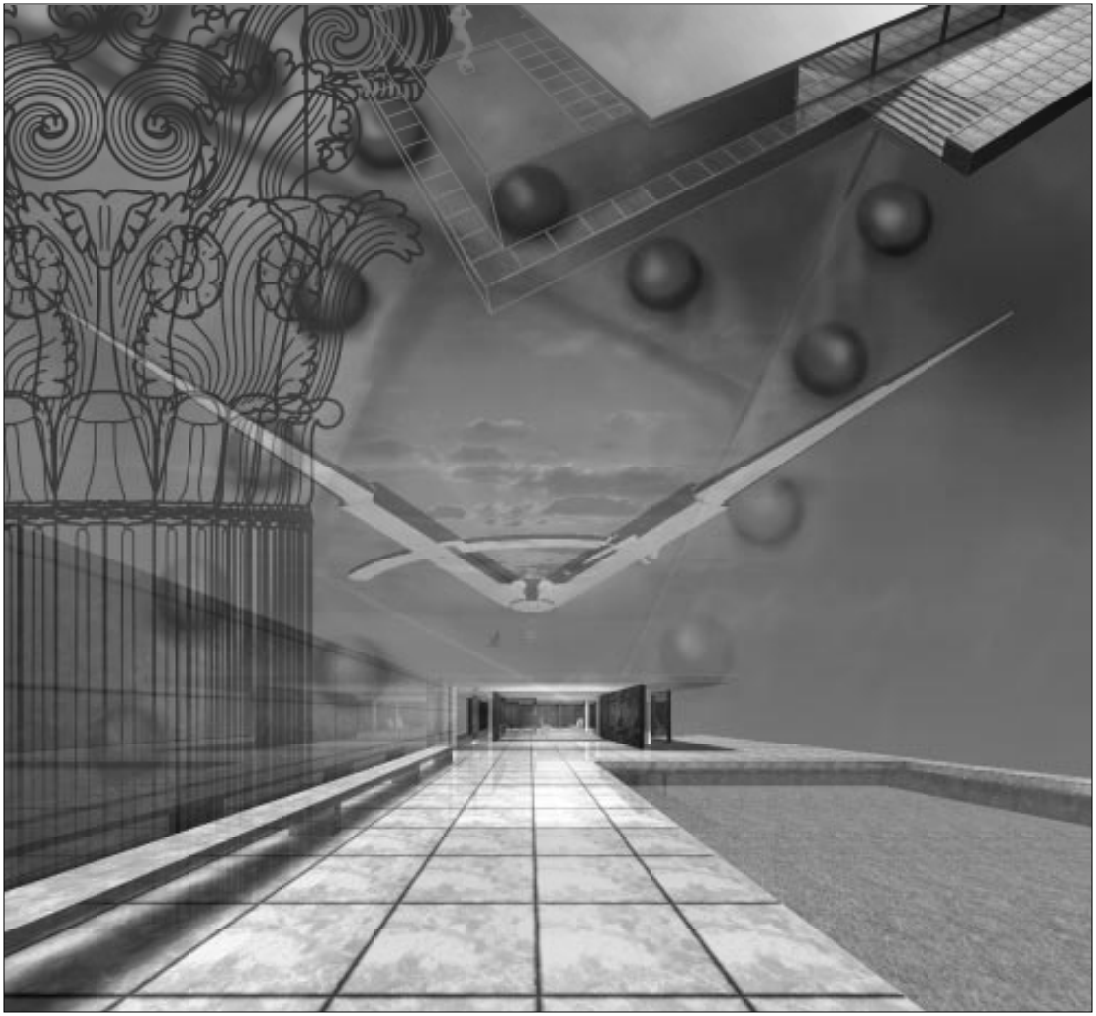
# DenebaCAD



DENEBA

## Evaluation Guide
for Mac OS

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## DenebaCAD Evaluation Guide


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# CONTENTS

# INTRODUCTION

Welcome to DENEBACAD, a new perspective for CAD on Mac OS™ systems.

DENEBACAD gives professionals in many disciplines the tools to get their work done faster and easier than ever before. DENEBACAD is an integrated solution that lets you produce 2D drawings, 3D models, and realistic renderings in one program.

You can open multiple windows to visualize a project from several views and perspectives. You can fine-tune the details in any window and see instant updates in all open windows. The fast rendering engine offers natural and artificial lighting, shading, reflectiveness, transparency, and surface mapping controls.

This is the DENEBACAD Evaluation Guide. This documentation accompanies evaluation versions of DENEBACAD. A manual with a comprehensive reference section is included with registered versions of DENEBACAD.

This chapter provides information on system requirements for running DENEBACAD and basic skills you should know.

## THINGS YOU SHOULD KNOW

### Recommended system requirements

For best performance with DENEBACAD, it is recommended that your system should contain the following minimum software and hardware components:

- A Mac OS compatible system with a PowerPC™ processor, running Mac OS System 7.5 or later
- 16 MB of RAM available to DENEBACAD (minimum)
- 32 MB of RAM available to DENEBACAD (recommended)
- 20 MB of free hard disk space available for installation
- A CD-ROM drive
- A 24 -bit color video display capable of 800 x 600 dpi resolution or higher

### Mac OS skills

You should be familiar with the basics of using the Mac OS operating system. If you need more information on any of the following topics, please refer to your Mac OS documentation.

- Using the mouse
- Opening, closing, resizing, and moving windows and using scroll bars
- Opening and saving files using the standard directory dialog box

## Conventions used in the documentation

The DENEBACAD documentation uses certain terms and syntax when describing mouse actions, commands, and keyboard keys.

**Click** To quickly depress the button on the mouse (or other pointing device) and release. Often, clicking is done at a specific location after moving the pointer to the location.

**Double-Click** To click the mouse button twice in a row with almost no pause between clicks. You must not move the mouse between clicks.

**Drag** To press the mouse button and move the mouse while the button is held down. Usually the button is released when the pointer has been moved to the location you want.

**Press** To hold down the mouse button while the pointer is over a specific location. This term is also used to describe the action of using a keyboard key.

**Select** To drag the mouse over text or click on a window, icon, object, or option in a dialog box.

## Choosing commands

To execute many procedures in DENEBACAD, you choose a command from a menu, submenu, or pop-up menu. Choosing a command is accomplished the same way in nearly all Mac OS software. If you are not familiar with using menu commands, read the procedures below to learn the necessary steps.

### To choose a command in a menu

1. Move the pointer to the menu title in the menu bar.
2. Press the mouse button. The menu drops down from the menu bar.
3. Move the pointer to the menu command you want to choose and release the mouse button.

### To choose a command in a submenu

1. Move the pointer to the menu title in the menu bar and press the mouse button. The menu drops down from the menu bar.
2. Drag to the submenu that contains the command you want to choose. The submenu expands when the pointer is on the submenu name.

   • You can identify a submenu by the small triangular indicator (▶) that appears at the right of the submenu name.
3. Drag into the submenu to the command that you want to choose. Release the mouse button when the command name is highlighted.

## Choosing menu commands

Follow the steps illustrated here to choose commands in menus and submenus.

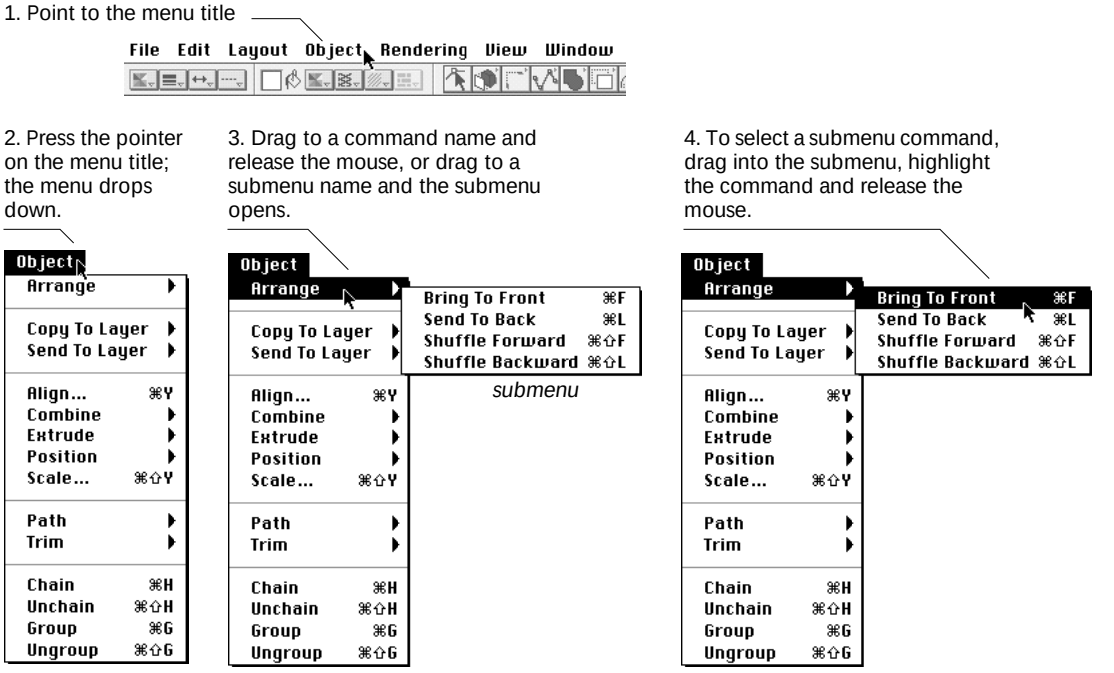



*Fig. 1 Choosing menu commands*

## Activating windows

Because you can display multiple views of a drawing on screen, you often need to make a particular window active before you can use a command or tool. For example, to change your view of a rendering, you need a Render mode window to be active, and then you can use one of the Focal Point tools.

To activate a window, simply click anywhere in the window, or on the edge of the window.

Only one drawing window is active at any time. The active window appears in front of other drawing windows. Windows that are not active appear gray or dimmed.

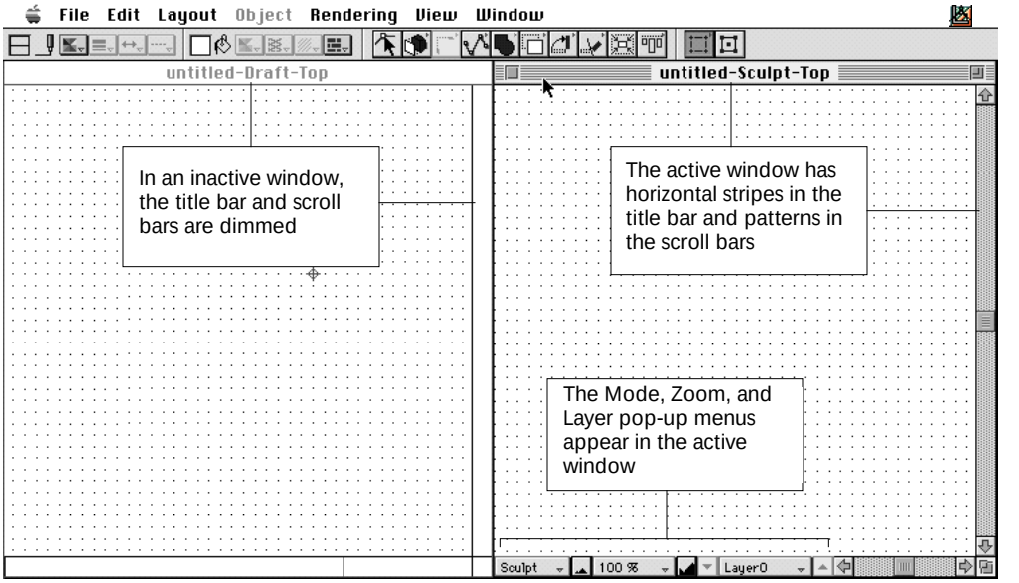



*Fig. 2 Active and inactive windows*

## KEYBOARD KEYS AND COMMAND SHORTCUTS

When you can execute a command by pressing keyboard keys, this is referred to as a *keyboard shortcut* or *command shortcut.*

To use most command shortcuts, you press a special key at the same time you press a letter or number key on the keyboard. This manual uses standard names and abbreviations for special keys. However, the keyboard that you use might have different labels.

The following are special keys you can use for command shortcuts:

**Command** This keyboard key is labeled "Command," or marked with a propeller ( ⌘ ) or Open Apple symbol. In menus, the propeller symbolizes the Command key.

**Control** The keyboard key labeled "Control" or "Ctrl." In menus, a caret ( ^ ) symbolizes the Control key.

**Option** The keyboard key labeled "Option." In menus, an option symbol ( ⌥ ) designates the Option key.

**Shift** This key is labeled "Shift" on most keyboards. This is the key you press to type capital letters. In menus, an upward pointing arrow ( ⇧ ) designates the Shift key.

# INSTALLING DENEBACAD

Before installing any Mac OS program, it's a good idea to disable all unnecessary system extensions. You can disable all extensions by restarting your computer and holding down the Shift key until the message "Extensions off" appears.

*Note:* If you do not have a valid DenebaCAD serial number see the Readme file included on the CD.

### To install DENEBACAD

1. Double-click the Installer icon on the first DENEBACAD floppy disk or CD-ROM.
2. An installer screen appears. Click Continue.
3. A scrolling window displays important information about the program. Be sure to read the text, and then click Continue.
4. An installation dialog box appears. Use the options in this dialog box to configure the installation of DENEBACAD on your computer (*figure 4*).
5. Select Easy or Custom installation from the pop-up menu at the upper left.

   ▲ Choose Easy unless you know that you want to install only certain components of DENEBACAD.

   ▲ If you choose Custom, the scrolling list displays the DENEBACAD components you can install. Select check boxes for the items you want to install.
6. Select a folder in which to install the DENEBACAD program.

   ▲ Select an available disk, and then choose Select Folder in the pop-up menu under Install Location. A directory dialog box appears. Highlight the name of the folder where you want to install DENEBACAD, and then click Select.
7. Click Install to begin installation. A window shows you the progress of installation. When installation is complete, a message appears. Click Quit to leave the installer.

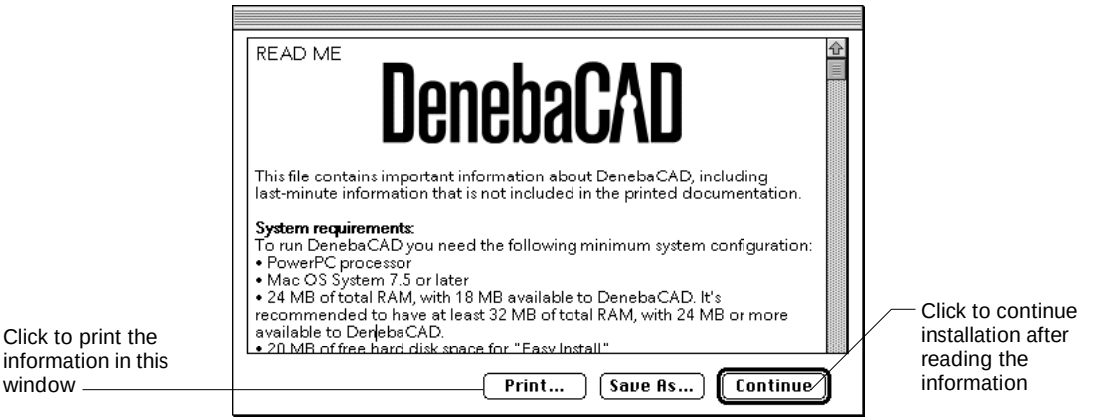



*Fig. 3 Information window*

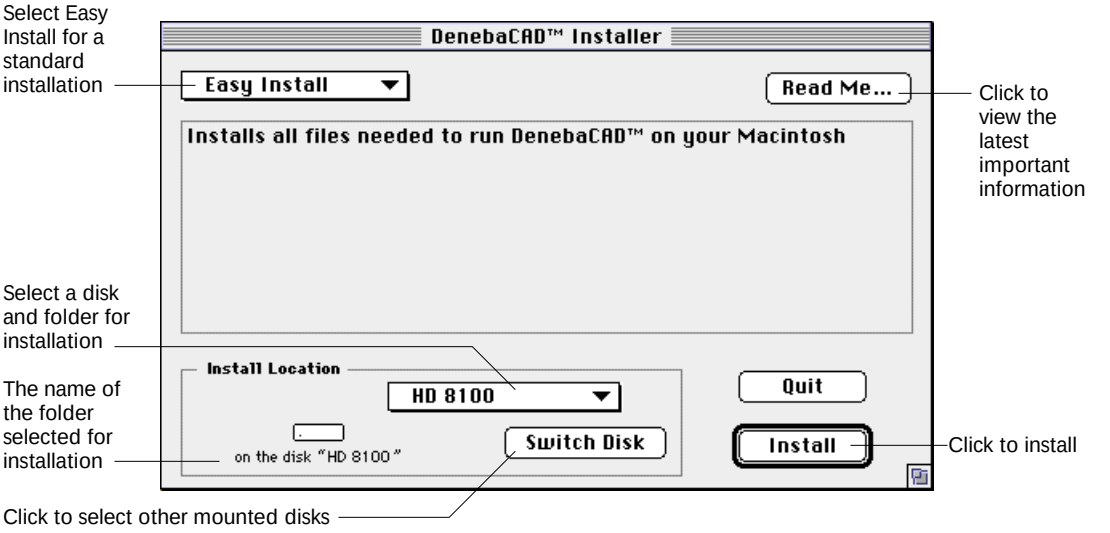
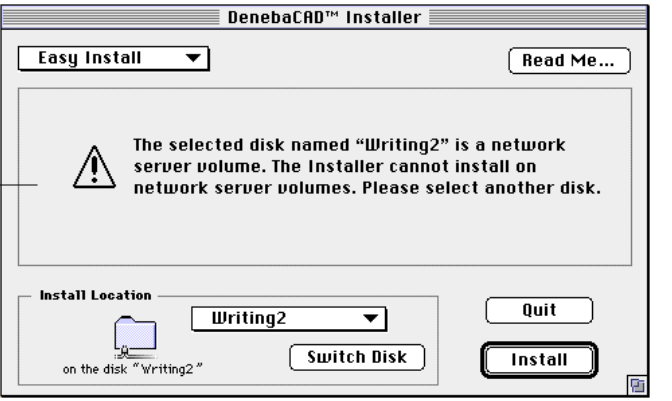
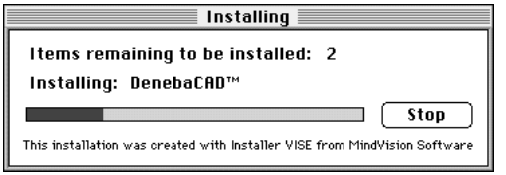



*Fig. 4 Installation dialog box options*

### Installation messages

If you select a network volume as the Install Location, a message tells you that you cannot install DENEBACAD on a network server volume. You should select a disk that is part of your computer and not located on a network.




This window lists the number of items to be copied and the bar indicates the progress of installation


A message box appears when DENEBACAD has been installed successfully. Click Quit to leave the installer

*Fig. 5 Installation messages*

# LAUNCHING DENEBACAD

To start DENEBACAD, double-click the DENEBACAD icon in the DENEBACAD folder.

▲ The start-up screen appears while DENEBACAD is loading.

▲ *Note:* If DENEBACAD can't locate your Materials folder, a directory dialog box appears asking you to locate your Materials folder. This might happen if you misplace the folder, or perform a custom installation of DENEBACAD without installing materials.

▲ When DENEBACAD finishes loading, it creates a new document. The untitled document appears in a Draft mode window in Top view.

▲ If you double-click a DENEBACAD document icon to launch the program, the selected document appears, in the mode that was active when the document was saved.

