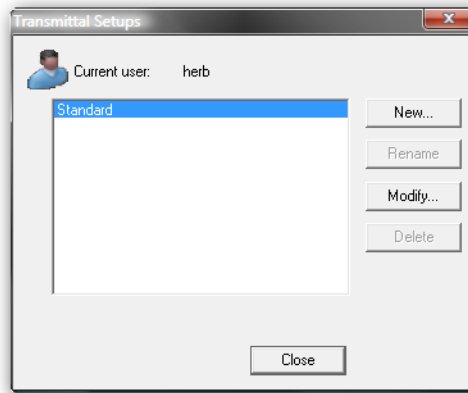
3. Click **New** to create a new setup.

4. Notice the New Transmittal Setup dialog box. Enter a name that describes the new setup.
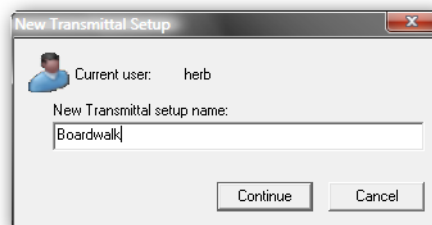
For this tutorial, enter "Boardwalk," and then click **Continue**.

5. Notice the Modify Transmittal Setup dialog box. The settings in the dialog box allow to you to specify the following options:

   • Compress the transmission package in a ZIP file or in a folder that is copied to a specified location.

   • Specify if the folder structure of the transmission files should be organized according to a logical hierarchy, inserted in a single folder, or copied to the addressee's computer as-is. If a FTP or HTTP destination is specified, the transmission package uses the single folder option.

   • Include *.shx* fonts used in the drawing, add password protection to the trans-mission package, set the default printer to "None," and send an email auto-matically after the transmission package creation is completed.
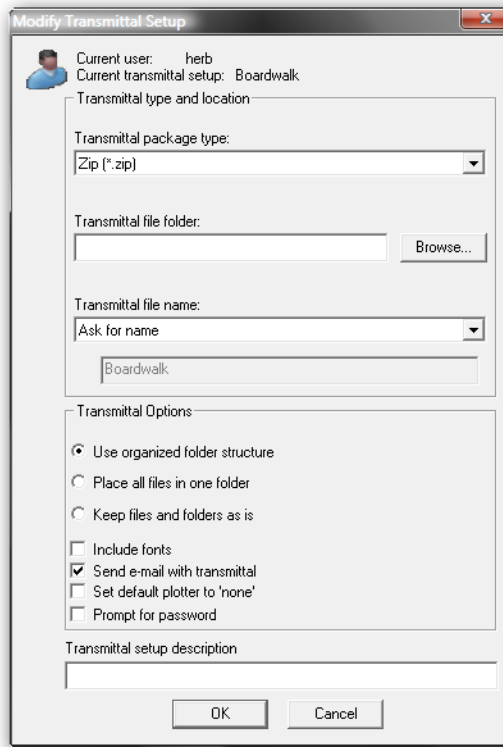
After creating the transmission package, you can make it available through the Internet or you can send it to other users as an email attachment. If you send the transmission package via email, there is an option in the Modify Transmittal Setups dialog box that launches the default system email application automatically, and attaches the transmission report file.

> **TIP** Full (absolute) paths of dependent files are converted to relative paths or "no path" to ensure that the drawing file can identify the dependent files.

6. Change options, and then click **OK**.

7. Notice that the Transmittal Setups dialog box now contains the new setup.
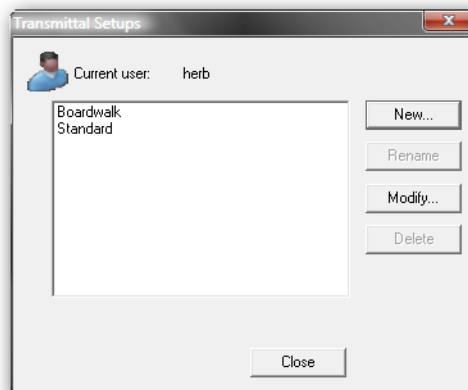
Click **Close**.

8. In the Create Transmittal dialog box, choose **Boardwalk**, and then click **OK**.

(If you chose the "Ask for name" option in the transmittal setup, then progeCAD prompts you for the file name and folder at this point.

If you chose the "Send email with transmittal" option, then progeCAD creates a new message with your default email software, and adds the transmittal package as an attachment.)

Emailing Drawing Files (Send)

progeCAD can send the current drawing as an attachment to email messages. From the **File** menu, select the **Send** command.

progeCAD creates a new message in your computer's default email software, and adds the current drawing as an attachment.

Creating Symbols

With the walls in place, it's time to populate the interior of the apartment. You need to add doors and windows, as well as furnishings. In this chapter, you create a door and a window from scratch, and then turn them into blocks (symbols), which are placed into walls. Later in this chapter, you learn how to insert ready-made symbols of furnishings and how to access symbol libraries on the Internet.

The fixed amenities include the bathroom fixtures, kitchen arrangement, and the fireplace. These remain part of the unit when it is purchased and sold.

Non-fixed amenities are provided by the purchaser; they are shown only to give an idea of how parts fit in the living spaces. These include the living room furniture, dining table and chairs, and bed.

In This Chapter

- **Rectangle** command
- **Insert** command
- **WBlock** command
- **Arc** command
- **DdInsert** command
- ALE block librarian
- **Block** command
- **ExpBlocks** command

Create Door and Window Symbols

When you need to draw the same kind of thing over and over, it makes sense to use *blocks*. Blocks are symbols. Floor plans, for instance, use lots of doors. Looking at our floor plan, it looks like about eight doors are needed.

Instead of drawing eight doors, it makes sense to draw one style of door, and then make copies of it. You could use the Copy command, but doors are made of several lines and arcs, and it can be tricky copying groups of lines. It's better to turn the group of entities into a single entity. You do this with the **Block** command.

To plan the doors, I printed out the drawing, and then sketched in the doors that are needed.
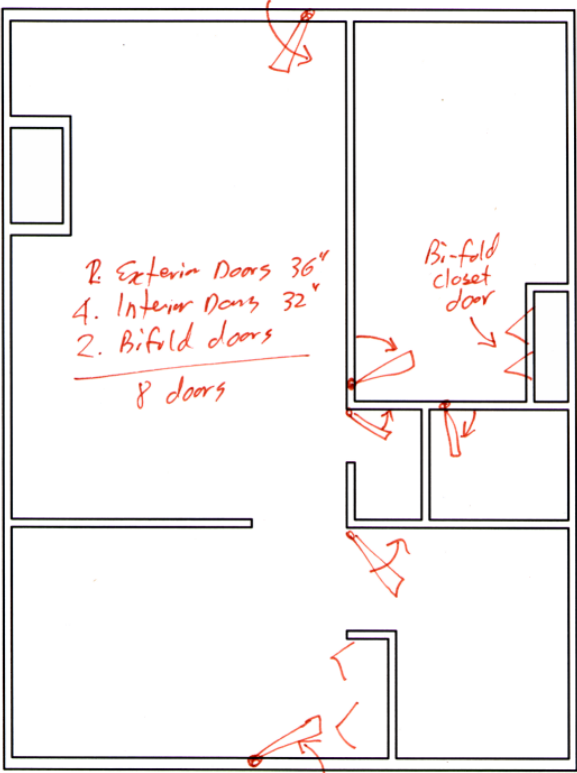
Looking closely, there are at least two kinds of doors: regular doors for entering rooms, and bi-fold doors for closets. Among entrance doors, exterior doors are usually larger (36") to make it easier to move furniture in and out, while interior doors are usually smaller (32") so that they take up less space. In summary, the following types of door are needed:

Type of Door	Number Needed
36"-wide exterior door	2
32"-wide interior door	4
Bi-fold door	2

That means you need three blocks, one for each type of door. Creating and placing symbols takes these steps:

1. Draw the symbol with the **Line**, **Arc**, and other drawing commands. (In this book, I use the term *symbol* for entities before they are turned into blocks.)
2. Convert the symbol into a block with the **Block** command.
3. Insert the block into the drawing with the **Insert** command.

(If you had problems completing the previous tutorial, the exterior walls drawing is available as *2InteriorWalls.dwg*.)

Step 1: Draw Door Symbols (Rectangle, Arc)

Door symbols consist of thin rectangles that represents doors, and arcs that show the *swing* of doors. Doors are usually shown open at 45 degrees. Symbols are often simplified representations of physical objects, so the door lacks details, such as hinges and knobs.
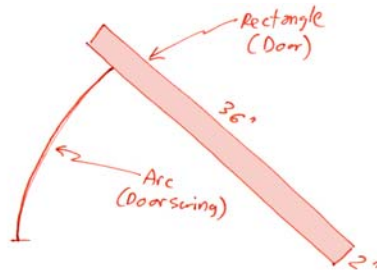
Here's is how to draw the first door, the 36" -wide, 2"-thick exterior door symbol.

1. In progeCAD, press **Ctrl+N** to start a new drawing. (If you are prompted for a template drawing, press **Enter** or respond with none.)
2. Turn on SNAP, GRID, and LWT modes by double-clicking the words on the status bar.
3. To draw the 36" x 2" door rotated at 45 degrees, use the **Rectangle** command, as follows.

   a. From the **Draw** menu, select **Rectangle**.

   (Alternatively, enter **rectangle** at the command prompt, or select the Rectangle button from the Draw 2D toolbar.)

   Command: **rectangle**

   b. Specify the rotation option, "r". progeCAD will ask later for the angle, after you specify the rectangle's size.

   Chamfer/Elevation/Fillet/Rotated/Square/Thickness/Width/
   <Select first corner of rectangle>: **r**

RECTANGLE COMMAND

Draws rectangles.

Command: rectangle

Aliases: rectang, rect, rec

Menu: Insert | Rectangle

Toolbar: Draw 2D | Rectangle

Options: Chamfer
Elevation
Fillet
Square
Thickness
Width

ALTERNATIVES

Polygon draws regular polygons.

Line draws lines.

c. Indicate the two opposite corners of the rectangle:

Chamfer/Elevation/Fillet/Square/Thickness/Width/<Select first corner of rectangle>: **0,0**
Other corner of rectangle: **36,2**

d. And then specify the angle of 45 degrees:

Rotation angle for rectangle <0.0000>: **45**

(If you cannot see the entire rectangle, use the **Zoom Extents** command:

Command: **zoom**
Zoom: In/Out/All/Center/Extents/Left/Previous/Right/Window/<Scale (nX/nXP)>: **e**

2. Other corner:
(36,2)

1. Start of rectangle:
Origin (0,0)

3. Rotation angle:
45 degrees
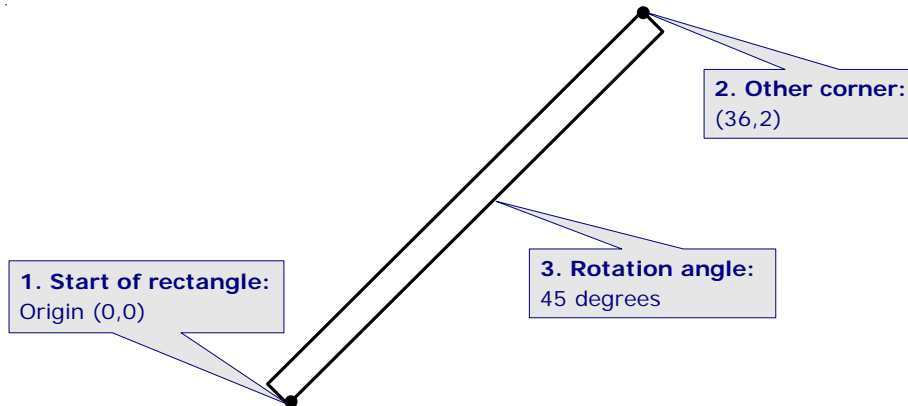
4. Indicate the door swing with the **Arc** command:

a. This arc is defined by its starting point, center point, and angle of rotation:

Command: **arc**

b. Use entity snap modes to start the arc at the end of the rectangle, as follows:

Start of arc: **end**
Snap to endpoint of: *(Pick point 1.)*

c. The center of the arc is the origin of the door, (0,0). Use the **Center** option, and
then specify the origin, as follows:

Angle/Center/Direction/End/Radius/<Second point>: **c**
Center of arc: **0,0**

d. The end of the arc is defined by the angle. progeCAD measures angles
counterclockwise, so you need to specify a negative angle of 45 degrees:

Angle/Length of chord/<End point>: **a**
Included angle for arc: **-45**

ARC COMMAND

Draws arcs.
Command: **arc**
Alias: **a**
Menu: **insert | Arc**
Toolbar: **Draw 2D | Arc**
Options: **Start**
          Center
          Angle
          Direction
          Radius
          End
          Follow
          Length of Chord

ALTERNATIVES

Polyline draws polylines and polyarcs.
Ellipse draws elliptical arcs.

Eighteen Ways to Draw an Arc

Arcs are one of the hardest entities to draw with CAD, because it can be difficult to determine ahead of time how they will turn out. And, because there are 18 ways to draw an arc! In general, progeCAD works with any 3 of the eight parameters illustrated by the figure below:
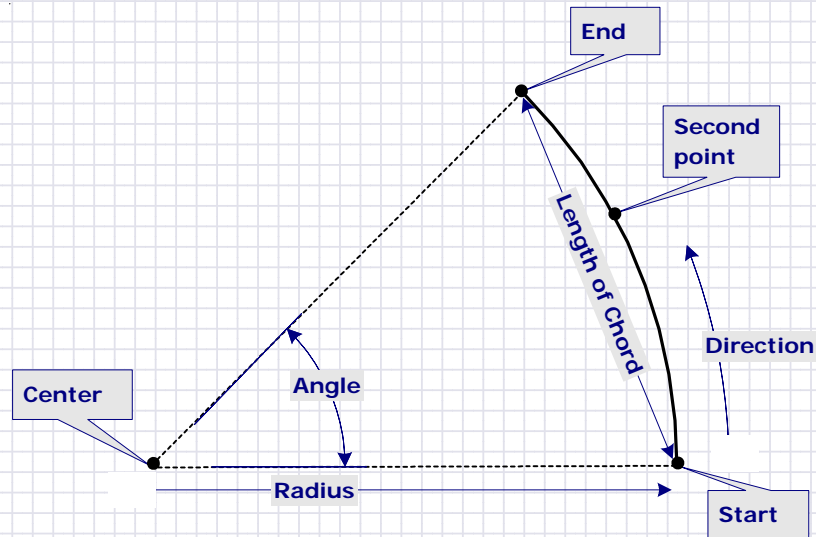
Arc Drawing Methods

The **Arc** command initially prompts you for the location of the arc's center or start points. After that, it prompts you for two other parameters, as listed by the table below. After you draw one arc, progeCAD has two more prompts that connect the new arc to the previous one: **Use last point** and **Follow**.

Initial Prompt	Second Prompt	Third Prompt
Center	Start	Angle
		Length of chord
		End point
Start of arc	Angle	Center
		End point
	Center	Angle
		Length of chord
		End point
	Direction	End point
	End	Angle
		Direction
		Radius
		Center point
	Radius	Angle
		End point
	Second point	End point
Use last point	End of arc	
Follow	End of arc	

1. Start of arc:
ENDpoint snap

3. Angle of arc:
-45 degrees

2. Center of arc:
Origin (0,0)
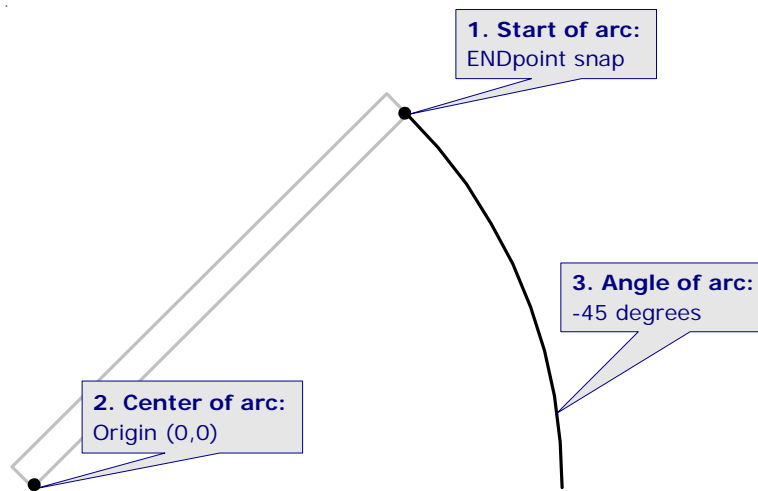
With the symbol drawn, it should look like this:

36" Door

Door Swing

Location of Hinge
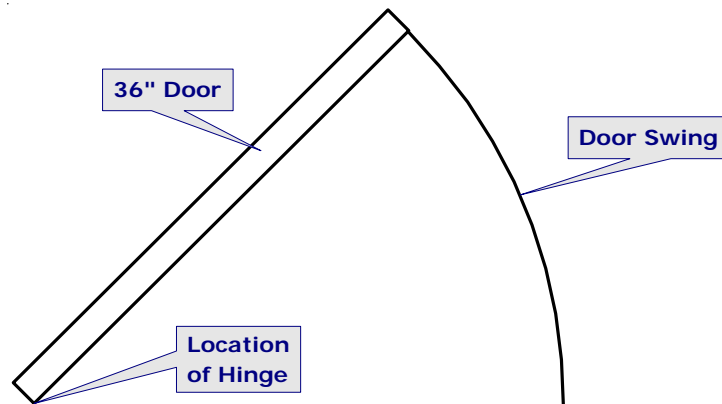
The next step is to turn the symbol into a block, so that it can be reused in this and other drawings.

Step 2: Convert to Blocks (Block)

The **Block** command converts collections of entities into a single entity, called a "block." Here's how to do it.

1. From the **Draw** menu, select **Block**, and then choose **Make**.

    (Alternatively, enter the **block** command, or click the Create Block button on the Tools toolbar.)

        Command: **block**

    Notice the Block Definition dialog box. To create a block, progeCAD needs to know three things: (1) a name for the block; (2) its base point; and (3) the objects that make up the block.

Block Definition dialog box with Name field, Base point section (Pick point, X: 0, Y: 0, Z: 0), Objects section (Select objects, Retain, Convert to block, Delete, 0 object selected), OK and Cancel buttons.
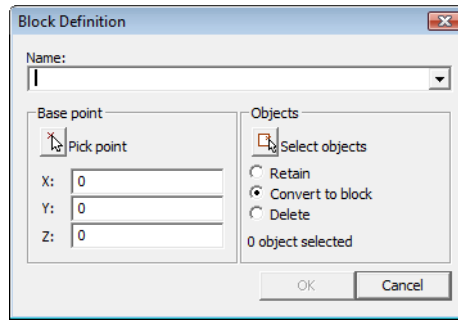
2. Blocks are known by their names. To help sort large collections of blocks, it makes sense to name blocks logically.

   For instance, I've named this block "door-36-exterior" — a name goes from the general ("door") to the specific (width and type). In this way, all the door blocks are listed together.

   > Name: **door-36-exterior**

3. When you later place the block in the drawing, the point you pick is called the "insertion point." You need to tell progeCAD where that insertion point needs to be.

   Most often, you would click the **Pick Point** button, and then pick a point that makes sense, like one corner or the center of the symbol.

   For this door, pick the location of the hinge, which just happens to be at the origin:

   > X: **0**
   > Y: **0**
   > Z: **0**

4. Now you get to select the entities, i.e., the rectangle and the arc.

   a. Click the Select Entities button.

   b. Notice that the dialog box disappears (temporarily), and that progeCAD is prompting you in the command bar:

   > Select entities: **all**
   > Select entities: *(Press Enter to return to the dialog box.)*

   Back in the dialog box, progeCAD reports the number of entities it found:

   > 2 objects selected

5. The last step is to decide on the fate of the symbol:
   - **Retain** — creates the block definition, and retains the symbol in the drawing.
   - **Convert to block** — creates the block definition, and inserts it into the drawing.
   - **Delete** — creates the block definition, and erases the symbol.