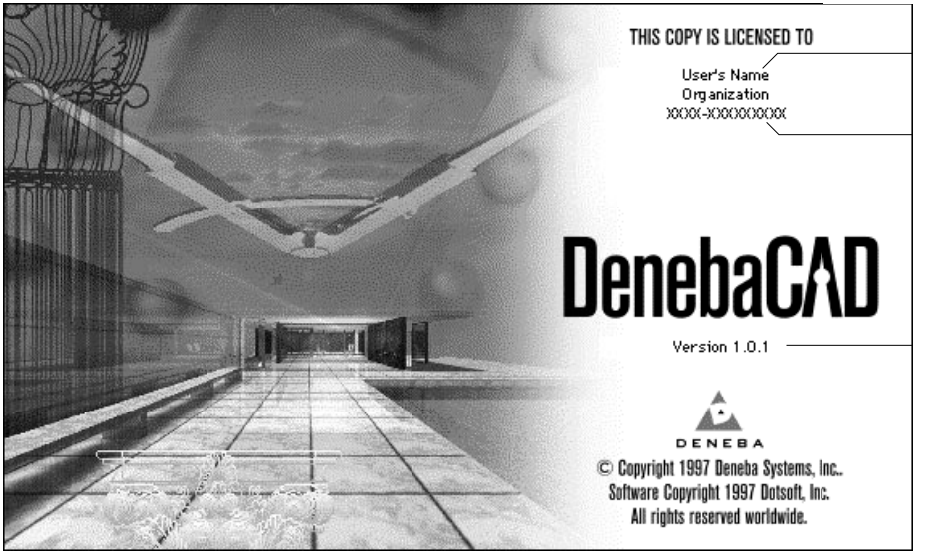



*Fig. 6 DENEBACAD start-up screen*

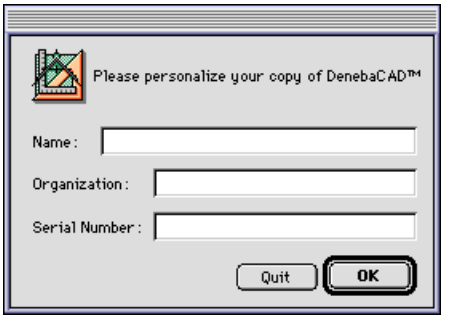
## PERSONALIZING DENEBACAD

The first time you launch DENEBACAD after installing the program, you need to type your name and evaluation serial number in the personalization dialog box. This procedure has to be performed only once (unless you throw away the DENEBACAD Prefs file).

Keep your evaluation serial number in a safe place. You need it to personalize DENEBACAD again if you reinstall the program.

**To enter personalization information**

1. When you launch DENEBACAD the first time, the personalization screen appears.
2. Type your name and organization in the text boxes. In the Serial Number box, type the evaluation serial number.
3. Click OK to personalize DENEBACAD.




# ENDING A WORK SESSION

When you finish working and want to stop running DENEBACAD, use the Quit command.

◆ **To stop running DENEBACAD:** Choose Quit in the File menu or press Command+Q.

### Saving documents before quitting

If any unsaved documents are open when you choose the Quit command, a message box appears and prompts you to save the documents.

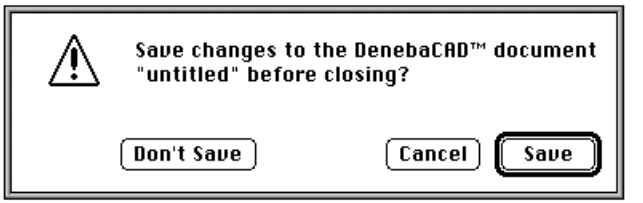
▲ If you want to continue working in DENEBACAD without quitting the program, click Cancel to close the message box.

▲ If you do not want to save changes to any open documents, click Don't Save.

◆ **To save document changes:** Click Save in the message box.

▲ DENEBACAD saves any open documents that have been saved at least once.

▲ If a document has not been saved, a directory dialog box appears. To save the document, type a name in the text box, select a location, and click Save.



# CONCEPTS OVERVIEW 1

DENEBACAD is the first fully integrated Architectural design environment for Mac OS computers. The DENEBACAD environment includes 2D drafting, 3D modeling, rendering, architectural publishing, and animation features in one application.

DENEBACAD is programmed to increase designers' productivity. By developing your projects in DENEBACAD, you can print and plot working drawings, generate animated walkthroughs, interact with virtual reality scenes, and export project management information to spreadsheet or database applications.

- You can design exclusively in 3D mode, or start in 2D and extrude into 3D.
- Once you create a 3D model, DENEBACAD lets you view it from any perspective in Render mode.
- You can also extract plans, sections, and facades from the 3D model to use as the working drawings.
- You create a structure once, and DENEBACAD makes it possible to move information between 2D and 3D without additional work.

## WORKING MODES FOR DESIGN AND RENDERING

DENEBACAD offers three integrated working modes: Draft, Sculpt, and Render. Each mode provides an appropriate tool set to streamline the design, modeling, and rendering processes.

You can open three windows for each DENEBACAD document that you open. Each window can be set to any working mode.

### Draft mode

Draft mode lets you create working drawings, including notes, dimensions, material tracking, and data analysis information.

In Draft mode, the work space contains up to 256 two-dimensional layers in each of six orthogonal views. You can picture this as if each view is a drawing pad of 256 acetate sheet layers.

### Sculpt mode

Sculpt Mode lets you design and model in 3D. To create 3D objects, you can extrude 2D objects from Draft into Sculpt mode. You can also draw 3D objects directly in six orthogonal views in Sculpt mode. You can use up to 256 total layers to organize 3D objects in Sculpt mode.

### Render mode

DENEBACAD's rendering options provide an unparalleled medium for displaying and viewing your 3D environment.

After establishing a point of view using horizontal and vertical "camera" tools, you can create snapshots or movies of your projects, complete with textured surface materials, lighting controls, and motion.

DENEBACAD lets you view a scene as Wireframe, Hidden Line, and Solid renderings. You can save renderings as still images, QuickTime™ movies, and QuickTimeVR™ environments. In addition, you can render in Stereoscopic 3D; this type of two-color-shifted rendering can be used to create a true sense of spatial perspective in scenes that users and audiences can view with 3D glasses.

## RELATIVE VIEWS

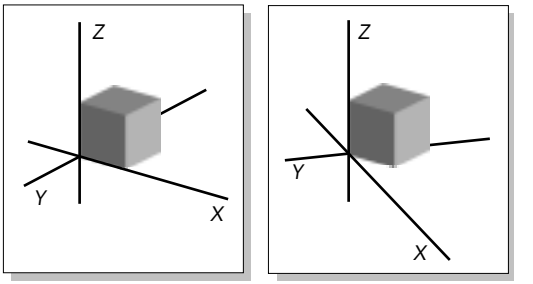
Fundamental to the operation of DENEBACAD is the use of Relative Views. In Sculpt mode, a Relative View lets you change the coordinate system to create a new drawing plane aligned with any surface. This allows precise drawing control in any environment and situation.

In DENEBACAD, you work in an infinite cubic environment. Normally, you view the environment from one face of the cube. With Relative Views, you define a new relative position from which to view the cubic environment.

For example, you can shift the Relative View so that your point of reference — the computer screen — is parallel to the slope of a roof. Then, drawing a skylight in the roof is as simple as drawing a rectangle. You can work in 3D the same way you would work in a 2D drawing.

The drawing plane is always parallel to the drawing screen. When you create a Relative View, you shift the drawing plane so the surface of any object can be viewed in real size and parallel to the drawing screen. You can always work on objects from an accurate vantage point.

Because DENEBACAD can adjust the point of reference to match the slope or angle of any object, the Relative View feature eliminates the need to calculate slopes and angles.




Coordinate axes in the default view

Coordinate axes in a relative view

*Fig. 7 Default and relative views*

## EXTRUSION METHODS FOR 3D MODELING

You can create 3D objects easily by extruding 2D shapes using one of three extrusion methods in DENEBACAD.

**Linear** This method extrudes an object by extending it along a single axis between two defined planes.

**Spin** This method extrudes an object by revolving it around a defined axis.

**Sweep** This method extrudes an object by extending it along an open or closed path.

You can create 3D objects by automatic extrusion while drawing in Sculpt mode or by selecting a 2D object and extruding it.

## EXTRUSION PLANES

You can define two extrusion planes to limit the projection of 2D objects into 3D space. For example, by setting a pair of extrusion planes at ground level and at 10 feet, you can draw a polyline to form 3D walls 10 feet tall.

If you set one extrusion plane four meters high and a second extrusion plane seven meters high, and draw a circle, this generates a cylinder three meters high placed four meters above the floor.

Extrusion planes can also be used to create flat objects contained in them. For example, use the Extrusion Format buttons to specify End Cap only extrusion format, and then draw a circle. The result is a disk floating in space.

Extrusion planes are primarily used to define the extent of objects that are extruded with the Linear extrusion method. Extrusion planes can be set up and used in Draft and Sculpt modes.

- You define extrusion planes for any view from two other views. For example, to set the height of objects you draw in Top view, DENEBACAD switches temporarily to Front view or Left view when you define extrusion planes.
- Extrusion planes can be parallel to each other, or angled independently.
- Extrusion planes are usually parallel to the view plane but may also be angled to it. The view plane is always parallel to the computer screen.

Within the 3D space, you can define, for example, that the Top view plane is horizontal, that the Front view plane is vertical and that the Left view plane is also vertical.

An extrusion plane parallel to the Top plane is horizontal and therefore perpendicular to the Z axis.

An extrusion plane angled with respect to the active view plane will be perpendicular to the view plane from where it was defined.

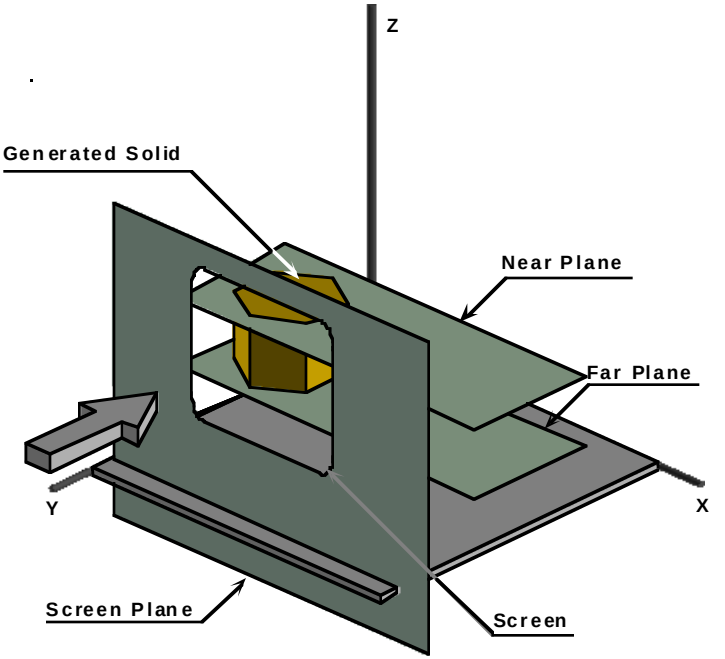



*Fig. 8 Orientation of extrusion planes relative to the viewer*

## Defining extrusion planes

In general, the extrusion planes are perpendicular to the view plane from where they were defined.

- If you are using a horizontal view (Top or Bottom) and define the extrusion planes from a frontal view (Front or Back), the extrusion planes will be angled with respect to the horizontal plane and perpendicular to the front plane (X-Z plane).
- If you are using a horizontal view (Top or Bottom) and define the extrusion planes from a side view (Left or Right), the extrusion planes will be angled with respect to the horizontal plane and perpendicular to the side plane (Y-Z plane). These planes will determine the upper and lower limits of the projected objects.
- If you are using a front view (Front or Back) and define the extrusion planes from a horizontal view (Top or Bottom), the extrusion planes will be angled with respect to the frontal plane and perpendicular to the horizontal plane (that is, vertical). These extrusion planes will determine the front and back limits of the projected objects.
- If you are using a side view (Left or Right) and define the extrusion planes from an horizontal view (Top or Bottom), the extrusion planes will be angled with respect to the lateral plane and perpendicular to the horizontal plane — that is, vertical. These planes will determine the left and right limits of the projected objects.

## EXTRUSION FORMATS

DENEBACAD lets you generate three-dimensional objects in several ways without altering the extrusion planes.

**Projected** When active, the extruded objects are projected between the two extrusion planes. This is the result of selecting the Sides Extrusion Format.

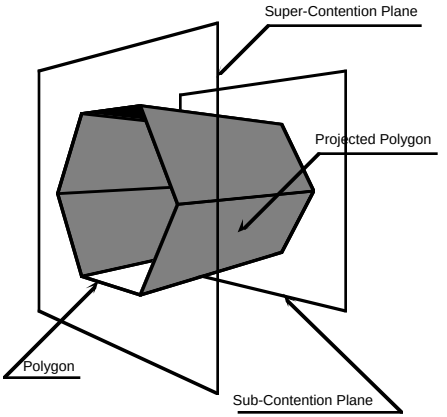



*Fig. 9 Projected extrusion*

**Super-contention** When active, the objects will be projected as contained in the extrusion plane nearest to the observer. This is the result of selecting the Front Cap Extrusion Format.

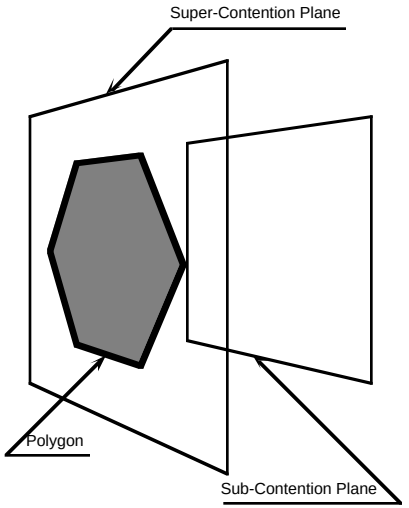
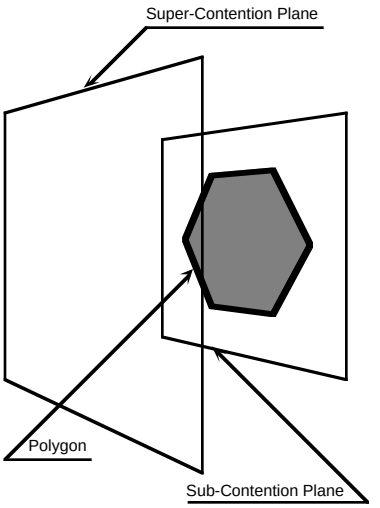



*Fig. 10 Super-contention extrusion*

**Sub-contention** When this is the active projection, the objects will be projected as contained in the extrusion plane farthest from the observer. This is the result of selecting the Back Cap Extrusion Format.




Extrusion formats are additive, so you may generate solid appearing objects by activating all three Extrusion Format buttons. The following figure illustrates several possible combinations of extrusion formats.

*Fig. 11 Sub-contention extrusion*

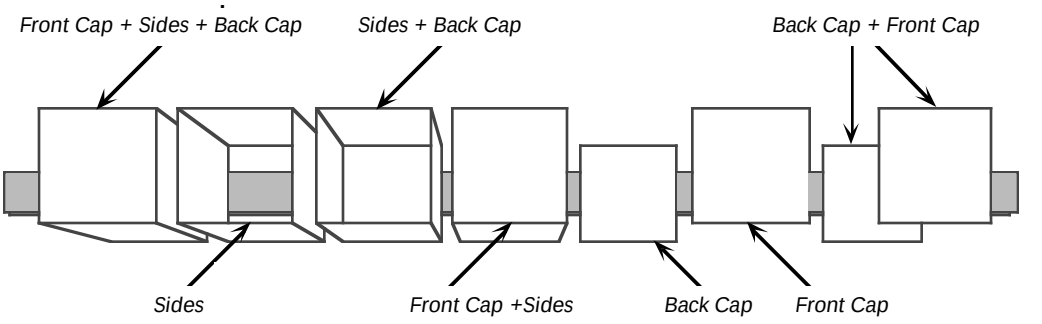



*Fig. 12 Combinations of extrusion formats*

# INTERFACE OVERVIEW 2

DENEBACAD's drawing environment makes the program easy to use and easy to adapt to your working methods. Items such as shortcut buttons, context-sensitive prompts, smart pointers, and status readouts help new users learn the program quickly and help advanced users become more productive.

This chapter presents an overview of the DENEBACAD work environment, including brief descriptions of interface elements, tools, and drawing modes.

## WINDOWS AND TOOLBARS

You can customize many aspects of the DENEBACAD interface (*figure 13*), including the display of toolbars, information fields, and multiple windows.

By default, DENEBACAD displays only the toolbox, the Attributes bar and Action buttons, and the Help bar. You can also display optional toolbars that offer quick access to snap settings, position data, and drawing and viewing options.

This section briefly described the features of the optional interface features and tells you how to display them.

◆ **To display or hide toolbars:** Choose the Toolbars command in the Layout menu. In the Toolbars dialog box, select the options you want to display and click OK. When you first start using the program, it's recommended that you select all the toolbar items to display.

DENEBACAD displays the same elements each time you launch the program, until you change the setup.

### Attributes bar

The Attributes bar contains buttons you can use to select attributes such as colors, patterns, arrowheads, and Surface materials. You can also use the attributes buttons to open the Pen and Fill palettes.

### Action buttons

Action buttons appear as an option to the right of the Attributes buttons. Action buttons provide shortcuts for the commands in the following submenus: Extrude, Trim, Path, Combine, and Position. There are Action buttons for the Reshape, Scale, and Align commands also.

### Info bar

The Info bar shows the absolute and relative coordinates of the pointer and the drawing vector. This data appears when you create or modify objects and when you set extrusion planes, Relative Views, and other items.

By pressing Tab, you can enter coordinates directly in the text boxes in the Info bar.

### Status bar

The pop-up menus on the Status bar show the current viewing mode, layers display, Clipping plane, extrusion planes, and Relative View, and let you select new settings.

### Extrusion Format buttons

The Extrusion Format buttons set constraints for extruding objects as solids, or as sides, faces, or any combination.

### Toolbox

The toolbox is a floating palette containing tools for creating objects, dimensioning, zooming, and panning.

### Help bar

The Help displays the names of tools and commands when the pointer is on an item, and gives feedback on procedures. The amount of available memory is displayed at the right end.

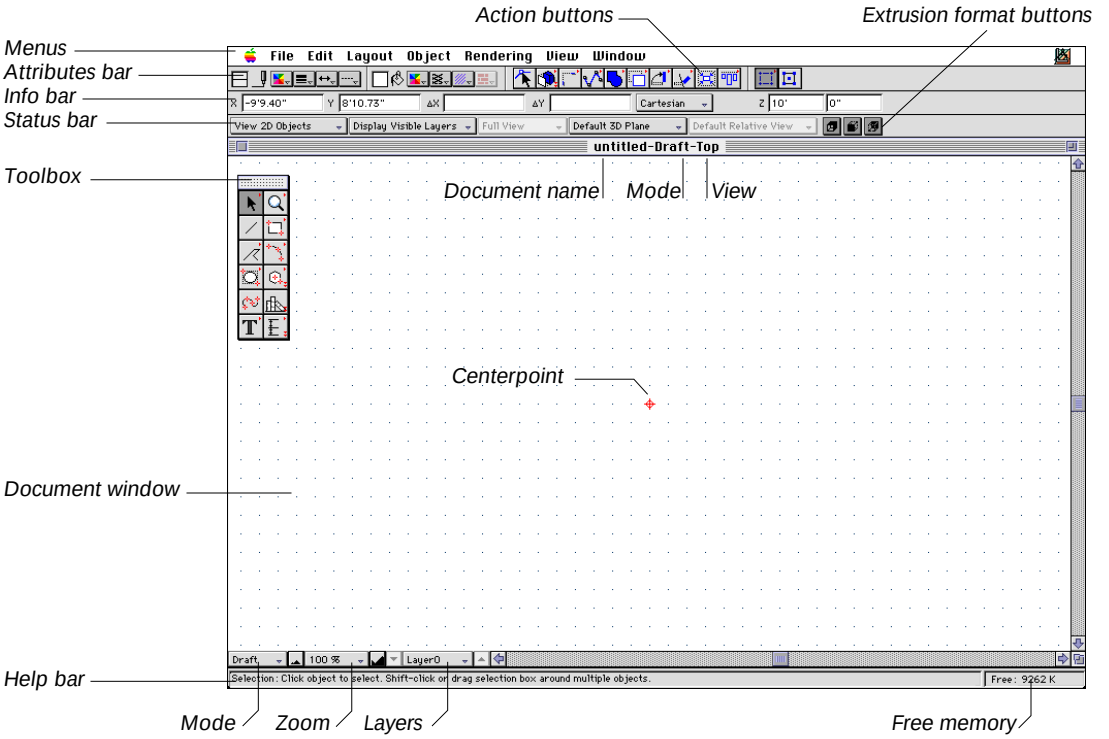


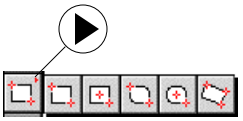

*Fig. 13 Main elements of the DENEBACAD interface*

# DRAFT, SCULPT, AND RENDER TOOLS

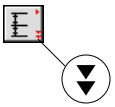
DENEBACAD's toolbox is a floating window that displays DENEBACAD's tools for drawing and rendering. Additional tools appear on toolbars that pop out like drawers from the main toolbox.

### Tool icons

A single arrow at the upper-right in a tool icon indicates that a toolbar will open if you press the icon. Similar tools are grouped on a pop-out toolbar. Drag into the toolbar to select a specific tool. The last tool that you select appears in the home position in the toolbox.



*Toolbar*



When a tool icon has two arrows pointing down, the tool has options you can configure. Double-click the icon to open an options dialog box.

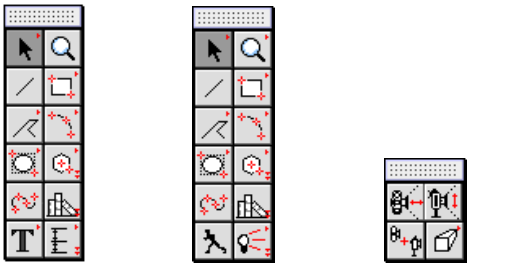
### Toolbox modes

The toolbox appears in one place on the screen, though the tools in the toolbox change for Draft, Sculpt, and Render modes.

The toolbox changes when you activate a window set to a different mode, or change the mode of the active window.

- In Draft mode, the toolbox contains tools for 2D drawing, dimensioning, and placing text.
- In Sculpt mode, 3D drawing tools, the Light Source tools, and the Camera Path tool appear in the toolbox.
- In Render mode, the Focal Point tools for setting point of view, and tools to change the rendering perspective appear in the toolbox.

To avoid selecting the wrong tool, set the drawing mode by clicking the window you want to draw in before you select a tool from the toolbox.



Draft mode tools

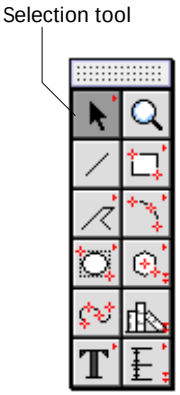
Sculpt mode tools

Render mode tools

*Fig. 14 Tools in Draft, Sculpt, and Render modes*

## SELECTING TOOLS

When you start DENEBACAD in Draft or Sculpt mode, the Selection tool is active. To use another tool, click the tool's icon. The icon of the active tool is shaded.




To select a tool located on a pop-out toolbar, press the home icon (the tool visible in the toolbox) to open the toolbar, drag to the tool you want to use and release the mouse. The tool you select stays in the home position for the toolbar.

To move the toolbox, drag the window by its title bar. If you use Mac OS 8 or have the Window-Shade control panel installed, you can roll up the toolbox and other windows into their title bars by double-clicking the title bars (if this setting is selected in the Window Shade or Appearance control panel).

## USING DRAWING TOOLS

To draw objects, select a drawing tool from the toolbox and follow the prompts displayed in the Help bar. You draw 2D objects in a Draft mode window and 3D objects in a Sculpt mode window.

With most drawing tools, you can click the mouse to set an object's creation points. With the Circle by Radius tool, for example, you click to set the center of the circle and click again to set the circle's radius. Or, you can drag the mouse from the first point to the second one.

When the Info bar is displayed, you can enter numeric data when creating objects. Press the Tab key to enter X, Y, or Z coordinates (if the Cartesian coordinate system is selected); click the check mark button or press Return to set the first point. Or, press Enter and then enter relative coordinates for subsequent points. To move from one field to another in the Info bar, press Tab.

# TABLE OF DENEBACAD TOOLS

The following table lists every tool that appears in the toolbox in Draft, Sculpt, and Render modes. The tools are listed in alphabetical order and according to the toolbars in which they are grouped in the toolbox.

| Toolbar | | Tool names | Functions | Modes |
|---|---|---|---|---|
|  | Arc | Arc Radius<br>Arc 3 Points<br>Arc Elliptical | Draw arcs from center to vertex, from 3 points, or from 5 points | Draft, Sculpt |
|  | Camera Path | Camera Path | Set waypoints and camera vectors for rendering walk-throughs | Sculpt |
|  | Curve | Curve | Draw open and closed objects with smooth curves | Draft, Sculpt |
|  | Dimensions | Chain Dimension<br>Baseline Dimension<br>Leader Dimension<br>Constrained Dimension<br>Angle Dimension<br>Leader Text | Dimension 2D objects, place baseline, chain, or angular dimensions and single or double leaders in drawings | Draft |

| Toolbar | | Tool names | Functions | Modes |
|---|---|---|---|---|
|  | Ellipse | Ellipse Diagonal<br>Ellipse Center to Corner<br>Circle Radius<br>Circle 3 Points<br>Ellipse 3 Points | Draw ellipses and circles from corner to corner, from center to corner, by radius, or by setting 3 points | Draft, Sculpt |
|  | Focal Point | Horizontal Focal Point<br>Vertical Focal Point<br>Dual Focal Point | Set the horizontal viewpoint, vertical viewpoint, or both at once for renderings | Render |
|  | Library | Library | Place Library Objects and Symbols in drawings | Draft, Sculpt |
|  | Light | Directional Light<br>Omnidirectional Light | Place light sources for illumination in solid renderings | Sculpt |
|  | Line | Line | Draw lines and planes | Draft, Sculpt |
|  | Polygon | Polygon Vertex<br>Polygon Midpoint | Draw closed, multiple-sided objects from the center to a side or vertex | Draft, Sculpt |
|  | Polyline | Single Polyline<br>Double Polyline | Draw open polygons with or without offset edges | Draft, Sculpt |
|  | Rectangle | Rectangle Diagonal<br>Rectangle Center to Corner<br>Rounded Rectangle Diagonal<br>Rounded Rectangle Center to Corner<br>Rectangle 3 Points | Draw rectangles, rounded rectangles and cubes from corner to corner, center to corner, or from 3 corner points | Draft, Sculpt |
|  | Render View | Perspective Render<br>Isometric Render | Select standard or isometric perspective for renderings | Render |
|  | Selection | Selection<br>Point Selection | Select objects and text<br>Select points within objects | Draft, Sculpt |
|  | Text | Text<br>Text Rotated | Place text with horizontal or angled baselines | Draft |
|  | View | Zoom<br>Pan | Increase or decrease view magnification and scroll the active drawing | Draft, Sculpt |

# DRAWING AND RENDERING MODES

When you launch DENEBACAD or create a new document, a Draft mode window appears. To display other views of a project, or to work with 3D objects and renderings, you can open additional windows in Draft, Sculpt, and Render modes.

**Draft mode** is the 2D drawing mode. A Draft mode window shows a Top, Bottom, Left, Right, Front, or Back view of a 2D document.

**Sculpt mode** is the 3D drawing and modeling mode. A Sculpt mode window shows 3D objects from Top, Bottom, Left, Right, Front, or Back view.

*Note:* You can use display options to view 2D and 3D objects in Draft or Sculpt modes.

**Render mode** is the 3D visualization mode. A Render mode window displays a Wireframe, Hidden Line, or Solid rendering. You also use Render mode to generate animated walk-throughs, QuickTime movies, and QuickTime VR scenes.

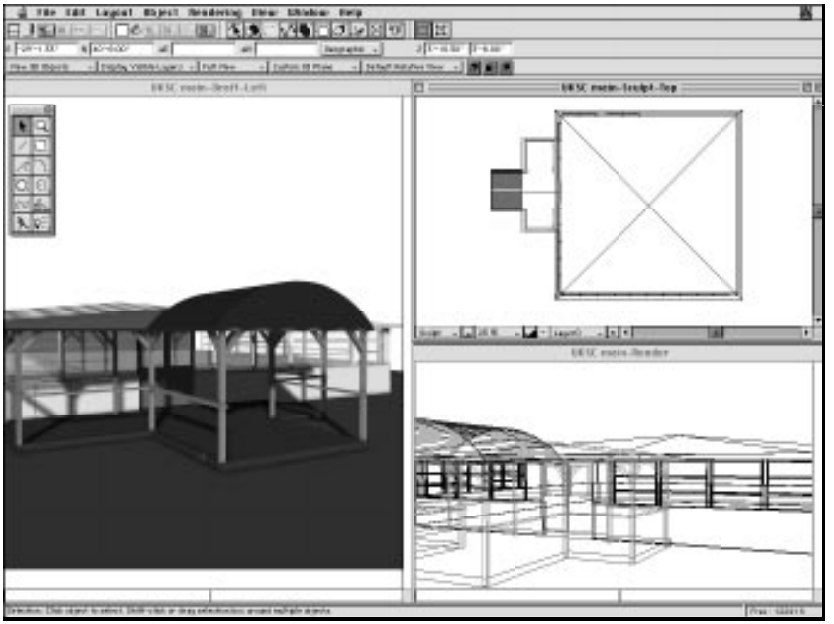
### Displaying multiple modes

To see multiple views of a project, you can open up to three windows and set them to any DENEBACAD view or mode. For example, you can use two Sculpt windows to view 3D objects from Front and Top views while you also view the finished model with Surface materials and lighting in a Render window.

- **To display the Draft mode window:** If the Draft mode window isn't already open, choose Show Draft in the Window menu. If this window is already displayed, the command reads Hide Draft.
- **To display the Sculpt mode window:** In the Window menu, choose Show Sculpt. If this window is already displayed, the command reads Hide Sculpt.
- **To display the Render mode window:** In the Window menu, choose Show Render. If this window is already displayed, the command reads Hide Render.
- **To change the display mode of a window:** Choose the new display mode from the pop-up menu at the bottom-left corner of the active window.


*Fig. 15 Mode pop-up menu*

When you use multiple windows, DENEBACAD arranges the windows so they don't overlap. You can choose the Tile command or press Command+T to arrange windows neatly on screen.



## APPLYING ATTRIBUTES TO OBJECTS

In Draft and Sculpt modes, you can assign a variety of attributes to objects. You can apply Pen and Fill colors, hatch patterns, and Pen styles to most objects. You can apply full-color surface materials to 3D objects for Solid renderings.

The buttons on the Attributes bar let you access palettes containing all Pen and Fill attributes.

When a particular attribute can't be applied to a selected object, or can't be used in the current mode, the attribute control is dimmed on the Attributes bar.

*Note:* The Attributes bar is always displayed when DENEBACAD is running.

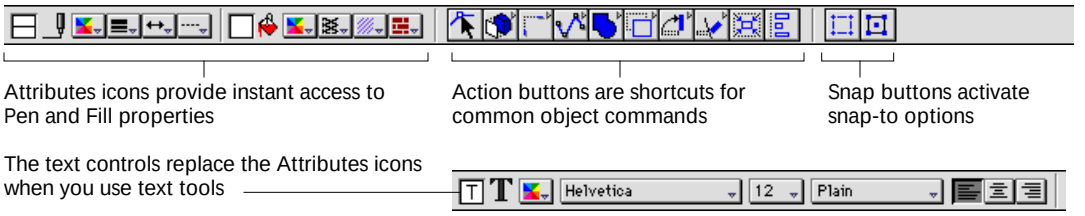


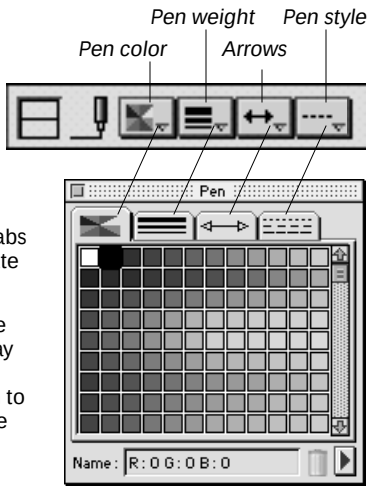

*Fig. 16 Attributes bar and Text toolbar*

## PEN CONTROLS ON THE ATTRIBUTES BAR

The Pen controls let you apply Pen color, Pen weight, Pen style, and Arrows to objects. You can use the buttons to apply attributes, or to open the Pen palette.

You can set Pen attributes before or after you create an object. The current attributes apply to new objects you create. When you change the Pen attributes of a selected object, the current attributes for new objects remain the same.

The Pen attributes conform to the AIA pen color and weight standards. When you render a 3D object with the Hidden Line rendering command, the object's Pen color becomes the 3D object's outline color.




Pen attribute buttons (above) correspond to tabs in the Pen palette (right)

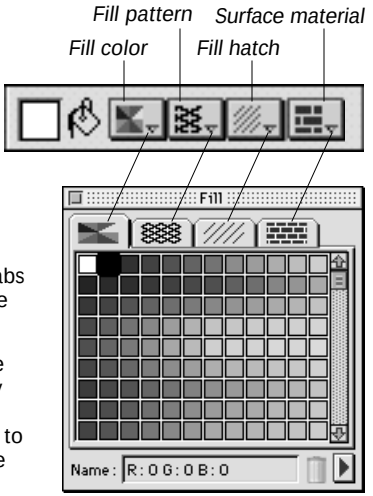
You can tear the Pen palette away from any Pen attribute button to keep the palette open

## FILL CONTROLS ON THE ATTRIBUTES BAR

The Fill controls let you apply Fill colors, Fill patterns, and Fill hatches to 2D objects, and apply Surface materials to 3D objects. You can use the buttons on the Attributes bar to apply attributes, or to open the Fill palette.

Surface materials are used only in Solid renderings. In quick Hidden Line renderings, a 3D object's Pen color becomes its outline color. In Solid renderings, an object's Pen color becomes its surface color if the object does not have a Surface material (or if the Materials option is not selected in the Rendering Options dialog box).

You can set Fill attributes before or after you create an object. The current attributes apply to new objects you create. When you change the Fill attributes of a selected object, the current attributes for new objects remain the same.




Fill attribute buttons (above) correspond to tabs in the Fill palette (right)

You can tear the Fill palette away from any Fill attribute button to keep the palette open

# GETTING STARTED 3

This chapter presents an overview to help you become familiar with the tools and commands you use to create objects in DENEBACAD. It includes an introduction to commands that you can use to set up documents, drawing scale and the drawing grid, and introduces the Snaps menu, a versatile constraint system that helps you control any drawing tool or operation you perform on objects in DENEBACAD.

## SETTING UP THE DRAWING ENVIRONMENT

To start using DENEBACAD, double-click the DENEBACAD application icon.

When DENEBACAD begins running, it opens a single Draft mode window, with *"Untitled-Draft-Top"* displayed in the window's title bar.

### Setup for exploring DENEBACAD

To be sure that DENEBACAD is set up in the best way for exploring the program, the next section describes how to customize the drawing environment, before introducing basics of drawing in DENEBACAD.

### Displaying optional toolbars

By default, DENEBACAD presents an almost empty drawing interface on screen. When you first start the program, the toolbox appears on the left, and the Attributes bar and the Action buttons appear in a row at the top of the screen under the menu bar.

Several optional toolbars are available as well. The *Status bar* provides shortcuts for changing your view of a drawing. The *Info bar* shows drawing data and lets you create objects using the keyboard. The *Help bar* displays a variety of helpful information about DENEBACAD's tools, commands, and procedures.

When you first begin using DENEBACAD, it's a good idea to display the optional toolbars. Once you become familiar with the shortcuts they provide, you can decide whether to continue to display all or some of them.

#### To display optional toolbars

1. Choose Toolbars in the Layout menu. The Toolbars dialog box appears.


2. To display all toolbars, select the four check boxes so that each one has *X*, and then click OK.

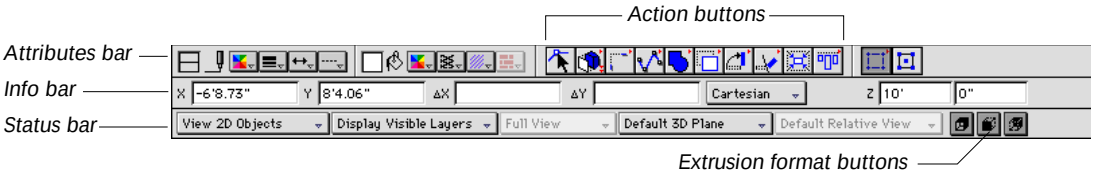



*Fig. 17 The Attributes bar, Action buttons, Info bar, and Status bar*

## SETTING PREFERENCES

To configure DENEBACAD for the introduction to drawing in this chapter, use the Preferences command to set up drawing and display preferences.

The settings recommended here are not necessarily the settings you would prefer. Once you understand the basics, use the Preferences command to customize the work environment to your liking. DENEBACAD applies the Preferences settings to all documents, and you can change these settings at any time.

### To set up drawing preferences

1. Choose Preferences in the Edit menu. The Preferences dialog box opens.
2. Select the check boxes for the options on the General tab to match the illustration (*fig. 18*).
3. Click OK to close the Preferences dialog box.

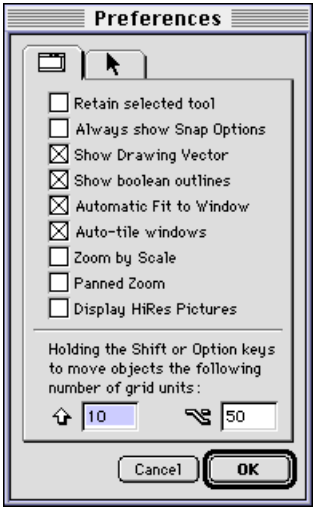



*Fig. 18 The General tab in the Preferences dialog box.*

## DRAWING SCALE AND GRID SETTINGS

Before you begin any new project, you'll want to check the settings for measurement units, drawing scale, and the drawing grid that appears in the Draft and Sculpt drawing windows.

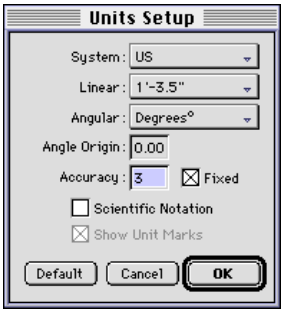
- Once you create a document with the settings you want to use often, you can save the document as a template. Then you can open this document to work with the same settings. Use the Save As command to save it

with a new name when you start a new project.

### To set up measurement units

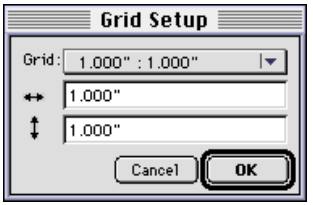
1. Choose Units Setup in the Layout menu. The Units Setup dialog box appears.
2. Set the dialog box to match the following illustration and click OK.

   ▲ To make the new settings the default settings for all new documents, click Default.




### To set up the drawing grid

1. Choose Grid Setup in the Layout menu. The Grid Setup dialog box appears.
2. In the Grid pop-up menu, choose 1" by 1" and click OK to implement the grid settings.




*Note:* The grid setting specified here is appropriate for drawing objects like doors, as described in the following tutorial lesson. For larger-scale drawings, you will want to use a larger grid setting.

## BASICS OF DRAWING OBJECTS

*Objects* are geometric shapes which form the basis of almost all drawings. In DENEBACAD, you can easily draw 2D objects, including lines, polylines, ellipses, arcs, curves, and rectangles. And you can just as easily create 3D objects, including cubes, columns, walls spheres, planes, arches, and more complex shapes.

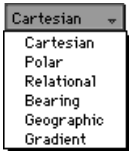
You can use any of the following methods to draw objects with 2D (Draft) and 3D (Sculpt) tools:

**Click or drag** Select a drawing tool, position the pointer in the drawing area, and click or drag the mouse to set the points that define an object.

**Enter coordinates** Select a drawing tool, press Tab, and type coordinates or dimensions for the object in the text boxes in the Info bar. After you type coordinates for the first creation point, press the Enter key and then type relative coordinates to set subsequent creation points.

**Using existing objects** You can also create objects by using other objects. To do this, select an existing object and choose the Duplicate, Mirror, Rotate, Copy, Array, or Combine command to create new objects.

## COORDINATE SYSTEMS




Coordinate Systems menu in the Info bar

You can use several coordinate systems in DENEBACAD.

You select a coordinate system from the pop-up menu in the Info bar. The values that appear in the Info bar when you draw with the mouse reflect the current coordinate system. Also, you work in the current coordinate system if you type coordinates in the text fields in the Info bar to create objects.

You can use the six coordinate systems in DENEBACAD. The coordinate systems are briefly described here. You can learn more about the coordinate systems by choosing each one in the Info bar and looking at how the text boxes in the Info bar change, and the coordinates that appear when move the pointer and draw objects.