BLOCK COMMAND

Creates blocks from entities.

Command: **block**

Alias: **-b**

Menu: **Tools | Create Block**

Toolbar: **Tools | Create Block**

Options: **Name**

          Insertion Point

ALTERNATIVES

Group groups entities together.

WBLock exports blocks and drawings to .*dwg* files.

Choose the Convert to Block radio button.
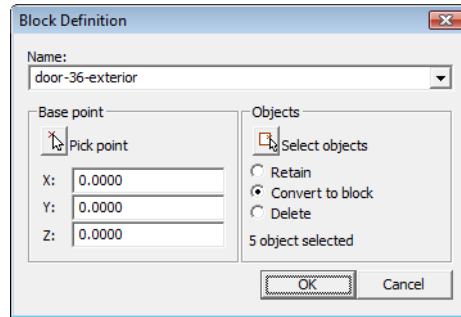
6. Click **OK** to end the command.

7. Save the drawing with the **SaveAs** command, using the name "Door-Exterior-36.dwg."

Sharing Blocks (ExplBlocks)

You created the block in one drawing, but need to use it in another. Sharing blocks between drawings is a easy as copy'n paste. (If you had difficulty creating the 36" exterior door block, you can access it from the *door-exterior-36.dwg* file.)

1. Open the Explorer to the blocks section with the **b** alias, a shortcut for the ExpBlocks command.

2. In Explorer, select the "door-36-exterior" block, and then press **Ctrl+C**. This copies the block to the Clipboard.
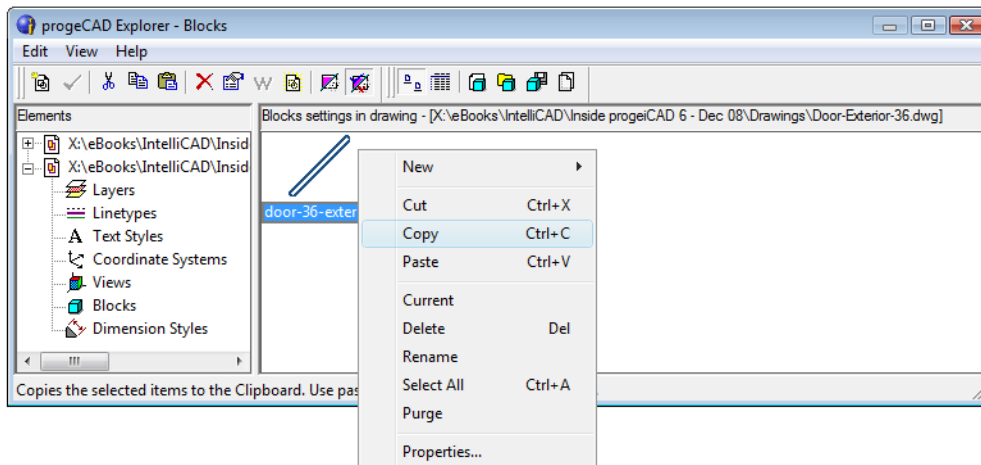
2. In Explorer's Elements pane, select to the *2InteriorWalls.dwg* file.

3. Select **Blocks**, and then press **Ctrl+V**. (Or, right-click and then choose **Paste**.)
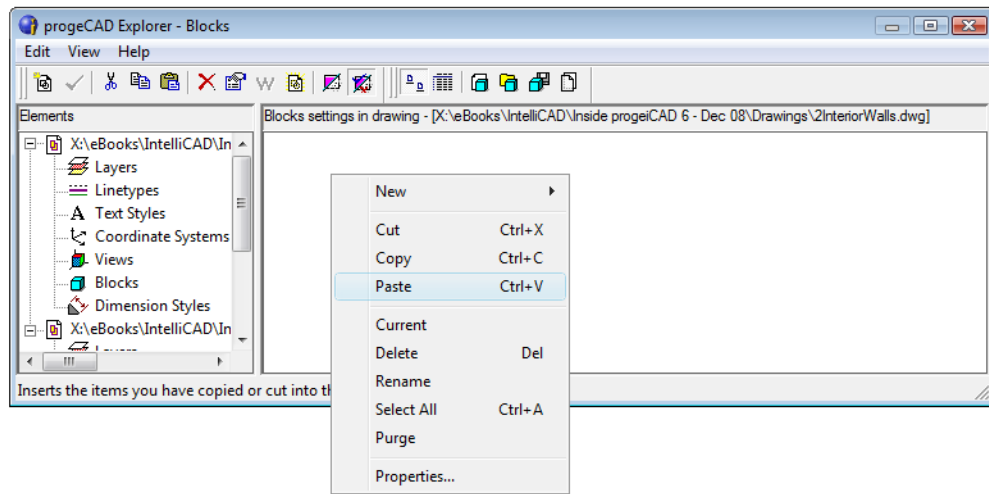
The block definition is added to the drawing, and appears in Explorer.

Step 3: Insert Blocks (Insert)

You now place the door block in the floor plan drawing. Technically, you insert the block *definition* to create block *instances*. There are two ways to place blocks in drawings:

- With the Explorer.
- With the **Insert** command.

We'll use both methods, one for each block insertion.

1. First, change the working layer to "Assets-Fixed."
2. The first door to be inserted is the entrance from the hallway, the door at the bottom of the floor plan. It's helpful to zoom to it, so use the **Zoom Window** command:

   Command: **z**

   Zoom: In/Out/All/Center/Extents/Left/Previous/Right/Window/<Scale (nX/nXP)>: **w**

   First corner: *(Pick a point.)*

   Opposite corner: *(Pick a point.)*

3. On the Explorer's toolbar, click the **Insert** button.
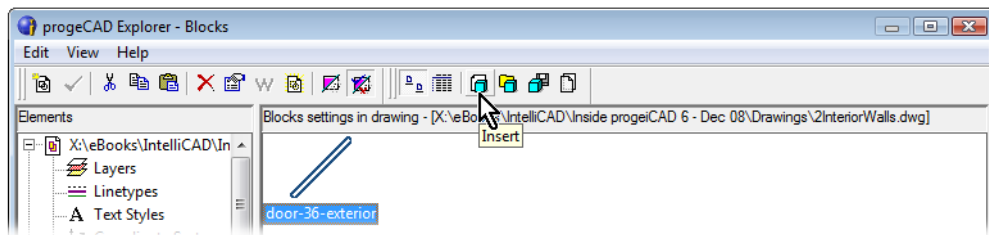
4. On the command bar, progeCAD prompts you:

   Command: _insert

   ? to list blocks in drawing/BROWSE/EXPLORE/<Block to insert> <DOOR-36-EXTERIOR>:
   DOOR-36-EXTERIOR

Notice that progeCAD runs the Insert command and fills in the name of the block for you.

5. Use the **NEArest** entity snap to help you insert the block accurately:

    Multiple blocks/<Insertion point for block>: **nea**
    Snap to nearest of: *(Pick a point on the wall.)*
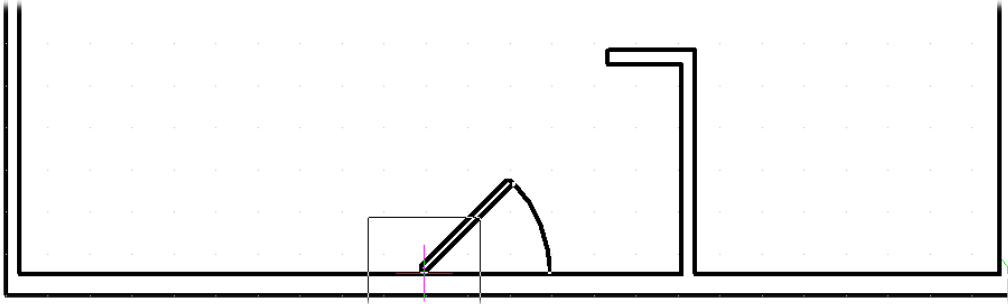
6. You have the option of scaling and rotating blocks during insertion, but this time you don't do this. Press **Enter** at each of the following prompts:

    Corner/XYZ/X scale factor <1.000000>: *(Press **Enter** to accept the default value.)*
    Y scale factor: < Equal to X scale (1.000000)>: *(Press **Enter** to accept the default value.)*
    Rotation angle for block <0>: *(Press **Enter** to accept the default value.)*

7. The door block is inserted. But there is the matter of the door frame: it doesn't exist. Draw the door frame as a pair of lines, using the appropriate entity snaps:

    Command: **line**
    Start of line: **int**
    Snap to intersection of: *(Pick at the door's hinge point.)*
    Angle/Length/<End point>: **per**
    Snap to perpendicular of: *(Pick a point on the opposite wall line.)*
    Angle/Length/Follow/Undo/<End point>: *(Press **Enter** to end the command.)*
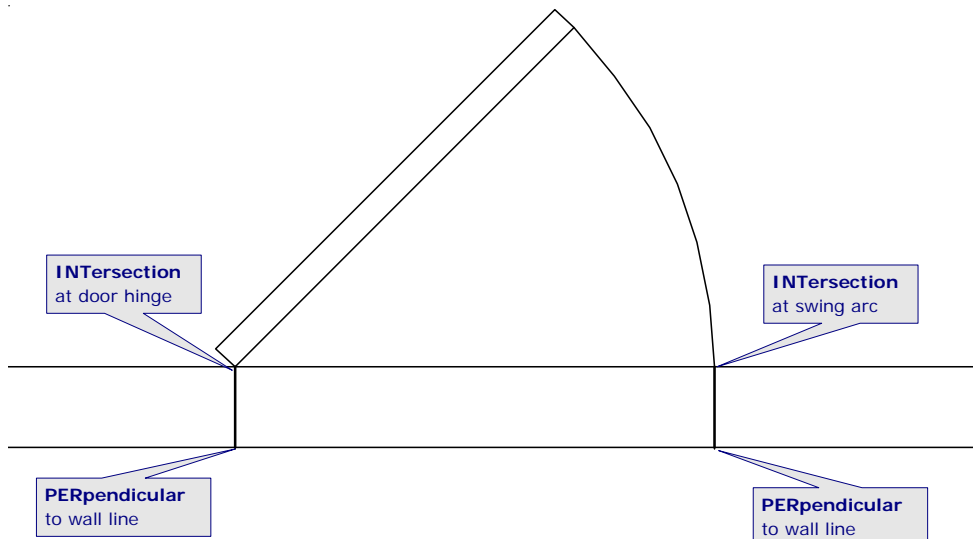
8. Draw the line for other side of the frame:

Command: **line**
Start of line: **int**
Snap to intersection of: *(Pick at the door's swing arc.)*
Angle/Length/<End point>: **per**
Snap to perpendicular of: *(Pick a point on the opposite wall line.)*
Angle/Length/Follow/Undo/<End point>: *(Press **Enter** to end the command.)*

9. Use the **Trim** command to cut away the unneeded portion of the wall.

Command: **trim**
Select cutting entities for trim <ENTER to select all>: *(Pick one door line.)*
Entities in set: 1
Select cutting entities for trim <ENTER to select all>: *(Pick other door line.)*
Entities in set: 2
Select cutting entities for trim <ENTER to select all>: *(Press **Enter** to end selection.)*
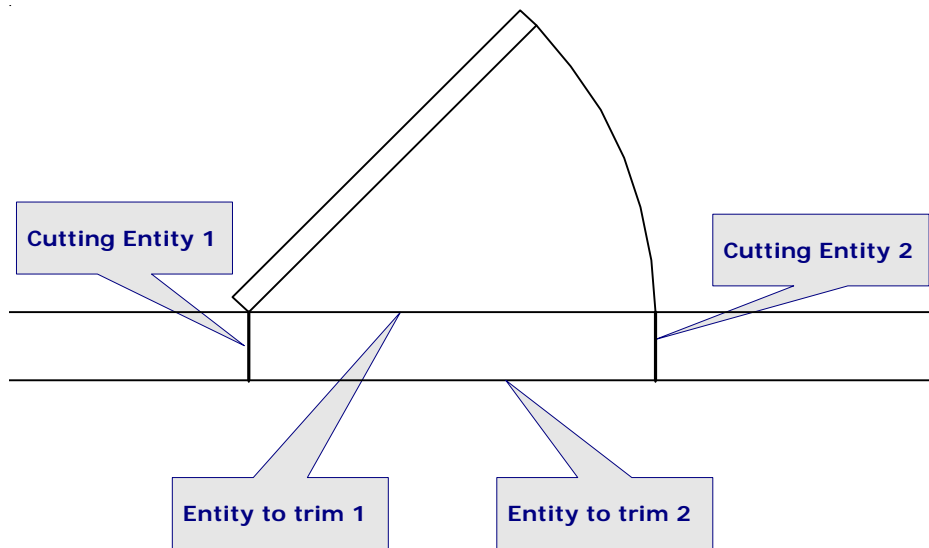
Edge mode/Fence/Projection/<Select entity to trim>: *(Pick one wall line.)*
Edge mode/Fence/Projection/Undo/<Select entity to trim>: *(Pick other wall line.)*
Edge mode/Fence/Projection/Undo/<Select entity to trim>: *(Press **Enter** to end command.)*
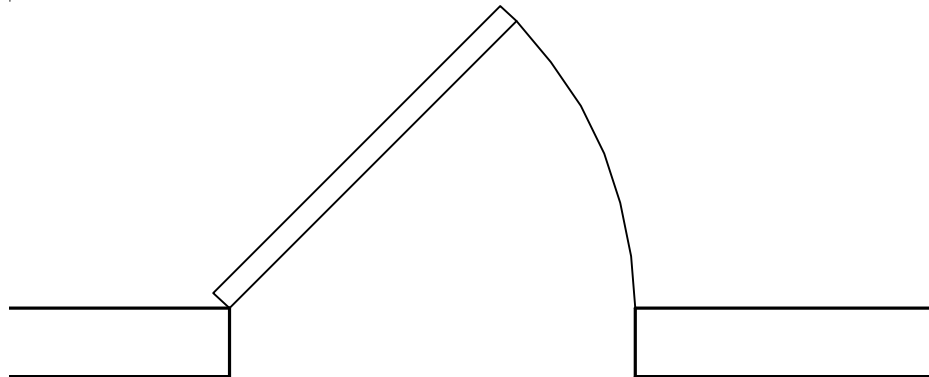
Don't forget to create and insert blocks of the interior doors (32" x 2") and the bi-fold closet doors (24" x 1"). They are both available as *Door-Interior-32.dwg* and *Door-Bifold-48.dwg*.

Inserting Blocks Upsidedown

Now insert the same block, but at the other end of the apartment: the door to the patio. This door swings in, so that when open, the door is against the living room wall. This time, use the **Insert** command, as follows:

1. Start the **Insert** command.

   (Alternatively, from the **Insert** menu, select **Insert Block**, or from the Draw 2D toolbar, select Insert Block.)

   > Command: **insert**

2. progeCAD prompts you for the name of the block. Because this is the same block as before, you can press **Enter** to accept the default name.

   > Command: **insert**
   >
   > ? to list blocks in drawing/BROWSE/EXPLORE/<Block to insert> <DOOR-36-EXTERIOR>: *(Press Enter to accept default.)*

3. Use **NEArest** entity snap to locate the insertion point on the inner wall line.

   > Multiple blocks/<Insertion point for block>: **nea**
   >
   > Snap to nearest of: *(Pick a point on the inner wall line.)*

> ┌─────────────────────────────┐
> │ **INSERT COMMAND** │
> │ *Inserts blocks with prompts in the command bar.* │
> │ Command: **insert** │
> │ Options: **Name** │
> │   **Scale Factors** │
> │   **Rotation Angle** │
> │ **ALTERNATIVES** │
> │ **DdInsert** inserts block via dialog box. │
> │ **Xref** inserts drawing files. │
> │ **XBind** converts xrefs into blocks. │
> └─────────────────────────────┘
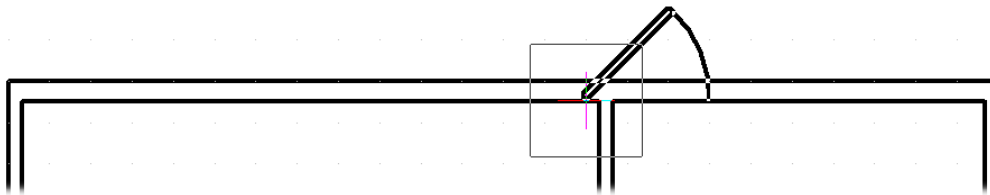
Notice that progeCAD ghosts an image of the block. That allows you to see where the block will end up.

4. The block is not, however, positioned correctly. The block needs to be flipped horizontally and vertically. progeCAD does this when you specify negative scale factors, as follows:

   > Corner/XYZ/X scale factor <1.000000>: **-1**
   >
   > Y scale factor: < Equal to X scale (-1.000000)>: **-1**
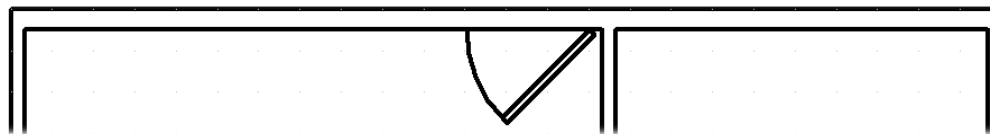
5. Specify zero for the rotation angle:

   > Rotation angle for block <0>: **0**

6. Use the **Line** and **Trim** commands to cut out the door opening, and then save the drawing.
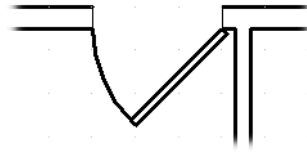
Add Window Blocks (Unit Blocks)

The apartment needs windows. The sketch shows their location and length.
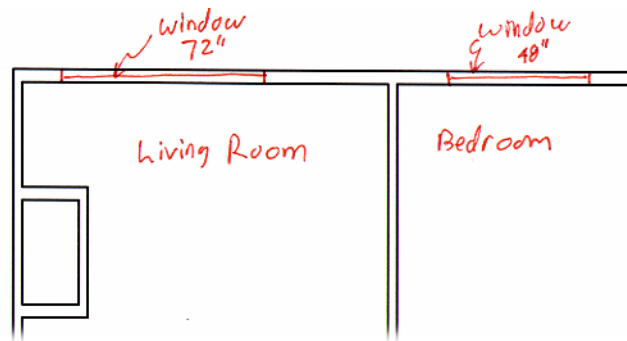
Window symbols typically look like this:
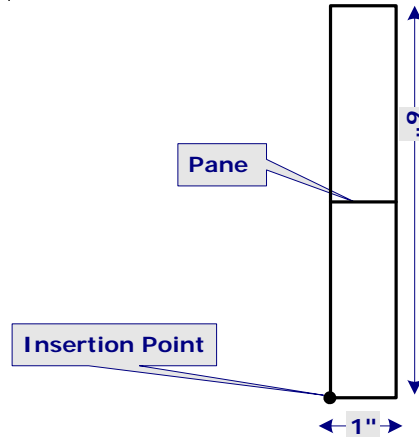
The center line represents the window pane; the other lines are the outline of the window frame.

The symbol is 6" tall, because this window fits the 6" exterior walls. But why is it just 1" wide?

Because it is a *unit block*. Unlike the door block, whose size does not change, unit blocks change their size upon insertion. For example, the windows you need are 72" and 48" long.

By drawing the block 1" long, the scale factor determines the inserted length. Need a 72" window? Specify 72 as the x-scale factor. It's that easy.