### Cartesian

The Cartesian system is based on a grid divided into four quadrants with a horizontal X axis and a vertical Y axis in Top view.

The Centerpoint in the Cartesian system is the intersection of the horizontal and vertical axes at coordinates 0, 0. To specify the location of a point, you can type a pair of distances measured along the X and Y axes in the Info bar. The coordinates can be absolute, expressed as the distances along the X and Y axes from the Centerpoint; or relative, expressed as the change in distance (ΔX, ΔY) from one point to another.

### Polar

When you use the Polar coordinate system, you can set the length of a line segment (labeled ΔD in dialog boxes) and its angle (labeled ΔA) directly. Each quadrant in the coordinate system encompasses 90 degrees.

### Relational

The Relational system expresses all distances as relative to the last creation point or edit point on an object.

### Bearing

In the Bearing coordinate system, a measurement is expressed as a distance and a bearing. Coordinates are expressed in geographic direction — North, South, East, West — to a fixed point or to the Centerpoint of the document.

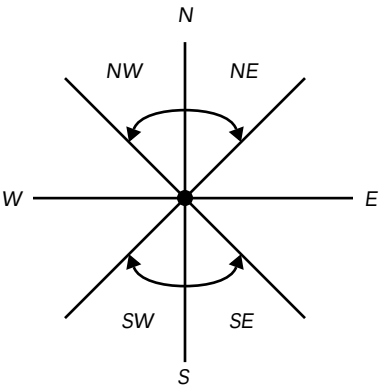



*Fig. 19 Bearing coordinate system*

### Geographic

The Geographic system can also be called the Linear or Reticular coordinate system. The Geographic system is a cartesian system based on a grid divided into four quadrants with a horizontal E (East) axis and a vertical N (North) axis. This system is similar to the horizontal X axis and vertical Y axis of the Cartesian coordinate system. Coordinates are positive or negative values, with positive N and E values for points above and to the right of the Centerpoint, and negative N and

E values for points below and to the left of the Centerpoint (*fig. 20*).

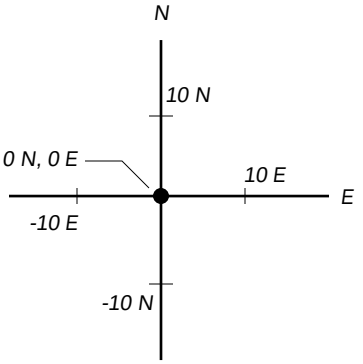



*Fig. 20 Geographic coordinate system*

## Gradient

When you use the Gradient coordinate system, the location of a point on the grid is expressed as a distance from the Centerpoint (or from the last creation point) and an angular measurement. Each quadrant in this system encompasses 100 degrees.

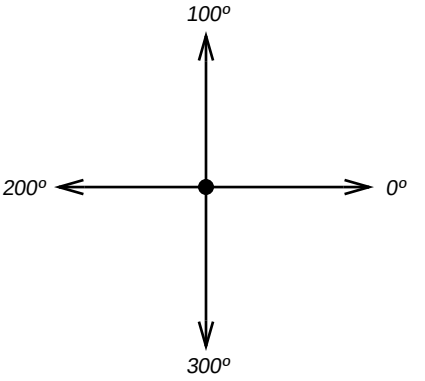



*Fig. 21 Gradient coordinate system*

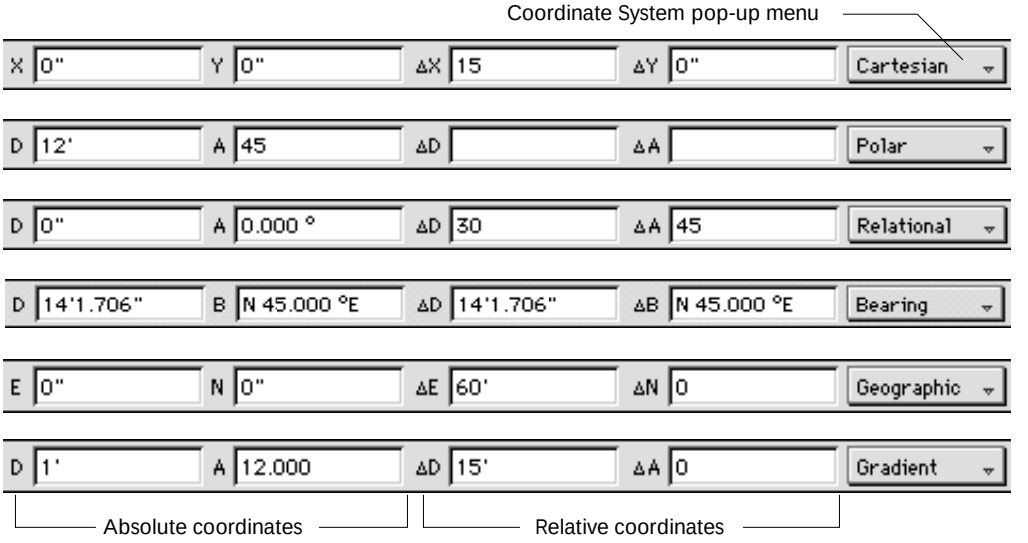



*Fig. 22 Info bar data and coordinate systems*

# DRAWING WITH THE MOUSE

Drawing with the mouse is the most common method for creating objects in DENEBACAD.


To draw objects with the mouse, you have a choice between two methods:

- You can click to place the points that define an object, such as the endpoints of a line segment.
- You can press the mouse button to set the first creation point, and drag to the location of the second and subsequent points.

Both methods work with most drawing tools. The exceptions are tools that define objects with more than two points, such as the Rectangle Rotated and Circle 3 Points tools. With these tools, you drag from the first to the second creation point, and then click to set the third point.

When you draw by dragging the mouse, you usually complete an object in one step. You can do this when you don't need to type numeric values in the Info bar to set creation points.

When you draw objects by clicking the mouse to set the first point, you are able to enter coordinates or distances numerically for subsequent points, as described in the next section.

#### To draw a line by clicking endpoints

1. Select the Line tool from the tool box.
2. To set the first point, click in the drawing window.
3. Move to the position of the second point. The line segment appears to stretch, or "rubber band," as you move the pointer.
4. Click to set the second point and finish the line. The line segment appears with the current Pen color and Pen weight, and is selected.

#### To draw a line by dragging between endpoints

1. Position the pointer where you want the line segment to begin
2. Press and hold the mouse button and drag to the second endpoint of the line. When you release the mouse button, the line segment appears with the current attributes, and the line is selected.

## DRAWING WITH THE INFO BAR

You can draw an object by using the text boxes in the Info bar to specify coordinates for the object's creation points. Drawing an object by typing values in the Info bar can be the best method to use when you want to set an object's location and size numerically.

When you draw using the mouse, notice that the numbers in the Info bar change as you move the pointer.

In the Cartesian coordinate system, for example, the X and Y fields show the pointer's position as X and Y coordinates with the Centerpoint at 0, 0. After you set an object's first creation point, the ΔX and ΔY fields show the pointer's distance from the previous creation point.

When you use a coordinate system that displays distance and angle, you can control the length

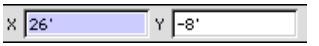
and angle of the drawing vector for any operation.

When you draw an object using the Info bar, you type in the same values that would appear if you were moving the pointer with the mouse and clicking to set object creation points.

### To draw a line by entering coordinates

You can draw a line by setting the coordinates of the first point and specifying the length of the line.

1. Be sure the Cartesian coordinate system is selected in the Info bar, then select the Line tool in the toolbox.
2. Press the Tab key to highlight the X field in the Info bar.
3. Type the X coordinate of the first endpoint. This is the distance horizontally from the Centerpoint to the starting point of the line.
4. Press Tab to highlight the Y field. Type the Y coordinate of the first endpoint. This is the distance vertically from the Centerpoint to the starting point of the line.
5. Press RETURN to set the first endpoint at the X,Y coordinates.
6. Press Tab to highlight the ΔX field in the Info bar, and type the line's length on the X axis (the horizontal distance).
7. Press Tab to highlight the ΔY field, and type the line length on the Y axis (the vertical distance). The line appears with the current attributes and is selected.




X and Y values set the first creation point relative to the Centerpoint.


ΔX and ΔY values set the second creation point relative to the first creation point.

*Fig. 23 Typing creation point coordinates*

### Alternate method for using the Info bar to create an object

1. Select the Line tool in the toolbox.
2. Press the Tab key to highlight the X field in the Info bar.
3. Type the X coordinate of the first endpoint, the distance horizontally from the Centerpoint to the starting point of the line.
4. Press Tab to highlight the Y field. Type the Y coordinate of the first endpoint, the distance vertically from the Centerpoint to the starting point of the line.
5. Press Enter to set the first point at the X, Y coordinates.
6. Notice that the ΔX text box is highlighted. Type the length of the line on the X axis in the text box.
7. Press Tab to highlight the ΔY field, and type the line length on the Y axis (the vertical distance). The line appears with the current attributes and is selected.

### To draw a horizontal line from the Centerpoint

This procedure gives an example of drawing a 50-foot line horizontally from the Centerpoint.

1. Select the Line tool. Press Tab to highlight the X field in the Info bar.

Line tool

2. To set the first endpoint at the Centerpoint, type 0 in the X field.
3. Press Tab, type 0 in the Y field, and then press Enter.
4. To complete a 50-foot horizontal line, type 50' in the ΔX box, press Tab, type 0 in the ΔY box, and then press Return.

## MODIFYING PROPERTIES OF OBJECTS

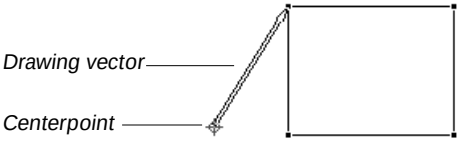
You can use the Properties Manager to view and edit the properties of an object. The Properties Manager is a floating palette, so you can keep it open while you work.

◆ **To open the Properties Manager:** Choose Properties Manager in the Window menu, or double-click an object you want to edit.

The Properties Manager shows position data and other information for a selected object. When you create an object, it is selected and its data appears in the Properties Manager. To work with a different object, click the object you want to select with the Selection tool.

The Properties Manager contains four tabs. When it first appears, the Coordinates tab is in front. The Coordinates tab displays information and position data for a selected object.

- The top of the tab displays and lets you edit an object's name.
- The pop-up menus under the name let you choose relative or absolute position data, and the coordinate system to be used.
- Under the pop-up menus, text boxes show data on the object's creation points. If the Show Vector option is selected, the drawing vector appears in the drawing. When you use absolute position data, DENEBACAD displays the vector that defines the point's position from the Centerpoint.




The drawing vector appears when you select an object and select Show Vector in the Properties Manager

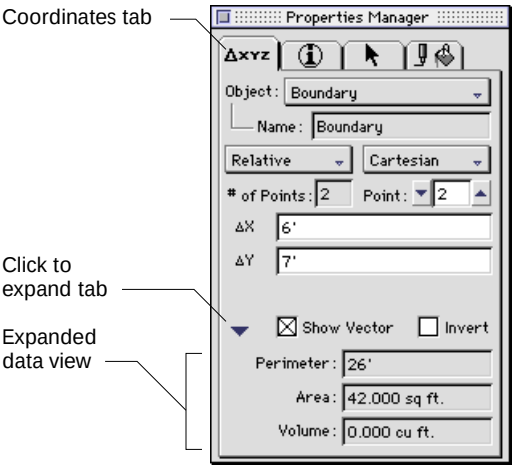



*Fig. 24 The Coordinates tab in the Properties Manager*

**To edit a line in the Properties Manager**

This procedure is an example of using the Properties Manager to change an object's size or coordinates.

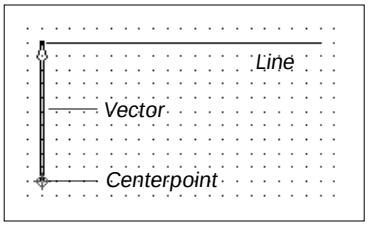
1. With a line selected, choose the Properties Manager command in the Window menu. The Properties Manager appears.
2. Click the Coordinates tab to bring it to the front, if necessary.


Coordinates tab

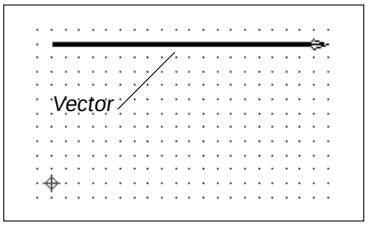
3. Choose Relative and Polar from the pop-up menus under the Name box. This sets the coordinate and measurement systems.
4. The values under the pop-up menus are the length (ΔD) and angle (ΔA) of the line. Type a new length in the ΔD box and press Tab to change the line length. Type a new angle in the ΔA box and press Tab to change the line's angle.
5. To view area, perimeter and volume data for a selected object, click the arrow at the lower-left to expand the Properties Manager.
6. When you finish editing an object, click the Close box if you want to close the Properties Manager.

When you select point 1, the Properties Manager reports the absolute distance from the Centerpoint to the object, shown by the drawing vector.

When you select point 2 of a line, Relative position data and the drawing vector show the length from the first to the second creation points, which is the length of the line.




*Point 1 selected*



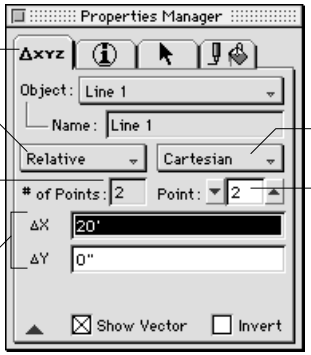

*Point 2 selected*

With a line selected, the Coordinates tab in the Properties Manager displays data about the line's creation points

Choose Relative or Absolute position data

Total number of creation points

Data displayed conforms to the selected coordinate system




Choose a coordinate system

Specify a creation point to edit

Vector data shows the location of the selected point

*Fig. 25 Editing position data in the Properties Manager*

# SNAPS MENU CONSTRAINTS

As shown in the next section (see “Drawing with snap constraints” on page 33), the Snaps menu provides constraints that let you precisely align objects by controlling the pointer and the drawing vector, without having to specify angles or distances numerically.

## USING THE SNAPS MENU

The Snaps menu pops up in the drawing area when you press the Control key and the mouse button at the same time.

In most cases, you indicate which object should be the reference for the constraint by pointing to an object when you press Control. In descriptions of the constraints in this section, the object you point to when you open the Snaps menu is called the *reference* object.

You can also make the Snaps menu appear whenever you press the mouse button to draw an object (without pressing the Control key). To do this, select the check box labeled “Always Show Snap Options” on the General tab in the Preferences dialog box.

You can combine constraints and use different constraints to set each creation point when you draw objects. Also, snap constraints can be used for all DENEBACAD operations, not just object creation.

Constraints can be applied to the positioning of planes when you define extrusion planes, clipping planes, and Relative Views. For example, when you use the Define Frontal 3D Plane command, you can choose Included in the Snaps menu to align the plane with the slope of a roof gable.

The Snaps menu lets you constrain the drawing vector and pointer in many ways. Listed below are all the constraints in the Snaps menu. If a constraint has a keyboard shortcut, it appears after the constraint name.

### Free Mouse

This option frees the pointer and drawing vector from any constraint that is in effect. In other words, Free Mouse is not a constraint itself; it restores the pointer to free movement.

### Click

Choosing Click is a substitute for clicking the mouse during a procedure.

### Snap Points (Spacebar)

This constraint displays the Snap points of the reference object. When an object’s snap points are displayed, the vector snaps to the closest point.

### Included (I)

The Included constraint confines the pointer or vector to the path of the object at the pointer; if you move the pointer beyond the object, the constraint continues as if the object or segment extended through the entire drawing.

### Parallel (P)

Choosing Parallel constrains the vector to a path parallel to the object at the pointer. If you choose Parallel before setting an object’s first creation point, a line appears at the pointer, parallel to the object you indicated. Move the pointer and then click to establish the offset from the object. An Offset dialog box appears so you can confirm the

offset, or type a new value in the text box. Click OK to set the offset. A small cross appears where you click, and the pointer is constrained to the parallel line at the specified offset distance.


### Perpendicular (N)

This constraint snaps the pointer or vector to a path perpendicular to the object you indicate (the object at the pointer) when you choose the constraint.

### Tangent (T)

Tangent constrains the vector to be tangent to a circle or circular arc. After setting the first endpoint of a line, choose Tangent and the vector snaps to the perimeter of the indicated circle. This constraint is not available unless the pointer is on a circle or circular arc.

### Direction (D)

The Direction constraint snaps the vector to the angular direction that you establish before choosing the constraint. For example, as you move the pointer away after setting the first creation point of a polyline, you establish a direction. If you choose Direction or press D, the vector locks to the established direction.

### Center (C)

Snaps the pointer and vector to the center of the reference object or segment.

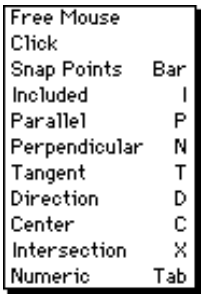
### Intersection (X)

Aligns the pointer and vector to the intersection of two separate objects, such as the point at which two lines meet or cross.

### Numeric (Tab)

Choosing Numeric is the same as pressing Tab during an operation: it lets you enter position data in the Info bar during an operation. For example, you might click to set the first creation point of an object, and then type numeric values to set the length and direction of the vector to complete an object.

When you open the Snaps menu, not all Constraints are available at all times. The available constraints depend on the position of the pointer and the type of operation in progress.




## DRAWING WITH SNAP CONSTRAINTS

DENEBACAD provides powerful and flexible "snap-to" drawing aids. In addition to the snap-to-grid and snap-to-object features, DENEBACAD offers a unique Snaps constraint system that you can use to control the drawing vector in all operations.

You apply the Snaps constraints using the Snaps pop-up menu that appears at the pointer location in a drawing, or by pressing a shortcut key (*fig. 26*).

The Snaps menu lets you constrain the drawing vector and pointer in many ways. You can draw parallel to an object, tangent to an object, aligned to an object's center, sides or snap points, perpendicular to an object, at a constant angle, and aligned to an object's snap points.

You can combine constraints and use different constraints to set each creation point when you draw objects. In this way, the Snaps menu gives you precise control over any drawing situation.

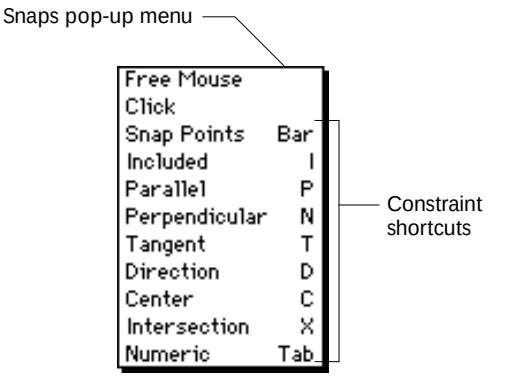



*Fig. 26 The Snaps menu*

### To use the Snaps menu

1. Select the drawing tool you want to use.
2. Hold down the Control key and press the pointer on an object that you want to snap to. The Snaps menu appears in the drawing area.
3. Drag into the Snaps pop-up menu to choose a snap constraint.

   ▲ The constraints that are available in the Snaps menu depend on what you click in the drawing.

   ▲ Choose Free Mouse to release a constraint that is in effect.
4. Click or drag the mouse to draw an object. The constraint you selected controls the drawing vector and pointer.

### To snap endpoints of lines

This procedure gives an example of snapping the creation point of a new object to the endpoint of an existing line.

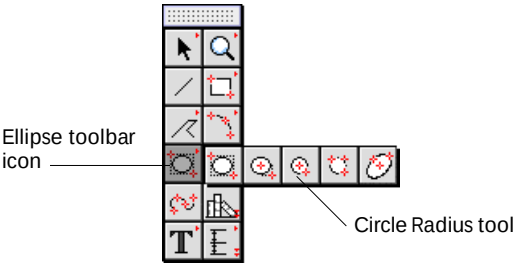
1. Select the Line tool and draw a line. Click away from the line to deselect it.
2. Select the Line tool again. Point to the first line, press the Control key and hold down the mouse button at the same time. The Snaps menu appears.
3. Drag into the Snaps menu and choose Snap Points. The line's snap points appear as small handles (*fig. 27*).
4. Click the snap point at one end of the line. The first point of the new line snaps to the point you clicked.
5. Move the mouse and click anywhere to set the second endpoint of the new line.