Follow these general steps to create the window block and insert it in the drawing:

1. Start a new drawing.
2. Draw the 1 x 6 window symbol with the **Line** command.
3. Convert to a block, naming it "Windows-Exterior." Use the lower left corner as the insertion point.
4. Save the drawing as "Window-Exterior."
5. Using Explorer, copy and paste the block to the floor plan drawing.
6. This time, use the **DdInsert** command to place the window blocks. The command displays a dialog box for specifying the insertion parameters.
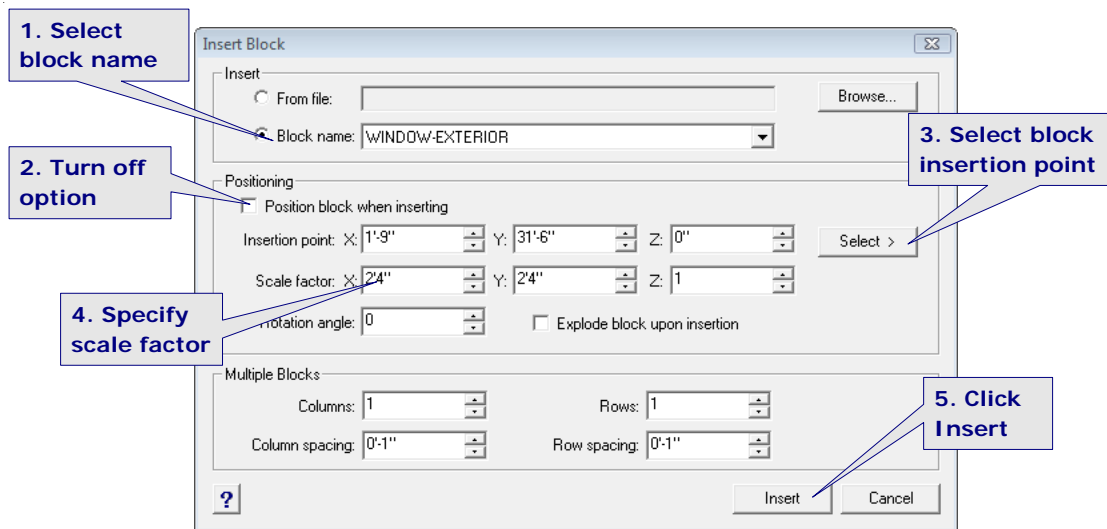
From the **Block name** droplist, select the name of the block, "Window-Exterior."

Turn off the **Position block when inserting** option. When on, you specify the position in the drawing; when off, you specify parameters in the dialog box.

7. Click the **Select** button. In the command bar, progeCAD prompts you to pick the insertion point. Use the **NEArest** entity snap to aid you:

> Select insertion point: **nea**
> Snap to nearest of: *(Pick a point on the inner wall line.)*

8. Change the **Scale Factor: X** to **72**. This stretches the block to 72".
9. Click **Insert**. The window appears in the drawing at the correct location and width.
10. Repeat for the bedroom window. The final result should look similar to the figure below.

11. Save the drawing.

ALE Block Librarian

progeCAD includes a sophisticated block libriarian for managing symbols on your computer. ALE lets you work with categorized blocks, previews them, and lets you preset the scale factor and rotation angle.
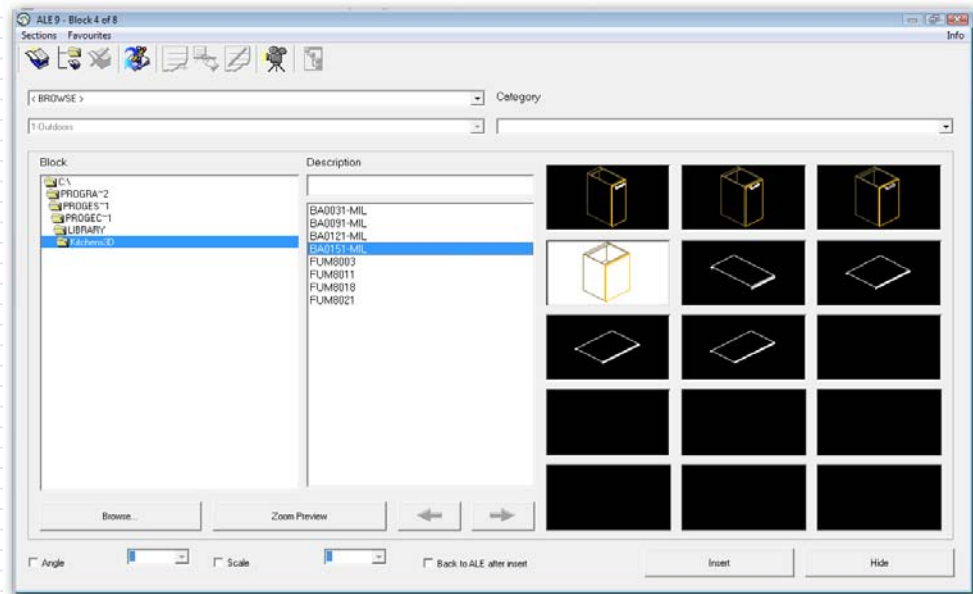
To access ALE, enter the **ale** command at the 'Command:' prompt.

Insert Blocks from Disc

The power behind blocks is that you don't need to create them from scratch. Instead, you can use libraries of blocks created by others. For example, the following blocks are stored in drawing files found in the *PlumbingBlocks* folder provided with this book:
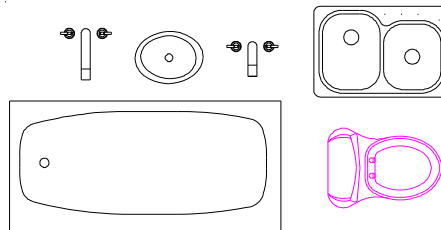
Follow these steps to insert blocks stored in drawings on disc:

1. Use the **i** alias to open the Insert Blocks dialog box.
2. In the Insert section, select **From File**.
3. Click **Browse**, and then go to the *PlumbingBlocks* folder.

3. Select the *Tub-Topview.dwg* file. Notice the preview window lets you see the block in the drawing.
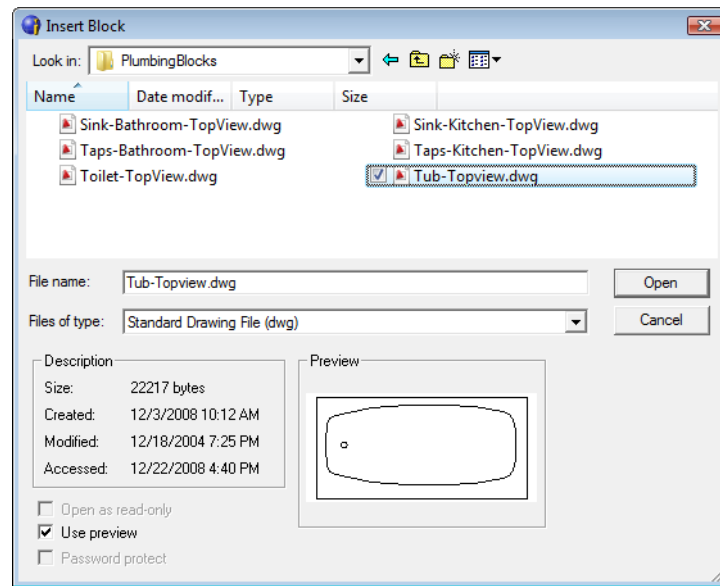
4. Click **Open**, and then place the block in the bathroom by clicking **Insert**.

    Finish the drawing with the fixed assets in the kitchen and bathroom. (Kitchen counters are often 30" deep.) When you are done, the floor plan should be something like the figure below.
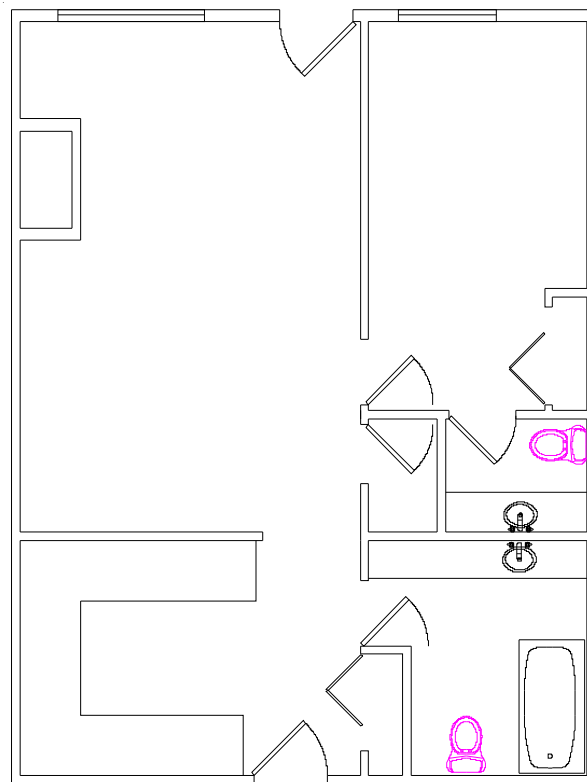
TIPS Any progeCAD *.dwg* drawing file can be used as a block in drawings; it doesn't just need to be drawings of symbols.

Just as you can open drawings from disc, you can also save blocks to disc. Use the **WBlock** command to save
• Selected entities.
• A specific block.
• Two or more blocks.
• The entire drawing.

Obtain Blocks from the Internet

Numerous companies and government agencies provide pre-drawn symbols free from the Internet. Companies often provide symbols free in the hope that you will specify their products, boosting sales. Government agencies provide them free to encourage standardization.

You can find CAD symbols on the Internet by searching Google with terms like "CAD symbols free" and "DXF blocks free." See the boxed text for sources of symbols.

Symbol Sources on the Internet

Ceco.net has a set of three thousand symbols for sale, but also a selection of free blocks from www.ceco.net. Click on Free Blocks of the Month.

Kohler has 2D and 3D symbols in DXF, DWG, and i-drop formats freely available from www.us.kohler.com/tech/cadsymbol/cadsymbol.jsp. The company specialized in residential and commercial fixtures for bathrooms.

You can download groups of symbols, or all of them. All 2D symbols is a 6MB download from www.us.kohler.com/tech/dxffile/All2d_dwg.exe. For plan views, use the drawing files that have "pln" in their name. Illustrated below is *1157pln.dwg*:
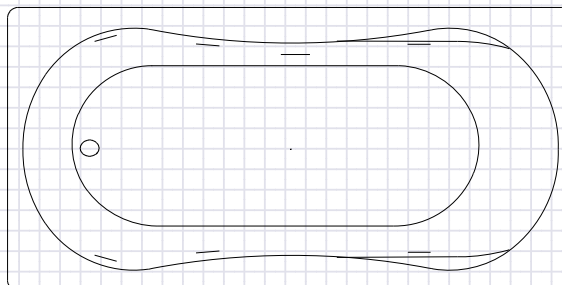
To figure out which file you need, refer to Kohler's "View Symbol List" at www.us.kohler.com/tech/cadsymbol/symbols.jsp?module=All. For example, model numbers up to K-1600 are bathtubs; K-4000 series are bidets; K-1000 series are body spas; K-5000 are drinking fountains; and so on.

THE BOLD LOOK
OF **KOHLER.**

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The Kohler i-drop Product Library allows you to easily "drag and drop" KOHLER product symbols and associated data files directly into i-drop aware Autodesk products that are 2004-based versions or newer. This includes products such as AutoCAD, Architectural Desktop (ADT), and Autodesk Building Systems (ABS). To take full advantage of the library's product specification and reporting features as well as the object-oriented features of ADT and ABS, be sure to install the Marinsoft PartDrop plug-in.

▸ How to use the Kohler i-drop Product Library

▸ Access the Kohler i-drop Library (installation of plug-ins required)

View/Download Symbols (self-extracting zip files)

2D Symbols

All Symbols	DWG (6000 kb)	DXF (5300 kb)	View symbol list
Baths	DWG (749 kb)	DXF (705 kb)	View symbol list
Bath & Shower Modules	DWG (103 kb)	DXF (82 kb)	View symbol list
Bathroom Sinks/Lavatories	DWG (763 kb)	DXF (694 kb)	View symbol list
Bidets	DWG (104 kb)	DXF (102 kb)	View symbol list
BodySpa Systems	DWG (272 kb)	DXF (267 kb)	View symbol list
Faucets	DWG (284 kb)	DXF (183 kb)	View symbol list
Furniture	DWG (284 kb)	DXF (183 kb)	View symbol list
Kitchen Sinks	DWG (1200 kb)	DXF (1100 kb)	View symbol list

Kit3D Models has 50 free furniture models available at www.kit3dmodels.com. Click on Free Download. The 3D symbols from this Spanish Web site are of modern furniture.

Thomas Register has millions of CAD drawings from thousands of manufacturers at www.thomasregister.com. The symbols are specific to manufacturing products, such as air conditioning units, braking systems, and electronic controls. You have to register with the site before being able to access symbols.

SteelCase provides a difficult-to-access software catalog of their office furnishings at www.steelcase.com/na/furniture_symbol_library_FSL.aspx?f=25196 . It requires that you register your mailing address with SteelCase, download a large software program, which runs in AutoCAD or MicroStation.

Human Figure Library at www.sum-cd.com has DXF files of people, cars, trees, and so on. A selection of 20 people files is available free. The text at the Web site is mostly Japanese (I think) but there was enough English for me to navigate around.

DesignCenter Online is Autodesk's free Web-based collection of blocks of architecture, mechanical, and electronics. Symbols are provided by CAD Easy <www.cadeasy.com> and are saved in DWG format, or dragged directly into drawings using *i-drop*. It seems to me that you can only access this site if you use AutoCAD's **ADCenter** command's DC Online feature, illustrated below.
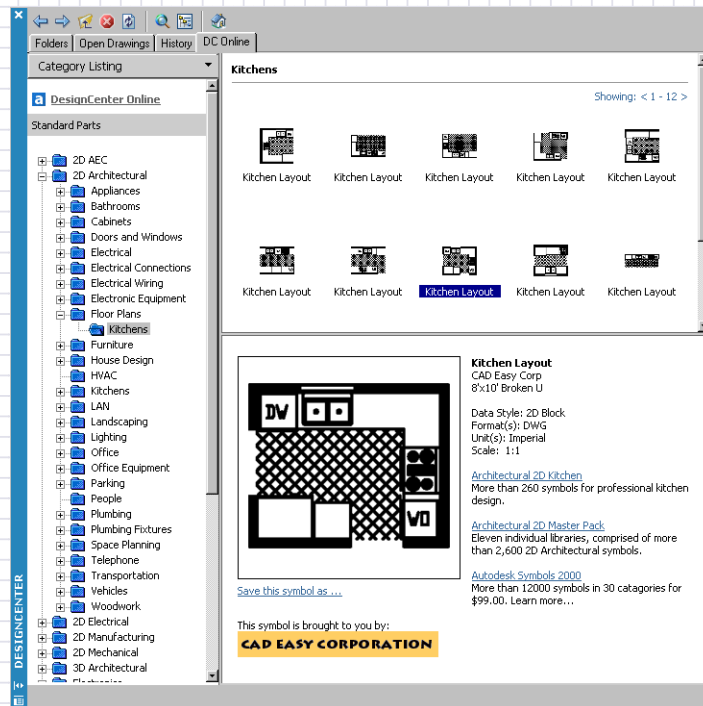
SteamCad is a collection of American steam locomotives drawn in AutoCAD at steamcad.railfan.net. All are free. Included in the collection is my favorite steam locomotive, the Shay. Site is slow loading and downloading.
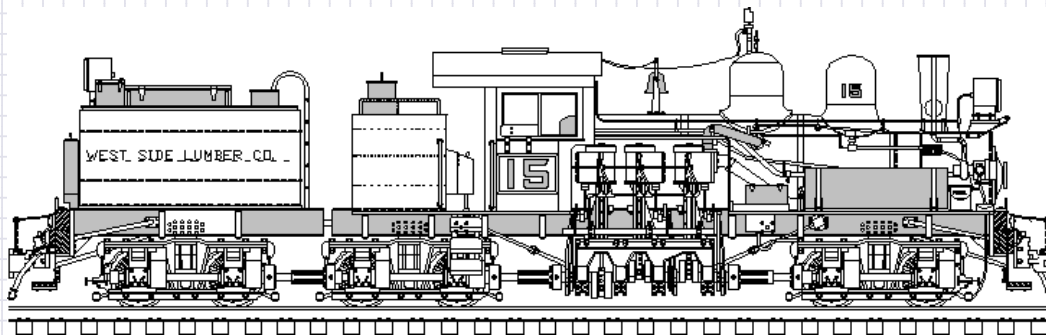
Placing Text

With the drawing of the walls and blocks largely complete, you need to add text to provide explanations of the apartment's features. This chapter describes how to place and edit text. It also shows how to add drawing borders and title blocks.

Important parts of drawings are labeled. Typical labels for floor plans include identification of the rooms and other areas. The figure below illustrates text to be added to the drawing.
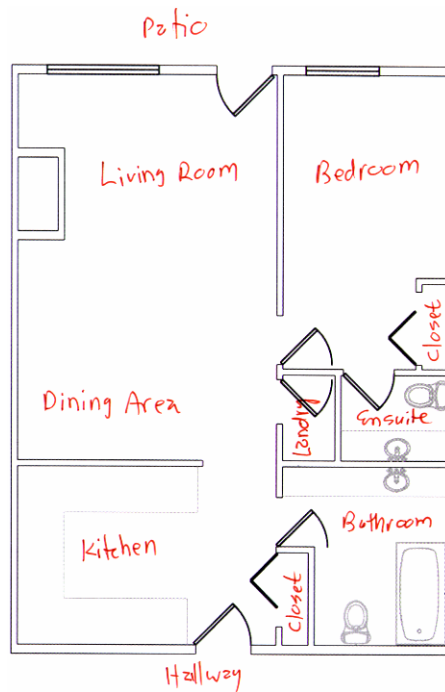
In This Chapter

- **Text** command
- **MText** command
- **MSpace** command
- **Style** command
- **DdEdit** command
- **PSpace** command
- **DText** command
- **MView** command

Placing Text (Text)

Begin by labeling the living room with the **Text** command.

1. Use the **Layer** command to create a new layer, "Text." Ensure the layer is current.

2. Start the Text command: from the **Draw** menu, select **Text**, and then choose **Single Line Text**.

   (Alternatively, click the Text button on the 2D Draw toolbar, or enter the **text** command in the command bar.)

   Command: **text**

3. Don't let the command's many options concern you. For now, just click anywhere along the left edge of the living room area.

   Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>: *(Pick a point.)*

4. Next, progeCAD wants to know the height of the text.

   Height of text <0">: **12**

TIP Q: How tall should text be? A: Tall enough to be legible when plotted.

The standard for text in drawings is 1/4", while headings are a bit taller at 3/8". If you were to use those numbers for text in this drawing, however, the text would be too small.

Instead, you need to make the text bigger, so that it appears 1/4" tall when printed out. How much bigger?

Recall that the **Print** command's dialog box reports the scale factor (see Chapter 9). progeCAD scales the drawing smaller so that it fits the page. For this drawing, the scale factor reported by the Print command is 1" = 4' or 48x smaller.
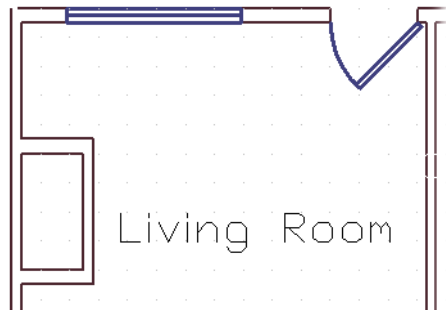

For 1/4"-tall text to be legible, it has to be 48x larger, or 48 x 0.25" = 12" tall.

---

5. Text can be placed at any angle. This text is horizontal, 0 degrees.

   Rotation angle of text <0>: *(Press **Enter** to accept default value.)*

6. Finally, you get to enter the text:

   Text: **Living Room**

TEXT COMMAND

Places lines of text.

Command: **text**

Aliases: **-t, tx**

Menu: **Insert | Text**

Toolbar: **Draw 2D| Text**

Options: **Style**

        Justification

        Height

        Rotation

ALTERNATIVES

MText places paragraphs of text.

DText repeats the 'Text:' prompt.