*Fig. 27 Snap points on a line*

**To draw using two Snaps constraints**

1. Select the Circle Radius tool. If the Circle Radius tool icon isn't visible in the toolbox, press the Ellipse toolbar icon to open the pop-out toolbar, and then drag to the Circle Radius tool.




2. Point to the intersection of the two lines you drew before, press Control, and choose Intersection in the Snaps pop-up menu.

   ▲ If Intersection is dimmed so you can't select it, be sure you point to the exact intersection of the two lines.

3. Press the pointer on one of the line segments and press the Control key. The Snaps menu appears.

4. In the Snaps menu, choose Center. The edge of the circle snaps to the center of the line segment. This completes the circle and leaves it selected.

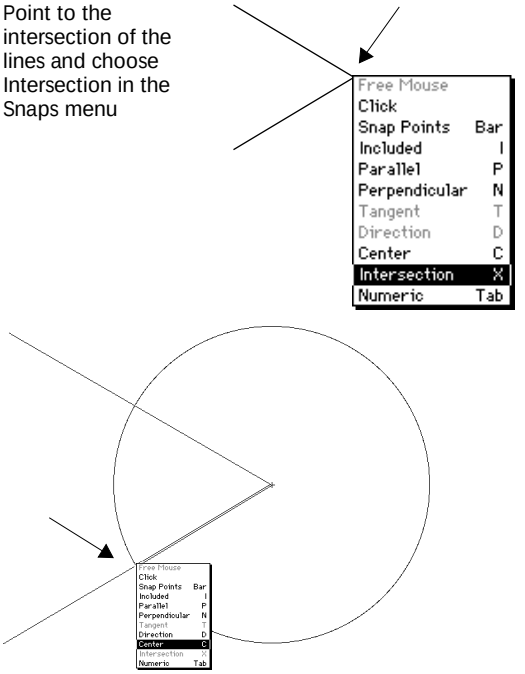


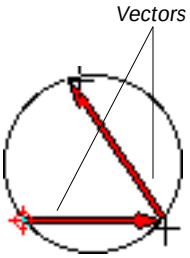

*Fig. 28 Snapping a circle to two lines*

## THE DRAWING VECTOR AND SNAPS

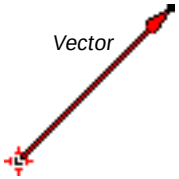
The drawing vector is a gray arrow that appears when you create objects and perform most operations, including Rotate, Mirror, Array, Define 3D Plane, and Define Clipping Plane.

The drawing vector indicates the distance from one point to another by its length. It also indicates angle or bearing. Because the Info bar and the Properties Manager display the drawing vector's length and angle, and because you can control the vector with the Snaps menu constraints, the drawing vector is one of the most powerful drawing aids in DENEBACAD.

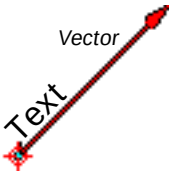
The illustrations on this page show the drawing vector as it appears when you create objects and perform other operations.



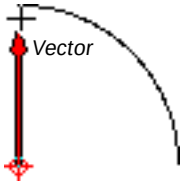
Circle 3 Points



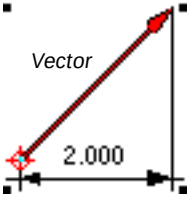
Line



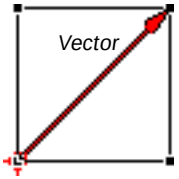
Text



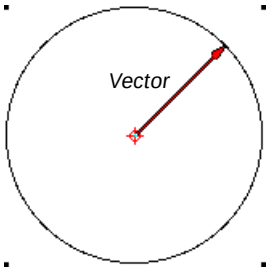
Radius Arc



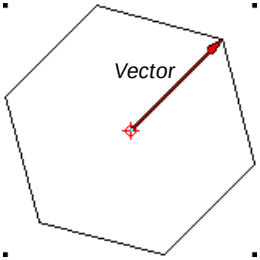
Dimension



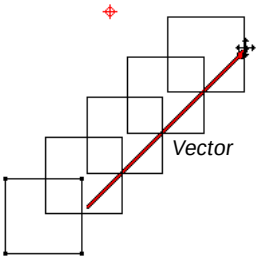
Rectangle



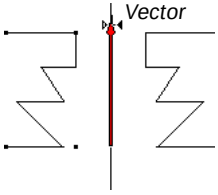
Circle



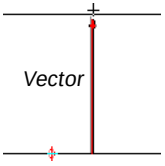
Polygon



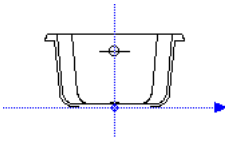
Linear Array



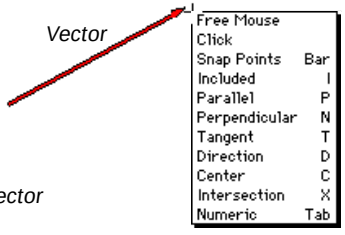
Mirror



Setting extrusion planes



Library



Drawing vector and Snaps menu

*Fig. 29 The drawing vector in common operations*

## CLEARING THE DRAWING WINDOW

To make sure you have enough room to continue drawing while you follow the examples in this chapter, you might want to clear all objects from the drawing window from time to time. To do this, Press Command+A to select all objects in the drawing. Then press the Delete or Backspace key to remove all selected objects.

## MAKING COPIES OF OBJECTS

Another method you can use to create objects is making copies of existing objects. Several commands let you duplicate an object and also rotate, flip, or reposition the copy at the same time.

**To duplicate a rectangle**

1. Select the Rectangle Diagonal tool in the toolbox and drag the pointer to draw a rectangle.

Rectangle Diagonal tool

2. With the rectangle selected (selection handles should be visible), select the Duplicate command in the Edit menu.
3. A copy of the original rectangle appears in front of the original. Point to one side of the rectangle and drag it away. You can see that the original rectangle remains in position behind the copy.

### DUPLICATING OBJECTS BY PASTING

You can cut and then paste a selection in DENEBACAD the same as you would in any Mac OS application. In addition, DENEBACAD's Paste Special command lets you paste a selection in a variety of ways.

**To position a pasted object with Paste Special**

1. Draw or select an object in your drawing.
2. Choose Copy in the Edit menu. This places a copy of the object on the Clipboard.
3. Choose Paste Special in the Edit menu. The Paste Special dialog box appears. You can choose from three options for pasting:

▲ **Absolute** places the copy in front of the original, at the same absolute coordinates.

▲ **Relative** places the copy relative to the current view, at the center of the screen.

▲ **Cursor Position** lets you click to place the copy.

4. Select an option and click OK to paste the object. If you chose Cursor Position, click in the drawing to set the upper-left corner of the object's bounding box.

## COPYING AND MIRRORING OBJECTS

Mirroring an object lets you create a flipped copy or a mirror image of a selected object.

### To copy an object by mirroring

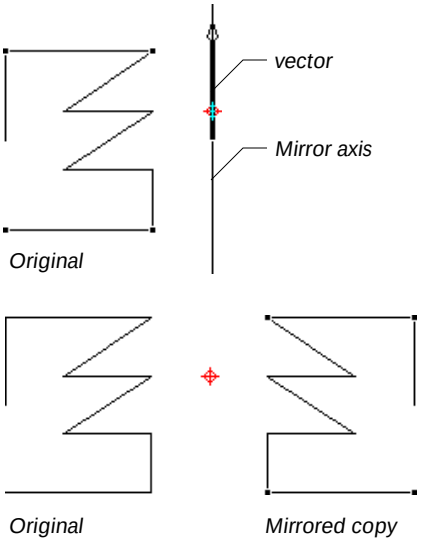
1. Select the Single Polyline tool from the Polyline tool bar.

Single Polyline tool

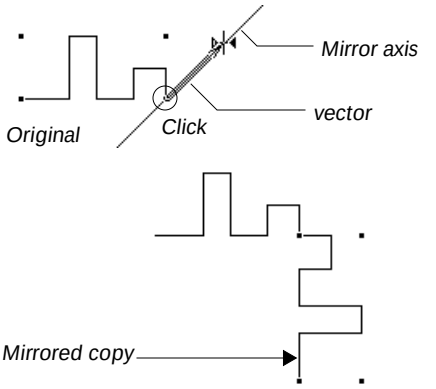
2. Click at several points in the drawing, and then double-click the last point to draw an irregular shape. The new polyline is selected.
3. Choose Mirror a Copy in the Position submenu in the Object menu. A vertical line, representing the mirror axis, appears at the pointer.
4. Move the pointer to position the axis horizontally, then click to set the horizontal position of the axis over which the object will be mirrored, or flipped.

   ▲ The copy will appear twice the distance from the axis to the original.
5. After you click, the drawing vector appears. Move the pointer to angle the axis away from vertical, if desired. The drawing vector indicates the angle and distance from the center of rotation, which is at the point you clicked in step 4.

   ▲ To snap the axis to 15° increments, press the Shift key as you position it.
6. Click to set the angle of the axis. A mirror copy of the original object appears and is selected.




Setting the mirror axis vertically (as above) results in a copy aligned horizontally with the original




Setting the mirror axis at a 45-degree angle results in a copy rotated 90 degrees from the original

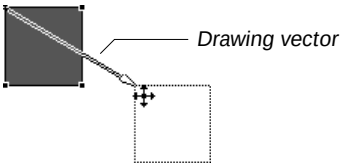
*Fig. 30 Using the Mirror a Copy command*

## COPYING AND MOVING OBJECTS

The Move a Copy command provides another method for you to copy an object and set the position of the copy in relation to the original.

### To copy and move an object

1. Draw a rectangle or select an existing object.
2. Choose Move a Copy in the Position submenu in the Object menu.
3. Click to set the reference point for moving the object.
4. When you move the pointer, the drawing vector shows the relative movement from the reference point, and an outline of the object shows where the copy will appear.
5. Click to set the position of the object copy.




The drawing vector shows the relative movement of the reference point

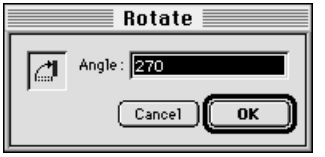
*Fig. 31 Using the Move a Copy command*

## COPYING AND ROTATING OBJECTS

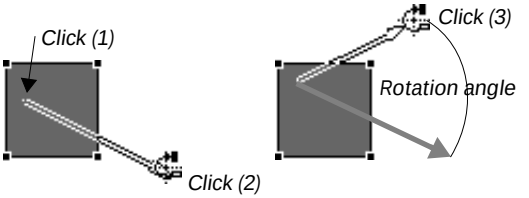
Rotating a copy is another method you can use to create an object by duplicating an existing one.

### To copy and rotate an object

1. Draw an object or select an existing one in your document.
2. Choose Rotate a Copy in the Position submenu in the Object menu.
3. Click in the drawing to enter the base point of rotation for the object.
4. When you move the pointer, the drawing vector shows the position of the pointer relative to the rotation point you clicked. Click in the drawing again to set the reference point and click a third time to input the rotation angle of the object.
5. A dialog box appears so you can confirm the rotation angle you have indicated.




▲ You can type a new angle in the text box, or click OK to use the existing value. When you click OK, the rotated duplicate object appears.




Click to set the center of rotation, then click to set the first vector and the second vector. The angle between the vectors is the angle of rotation

*Fig. 32 Using the Rotate a Copy command*

## DUPLICATING OBJECTS IN ARRAYS

You can duplicate and create new objects using the Linear Array and Polar Array commands. These commands let you create one or many copies and position the copies a set distance from one another.

With the Linear Array command, in Draft mode, you can specify the ΔX, ΔY, distances, as well as having the option of copying the objects by offset distance or inclusively.

In Sculpt mode, you can specify the ΔX, ΔY, and Δ Z distances, as well as having the option of copying the objects by offset distance or inclusively.

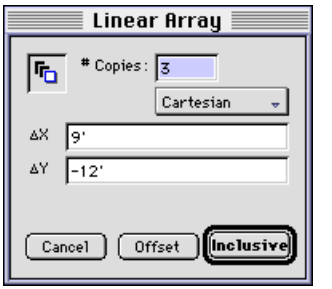
- The Offset Array option will create the number of copies you specify and place each copy a specified distance from the last one.
- The Inclusive Array option will place the specified number of copies within a specified distance.

**To use the Linear Array command**

1. Select an object you want to duplicate.
2. Choose Linear Array in the Position submenu in the Object menu.
3. The pointer becomes a four-way arrow. Click to set the reference point and move to set the array distance and angle, as shown by the drawing vector. You can also drag from one point to another.
4. The Linear Array dialog box appears. Type the number of copies to create in the "# Copies" text box.

   ▲ You can type new coordinate values in the text boxes and select a coordinate system from the pop-up menu.
5. Click a button in the dialog box to choose the Array method.

   ▲ Click offset to use the drawing vector as an offset distance and direction. Each copy appears the same distance away from the previous copy.
6. Click Inclusive to distribute the copies within the distance and direction specified by the drawing vector.



The drawing vector indicates the distance and angle for duplicating objects with the Array command

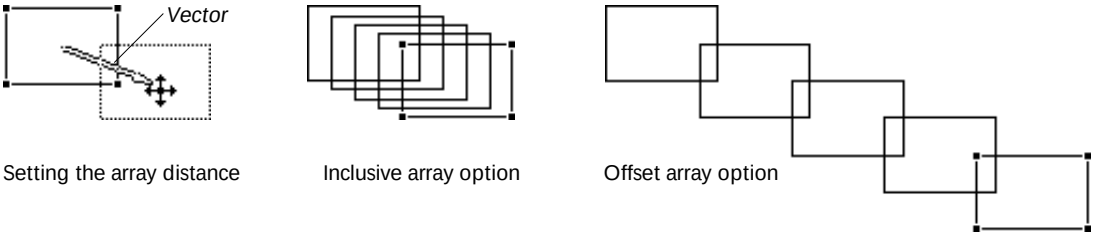



*Fig. 33 Using the Linear Array command*

# CREATING OBJECTS WITH BOOLEAN OPERATIONS

Boolean operations let you create complex shapes by combining multiple objects. You can choose from six combine operations: Unite, Intersect, Punch from Back, Punch from Back and Trim, Punch from Front, and Punch from Front and Trim.

## UNITING OBJECTS

The Unite command unifies selected objects into one new object.

### To unite objects

1. Draw several objects, such as circles, squares, polygons, or any other shapes that you like.
2. Make sure that the objects all overlap each other. If necessary, drag the objects to reposition them.
3. To apply a fill color to one or more of the objects, select the object. Press the Fill Color icon in the Attributes bar and drag to select the color to apply.

Fill color icon

*Note:* You can apply Fill colors to 2D objects only.

4. Press Command+A to select all the objects.
5. Choose Unite in the Combine submenu in the Object menu. A new object made from the union of the original objects appears. The new object takes on the attributes of the original backmost object.

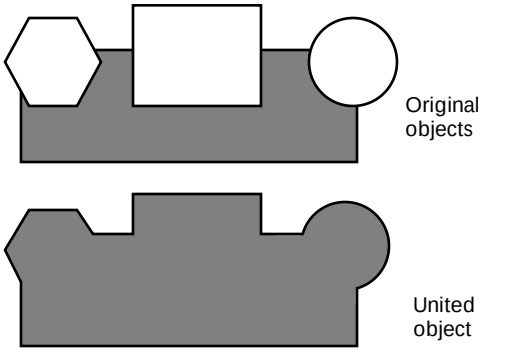



*Fig. 34 Objects combined with Unite*

## RESHAPING OBJECTS

After you create a Boolean object using a Combine submenu command, you can reshape the object by editing its original components. When you finishing editing the shapes and repositioning the component objects, DENEBACAD reapplies the Combine command that you had used to create the Boolean object.

**To reshape a Boolean object**

1. Select the Boolean object.
2. Choose Reshape in the Edit menu, or click the Reshape action button on the Attributes bar. DENEBACAD displays the original objects.

Reshape button

▲ Because the Preference option "Show Boolean outlines" is selected, the original objects have a black outline, and a red outline appears around the Boolean object.

3. Drag one of the component objects to a new position.

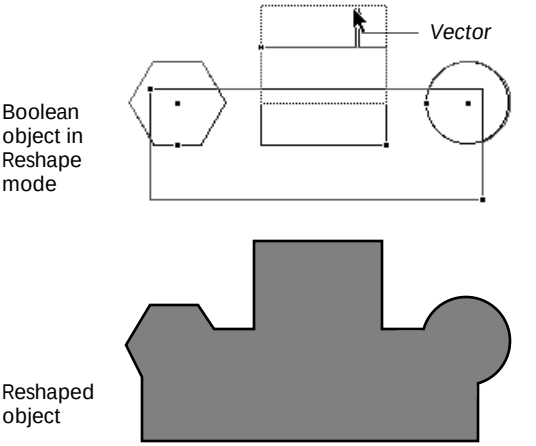



*Fig. 35 Reshaping a Boolean object*

4. Click outside of the objects. The Boolean object is reshaped based on the change to its component objects.

# ZOOMING IN AND OUT OF DRAWINGS

You can adjust the view of a drawing by zooming in to specific areas, and zooming out for an overview of the whole drawing.

Zoom tool

Use the Zoom tool to drag a box around an area, and DENEBACAD magnifies the area until it fills the drawing window. To zoom out, press the Option key and drag in the drawing. You can use the Zoom tool in Draft and Sculpt windows.

You can also change magnification with the Zoom bar at the bottom of the active window. Press the center of the bar to use the Zoom pop-up menu. You can quickly zoom by clicking the Zoom In and Zoom Out buttons. In the Zoom pop-up menu, you can choose Fit to Window, Home View, and a number of preset zoom scales.

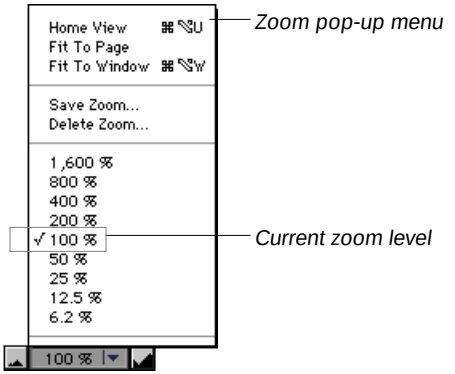



*Fig. 36 The Zoom bar*

# 2D AND 3D LIBRARY OBJECTS 4

Libraries offer a way of creating, organizing, and using items such as doors and windows, wiring and plumbing symbols, and roof and wall sections in your work. This chapter explores procedures for creating 2D and 3D Library Objects.

### Library Object types

There are two types of items you can store in the Library Palette: Symbols and Objects. You can create Library items in 2D, 3D, and combined 2D/3D.

The items described in this chapter are 2D and 3D Library Objects. As part of creating Library items, you'll learn how to use some tools not described in preceding sections.

To follow the exercises in this chapter, open the file named "Library.start" located in the Tutorial folder. This file has all the preferences set up correctly for the procedures described in this chapter. When you open the file, you should see a blank document in Draft mode.

## DRAWING A 2D DOOR

To create a combination 2D/3D Library Object, you usually start by creating the 2D object.

#### To draw a 2D frame

1. Select the Rectangle Diagonal tool.
2. Click to set one corner of the door rectangle. Press Tab and Type 3' for ΔX, press Tab again and type 6" for ΔY.
3. Press the Return key to set the points.

#### To draw 2D door jambs

1. Select the Rectangle Diagonal tool and snap to the outside corner point of the rectangle and draw a 2" wide by 6" tall door jamb.
2. To apply a Fill color to the door jamb, press the Fill Color icon in the Attributes bar and drag to select the Fill color you want to apply.
3. Make a mirror copy of the door jamb so there is one on the other side of the door. With the door jamb you just drew still selected, choose Mirror a Copy in the Position submenu in the Object menu. Snap to the center of the outlined rectangle to set the insertion point. Set the mirror axis so it is vertical using the Shift key to restrain its position.
4. Click the mouse when you have the axis set correctly. A mirror copy of the door jamb appears.

#### To draw a 2D door

1. Select the Rectangle Diagonal tool.
2. Snap to the mid point of the door jamb and click the mouse to set the insertion point.
3. Drag until you have a rectangle that is 1" thick (shown in the ΔX box) by 3' tall (shown as ΔY).

4. You can assign a Fill color to the door the same as you did with the door jambs. For now, assign the door the same Fill color as the door jambs.

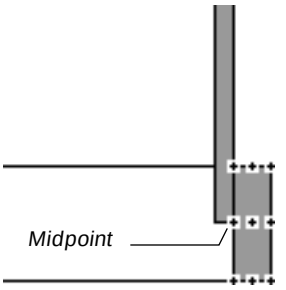



*Fig. 37 Drawing a door*

Now that you have drawn the door frame, jamb and door, you can draw the door swing.

### To draw a 2D door swing

1. Select the Arc Radius tool from the Arcs toolbar.

Arc Radius tool

2. Snap to the midpoint of the right side door jamb and click the mouse to set the center point of the arc.
3. Snap to the corner point of the door to set the first endpoint of the arc.
4. Set the second endpoint of the arc at the midpoint of the left side door jamb. You now have the door swing.

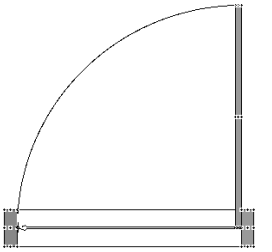


*Fig. 38 Drawing a door swing*

The last object to draw for the door is a rectangle that will be used as a Boolean object.

### To draw a 2D door's Boolean object

1. Select the Rectangle Diagonal tool from the Rectangles toolbar.
2. Snap a rectangle from the top-left edge of the left door jamb to the bottom-right edge of the right door jamb.

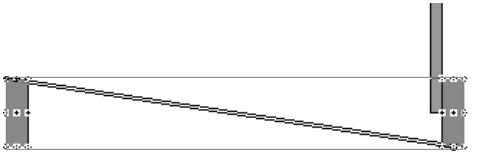


*Fig. 39 Drawing a door's Boolean object*

## MAKING A 2D DOOR A LIBRARY OBJECT

When you finish drawing a 2D door, you can convert it into a Library Object by using the Properties Manager.

### To create a Library Object

1. Select all the objects drawn for the door.
2. Choose Group in the Object menu. Handles appear around the door.

3. Choose Properties Manager in the Window menu to open the Properties Manager.
4. Click the Object Info tab to bring it to the front, if necessary. This tab lets you view information about the new group.
5. Type *3' Door* in the Name box to give the door a descriptive name, and then press Enter.
6. To convert the door to a Library Object, check the Library Type check box, and then click the Object radio button.
7. Check the Boolean check box. This makes the front object in the group, the rectangle drawn last, a subtractive shape. The rectangle will cut the opening needed when you place the door in a wall.

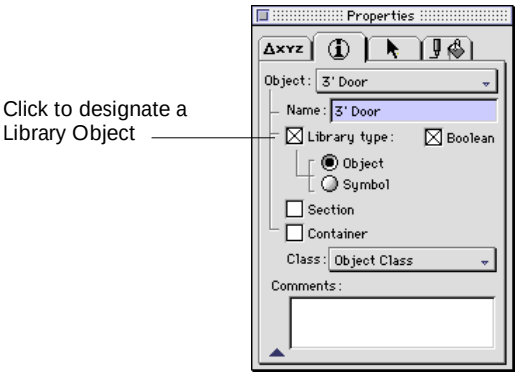



*Fig. 40 Designating a Library Object in the Properties Manager*

## Setting the insertion point

An insertion point is the point at which a Library Object snaps to objects in a drawing. When a Library Object is first defined, DENEBACAD places the insertion point at the center of the Library Object.

In the case of a door, an insertion point at the center of the door object is not useful. You can move the insertion point to the midpoint of the door jamb. This will center the door in any wall where you place it.

### To position a door's insertion point

1. Select the door Library Object.
2. Choose Reshape in the Edit menu, or click the Reshape Action button on the Attributes bar. The highlighted insertion point appears at the center of the Library Object.
3. Drag the insertion point to its new location at the midpoint of the outer edge of the left door jamb.

You have now completed a 2D door Library Object.

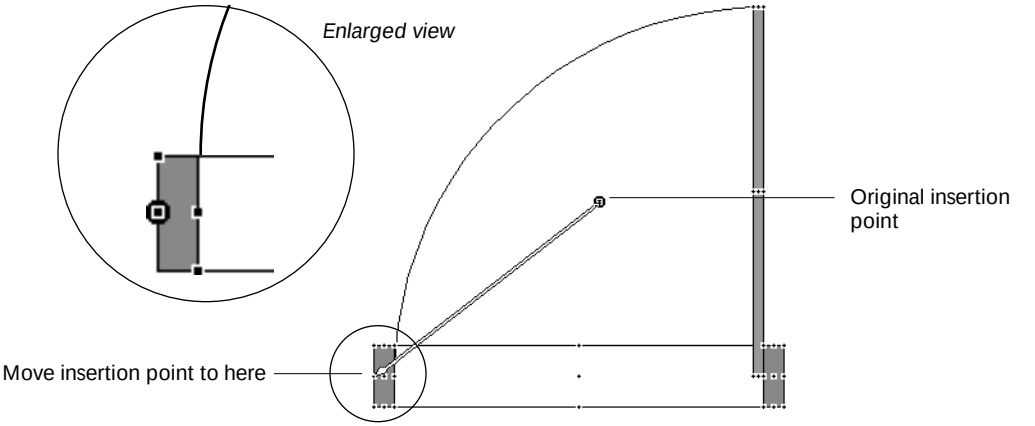



*Fig. 41 Setting the insertion point for a door Library Object*

# DRAWING AN ELEVATION OF A DOOR

In this part of the lesson, you're going to draw an elevation for the 3-foot-wide door you created.

### Using multiple windows

First, open a new drawing window in Sculpt mode.

#### To open a new window

1. Choose Show Sculpt in the Window menu. A new window in Sculpt mode set to Top View appears. DENEBACAD arranges, or "tiles" the windows so you can see both the Draft mode window and the Sculpt mode window.
2. Click the Draft mode window to make it active.
3. Choose Fit to Window in the Zoom pop-up menu, or press Command+Shift+W, so you can see the entire drawing.

#### To move the Centerpoint

Next we're going to move the drawing's Centerpoint (origin) to a new location.

1. Choose Set Position in the Centerpoint submenu.
2. Move the pointer to the left door jamb and snap to the lower-left corner. Click to set the new Centerpoint position.

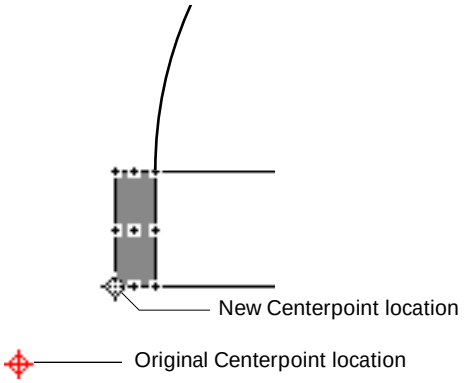



*Fig. 42 Changing the Centerpoint location*

### To change views

1. Choose Front in the View menu. The window displays the DENEBACAD workspace from the Front. The document window is empty because you haven't created any objects in Front view.
2. Make sure the Centerpoint is visible. If necessary, zoom in or out, or scroll until the Centerpoint is visible in the lower-left corner of the window.

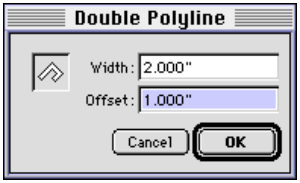
## Drawing the door's elevation

Let's now start drawing the elevation of the door To do this, you'll need to configure the Double Polyline tool.

### To configure the Double Polyline tool

1. Double-click the Polyline tool icon to configure its settings. The tool's dialog box appears.

Double Polyline tool




2. Type 2" in the Width box and 1" in the Offset box.
3. Click OK. This will allow you to draw the door frame at the same thickness as the door jambs in Top view.

### To draw an elevation of the door

1. Click 2 inches away in the X direction from the Centerpoint and 0 inches in the Z direction.
2. Move the pointer up 6'8" in the ΔZ direction and click to set the second point.
3. Move the pointer 3' in the ΔX direction and click to set the third point and complete the top of the door frame.
4. Move the pointer -6'8" in the ΔZ direction and double-click to complete the door frame.

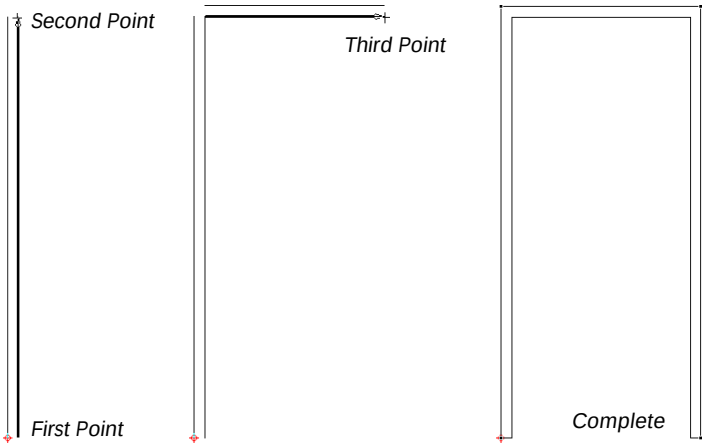



*Fig. 43 Drawing a door elevation*

### To draw the door

1. Select the Rectangle Diagonal tool.
2. Snap to the inner corner of the top-right corner of the door frame. Drag to create a rectangle that is -1" in the ΔX direction and -6'7" in the ΔZ direction.

# WORKING WITH 2D & 3D OBJECTS

We have now constructed the elements we need to build a 3D door. The next step is to extrude these 2D objects into 3D objects. To extrude a 2D object into 3D, you first define a set of extrusion planes.

## Using 2D and 3D display modes

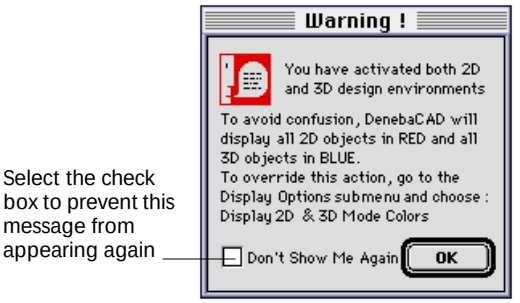
First, let's change the display mode of the Draft window in the Status Bar to "View 2D & Lock 3D Objects." This mode lets you see 3D information in the window set to Draft mode.

### To change the display mode

1. In the Display Mode pop-up menu at the left end of the Status bar, choose View 2D & Lock 3D Objects.


*Fig. 44 Using the View Options pop-up menu*

2. When you change the display mode, a message box appears to inform you that all 2D objects will now be displayed in red and all 3D objects will now be displayed in blue. Click OK to close the message window.




## DEFINING EXTRUSION PLANES

Now, you are ready to begin defining the 3D extrusion planes and extruding the objects that make up the 3D door.

### To define extrusion planes

1. Choose Define Vertical 3D Plane in the 3D Plane submenu in the Layout menu. Your view of the active window changes to Top View. The 2D door you drew should now be visible. A line extends from the pointer. The line is the first extrusion plane.
2. Move the extrusion plane so it aligns horizontally with the bottom face of the door, and then click. Move the mouse up; the drawing vector appears, extending from the line you just defined.
3. Align the line pointer with the top face of the door and click. The view of the window changes back to the original view, which is Front view.

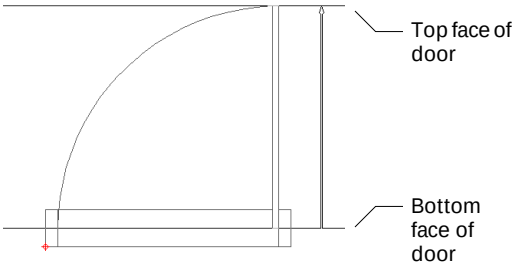



*Fig. 45 Setting extrusion planes*

### To extrude the door into 3D

You have just defined the depth of the door. The next step is to extrude the door.

1. Click the door to select it.
2. Choose Extrude Linear in the Extrude submenu in the Object menu. A blue rectangle appears in the Sculpt window. This means that a 3D object has been created from the 2D object.

Now you will repeat the operation, but this time we will set the parameters for the extrusion of the door jamb.

### To extrude the door jamb

1. Switch to Front view, and then choose Define Vertical 3D Plane in the 3D Plane submenu. DENEBACAD switches the document window to Top view.
2. Move the first extrusion plane, align it with the bottom face of the door jamb, and then click.
3. Move the second extrusion plane, align it with the top face of the door jamb, and then click.
4. Select the door jamb object.

5. Choose Extrude Linear in the Extrude sub-menu in the Object menu. A blue rectangle — the 3D doorjamb — appears in the Sculpt window. If you can't see the door jamb, choose Fit to Window in the pop-up menu in the Zoom bar.

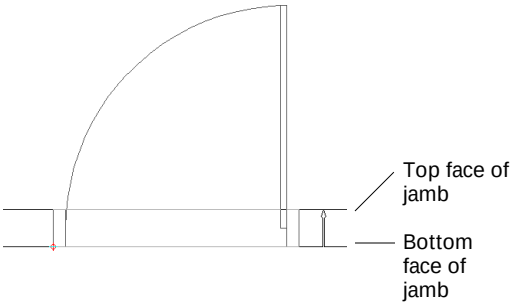



*Fig. 46 Setting extrusion planes for the door jamb*

## Creating a Boolean outline

The last object you will need to draw for the 3D door is a Boolean outline object. Just as we did for the 2D door, we need to create a rectangle that will cut an opening into the object that the Library Object it is placed into.

### To create the door Boolean object

1. Switch to Sculpt mode.
2. Choose Front in the View menu.
3. Select the Rectangle Diagonal tool.
4. Snap to the bottom-left corner of the door frame and drag up to the upper-right corner of the door frame and snap to the corner point. A blue rectangle now surrounds the perimeter of the 3D door. Because you drew the rectangle in Sculpt mode, it is extruded a based on the active set of extrusion planes. In this case, the active extrusion planes are equal to the depth of the door frame.

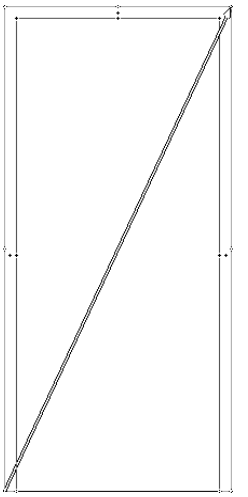
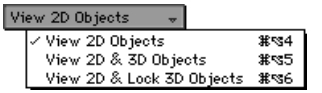


*Fig. 47 Drawing the Boolean object for a 3D door*

5. Before continuing, change the display mode in the Status Bar to View 2D Objects for the Draft mode window and View 3D Objects for the Sculpt mode window.




## Converting the drawing to a Library Object

Now that we have finished drawing a 3D door, we will convert it into a Library Object and link it to the existing 2D door Library Object.

1. Press Command+1 to change the Sculpt mode window to Top view.
2. Press Command+A to select all of the door objects.
3. Press Command+G, or choose Group in the Object menu, to group the door objects.

4. Choose Properties Manager in the Windows menu (if necessary) to open the Properties Manager.

5. On the Object Info tab, type the name for the door in the Name text box. Type *3' Door*, the same name that you used for the 2D door. By naming the two library groups the same, you link them together as one Library Object with both 2D and 3D characteristics.

6. To convert the group into a Library object, check the Library type check box and select the Object radio button.

7. To give the door a Boolean feature, check the Boolean check box.

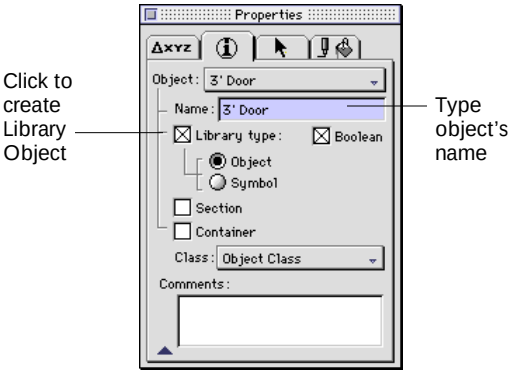



*Fig. 48 Designating a 3D door Library Object*

As before, the topmost object in the group, the rectangle drawn last, will be a subtractive shape. The rectangle will cut through the walls to which you add the door.

### To set the insertion point for the 3D door

The last step for creating the 3D library door is to set the insertion point. This point aligns the door with a wall opening.

1. In the Sculpt mode window, select the door Library Object.

2. Choose Reshape in the Edit menu, or click the Reshape button on the Attributes bar. The insertion point appears highlighted at the center of the Library Object.

3. Drag the insertion point to the midpoint of the left side of the left door jamb.

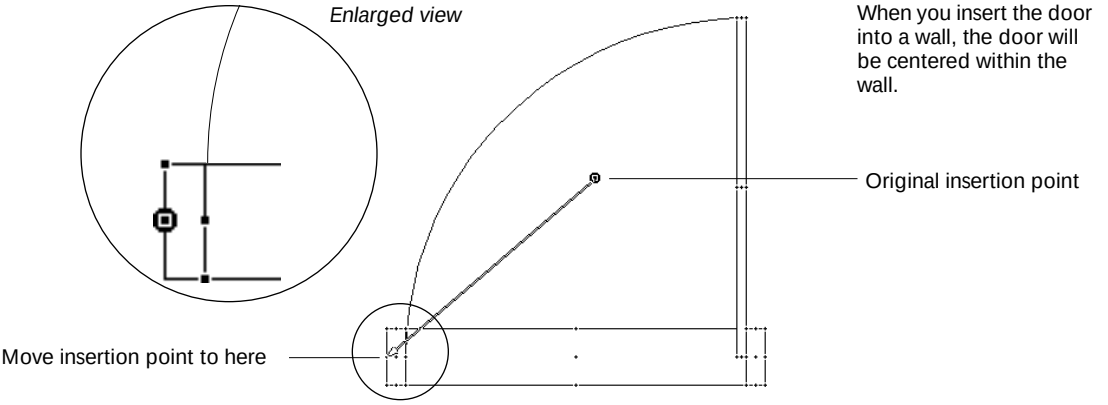



*Fig. 49 Setting the insertion point for the 3D Library Object*

## Saving a Library Object

You should now save the Library Object you have created so you can insert it into your next document.

### To save the document containing the door Library Object

1. Choose Save As in the File menu. In the directory dialog box, select the Libraries folder in the DENEBACAD folder.
2. Type “Doors” in the Save as text box. Any new doors you want to save can be created in this document following the same procedures.
3. Click Save to save the document.

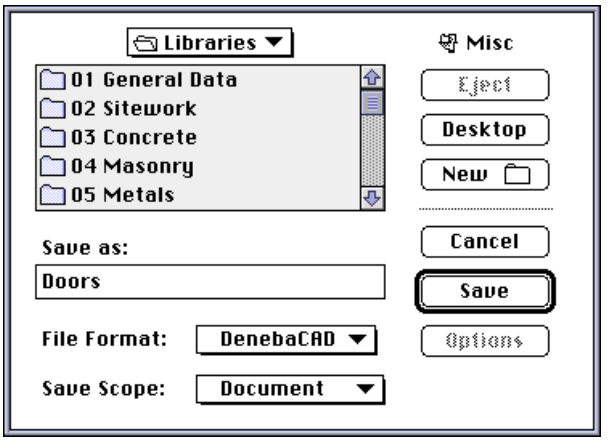



*Fig. 50 Saving the door Library Object*

This chapter demonstrates how you can place Library Objects, like the door completed in the previous chapter, into a new document. This technique lets you use Library Objects in as many documents as you like.

To follow the tutorial lesson in this chapter, open the file named "Library.inserts" located in the Tutorial folder in the DENEBACAD folder.

**To open the library document**

1. Launch DENEBACAD if it isn't running.
2. Press Command+O or choose Open in the File menu.
3. In the directory dialog box, open the Tutorial folder inside the DENEBACAD folder.
4. Select the file named "Library.inserts" and click Open. A blank document appears in a Draft mode window.

## DRAWING A STRUCTURE IN 2D

This section describes how to construct a simple building in DENEBACAD. The procedure uses a Library Object to quickly place a door in the drawing in 2D and 3D.

To draw a basic structure, you can draw simple shapes, and then combine the shapes into a more complex wall outline.

To see dimensions when you draw objects, be sure that the Info bar is displayed. If the Info bar is not visible, choose the Toolbars command in the Layout menu. In the Toolbars dialog box, select the Info Bar check box, and click OK.