Leader places text with leader lines and arrows.

progeCAD places the text. But it looks kinda rough, as the figure below illustrates. The font, called "Txt," was designed in the early 1980s, when computers were slow. To make the display of text fast, all letters were made of straight lines; there are no rounded parts. For instance, the letter "o" is made of eight short lines.

To get nicer looking text, use a different font. progeCAD can use any font that works with Windows. (These are called "TrueType" fonts, and have an file extension of *.ttf.*) As well, progeCAD can use any font that come with AutoCAD, with the file extension of *.shx.*

Creating Styles (Style)

To use a different font in drawings, you use the **Style** command.

You may have noticed that the **Text** command lacked a "Font" option for specifying the font name. That's because fonts are selected with the **Style** command. You may be familiar with *styles* in documents: styles define fonts and their properties for headings, paragraphs, and so on — bold, underlined, superscript, and so on. progeCAD also uses styles to define the look of text.

To change the font used for the "Living Room" text, follow these steps:

1. In the command bar, enter **style**:

    > Command: **style**

    Notice that progeCAD displays the Text Style dialog box.
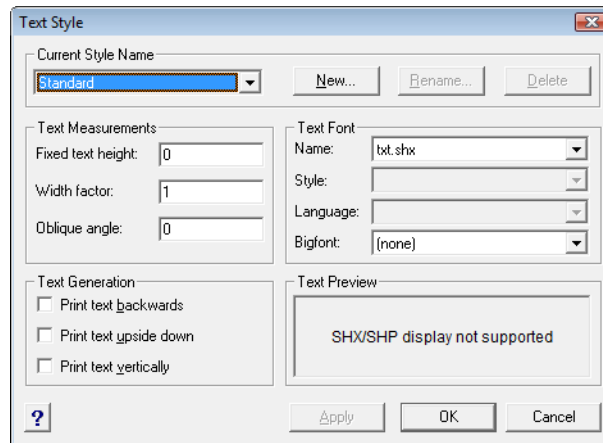
2. In the Text Font area, click the **Name** droplist.

    Select the "Arial" font, because this font is available on every computer. And it looks better than the Txt font.

3. Click **OK**. Notice that the "Living Room" text instantly changes to Arial font.

STYLE COMMAND

Creates and modifies text styles.

Command: **style**

Aliases: **st, font**

ALTERNATIVES

-Style creates styles at the command prompt.

**ExpStyles** displays the Explorer Styles window.

Controlling Text Styles

progeCAD provides three ways to create and change text styles:

- **Style** command displays a dialog box.

- **-Style** command prompts in the command bar; it is meant primarily for use in macros and LISP.)

- **ExplStyles** command displays the Explorer-Text Styles window.

All three do the same job; they just have different user interfaces. Try each out to see which you prefer. Here is what Explorer-Text Styles looks like:
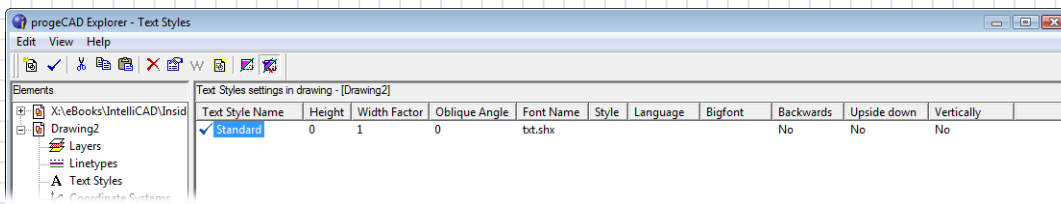
The options have the following meaning:

Height specifies the height of the text. When set to 0, the Text command prompts you for a height.

Width Factor makes characters wider (values over 1.0) or narrower (less than 1.0). Using a value of 0.85 allows you to squeeze in 15% more text per line, while keeping text legible.

Oblique Angle slants the characters forward (positive angles) or backward (negative). The old way of creating italic-looking text before italics became common in fonts.

Font name selects the font file from those available on your computer.

Backwards draws text backwards; used for printed circuit board drawings.

**Upside down** same as specifying 180 degrees for the rotation angle.

The following options are available only for those fonts that support the feature:

Vertically draws text vertically, one character below the next.

Style selects normal, bold, italic, or bold italic.

Language selects the language type.

Bigfont used for *.shx* fonts with Chinese and Japanese characters.

Living Room
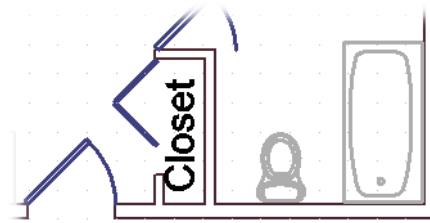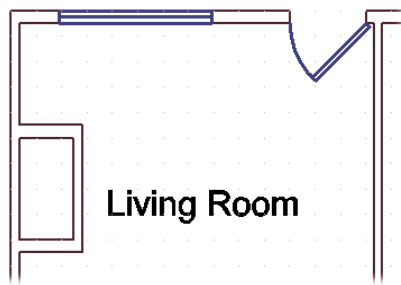
Closet

4. The closet labels are at right angles. Specify a rotation angle of 90 degrees, as follows:

> Command: **text**
>
> Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>: *(Pick inside closet.)*
>
> Height of text <1'-0">: *(Press Enter to accept default height.)*
>
> Rotation angle of text <0>: **90**
>
> Text: **Closet**

TIPS Text is just like other entities: you can use the **Move** and **Copy** commands to reposition text.

   Not all editing commands work on text; for example, you cannot use the **Stretch** or **Explode** commands.

Label Drawings (DText)

Repeatedly providing the options that don't change — height and rotation angle — gets annoying after a while. progeCAD has a second text command, called **DText** (short for "dynamic text"), that asks for them once, and then lets you repeatedly pick the insertion point to place the text. This command is excellent for placing labels in drawings.

1. To place the remaining room labels, enter the **DText** command in the command bar:

   > Command: **dtext**

2. The following prompts are identical to that of the **Text** command:

   > Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>: *(Pick a point in the drawing.)*
   >
   > Height of text <1'-0">: *(Press **Enter**.)*
   >
   > Rotation angle of text <0>: *(Press **Enter**.)*
   >
   > Text: *(Enter a room label, such as **Kitchen**.)*

Notice that the text appears, letter by letter, as you type. When you press **Enter**, the command does not end. Instead, the 'Text:' prompt repeats itself.

3. Position the cursor in another room, and then click. This indicates the new insertion point. Enter the room label.

   > Text: *(Move the cursor, and then click. Enter another room label, such as **Bedroom**.)*

> ## DTEXT COMMAND
>
> *Places lines of text, repeating the 'Text:' prompt.*
>
> Command: **dtext**
>
> Alias: **dt**
>
> Options: **Style**
>
>   **Justification**
>
>   **Height**
>
>   **Rotation**
>
> ## ALTERNATIVES
>
> **MText** places paragraphs of text.
>
> **Text** places lines of text.
>
> **Leader** places text with leader lines and arrows.

4. Finish labeling the floor plan: add text to the dining room, and so on.
5. When done, press **Enter** at the 'Text:' prompt to exit the command.

    Text: *(Press **Enter**.)*

When done, the drawing should look similar to the figure below. Save your drawing.
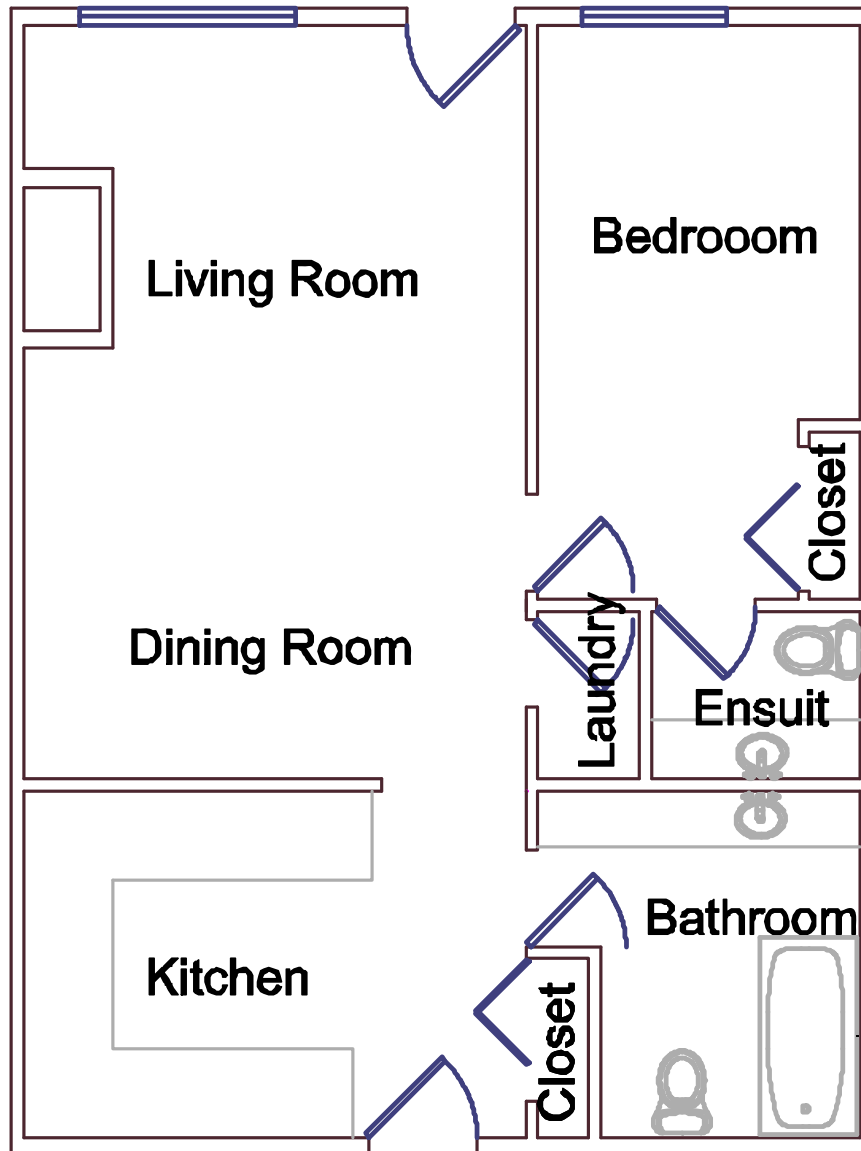
Editing Text (DdEdit)

Did you notice that "Ensuit" was misspelled? The **DdEdit** command allows you to change the wording of text, and correct mistakes. (DdEdit is short for "dynamic dialog editor.") You double-click misspelled words to correct mistakes.

Understanding Text Justification Modes

If you have difficulty placing text, it could be a problem with *insertion points* and *justification*.

Text entities have an *insertion point*, just like blocks. For text, the insertion point is where the text starts. To see the insertion point of text, select it: the green grip is located at the insertion point. You can use the **INSertion** entity snap with text.

.Living Room

But the insertion point can move around, depending on the *justification*. Justification tells progeCAD where to start the text. There are 15 justification modes, of which you use three most often:
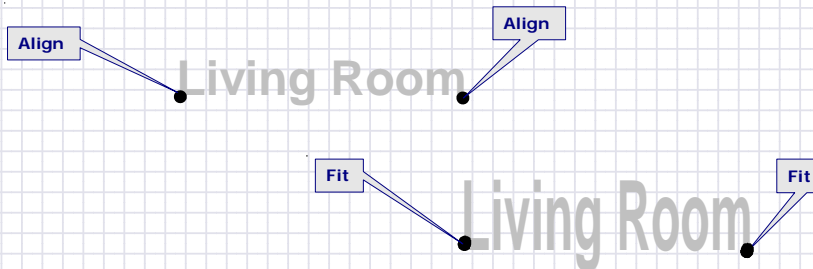
> **Left** — the default, a.k.a. "Start point."
>
> **Right** — for right-justified text.
>
> **Center** — for center-justified text.

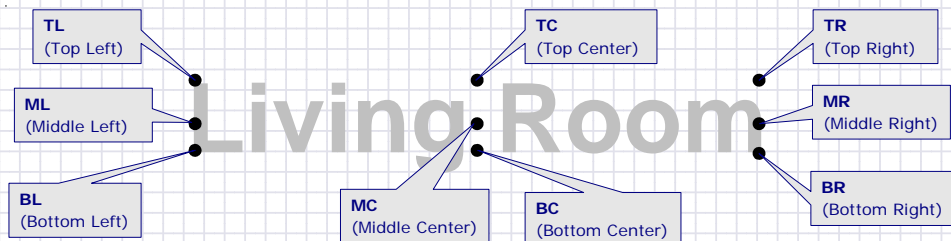Start Text (Left)   Living Room   Middle   Right

The others are Align, Fit, Middle, TL, TC, TR, ML, MC, MR, BL, BC, and BR, as illustrated below.

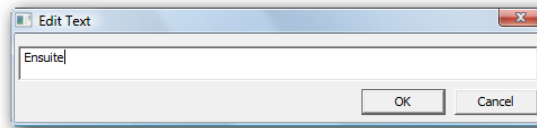**Align** (text keeps its width factor) and **Fit** (text keeps its height) justification modes:

Align   Living Room   Align

Fit   Living Room   Fit

And the many two-letter justification modes:

TL (Top Left)   TC (Top Center)   TR (Top Right)

ML (Middle Left)   Living Room   MR (Middle Right)

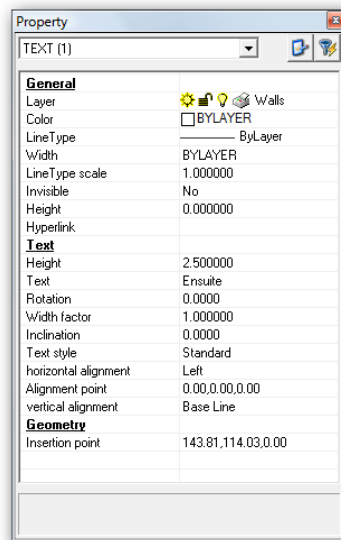BL (Bottom Left)   MC (Middle Center)   BC (Bottom Center)   BR (Bottom Right)

1. To edit the word, select it, and then double-click. Notice the Edit Text dialog box.

2. In the text box, change Ensuit to "Ensuite" and then click **OK**. Notice that the word is corrected in the drawing.
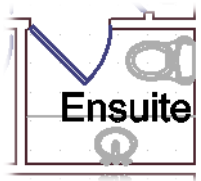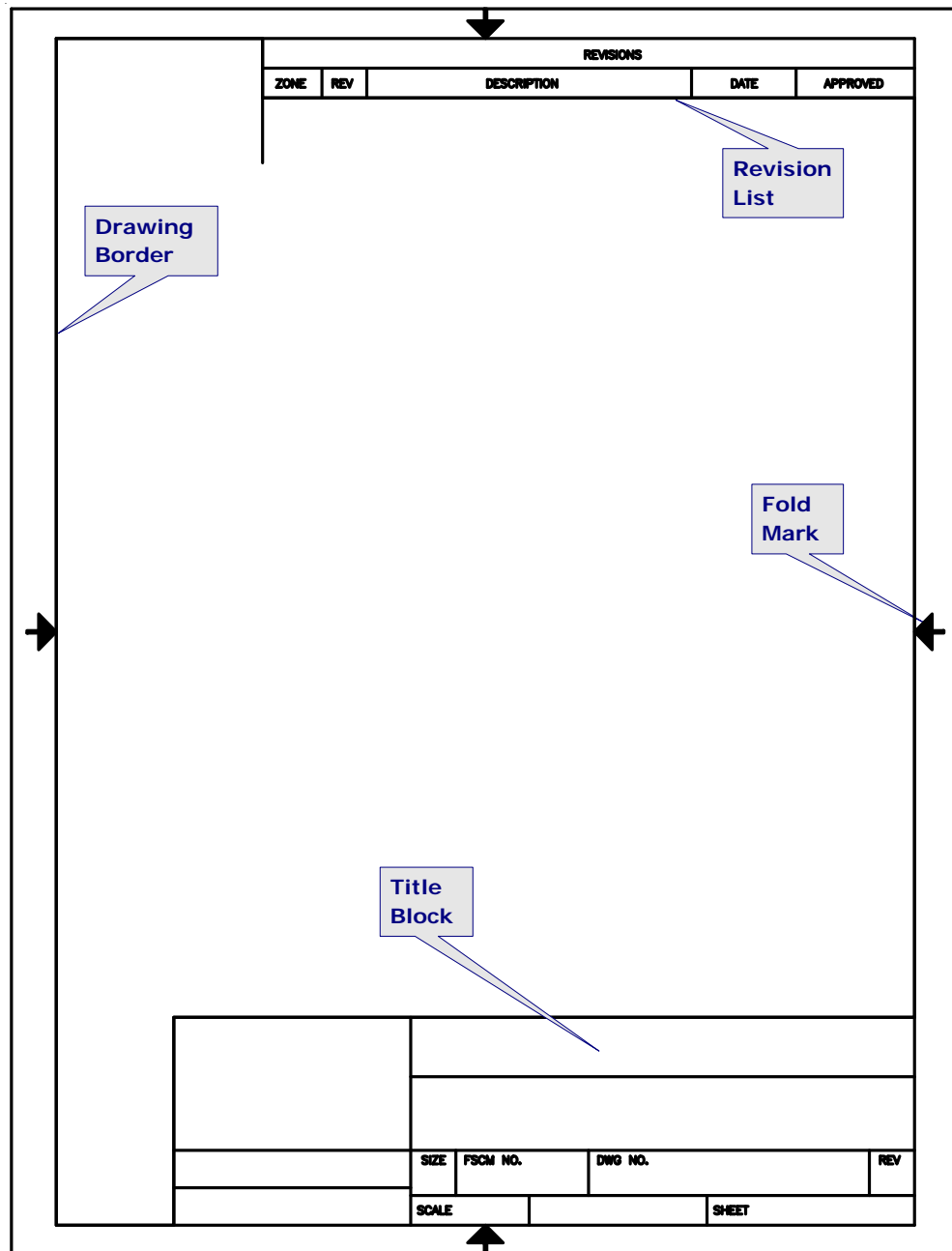
Borders and Title Blocks

Drawings are often drawn with *borders*. Borders are useful for identifying drawing areas — and give drawings a professional, finished look. Engineering and architectural firms some-times have their own drawing border standards; other times clients require the use of their borders. In addition, standards organizations such as ANSI (United States), DIN (Germany),

ISO (international), and JIS (Japan) specify the design of drawing borders.

For instance, the figure below illustrates the ANSI standard border for A-size portrait drawings.

Drawing Border shows the limits of the drawing.

Revision List provides a record of changes made to the drawing.

Fold Mark indicates where to fold the paper in halves.

Title Block records information about the drawing, such as its title, the date it was drawn, and the name of the drafter.

Borders are available in a range of standard sizes. The smallest is called "A", which you may recognize as the 8-1/2 x 11" letter-size paper used in many offices. Each larger size doubles in area:

Size	Dimensions
A	8.5 x 11" (landscape) and 11 x 8.5" (portrait)
B	11 x 17" (landscape only)
C	17 x 22"
D	22 x 34"
E	34 x 44"

The equivalents in metric ranges from A4 (smaller) through A0 (larger).

Layout Drawings (MView)

You could draw the border around the drawing, but that leads to scale issues. As with text, the border has to be scaled to fit the size of paper. Unlike text, the border changes, depending on the size of paper the drawing is printed on.

progeCAD provides a second way of looking at drawings, called "layout mode." Until now, you have been drawing in model mode. At the bottom of the drawing area are three tabs: Model, Layout1, and Layout2.
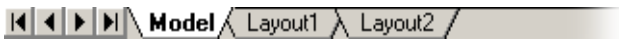
As their names suggest, Model mode is for creating and editing the model (drawing), while Layout mode is for laying out the drawing in preparation for plotting. The number "1" next to Layout (Layout1) is an indication that models can have numerous layouts. (Layouts are the new name for "paper space.")