**To draw shapes for the building's shell**

1. Select the Rectangle Center to Corner tool in the toolbox.


Rectangle Center to Corner tool


Circle Radius tool

2. Draw a rectangle measuring 19 feet by 14 feet. In the Info bar, 19' appears in the ΔX text box and 14' appears in the ΔY text box when you drag the pointer.
3. Select the Circle Radius tool.
4. Draw a circle, setting the center of the circle at the lower-right corner of the rectangle, and extending the circle until its edge is at the midpoint of the right side of the rectangle.

   ▲ You can use the Snaps menu to snap the circle's center and edge to the exact points on the rectangle.

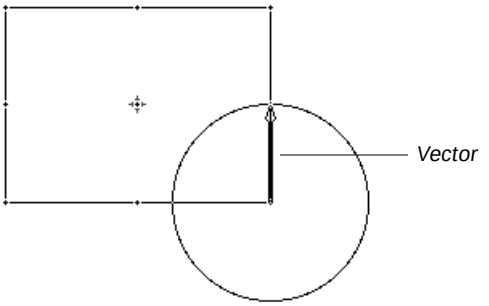



*Fig. 51 Aligning a circle to a rectangle*

**To combine objects to form the shell**

1. Select both objects.
2. Choose Unite in the Combine submenu in the Object menu. Or, click the Unite action button. DENEBACAD combines the selected objects into one outline.

Unite action button

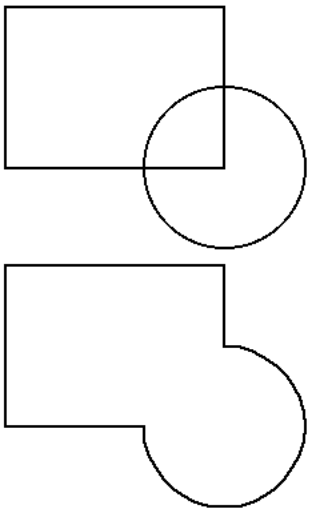
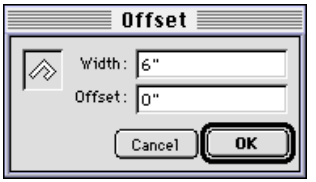



*Fig. 52 Combining object with the Unite command*

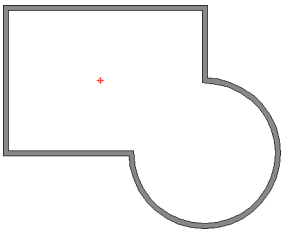
**To set wall thickness using Offset**

1. Choose Offset in the Path submenu in the Object menu. Or, click the Offset action button. The Offset dialog box appears.

Offset action button




2. Type 6" in the Width box and click OK.



Object offset 6"

3. To apply a Fill color to the new walls, press the Fill Color icon in the Attributes bar. Drag to select a Fill color to apply.

## LOADING LIBRARY OBJECTS

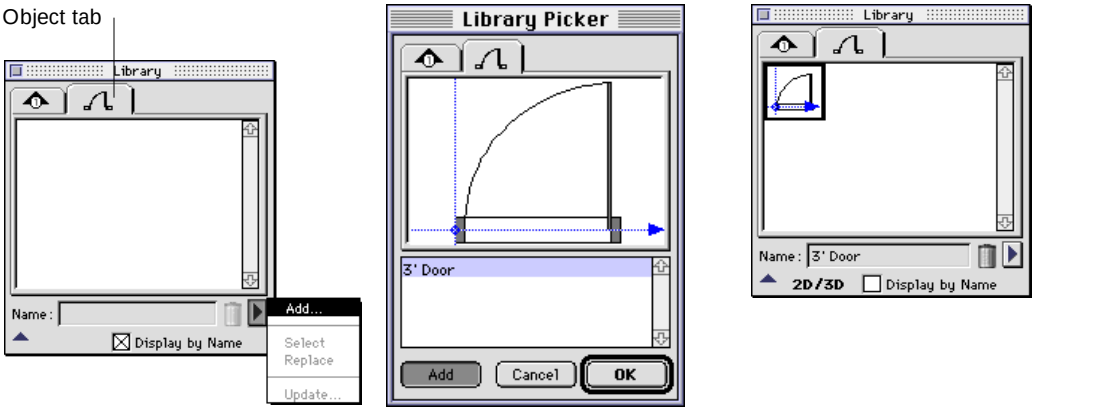
After completing the structure's footprint by building the exterior walls, you can now load the Library Object that will be used to add doors to the building. The procedure used to create the Library Objects is described in the previous chapter.

### To open a document and select a Library Object

1. Double-click the Library tool in the toolbox. The Library palette appears.

Library tool

2. Click the Object tab to bring it to the front, if necessary.
3. Choose the Add command in the pop-up menu in the Library palette.
4. In the directory dialog box, open the folder where you stored the “Doors” file you created earlier. If you didn’t create this file, open the Tutorial folder inside the DENEBACAD folder. Select the file named “Doors” and click Open. The Library Picker appears. The Library Picker displays and lists all Library Objects contained in the DENEBACAD document that you selected.
5. Select the object named “3' Door” and click Add. This adds the selected Library Object to the current document.
6. Click OK to close the Library Picker.




Choose the Add command in the Library palette pop-up menu.

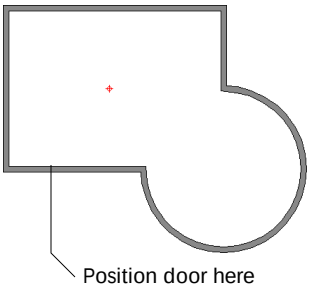
The Add button in the Library Picker adds the selected item.

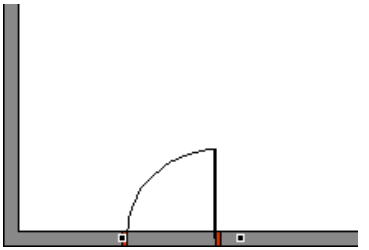
The selected Library Object appears in the Library palette.

*Fig. 53 Using the Library palette and Library Picker dialog box*

### To place the door object from the Library

1. The Library palette displays a thumbnail sketch of the door object. Click “3’ Door” to select it.
2. Move the pointer into the drawing. A grayed image of the door’s bounding box follows the pointer. Point to the exterior wall at the bottom of the drawing and click at the center of the wall. A drawing vector appears, showing the orientation of the Library Object.
3. Press Shift to constrain the pointer movement and drag to the right in a straight line. This places the door in the wall and sets its orientation.



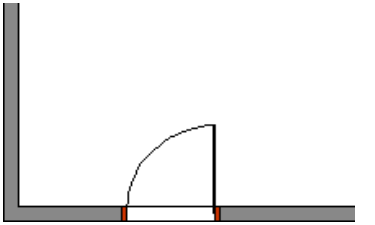
Closeup of door in wall

*Fig. 54 Positioning the door*

4. Press Shift and click the wall with the Selection tool to select both objects.

Unite action button

5. Choose Unite in the Combine submenu in the Object menu, or click the Unite action button. The door makes a cutout in the wall. Also, only one set of selection handles appears around the entire object, because the door and the wall have become one Boolean object.



Cutout of door in wall

## EXTRUDING FROM 2D INTO 3D

When you have a structure in Draft mode, you create a 2D building plan. You can then set up extrusion planes and extrude the 2D objects into a 3D model.

A linked 2D/3D Library Object, like the door added to the drawing in the previous section, will appear correctly in the 3D model when you extrude the wall containing the door.

**To define extrusion planes**

1. Choose Define Frontal 3D Plane in the 3D Plane submenu in the Layout menu. The drawing window changes to Front view.
2. A line extending from the pointer indicates the first plane that will limit extrusions. Align the line with the Centerpoint of the drawing and click to set the extrusion plane.
3. Move the pointer 10 feet up on the Z axis (until 10' appears in the ΔZ box in the Info bar). Click to set the second extrusion limit for a 10-foot-high wall. The drawing window changes back to Top view.

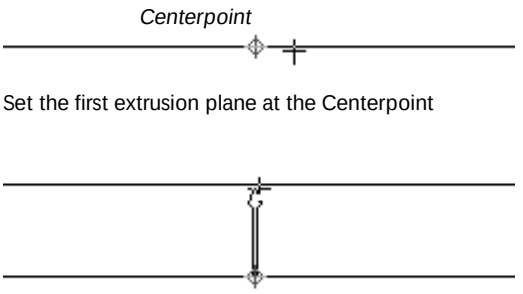



*Fig. 55 Setting extrusion planes*

**To extrude the walls**

1. Select the building in the drawing window.
2. Choose Extrude Linear in the Extrude submenu in the Object menu. DENEBACAD extrudes the selected objects according to the extrusion planes set up in the previous procedure. This creates a 3D structure from the 2D objects, and the original 2D objects remain in the Draft mode drawing window.

## DISPLAYING SCULPT AND RENDER MODES

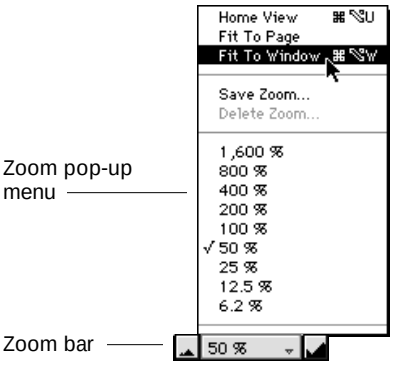
You can open additional windows when you want to view a design from more than one point of view or when you want to work in more than one mode. Objects that you draw in Draft mode and then extrude appear when you open a window set to Sculpt mode or Render mode.

**To open Sculpt and Render mode windows**

1. Choose Show Sculpt in the Window menu. A new window set to Sculpt mode appears.
2. Choose Show Render in the Window menu. A new window in Render mode appears.
3. Choose Tile in the Window menu. DENEBACAD arranges the windows so all three are visible, with the Draft mode window along the left of the screen, the Sculpt mode window at the upper right, and the Render mode window at the lower right.

   ▲ This step is not necessary when the preference "Auto-tile windows" is selected.

4. Click in the Draft mode window to make it active. Choose Fit to Window in the pop-up menu in the Zoom bar.




5. Click in the Sculpt mode window. Choose Front in the View menu. Press Command+Option+W to set the zoom so you can see all of the drawing. The window shows the wall elevation, including the door library object.

# RENDERING A 3D SCENE

You can set the focal point, or "camera" position, to view a rendered drawing in the Render mode window. When you change the focal point, DENEBACAD renders the scene from the new point of view.

### To set the horizontal focal point

1. Click in the Render window. The toolbox changes to display four tools that are used in Render mode.


Render mode tools

2. Select the Horizontal Focal Point tool. The view in the Render window changes to show the drawing from Top view in Sculpt mode.

Horizontal Focal Point tool

3. Adjust the zoom if necessary so you can see the entire drawing in the window. You might want to zoom out slightly so you can see some of the area outside the building.
4. A symbol with an extended red line appears in the drawing (*figure 56*). The symbol is the horizontal *focal point* from which an observer views the rendered scene. The red line is the direction of view. Click in the drawing to set a new focal point.
5. Two lines appear angled outward from the point you clicked. The area inside these lines is the field of view. The drawing vector between the lines indicates the angle of view. Click to set the view angle. The window displays a rendered scene based on the new horizontal focal point.

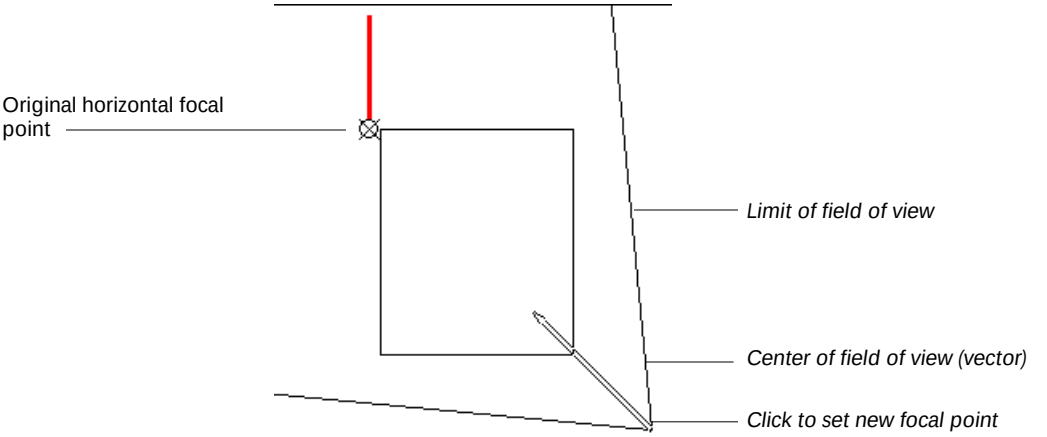
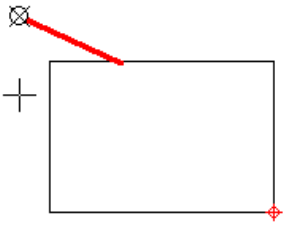



*Fig. 56 Setting the focal point*

**To set the vertical focal point**

1. Select the Vertical Focal Point tool. The Render mode window changes to show the building elevation from Front view in Sculpt mode. A symbol with a red line appears in the drawing. The symbol represents the height of an observer; the line shows the direction of view.
2. Move the pointer to adjust the vertical view position. As the pointer moves, a large cross shows the new vertical position. The cross moves only up and down in the drawing. Click to set the new vertical position.
3. Two lines appear angled outward from the new vertical focal point to indicate the vertical field of view. A drawing vector extends from the new position and indicates the vertical angle of view. Move the pointer to adjust the view angle, and then click to set the new vertical focal point. The window displays a rendered scene based on the new vertical focal point.



Move the pointer to set the height of the view with the Vertical Focal Point tool.

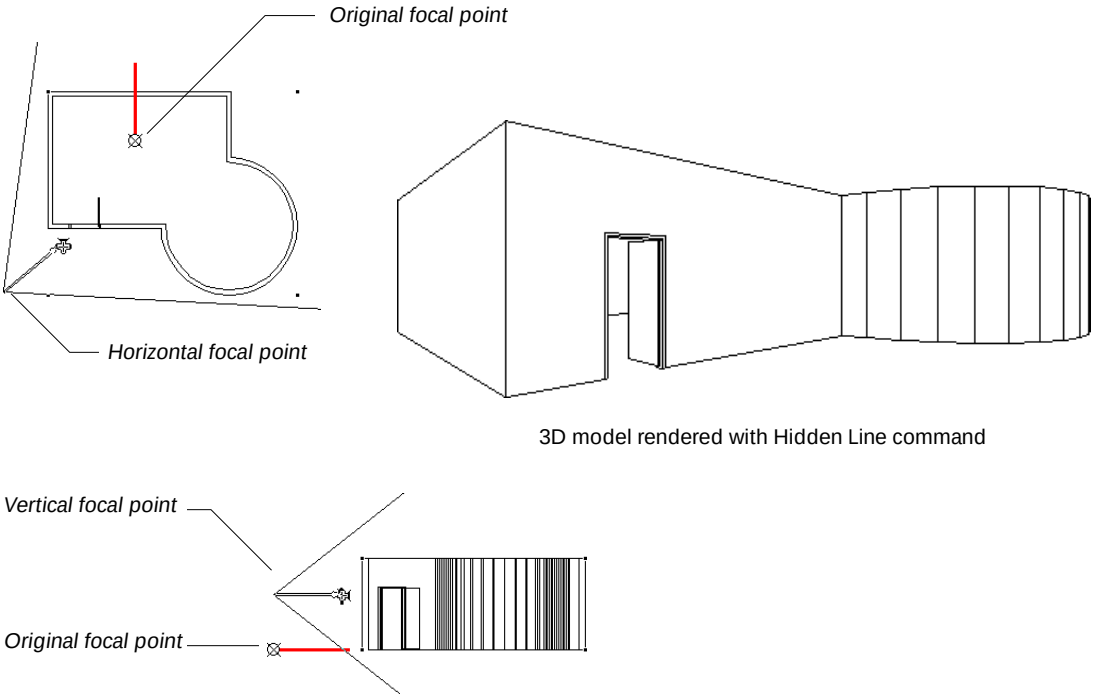



*Fig. 57 Focal points and Hidden Line rendering*

## HIDDEN LINE RENDERING

A final step in the visualization process is to view a Hidden Line rendering of a project you design.

**To view a Hidden Line rendering**

1. Make sure the Render mode window is the active window.
2. Choose Hidden Line in the Rendering menu. DENEBACAD performs the calculation to remove hidden lines, which can require several seconds, depending on the complexity of the drawing, and then displays the rendered scene.

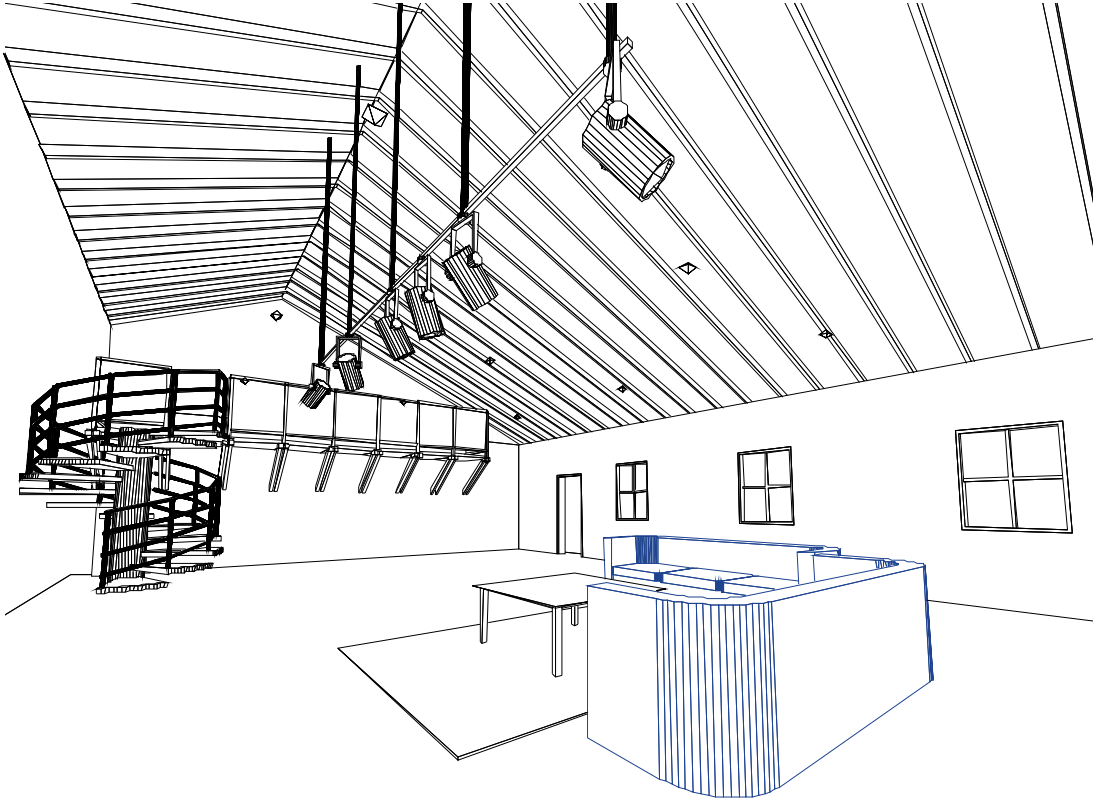


*Fig. 58 A Hidden Line rendering*

This chapter shows you how you can create a hip roof using the Sweep extrusion method.

The tutorial shows you how to convert a shape into a sweep section, and then how to use the Sweep command to extrude the sweep section along a path.

### Opening the tutorial document

To follow this tutorial, open the file named "Roof.start" located in the Tutorial folder in the DENEBACAD folder.

#### To open the document

1. Launch DENEBACAD if it isn't running.
2. Press Command+O or choose Open in the File menu.
3. In the directory dialog box, select the file "Roof.start" in the Tutorial folder and click Open. DENEBACAD opens the file.
4. Choose Toolbars in the Layout menu, select all the options, and then click OK in the Toolbars dialog box.

## CREATING A HIP ROOF USING A SWEEP EXTRUSION

In this section, you'll learn how to create a sweep section. In the Sweep extrusion method, you use one or more objects as a sweep section, which is a cross-section of the extruded object.

### CREATING A SWEEP SECTION

You can make a sweep section from any objects, including a single object or many objects that have been unified with the Group command. In the tutorial document, the sweep section object has been created. The following procedure shows you how to designate the existing object as a sweep section.