> **MVIEW COMMAND**
>
> *Creates viewports in layouts to reveal model space entities.*
>
> Command: **mview**
>
> Alias: **mv**
>
> Menu: **View | Layout Viewports**
>
> Toolbar: **View | Layout Viewports**
>
> Options: **ON, OFF**
>
> **Fit**
>
> **2, 3, 4**

You can create layouts for different sizes of drawing and types of plotters. For example, you might want one layout for plotting to your A-size laser printer, and a second for plotting to your E-size color inkjet plotter. Here's how to use layouts:

1. Switch to layout mode by clicking the **Layout1** tab.

   The drawing goes blank, but don't worry: the drawing is not missing; rather, it is "hidden" in model space until you chose to reveal it by creating a viewport into model space.

2. Draw a border and title block using the dimensions illustrated below. Draw as much or as little of the border as you wish.

Or, use the **DdInsert** command to place the *ANSI A Portrait.dwg* file as a block. Use the default settings to insert the block: insertion point 0,0; scale factors = 1; rotation = 0.
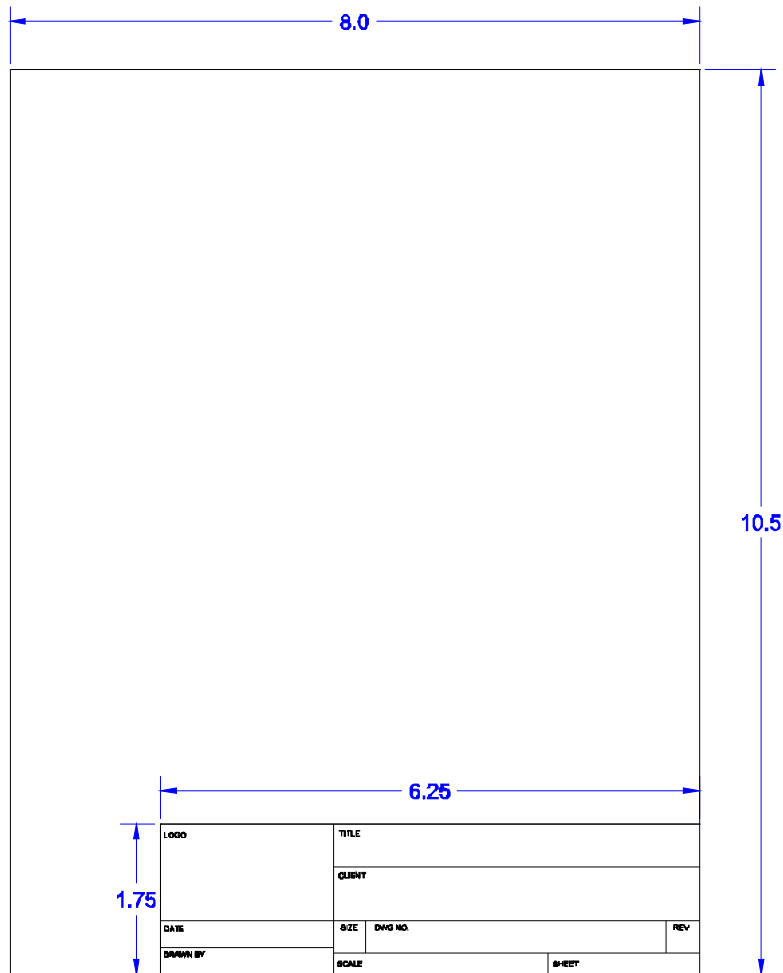

3. The next step is to create a viewport in this layout to reveal the model. But first, use the **la** alias (for the layer command) to create a new layer called "Viewport."

4. Use the **MView** command (short for "Make viewport"). From the **View** menu, select **Layout Viewports**.

    Alternatively, select the Layout Viewports button from the View toolbar, or enter **mview** in the command bar.
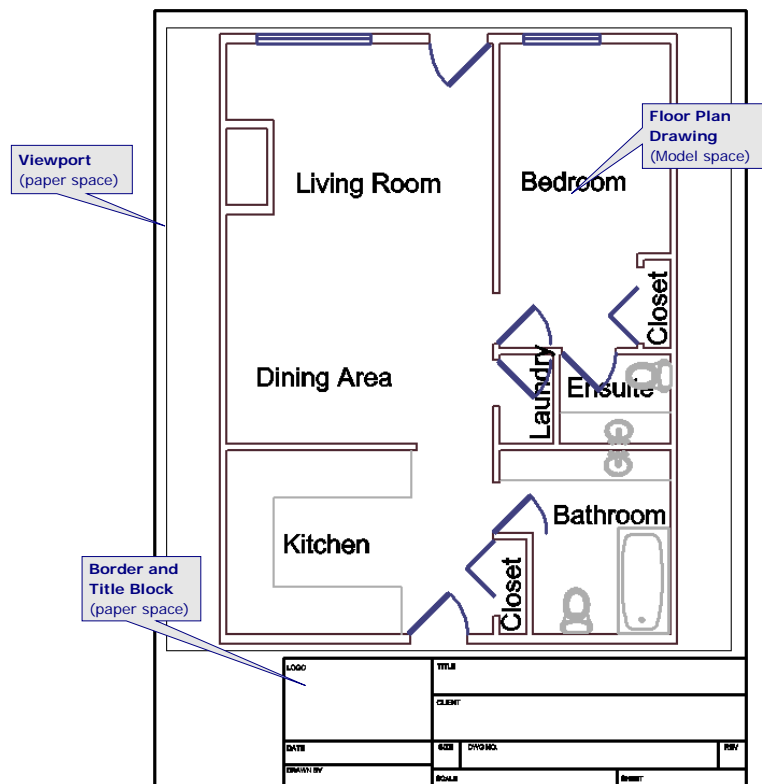
    Command: **mview**

5. progeCAD prompts you to pick two corners of a rectangle that forms the viewport.

    ON/OFF/Fit/2/3/4/<First corner>: *(Pick a point.)*
    Opposite corner: *(Pick another point.)*

    For an idea of where to pick the points, see the figure illustrated on the next page.

First
corner:

Viewport

Opposite
corner:
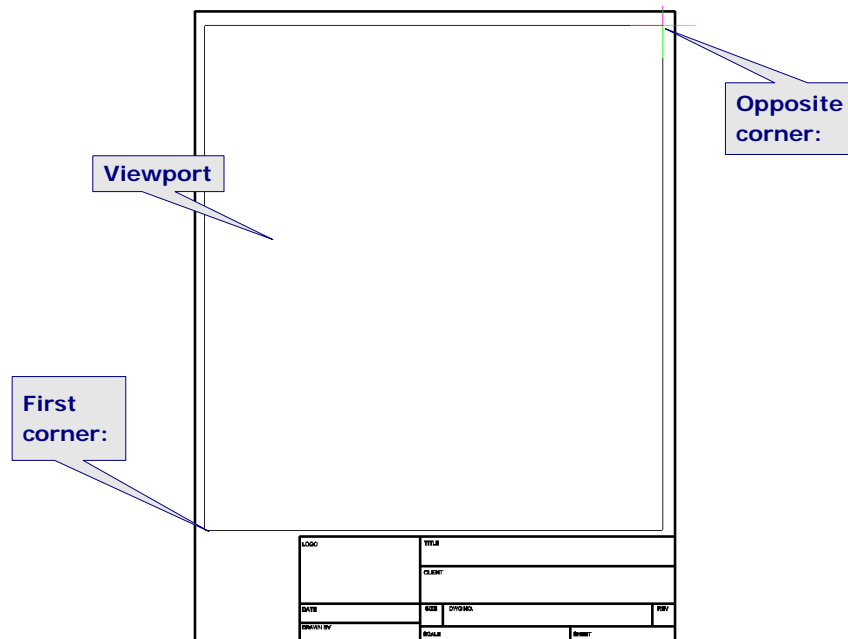
As soon as you pick the second corner, a magical event occurs: the floor plan reappears!

Viewport
(paper space)

Floor Plan
Drawing
(Model space)

Living Room

Bedroom

Dining Area

Laundry

Ensuite

Closet

Kitchen

Bathroom

Border and
Title Block
(paper space)

Closet

You are looking at two "overlapping" drawings. One is the floor plan in *model* space; the other is the border and title block in *paper* space (a.k.a. layout). The two spaces are independent of each other, linked by the viewport.

Paper space (layout mode) can be confusing to new CAD users, so here are some exercises to help you understand the differences.

1. Select the floor plan. Move the cursor over any part of the floor plan and click. Notice that progeCAD doesn't find any entities.

   Lesson: when in layout mode, model space entities are inaccessible. You can see, but you can't touch.

2. Select the border or title block. Notice that you can select them.

   Lesson: when in layout mode, paper space entities can be selected and edited.

3. Select the viewport rectangle. Notice that square green grips appear. Drag one of the grips so that the viewport changes its size. (See figure below.)

   Lesson: viewports are editable entities. You can move, copy, stretch, and erase them — as well as apply properties, such as color, linetype, and lineweight.
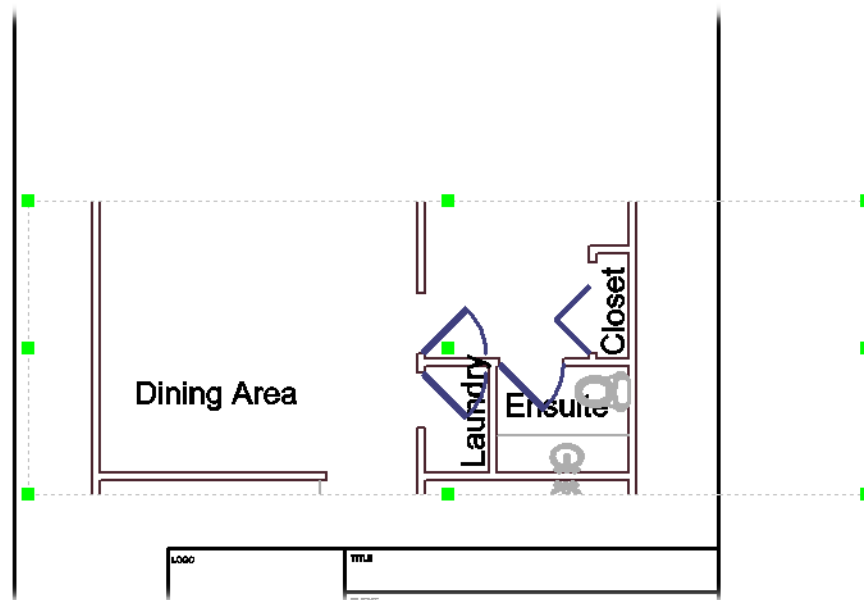
   Use the **U** command to revert the viewport to its original shape.

5. Use the **la** alias to freeze the "Viewport" layer. Notice that the viewport rectangle disappears, but that the floor plan drawing remains.

   Lesson: The viewport's properties are independent of the viewport content (the model space drawing).

6. Use the **Zoom Window** command to zoom into the title block. Notice that the floor plan model zooms along. Use the **Zoom Extents** command to revert the view.

   Lesson: You can zoom and pan in layout mode just as in model mode.

7. A button on the status bar lets you switch between the two spaces, model and paper.
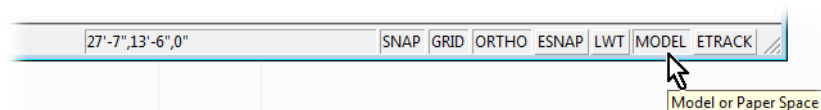
Click **PAPER**; notice that it changes to **MODEL**. (This action executes the **MSpace** command.)

8. Repeat the actions you used above:

   a. Select an entity in the floor plan: this time it is highlighted, because it is in model space.

   b. Select the border: this time progeCAD cannot find it, because the border is in paper space.

   c. Zoom into a part of the floor plan: this time the floor plan (in model space) becomes larger, but the border and title block remain unchanged (in paper space).

Zoomed-in Floor Plan Drawing (Model space)

Border and Title Block (paper space)
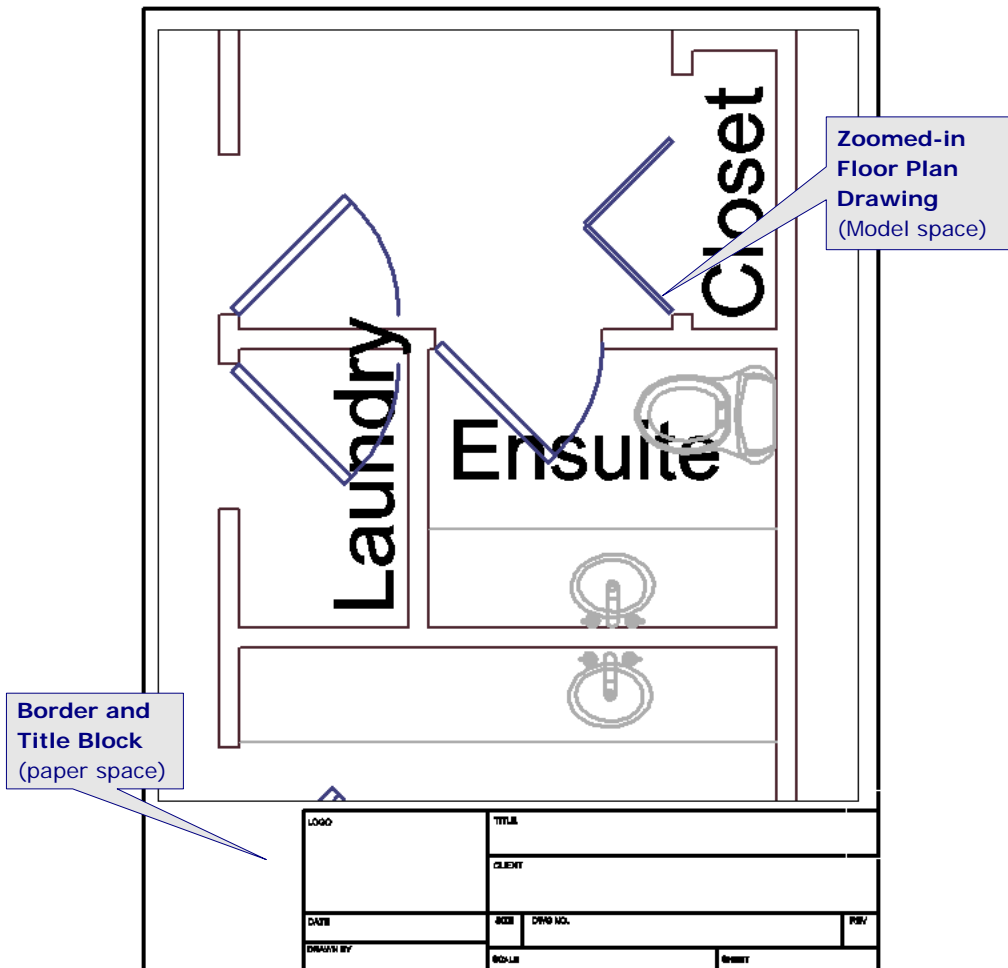
   d. Use the **Zoom Extents** command to return the view to normal.

9. Click **MODEL** on the status bar to return to paper space. (This executes the PSpace command.)

Place Paragraphs of Text (MText)

After learning about the Text and DText commands, you might be surprised to learn that progeCAD hosts yet another command for placing text.

MText (short for "multi-line text") is designed for placing paragraphs of text, such as notes. But it is especially useful for text that must contain a variety of text effects, such as multiple fonts, boldface, superscripted characters, and several colors.

1. To place notes in the drawing, start the **MText** command. From the **Draw** menu, select **Text**, and then choose **Multiline Text**.

   (Alternatively, click the Multiline Text button on the 2D Draw toolbar, or enter **mtext** in the command bar.)

   > Command: **mtext**

2. progeCAD prompts you to pick two points. These form the rectangle within which the paragraphs of text are fitted.

   > Multiline Text: First corner for block of text: *(Pick a point.)*
   >
   > Justification/Rotation/Style/Height/Direction/Width/<Opposite corner for block of text>: *(Pick another point.)*

First corner for block of text:

LOGO

DATE

Opposite corner for block of text:
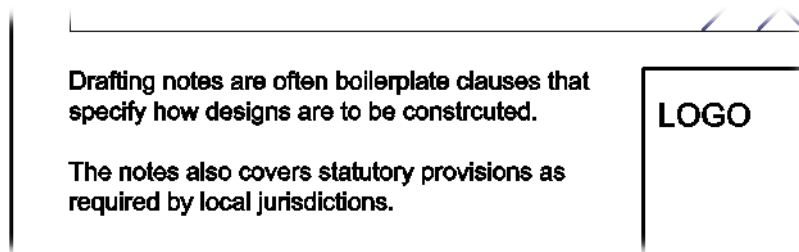
DRAWN BY
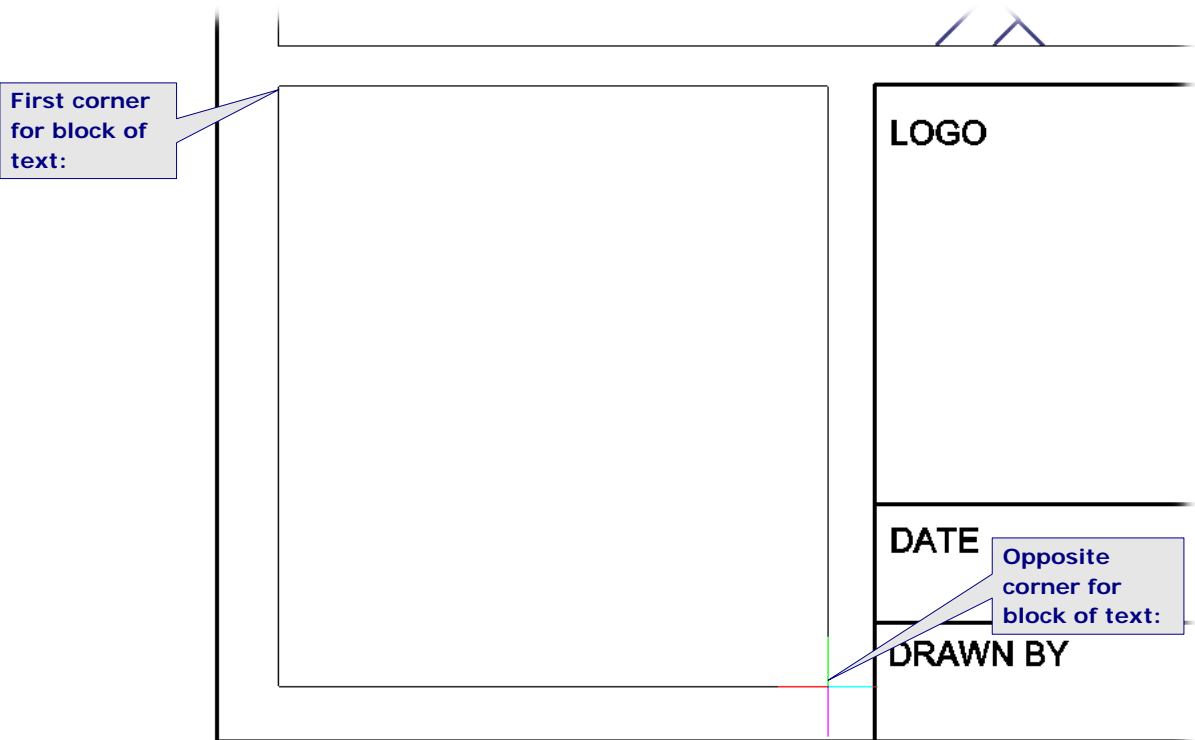
3. After you pick the second point, progeCAD displays the Multiline Text dialog box. It is here that you enter and format text.
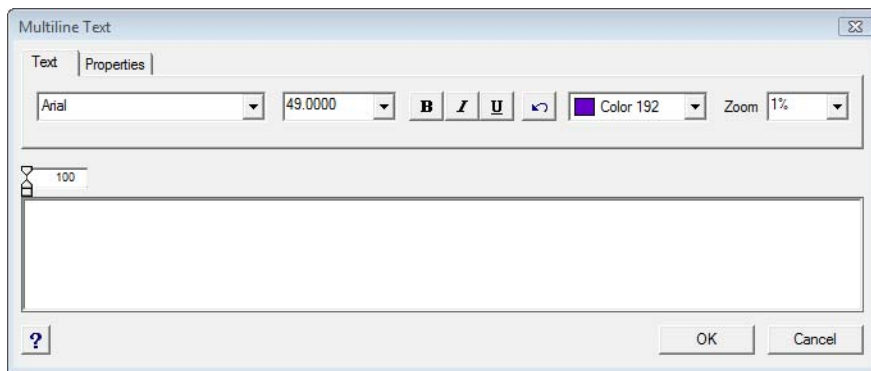
4. Enter text, such as the notes illustrated by the figure below.
5. Click **OK** to dismiss the dialog box, and place the text in the drawing.

6. Click the **Print Preview** button on the toolbar, or enter the **Preview** command. You see a different version of the drawing, depending on the mode in which you enter preview:

> **PAPER** – in paper space, both model and paper space (floor plan and border) are printed.
>
> **MODEL** — in model space, only the model (floor plan) is printed.

7. Print the drawing, if you wish.

TIPS To format a selection of mtext, highlight it (drag the cursor over the text), and then select the formatting option.

You can copy text from another source, such as a Word document, and then use **Ctrl+V** to paste the text in the mtext editor.

After mtext is placed in the drawing, you can edit it by double-click the mtext; in the Entity Properties dialog box, select **Full Editor**.

Text Tab

The options available in the Text tab are:

Font droplist select fonts. You can chose from any TrueType and AutoCAD font installed on your computer.

Height specifies the height of the text in inches. The droplist contains the default height of 0.2 units, as well as any recently-used heights.

**Bold**, **Italic**, **Underline** adds bolding, italics, and underlining to text.

Remember that the above options work only when you first highlight some or all of the text. If you forget to highlight it, it fails to change its font, height, or other effects.

Undo undoes the last action in the dialog box.

Zoom droplist enlarges and reduces the size of the text in the dialog box; does not affect the size of the multiline text placed in the drawing.

Properties Tab

Click the Properties tab for additional options:
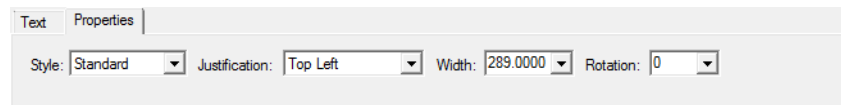
Style droplist selects a text style defined by the Style command. The properties set by the Text tab override the properties specified by the style.

Justification droplist specifies how the paragraphs of text fit within the mtext rectangle.

Width changes the width of the text block; enter another width, or select **No Wrap** (text is not constrained to the mtext rectangle).

Rotation rotates the entire block of text; select one of the preset angles, or enter a specific angle.

Adding Dimensions

The floor plan drawing is almost complete. All that's missing are *dimensions*, which indicate lengths and distances. progeCAD supports a number of different kind of dimensions.

Linear dimensions measure the distance between two points. Most are horizontal or vertical, but some might be rotated (measured at an angle) or aligned (measured along entities).

Radial dimensions measure the radius or diameter or curved entities, like arcs and circles.

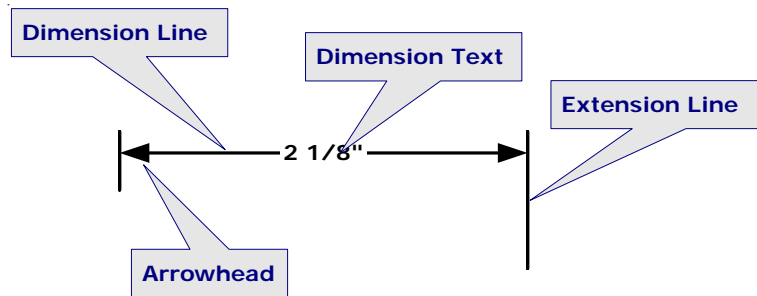**Angular dimensions** measure the angle between vertices.

Leaders are used to point text to entities.

In This Chapter

• **DimLinear** command
• **Dist** command
• **DimScale** command
• **List** command
• **DimStyle** command
• **Area** command

Linear dimensions consist of four parts:

- **Dimension line** indicates the distance between the two points being measured.
- **Dimension text** spells out the distance and its modifiers (if any).
- **Extension lines** link the dimension line with the points being measured.
- **Arrowheads** traditionally point to the ends of the dimension line.

Hence, you typically pick three points for each linear dimension: one pick for the starting point of each extension line, plus one pick that simultaneously locates the dimension line and the text. progeCAD figures out everything else: drawing the lines and arrowheads, and then measures the distance to generate the text.

Dimensions have *styles* like text, which let you define the look and placement of every dimension element.

Linear Dimensions (DimLinear)

You place linear dimensions like this:

1. Open the floor plan drawing in progeCAD.

   (If you were unable to complete the drawing to this point, open the *4Text.dwg* file.)

2. Click the **Model** tab to return to model space.

   (If necessary, create some breathing space around the drawing with the **Zoom 0.8x** command.)

3. Change to the "Dimensions" layer using one of these methods:

   - From the Entity Properties toolbar, click the **Layer Control** drop list, and then select "Dimensions."
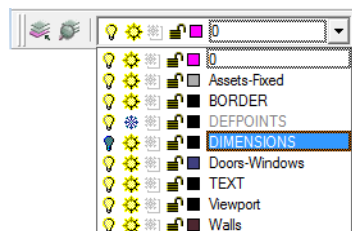
COMMAND SUMMARY

Dimensions linear distances.

: **dimlinear**

Aliases: **dli, dimlin**

Menu: **Insert | Dimensions | Linear**

Toolbar: **Dimensions | Linear**

Options: **Angle**

      Text

      Orientation

ALTERNATIVE

DimAligned dimensions along entities.

Dimensioning Drawings

progeCAD provides a number of commands for dimensioning entities.

Drawing Dimensions

DimAligned places dimensions aligned to entities.

DimAngular places dimensions that measure angles.

DimBaseline continues from a previous linear, angular, or ordinate dimension; the first extension line is the start of the baseline dimension.

DimCenter places center markers at the center of circles, arcs, and polyline arcs.

DimContinue continues from a previous dimension; the final extension line of the last dimension is the start of the continued dimension.

DimDiameter measures the diameters of circles, arcs, or polyline arcs.

DimLeader draws leaders.

DimLinear places horizontal and vertical dimensions.

DimOrdinate places ordinate dimensions.

DimRadius dimensions the radii of circles, arcs, and polyline arcs.

Editing Dimensions

DimEdit changes, rotates, and restore text, and obliques extension lines.

DimOverride overrides the style settings for selected dimensions.

DimTEdit changes the position and angle of dimension text.

In addition to the above commands, dimensions can also be edited using grips.

Handling Dimension Styles

DimStyle displays the Dimension Settings dialog box.

-DimStyle creates, changes, and saves dimension styles in the command bar.

Quick Dimensioning

To create and modify a series of dimensions quickly, you can use the **QDim** command. It is particularly useful for creating a series of baseline or continued dimensions, or for dimensioning a series of circles and arcs.

To start quick dimensioning, enter the **QDim** command, or choose **Dimension | Quick Dimension** from the menu. You are prompted to select objects for dimensioning; specify the position of the dimension line, or else choose from the options:

    [Continuous/Staggered/Baseline/Ordinate/Radius/Diameter/datumPoint/Edit]:

- Enter the **la** alias. When the Layer Properties Manager dialog box appears, select "Dimensions", and then click the **Current** ✔ button. Click **OK**.
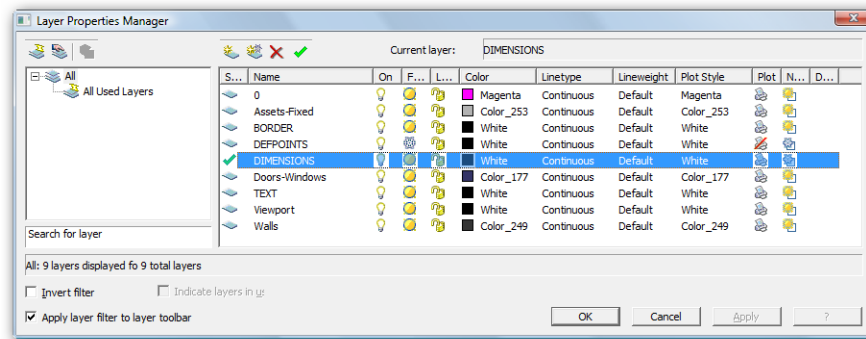
- Use the -**Layer** command's **Set** option to change the working layer to "Dimensions," as follows:

Command: **-layer**

Layer: ? to list/New/Make/Set/Color/Ltype/LWeight/ON/OFF/Plot/Freeze/Thaw/LOck/Unlock: **s**

Layer to set as current <0>: **dimensions**

Layer: ? to list/New/Make/Set/Color/Ltype/LWeight/ON/OFF/Plot/Freeze/Thaw/LOck/Unlock: *(Press **Enter** to exit the command.)*

4. Use the DimLinear command to dimension the outside wall on the left. From the **Dimension** menu, select **Linear**.

   (Alternatively, click the **Linear** button on the **Dimensioning** toolbar, or type **dimlinear** in the command bar.)

   Command: **dimlinear**

TIP   The **DimLinear** command creates horizontal, vertical, or rotated dimensions — based on the order of your pick points.
   progeCAD creates linear dimensions using two methods: by measuring the entity, or by measuring the distance between two pick points.

Step 1:
Select entity to dimension.

Step 2:
Position dimension line.

5. progeCAD prompts you to:
   - Select an entity; or
   - Pick the location of the first extension line.

   For this dimension, select the first option:

   ENTER to select entity/<Origin of first extension line>: *(Press **Enter**.)*

6. progeCAD asks which entity to dimension:

   Select entity to dimension: *(Pick the line.)*

7. Finally, you need to tell progeCAD where to locate the dimension line:

Angle/Text/Orientation of dimension line: Horizontal/Vertical/Rotated: *(Move cursor away from line, and then pick a point.)*

progeCAD draws the dimension, but look closely: can you see what's wrong? The arrowheads and dimension text appear to be missing. The problem is corrected with the **DimScale** dimension variable.

Dimension Scale (DimScale)

The problem is that dimensions need to be scaled, just like text. Scaling is performed with the **DimScale** dimension variable.

1. Use the **U** command to remove the dimension.
2. Enter the **DimScale** command, and use the same scale factor you used for text in the previous chapter.

   Command: **dimscale**
   New current value for DIMSCALE (0 or greater) <1>: **48**

3. Repeat the **DimLinear** command, and redimension the line. This time the text and arrowheads are visible.

But the dimension text is annoying: it is in decimal units and has four decimal places. For this architectural drawing, it should be in architectural units — feet and fractional inches. This problem is corrected with the **DimStyle** command.

Dimension Styles (DimStyle)

The look of dimensions is set with the **DimStyle** command. Change the style so that dimension text is shown in feet and fractional inches.

1. Start the DimSytle command. From the **Dimension** menu, select **Dimension Style**.

   (Alternatively, select Dimension Settings from the Settings toolbar, or enter **dimstyle** in the command bar.)

   Command: **dimstyle**

   Notice the Dimension Settings dialog box.

> **DIMSTYLE SUMMARY**
>
> *Creates and edits dimension styles.*
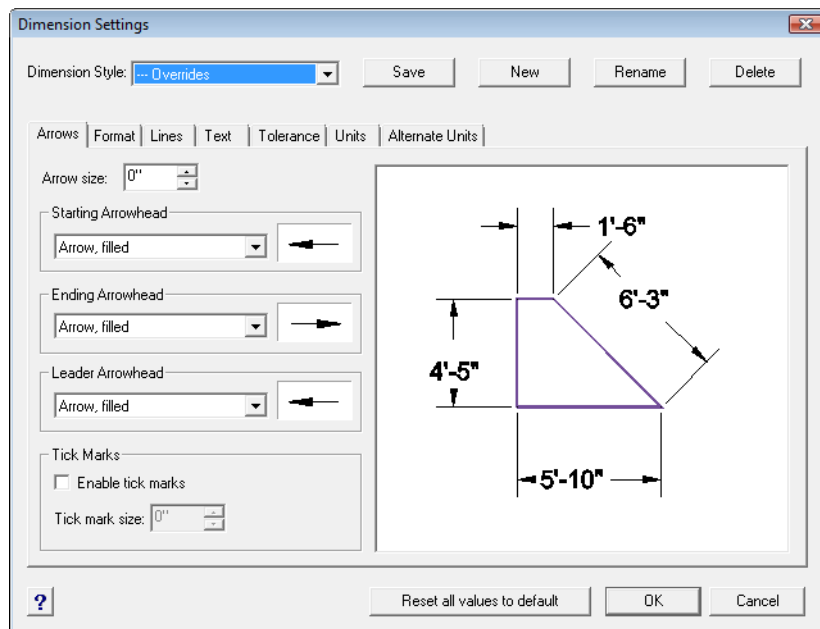> Command: **dimstyle**
> Aliases: **d, ddim, setdim**
> Menu: Settings | Dimension
>   Settings
> Toolbar: Settings | Dimension
>   Settings
>
> **ALTERNATIVES**
>
> **-DimStyle** controls dimension styles at the command line.
> **ExpDimstyles** displays the Explorer Dimension Styles window.

2. The enormous preview window lets you see the effect of changing the dimension style. Take a quick tour by clicking on each tab: Arrows, Format, Lines, and so on. This may well be the most imposing of all dialog boxes, but fortunately you need change only a few items.

   (If necessary, select "Standard" from the **Dimension Style** droplist; ignore the warning.)

3. Click the **Units** tab.
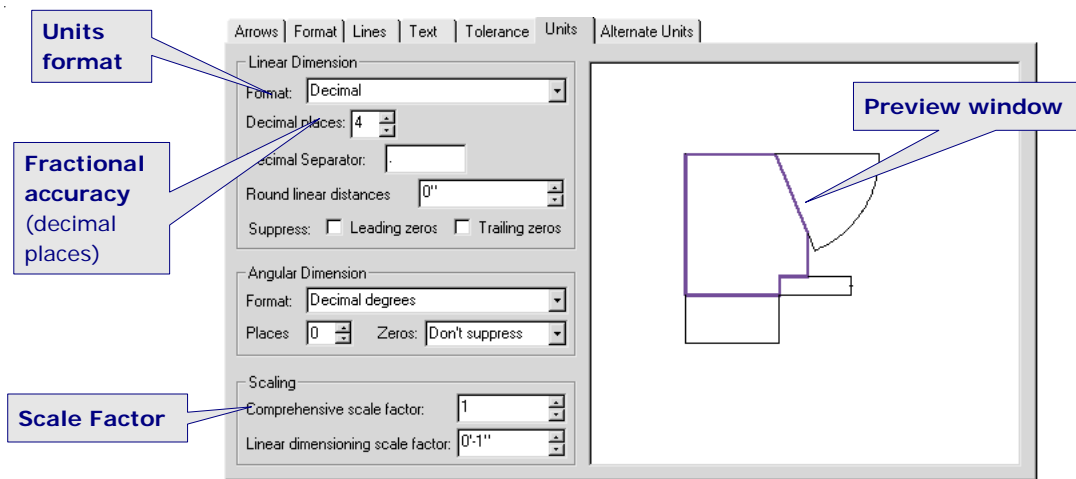
6. First, change the dimension scale factor back to 48, so that you can see the preview window in action: in the Scaling section, change **Comprehensive Scale Factor** to **48**.

TIP   The **Linear dimensioning scale factor** allows you to switch between Imperial (inches) and metric measurement systems.
   For example, this drawing is in feet-and-inches. If you were to change this scale factor to **25.4**, all dimensions would change to millimeters.

---

7. In the Linear Dimension section, click on the **Format** drop list, and then select "Architectural."

8. Reduce **Decimal Places** to **1**. That means that fractional inches are dimensioned to the nearest 1/2-inch.

9. Click **Save**. This forces the changes to be applied to the dimension style.

10. Click **OK**. Notice that dimension changes from 384.0000 to 32'.

Dimensioning Between Two Points

Let's continue dimensioning the drawing:

1. Repeat the **DimLinear** command. This time dimension the top of the floor plan. Use INTersection entity snap to ensure the dimension is placed between the two corners.
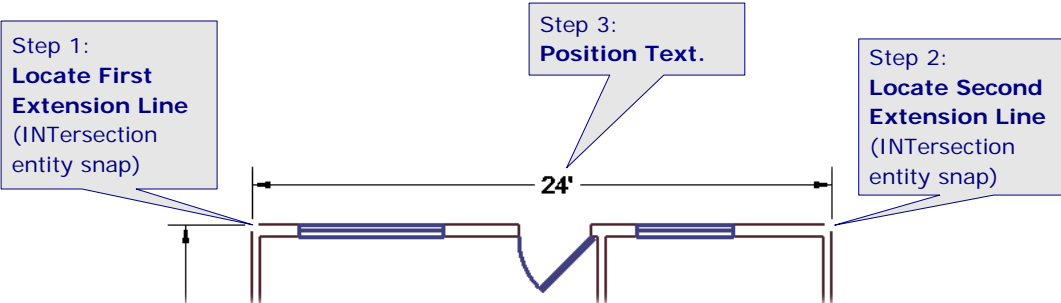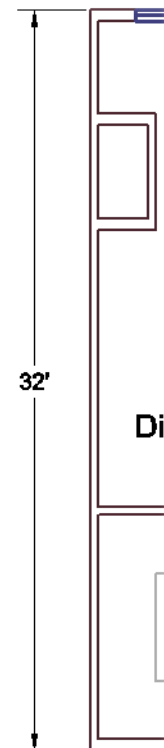
   > Command: **dimlinear**
   >
   > ENTER to select entity/<Origin of first extension line>: **int**
   >
   > Snap to intersection of: *(Pick point 1.)*
   >
   > Origin of second extension line: **int**
   >
   > Snap to intersection of: *(Pick point 2.)*
   >
   > Angle/Text/Orientation of dimension line: Horizontal/Vertical/Rotated: *(Pick point 3.)*

32'

Dir

Step 1:
Locate First Extension Line (INTersection entity snap)

Step 3:
Position Text.

Step 2:
Locate Second Extension Line (INTersection entity snap)

24'

Continue Dimensions (DimContinue)

The right side of the floor plan has four rooms, and would use four dimensions. Here is where the **DimContinue** command comes in handy: it continues from the previous dimension, placing the first extension line at the previous one, and locating the dimension line with the previous one. Try it now:

1. Dimension the bedroom with the **DimLin** alias. (Remember to keep INTersection entity snap turned on.)

2. Start the DimContinue command. From the **Dimension** menu, select **Continue**.

DIMCONTINUE COMMAND

Adds dimensions from the previous extension line.

Command: **dimcontinue**

Aliases: **dco, dimcont**

Menu: **Insert | Dimensions | Continue**

Toolbar: **Dimensions | Continue**

ALTERNATIVE

DimBaseline adds dimensions from the same starting point.

(Alternatively, click the Continue button on the Dimensions toolbar, or type **dimcontinue** in the command bar.)

> Command: **dimcontinue**

3. Pick the location of the next extension line:

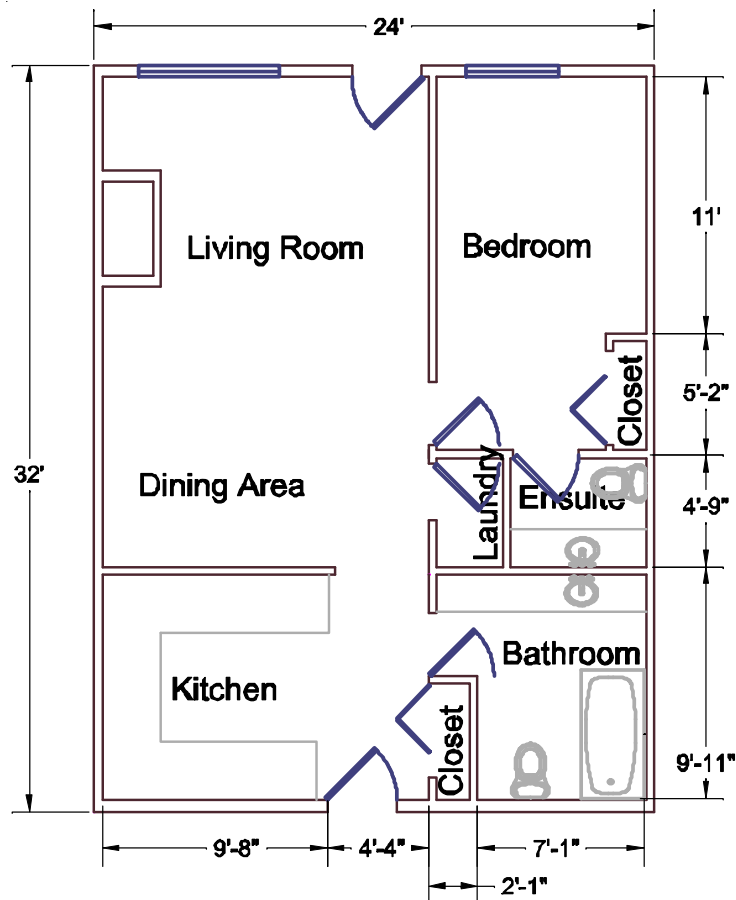   > Continue: ENTER to select starting dimension/<Origin of next extension line>: *(Pick a point.)*

4. Continue until the end of the wall:

    > Continue: ENTER to select starting dimension/<Origin of next extension line>: *(Pick another point.)*
    >
    > Continue: ENTER to select starting dimension/<Origin of next extension line>: *(Press **Esc** to exit the command.)*

5. Dimension the lower wall. When you are done, the drawing should look something like the figure illustrated below.

6. Save your work with the **QSave** command.

Other Methods of Measuring Drawings

The dimensioning commands are holdovers from the days when drawings were hand-drafted with pencils, straight edges, scale rulers, and erasers. Back then, drawings were mostly drafted to scale. Dimensions helped others read the drawing without needing to reach for scale rulers. ("Rulers" in plural, because multiple rulers were needed for different disciplines, such as architecture and engineering.)

Strictly speaking, dimensions are not needed in CAD drawings, because everything is drawn full-size. That 32'-wall truly is thirty-two feet long. Scaling occurs only when it comes time to print the drawing.

progeCAD includes commands for measuring distances, angles, and areas directly from drawings. Let's see how they work.

Measuring Distances and Angles (Dist)

The **Dist** command (short for "distance") measures distances and angles between two points. Use it to measure the length of the wall between the kitchen and dining area.

> **DIST COMMAND**
>
> *Determines the distance and angle between two points.*
>
> Command: **dist**
> Alias: **di**
> Menu: **Tools | Inquiry | Distance**
> Toolbar: **Inquiry | Distance**
>
> **ALTERNATIVES**
>
> **Area** reports the area and perimeter of closed entities.
> **List** reports the length and other data of entities.

1. Start the Dist command. From the **Tools | Inquiry** menu, select **Distance**.

    (Alternatively, click the Distance button on the Inquiry toolbar, or type the **dist** command.)

    > Command: **dist**

2. As with dimensions, it's a good idea to use entity snaps to ensure the measurements are 100% accurate. (If necessary, turn on ENDpoint entity snap.)

    progeCAD prompts you for two points:

    > Starting point for distance: *(Pick one end of the wall.)*
    > End point: *(Pick other end of the wall.)*

3. Notice that progeCAD reports back the following information.

    > **Distance = 10'-0"**, Angle in XY Plane = 0, Angle from XY Plane = 0
    > Delta X = 10'-0", Delta Y = 0", Delta Z = 0"

    The wall is ten feet long. (If you cannot see all the text in the command bar, press **F2** to display the text window.)

4. To see how angles work, find the diagonal distance from one corner of the apartment to the opposite corner. The result I got was (your result may differ):

    > Command: **dist**
    > Starting point for distance: *(Pick one corner.)*
    > End point: *(Pick other corner.)*
    > Distance = 38'-10", **Angle in XY Plane = 233**, Angle from XY Plane = 0
    > Delta X = -23'-4", Delta Y = -31'-0", Delta Z = 0"

The meaning of the various distances are illustrated by the following figure:
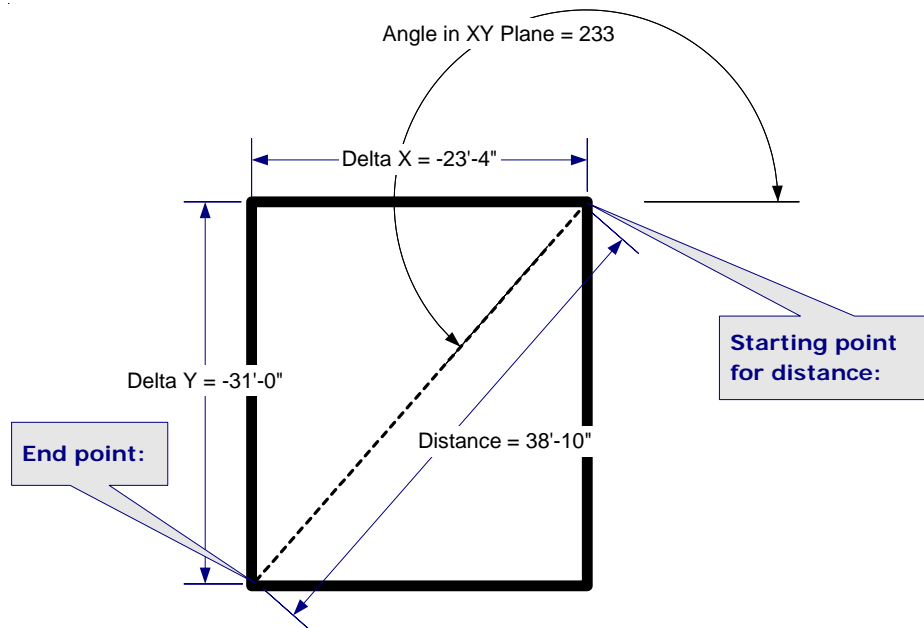
"Delta" means difference, or the net distance between the two points. **Delta Z** and **Angle from XY Plane** are used only in 3D space.

Measuring Entities (List)

The **List** command reports information about selected entities. It returns everything progeCAD knows about the entities; much of that information isn't terribly useful to everyday drafting; still, this command is handy when you can't figure out what's happening with some entity or another.

In the context of this chapter, **List** returns length, perimeter, circumference, and area data — the data varies, depending on the entity. Try out the command on several entities.

1. Start the List command. From the **Tools | Inquiry** menu, select **List Entity Info**.

   (Alternatively, click the List Entity Info button on the Inquiry toolbar, or type the **list** command in the command bar.)

   > Command: **list**

2. Select a line.

   > SCroll/SOrt/Tracking/<Select entities to list>: *(Pick a line.)*

LIST COMMAND

Reports everything progeCAD knows about entities.

Command: **list**

Aliases: **li, ls**

Menu: **Tools | Inquiry | List Entity Info**

Toolbar: **Inquiry | List Entity Info**

Options: **Scroll**

Sort

Tracking

ALTERNATIVES

Area reports the area and perimeter of closed entities.

Dist measures the distance between two points.

---

3. progeCAD displays its report in the text window, because the listing is longish:

```
---------- Line ---------------------------------------------------
     Handle: 55
Current space:          Model
     Layer: Walls
     Color: 256 (BYLAYER)
 Linetype: ByLayer
Lineweight:   30
From point:  X=   2'-6"  Y=   27'-0"  Z=   0"
 To point: X=   2'-6"  Y=   23'-0"  Z=   0"
   Length: 4'-0"
Angle in UCS XY plane:          270
Delta values: X=   0"  Y=   -4'-0"  Z=   0"
```

Highlighted is the length, four feet.

4. Select a block:

```
---------- Insert of Block ---------------------------------------
     Handle: 212
Current space:          Model
     Layer: Assets-Fixed
     Color: 256 (BYLAYER)
 Linetype: ByLayer
Lineweight:   30
Block name:  Tub-Topview
Insertion point:         X=   23'-7"  Y=   3'-4"  Z=   0"
X scale factor:          1
Y scale factor:          1
Z scale factor:          1
Rotation angle:          90
```

5. And select a piece of text:

```
---------- Text ---------------------------------------------------
     Handle: 6C1
Current space:          Model
     Layer: BORDER
     Color: 256 (BYLAYER)
 Linetype: ByLayer
Lineweight:   0
     Color: 256 (BYLAYER)
 Linetype: ByLayer
Lineweight:   0
Text style: Standard
Insertion point:         X=   1'-11"  Y=   13'-7"  Z=   0"
Text height:  1'-0"
Text value:    Dining Area
Rotation angle:          0
Generation:  Normal
Vertical alignment:      Baseline
Horizontal alignment:  Left
```

Measuring Areas (Area)

The **Area** command finds the areas of entities and spaces. Here's how to find the area of any room.

1. Start the **Area** command. From the **Tools | Inquiry** menu, select **Area**.

    (Alternatively, click the Area button on the Inquiry toolbar, or type the **area** command in the command bar.)
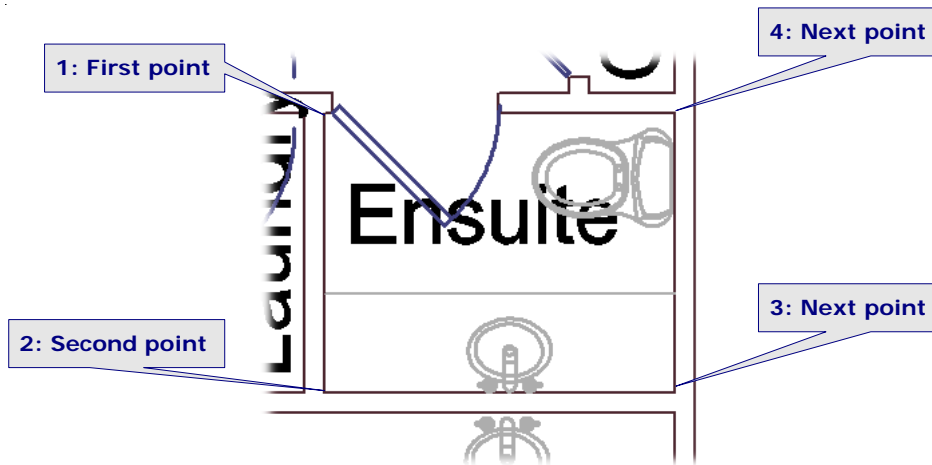
    > Command: **area**

2. With the assistance of INTersection entity snap, pick each corner of the Ensuite bathroom.

    > Entity/Add/Subtract/<First point>: *(Pick one corner.)*
    > <Next point>: *(Pick another corner.)*
    > <Next point>: *(Pick the third corner.)*
    > <Next point>: *(Pick the last corner.)*

3. Press **Enter** to end the measurements. progeCAD reports the results:

    > <Next point>: *(Press Enter to exit the command.)*
    > **Area =** 3920" sq. (**27' sq.**), Perimeter = 21'-0"

The bathroom has an area of twenty-seven square feet. This tells the subcontractor the amount of floor tile to order. And the perimeter of twenty-one linear feet is the length of baseboard material that will be needed.

TIP   The **Area** command's **Add** and **Subtract** options allow you to add other areas and remove "holes" to find a grand total.

The command's **Entity** option reports the area and perimeter of closed entities: circles, ellipses, polylines, polygons, traces, and planes. In the case of polylines, Area "closes" the start and end points to find the area.

It also works with open entities, but in then only the length is reported: lines, arcs, and splines.

AREA COMMAND

Measures the area and perimeter of entities and spaces.

Command: **area**

Alias: **aa**

Menu: **Tools | Inquiry | Area**

Toolbar: **Inquiry | Area**

Options: **Entity**
         Add
         Subtract

ALTERNATIVES

List reports the area and other data of entities.

Dist measures the distance between two points.

Interacting with Google Earth

With progreCAD, you can publish 2D drawings or 3D models to Google Earth. To do so, you first need know the position of the drawing on the earth's surface.

The method I consider the easiest is to first import a Google Earth area into progeCAD. The area is imported from the current Google Earth view into progeCAD as raster image. You use the raster image as a background for the drawing. More importantly, the image gives the geographical reference (longitude and latitude) of your drawing's position on the surface of the earth.

Starting Google Earth

Here are the steps involved:

1. Start Google Earth. (If necessary, download and install Google Earth from earth.google.com/download-earth.html.)

2. Maximize the Google Earth window. Do not minimize or close the Google Earth application at any time during this tutorial.

3. Zoom to the location on the earth's surface in which you want to put your drawing. Zoom in to capture enough detail of the location area. See explanation of Google Earth controls at right.

4. Be sure you are looking down perpendicularly to the earth's surface (the view has no rotation.)

Tilt

Pan

Zoom In
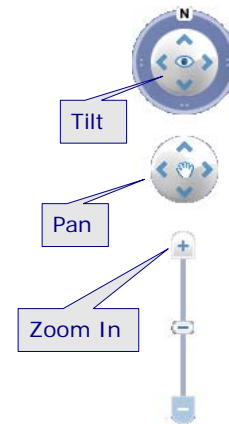
In This Chapter

—

• Setting up Google Earth
• Exporting to Google Earth

• Adjusting the image and drawing

At this point, you might be seeing something like the figure shown below.
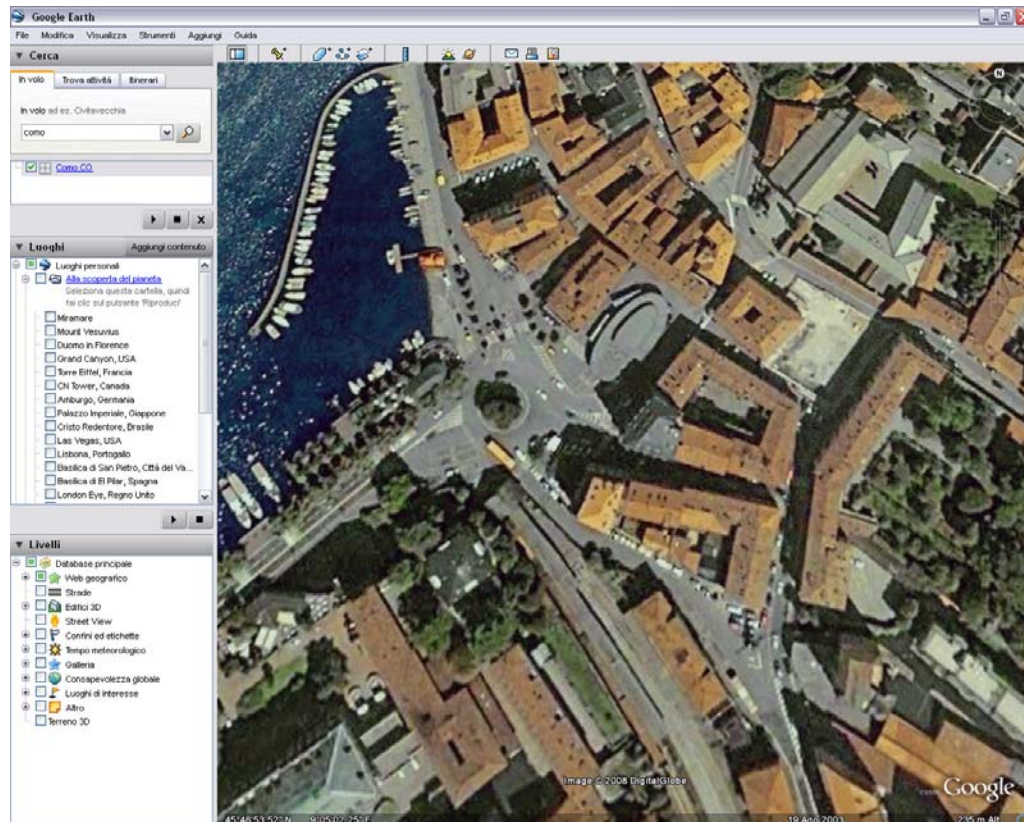
Importing Images to progeCAD

With Google Earth running and ready, it is time to copy the satellite view to your drawing in progeCAD. Follow these steps:

1. Start progeCAD, and then open your drawing. If it is a 3D model, then ensure it is in plan view.

2. Click the ImportGEimage button progeCAD's Google Earth toolbar. Alternatively, enter **ImportGEImage** in the command bar.

3. Set the drawing units in order to scale the image correctly.
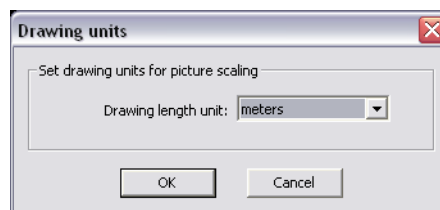
4. Click **OK**.

After a few seconds, the current Google Earth view is imported and placed at the origin (0,0) of the drawing. Notice that the image is black and white; this is a current limitation imposed by Google Earth.
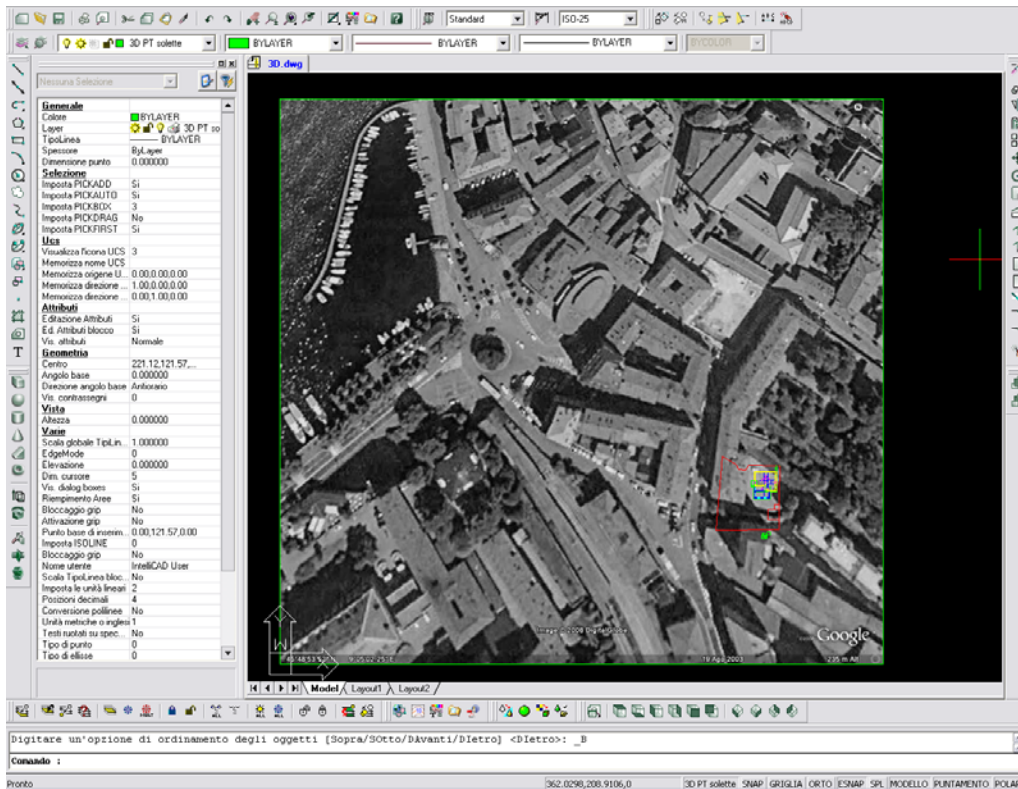
If you cannot see anything, use Zoom Extents to include all objects of the drawing.

Working With Google Earth Images

When working with images from Google Earth, the following suggestions may help.

Once the image is placed, you can use progeCAD's **Move** and **Rotate** commands to refine the drawings's position relative to the image. Do not adjust the position of the Google Earth's image!

The image displayed in the drawing is a JPG file and it is stored in the same directory as the DWG file.

Use the **DrawOrder** command to visually place the image underneath the drawing, like this:

1. From the **Tools** menu, choose **Draw Order**. (Alternatively, enter the DrawOrder command.)
2. Select the entity you wish to reorder, and then press Enter. If you are reordering above or under, select the entity you want the first entity to be above or below, and then press Enter.
3. Specify the new drawing order, and then press Enter.

Publishing Drawings to Google Earth

To publish drawings to Google Earth, follow these steps:

1. Click the PublishKML button on progeCAD's Google Earth toolbar, or enter the **PublishKML** command.
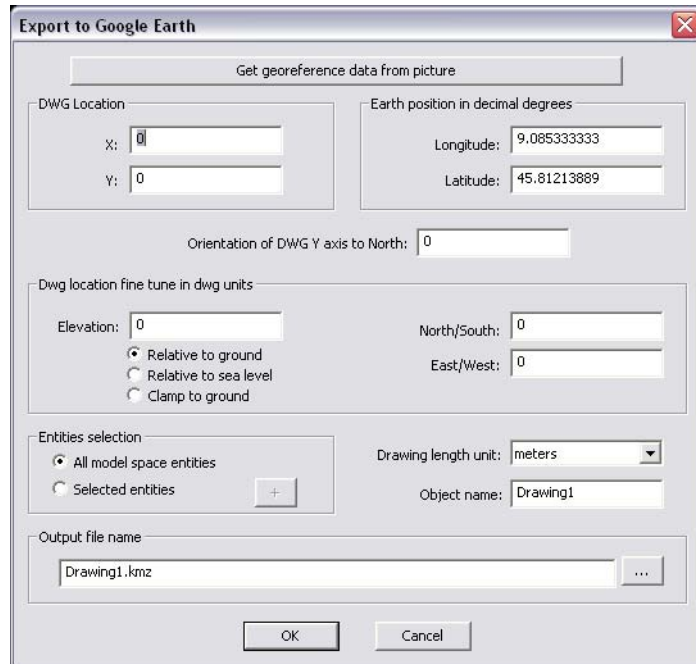
2. In the dialog box, click the **Get Georeference Data from Picture**.

3. Select the Google Earth image that you previously inserted. This picks up the coordinates.

4. If the position and rotation of a model relative to the image are correct, there's no need to modify the orientation of the Y axis to North. The output file name is taken automatically from the DWG file name and directory.

5. Click **OK**.

The KMZ file generated by export has the same name and it is stored in the same folder as the DWG file. (KMZ is short for "keyhole model compressed." Keyhole is the name of the company from which Google purchased Earth.)

TIP   Colors of published entities in Google Earth match progeCAD's object colors.

Toolbars, Menus, Command Shortcuts, Aliases & Buttons

this appendix illustrates the toolbars and menus included with progeCAD. It lists the default list of keyboard shortcuts and command aliases in alphabetical order. And it provides a list describing actions of mouse buttons.

Note that these interface items are specific to progeCAD.

In This Appendix

- Toolbars
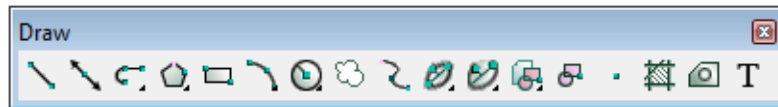- Command Aliases
- Menus
- Mouse Buttons
- Keyboard Shortcuts

Toolbars

Toolbars consist of groups of buttons that exectute commands and macros. Toolbars, macros, and help text are created and changed with the **Tools | Customize | Toolbar** command. The icons cannot be changed.

Dimensions
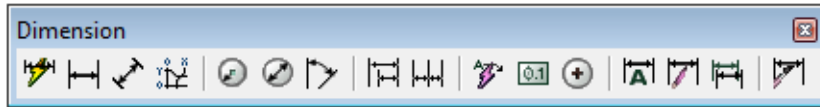
Draw 2D

DrawOrder
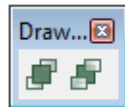
Entity Properties
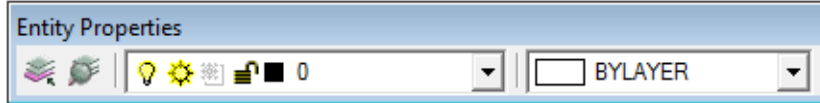
Entity Snaps
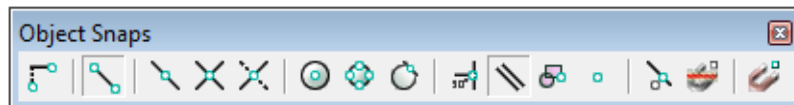
Express Tools

Express Tools – Layers

Google Earth
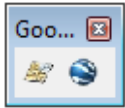
Groups

Inquiry

Insert

Layouts
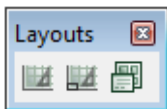
Modify

Modify II

Redlining
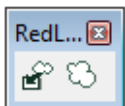
RefEdit

Reference
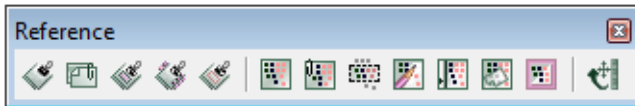
Render
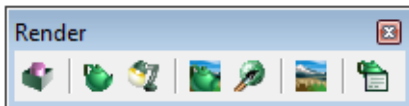
Shade

Solids

Solids Editing

Standard

Style

Surfaces

Text

UCS
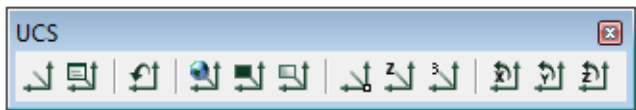
View
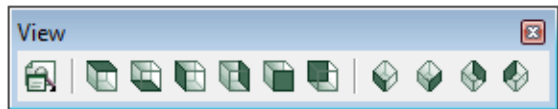
Viewports
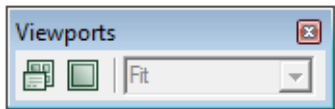
Zoom

3dOrbit

Menus

The menu bar, menu items, macros, and help text are created and changed with the **Tools | Customize | Menu** command. The "experience level" is selected with the **Tools | Options | General** command: advanced, intermediate, and beginner. Shown below are advanced menus.

File

New...	Ctrl+N
Open...	Ctrl+O
Close	
Acis In	
Save	Ctrl+S
Save As...	
eTransmit...	
Export...	
Convert PDF to DXF	
Acis Out	
Print Setup	
Plot Preview	
Plot...	Ctrl+P
Print style Manager	
Utility	▶
Send...	
1 ...- Dec 08\Drawings\4Text.dwg	
2 ...awings\3DoorsAndWindows.dwg	
3 ...awings\Door-Exterior-36.dwg	
4 ...08\Drawings\5Dimensions.dxf	
Exit	

Edit

Undo	Ctrl+Z
Redo	Ctrl+Y
Cut	Ctrl+X
Copy	Ctrl+C
Copy with base point	Ctrl+Shift+C
Copy Quick	Ctrl+D
Copy Quick with base point	Ctrl+Shift+D
Paste	Ctrl+V
Paste to original coordinate	
Paste as block	
Paste Special...	
Delete	Del
Select All	Ctrl+A
OLE Links...	
Find and Replace	

View

Redraw	
Regen	
Regen All	
Zoom	▶
Pan	▶
3D Orbit	▶
Viewports	▶
Named Views...	
3D Views	▶
Dynamic View	
Viewpoint Presets	
Workspaces...	
Paper Space Views	
Hide	
Shade	▶
Render	▶
Display	▶
Toolbars	

Insert

Block...	
Block from library	
External Reference...	
Attach raster image...	
Import RedLine	
Layout	▶
ACIS File	
Define Attributes...	
OLE Object...	
XRef Manager	
Hyperlink	

Format

- Layer...
- Layer State Manager...
- Set layer from Entity
- Colors...
- Fill
- Explore Linetypes

- Reference Grid   F7
- Snap   F9
- Entity Snap
- Draw Orthogonal   F8

- Explore Text Styles
- Text Style...
- Explore Dimension Styles
- Dimension Style...
- Point Style...
- Explore Blocks
- Explore Views

- Units...
- Elevation
- Thickness
- Drawing Limits

- Rename

Tools

- Spell Check
- Dwg Explorer
- Draw Order
- Inquiry
- Entity Data
- Audionote
- Extract Attributes...
- Data Extraction...
- In-place Block Edit
- Properties   Ctrl+1
- progeCAD Explorer   Ctrl+2
- Run Script...
- Record Script...
- End Script...
- Macro...
- Load LISP or SDS Application
- Named UCS...
- Orthographic UCS...
- New UCS
- Drawing Settings...
- Make Snapshot...
- View Snapshot...
- Tablet
- Customize
- Options...

Draw

- Line
- DLine
- Draw Traces
- Ray
- Construction Line
- Freehand
- Polyline
- 3D Polyline
- Polygon
- Rectangle
- Arc
- Circle
- Donut
- Plane
- Spline
- Ellipse
- Block
- Point
- Hatch...
- Boundary...
- Wipeout
- Revision Cloud
- Region
- Text
- 3D Entities
- Surfaces

Dimensions

- Quick Dimension
- Linear
- Aligned
- Coordinate
- Rotated
- Leader
- Radius
- Diameter
- Angular
- Baseline
- Continue
- Tolerance...
- Center Mark
- Make Oblique
- Edit Dimension Text
- Rotate Dimension Text
- Reposition Dimension Text
- Restore Text Position
- Save Style
- Restore Style
- Align Text
- Dimension Style
- Override
- Update

Modify

- Properties
- Change
- Match Properties
- Object ▸
- Clip ▸

- Delete
- Copy
- Mirror
- Offset
- Array

- Move
- Rotate
- Scale
- Stretch
- Edit Length

- Trim
- Extend
- Break
- Join
- Chamfer
- Fillet

- Flatten

- 3D Operation ▸
- Solids Editing ▸

- Explode

- Group...

- Hyperlink
- Open link

Image

- Attach Raster Image
- Image Management

- Image Quality
- Display Image Frame
- Image Correlate
- Draw Order
- Clip Image
- Transparency

- ECW/JPEG2000 ▸

Add-On

- Ale - Blocks Manager
- Raster to vector Wintopo

- Convert PDF to DXF

- O2C Converter ▸

- Google Earth ▸

Express

- Attach Raster Image
- Image Management

- Image Quality
- Display Image Frame
- Image Correlate
- Draw Order
- Clip Image
- Transparency

- ECW/JPEG2000 ▸

Window

- New Window

- Close
- Close All

- Cascade
- Tile Horizontally
- Tile Vertically
- Arrange Icons

- 1 4Text.dwg
- ✓ 2 Drawing2

Help

- Help   F1
- Printable Help
- System Variables
- Command reference
- Developer reference
- Help Render

- Tip of the Day
- Online Support
- FAQ

- progeCAD on the Web
- Buy professional version OnLine

- Check for progeCAD Update

- Register progeCAD

- About progeCAD

Keyboard Shortcuts

Keyboard shortcuts are abbreviations for commands. They consist of function keys, or holding down the Ctrl and Shift keys in conjunction with other keys. Keyboard shortcuts are created and changed with the **Tools | Customize | Keyboard** command.

Shortcut	Meaning	Command Executed
Function Keys		
F1[a][w]	Display on-line help.	HELP
F2[a]	Display Prompt History window.	PMTHIST
F3[a]	Toggle object snap mode.	ESNAP T
F4[a]	Toggle tablet mode.	TABLET T
F5[a]	Switch to next isoplane.	ISOPLANE
F6[a]	Toggle coordinate display.	COORDINATE T
F7[a]	Toggle display of the grid.	GRID T
F8[a]	Toggle ortho mode.	ORTHOGONAL T
F9[a]	Toggle snap mode.	SNAP T
F10	Toggle status bar.	STATBAR T
Control Keys		
Ctrl+A[a][w]	Select all entities in drawing.	SELGRIPS ALL
Ctrl+C[a][w]	Copy selected entities to Clipboard.	COPYCLIP
Ctrl+Shift+C	Copies entities with a basepoint.	COPYBASE
Ctrl+D	Copies large numbers of entities.	COPYQUICK
Ctrl+Shift+D	Copies with basepoint.	COPYBASEQUICK
Ctrl+E[a]	Switch to next isoplane.	ISOPLANE
Ctrl+N[a][w]	Start a new drawing.	NEW
Ctrl+O[a][w]	Open a drawing file.	OPEN
Ctrl+P[a][w]	Print the drawing.	PRINT
Ctrl+S[a][w]	Save the drawing.	QSAVE
Ctrl+T[a]	Toggle tablet mode.	TABLET T
Ctrl+V[a][w]	Paste from Clipboard into drawing.	PASTECLIP
Ctrl+X[a][w]	Cut selected entities to Clipboard.	CUTCLIP
Ctrl+Y[a][w]	Redo the last undo.	REDO
Ctrl+Z[a][w]	Undo the last command.	U
Ctrl+1[a]	Toggles the Properties palette.	PROPERTIES
Ctrl+2[a]	Displays the Explorer dialog box.	EXPLAYERS
Other Keys		
Del[a][w]	Deletes selected entities .	DELETE
Left Arrow	Pans left.	PAN L
Right Arrow	Pans right.	PAN R
Down Arrow	Pans down.	PAN D
Up Arrow	Pans up.	PAN U
Shift+Left Arrow	Pans left by a pagefull.	PAN PGL
Shift+Right Arrow	Pans right by a pagefull.	PAN PGR
Shift+Down Arrow	Pans down by a pagefull.	PAN PGD
Shift+Up Arrow	Pans up by a pagefull.	PAN PGU
Page Down	Pans down by a pagefull.	PAN PGD
Page Up	Pans up by a pagefull.	PAN PGU

[a] Shortcut key compatible with AutoCAD.   [w] Shortcut key standard in Windows.

Command Aliases

Aliases are one-, two-, and three-letter abbreviations for commands. They let keyboardists enter commands faster; it is far quicker to type "l" than to type "line".

Aliases can be used to map the names of commands from other CAD packages, such as AutoCAD, to progeCAD commands. For example, AutoCAD's "pedit" is mapped to progeCAD's EditPline command. Aliases cannot be used for command options, or for accessing options within dialog boxes.

Aliases are created and changed with the **Tools | Customize | Aliases** command.

Command	Alias(es)		Command	Alias(es)
A			**Dim**	
Align	al		DimAligned	dimali, dal
Aperture	ap		DimAngular	dimang, dan
Arc	a		DimBaseline	dba, dimbase
Area	aa		DimCenter	dce
Array	ar		DimContinue	dco, dimcont
AttDef	-at		DimDiameter	ddi, dimdia
AttDisp	ad		DimEdit	dimed, ded
AttEdit	-ate		Dimension	dim
AttExt	ax		DimLinear	dli, dimlin
			DimOrdinate	dimord, dor
B			DimOverride	dov, dimover
Backgrounds	background		DimRadius	dimrad, dra
Base	ba		DimStyle	dst, dimsty
BHatch	bh		DimTEdit	dimted
Blipmode	bm			
Block	-b		Dist	di
Boundary	bo, bpoly		Divide	div
-Boundary	-bo		DLine	dl
Break	br		Donut	do, bagel
Bsr_Stru	toolbar		DTtext	dt
			DView	dv
C			DxfIn	dn
Chamfer	cha		DxfOut	dx
Change	-ch			
Circle	c		**E**	
Config	options, rconfig, preferenc, pr		EditLen	lengthen, len
Copy	cp, co		EditPline	pedit, pe
Copylink	cl		Ellipse	el
			EntProp	ddmodify, ai_propchk, mo
D			Erase	e
DdAttDef	at		ESnap	-os, osnap, -osnap
DdAttE	ate		ExpBlocks	xb, bmake, b
DdChProp	ch		ExpDimstyles	ds
DdEdit	ed		ExpFonts	ddstyle, expstyle, expstyles
DdGrips	gr		ExpLayers	ddlmodes, la, explorer
DdInsert	i		Explode	x
DdRename	ren		ExpLtypes	ddltype, lt
DdRModes	rm		Export	exp
DdSelect	se		ExpUcs	dducs, uc
DdUnits	un			

Command	Alias(es)
ExpViews	ddview, v
Extend	ex
Extrude	ext
F	
Fillet	f, fi
Font	st, style
Freehand	sketch
G	
Grid	g
H	
Hatchedit	he
Hide	hi
I	
IdPoint	id
ImageAttach	iat
ImageClip	icl
InsertObj	io
Interfere	inf
Intersect	in
Isoplane	is
L	
-Layer	-la
Leader	le
Lighting	light, lights
Line	3dline, l
-Linetype	-lt
List	li, ls
LtScale	lts
M	
MatchProp	ma
Materials	rmat
Mesh	3dmesh
Mirror	mi
Move	m
MsnapShot	mslide
MSpace	ms
MText	mt, t
MView	mv
N	
New	n
NewWiz	ddnew
O	
Open	op, import
Orthogonal	ortho, or

Command	Alias(es)
P	
-Pan	p, -p
Parallel	o, offset
PasteSpec	pa
Plane	so, solid
Point	po
Polygon	pol
Polyline	pline, pl
PPreview	pre, makepreview, preview
-Print	-plot
Print	plot
PSpace	ps
Purge	pu
Q	
Qtext	qt
R	
Rectangle	rect, rec
Redraw	r
RedrawAll	ra
Regen	re
RegenAll	rea
Region	reg
ReInit	ri
Rename	-ren
Render	rr
Revolve	rev
Rotate	ro
S	
Save	sa
Scale	sc
Script	scr
Section	sec
SelGrips	selgrip
SetColor	col, ddcolor
SetDim	ddim, d
SetEsnap	os, ddosnap, ddesnap
SetRender	rpref
SetUcs	ucp, dducsp
SetVar	set
Shade	sha
Slice	sl
Snap	sn
Spell	sp
Spline	spl
SplinEdit	spe
Stretch	s
Subtract	su

Command	Alias(es)
T	
Tablet	ta
TbConfig	to
Text	-t, tx
Time	ti
Tolerance	tol
Torus	tor
Trim	tr
U	
Undelete	oops, oo
Union	uni
Units	-un
V	
Vba	vbaide
-View	-v, v
View	ddview
ViewCtl	vp, ddvpoint
Viewpoint	vpoint, -vp
VpLayer	vl
VPorts	vw, vport
VSnapshot	vs, vslide

Command	Alias(es)
W	
WBlock	w
WCloseAll	closeall
Wedge	we
WmfIn	wi
WmfOut	wo
X	
XBind	-xb
XClip	clip , xc
XLine	xl, infline
-XRef	-xr, xa, xattach
Z	
Zoom	z
3	
3dArray	3a
3dPoly	3p

Mouse Buttons

Mouse and puck buttons execute commands. Mouse buttons cannot be customized; puck buttons can be customized through the *tablet.lsp* program.

Button	Action
Objects Not Selected	
Left	Selects entities, menu items, toolbar buttons, and so on.
Shift+Left	Displays shortcut menu with entity snaps.
Right	Ends command; repeats last command.
Entities Selected	
Left (single click)	Deselects entities.
Left (double-click)	Displays entity properties dialog box.
Shift+Left	Displays shortcut menu with entity snaps.
Right	Displays shortcut menu with editing commands.
Wheel (scroll bars not displayed)	
Roll forward	Zooms in 1.25x.
Roll back	Zooms out 0.8x.
Wheel (scroll bars displayed)	
Roll forward	Pans up.
Roll back	Pans down.
Ctrl+Shift+Left	Pans left or right.
Shift+Left	Zooms in or out.
Other Commands	
Left	Starts and ends sketch mode (Sketch command).
Left	Enables real-time rotation (RtRot command).