#### To create a sweep section

1. After opening the file "Roof.start," you'll see a Draft mode window in Top view with four objects: three rectangles, and a smaller object to the left of the rectangles. To make the smaller object into a sweep section, you use the Properties Manager.

   ▲ If you do not see all of the objects when you open the file, you can choose Fit to Window in the Zoom bar or press Command+Option+W.

2. Select the Zoom tool in the toolbox and drag a box around the smaller object to the left of the rectangles. DENEBACAD zooms in on the area. This gives you a close-up of the object to be used as a sweep section.

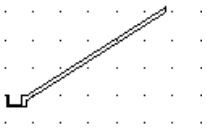


*Fig. 59 Sweep section close-up*

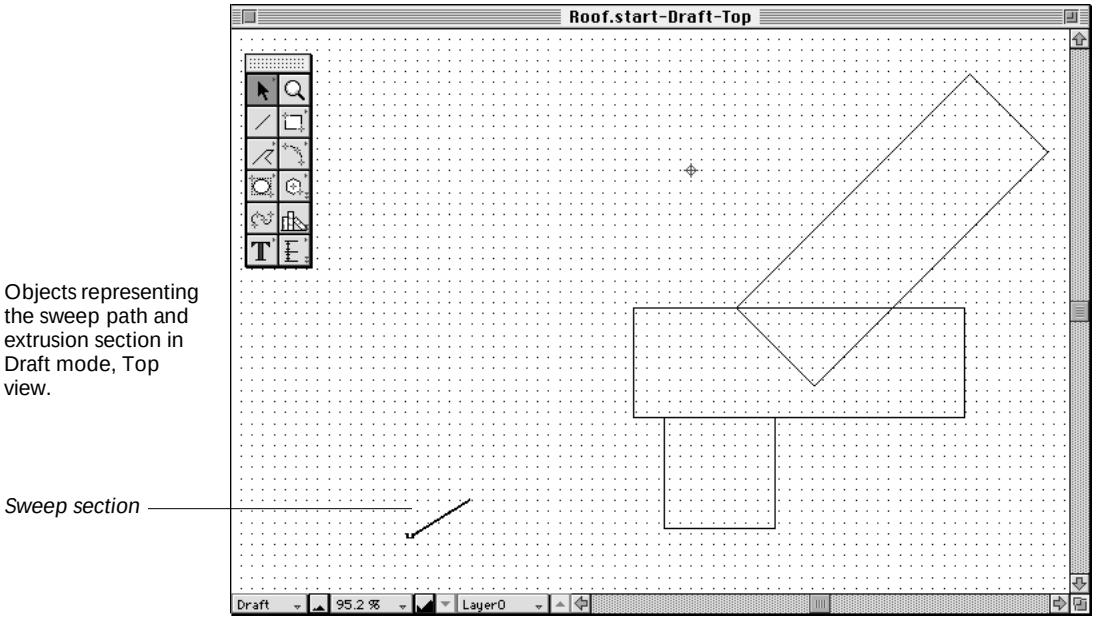



*Fig. 60 Roof.start tutorial*

3. Double-click the object. This opens the Properties Manager and selects the object. If necessary, move the Properties Manager so you can see the entire object.
4. In the Properties Manager, click the Info tab to bring it to the front.
5. To designate the object as a sweep section, click the Section checkbox. The sweep section will appear in the Sweep Sections palette.
6. In the Name text box, type “Hip roof.” When you name a sweep section, try to indicate its purpose. After typing the name, press Enter. The name you assigned to the section appears in the Object pop-up menu on the Info tab, because the object is selected.
7. To close the Properties Manager, click its close box at the upper-left corner in the title bar.

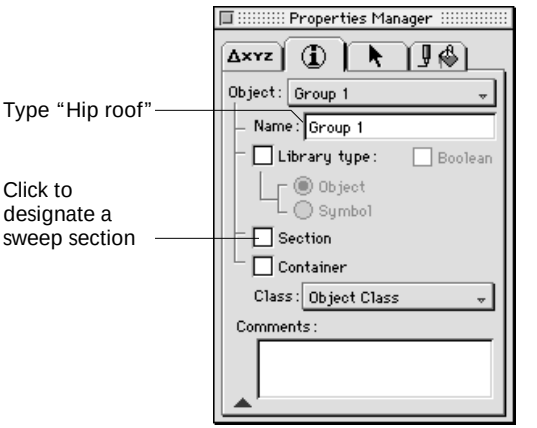



*Fig. 61 Info tab*

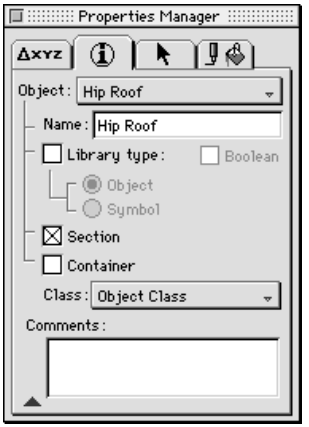



*Fig. 62 Sweep section settings*

By clicking the Section checkbox, you've made the object a sweep section.

## Setting the insertion point for a sweep section

Before you use a sweep section, you might need to adjust its insertion point. This is important because it affects the orientation of the sweep section along the extrusion path.

### To adjust the sweep section's insertion point

1. Click the Reshape action button on the Attributes bar. You can do this because the section remains selected when you close the Properties Manager. DENEBACAD puts the sweep section into Reshape mode. It also displays the object's creation points, and shows the sweep insertion point. The sweep insertion point appears as a black dot in the center of the object's sloping line.
2. Move the tip of the pointer until it's on the object. Press the Spacebar to activate the object's snap points. The makes the drawing vector snap to the end points of segments of the object.
3. Drag the insertion point to the point where the slope of the hip roof section meets the gutter (*figure 64*). The drawing vector snaps to this point as you drag toward it.

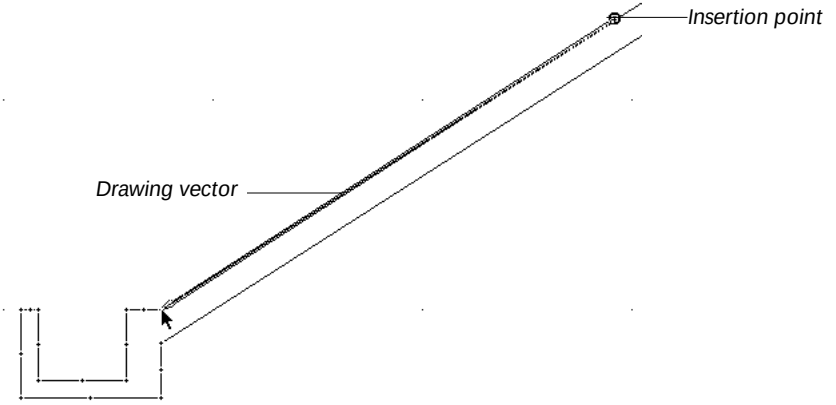



*Fig. 63 Dragging the sweep insertion point with snap points active*

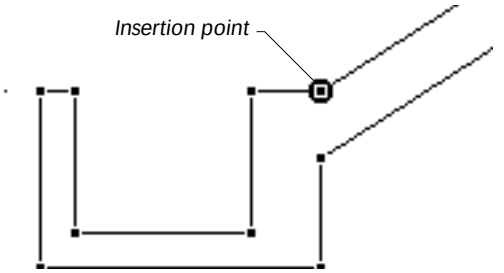


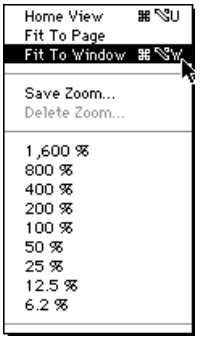

*Fig. 64 Sweep insertion in position*

## CREATING A HOUSE OUTLINE FOR THE ROOF MODEL

To create a roof you can extrude it directly on top of a house, using the perimeter of the house as a sweep path. The Unite command makes it easy to combine basic shapes into the outline of a structure.

### To create a sweep path

1. Choose Fit to Window in the Zoom pop-up menu. DENEBACAD increases the zoom level so you can see the rectangles and the sweep section.




2. Drag a box around the three rectangles (*figure 65*). DENEBACAD selects the rectangles, and a bounding box appears around each rectangle.

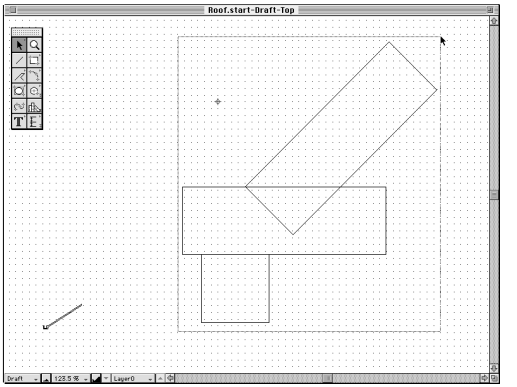



*Fig. 65 Selecting rectangles*

3. Press Command+U. This is a shortcut for the Unite command in the Combine submenu. DENEBACAD unites the rectangles into a single shape. This shape becomes the outline of a building that you will use as the sweep path for the roof.

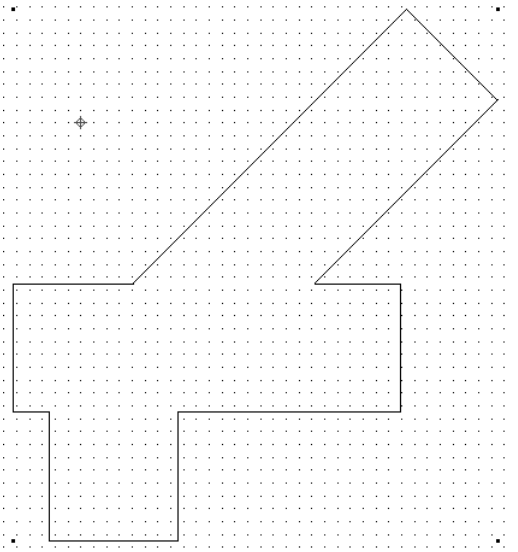


*Fig. 66 United object*

4. Deselect the united object by clicking in an empty area of the document.

5. Choose the Sweep extrusion method by selecting the Sweep action button in the Attributes bar.

   ▲ When you select the Sweep extrusion method, the Sweep Sections palette appears. This palette displays the sweep section designated earlier as "Hip roof." To see your objects clearly, you can drag the palette to move it out of the way.

6. Double-click the Sweep action button to open the Extrude Options dialog box.

7. Make the settings in the Extrude Options dialog box match the illustration (*figure 68*), and then click OK.

8. Select the house (the object you combined).

9. To sweep the section around the house, click the Sweep action button once. DENEBACAD sweeps the section around the selected path.

*Note:* You won't see the result of the sweep extrusion until you switch to Sculpt and Render modes as described later.

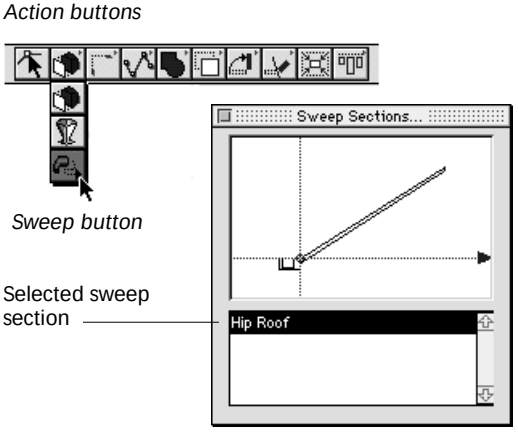



*Fig. 67 Sweep action button and Sweep Sections palette*

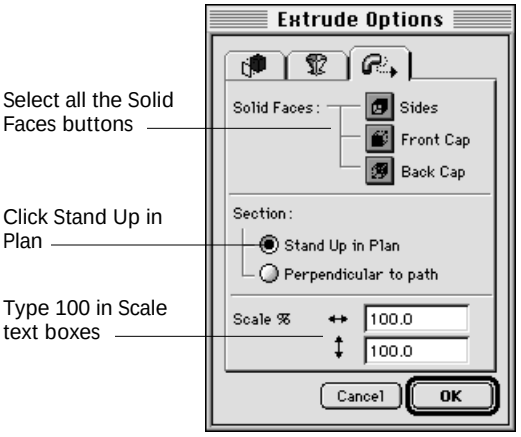



*Fig. 68 Sweep extrusion settings*

## OFFSETTING AND EXTRUDING WALLS

Extruding the roof with the Sweep method creates a 3D object. To make the house match the hip roof, you can turn the simple path into walls, and then extrude the walls into 3D to meet the roof.

This section explains how to offset the house outline to make walls, and how to give height to the walls using predefined extrusion planes.

### To offset an outline for walls

1. Select the house object, which became deselected after the Sweep procedure.
2. Press Command+Shift+O or choose the Offset command in the Path submenu in the Object menu. The Offset dialog box appears.
3. Make the value in the Width text box 3 31/32". Type 8" in the Offset text box. Click OK. DENEBACAD offsets the path 8 inches.

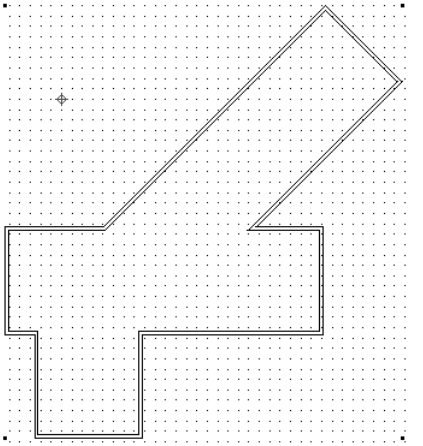



*Fig. 69 An offset path for walls*

### To extrude the walls

1. In the 3D Plane pop-up menu on the Status bar, choose the 8' High Wall, which is a predefined set of extrusion planes for Linear extrusions.

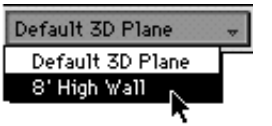



*Fig. 70 Predefined extrusion planes*

2. Select the house, and type Command+Shift+E, the shortcut for the Linear Extrude command. DENEBACAD extrudes the walls from 0' to 8' feet high into 3D space.

As mentioned previously, you won't see the results of the sweep and linear extrusions until you switch to Sculpt and Render mode in the next section of the tutorial.

## VIEWING THE HOUSE MODEL AND RENDERING

To see the results of the roof and wall extrusions, you can open Sculpt and Render mode windows.

In these windows, you can see now that the extruded walls meet the hip roof.

◆ **To open a Sculpt mode window:** Choose Show Sculpt in the Window menu.

◆ **To open a Render mode window:** Choose Show Render in the Window menu.

▲ Notice that DENEBACAD arranges the windows so you can see all three windows on-screen. This happens when the setting "Auto-tile windows" is selected in the Preferences dialog box.

In the window titled "Roof.start-Sculpt-Top," you can see the results of extruding the hip roof (*figure 71*).

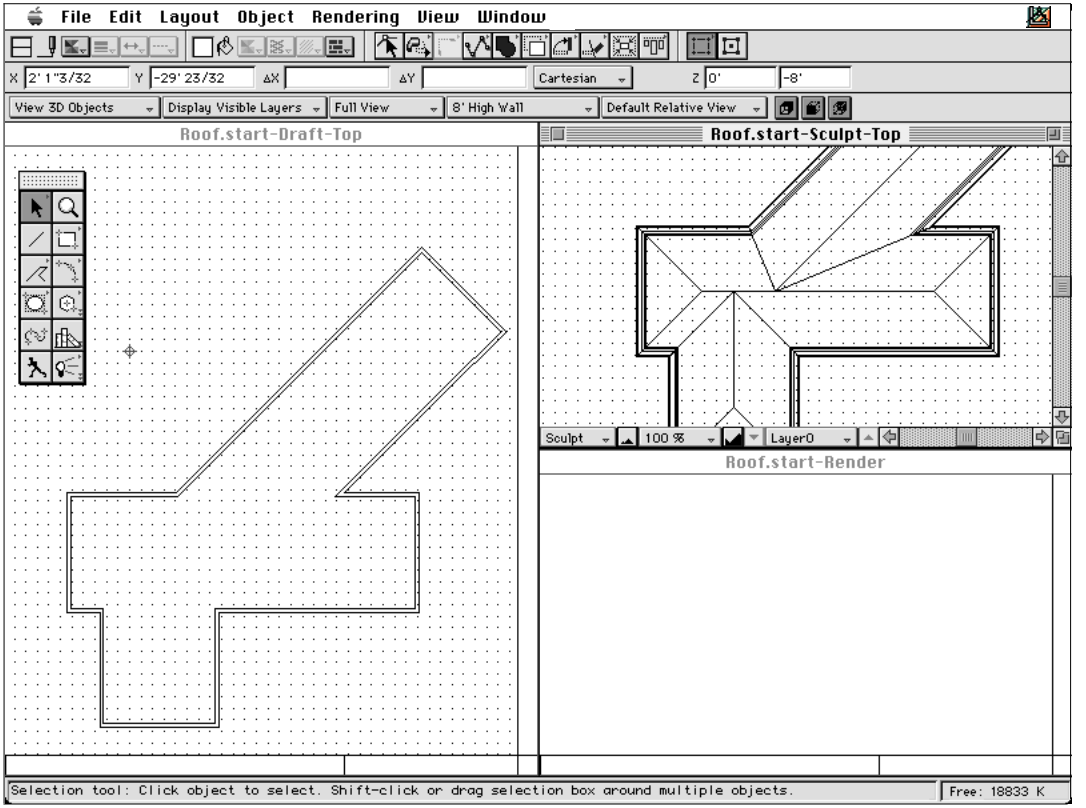



*Fig. 71 The roof visible in Draft and Sculpt windows*

## VIEWING A WIREFRAME RENDERING

Because the roof document now contains 3D objects, you can develop isometric and perspective renderings in the Render window.

By working in Render mode, you can view the building from any perspective. You can create your own perspectives by setting focal points wherever you want as you follow the procedures.

**To generate an isometric perspective**

1. Choose the Isometric Render tool in the Rendering toolbox. DENEBACAD creates an isometric Wireframe rendering in the Render window.

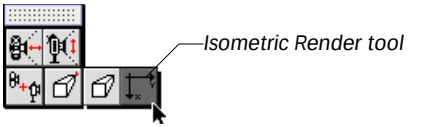
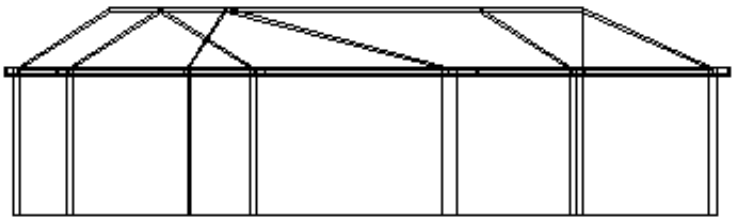
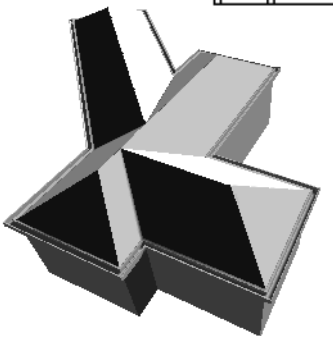



*Fig. 72 Render mode tools*

2. To center the rendering in the window, you can click the  zoom box at the far right of the Render window's title bar. This expands the window to fill the screen, and should give you a full view of the Isometric Wireframe rendering (*figure 73*).

Final Isometric Wireframe rendering





By using the Solid command in the Render menu and the Sunlight option in the Rendering Options dialog box, you can create a perspective Solid rendering of the house with shadows and lighting effects

*Fig. 73 Hip roof renderings*

This chapter is a tutorial that describes how to model a building from the ground up. The procedures include using drawing tools, extrusions, and Library Objects to create a structure. Then, you place lights and furniture, and apply surface materials to the model, before creating a final solid rendering.

### To open the model file

1. Launch DENEBACAD if the program isn't already running.
2. To open the file for this tutorial, press Command+O. In the directory dialog box, go to the Tutorial folder in the DENEBACAD™ folder.
3. Select the document named "Tutorial.start" and click Open.

DENEBACAD displays the document in a Draft mode window set to Top view. The options and grid settings are already configured for the project.

◆ **To display all toolbars:** If the Info bar and Status bar aren't displayed on screen, choose Toolbars in the Layout menu. In the Toolbars dialog box, select the four options, and then click OK.

## DRAFTING A DESIGN FOR MODELING

In DENEBACAD, you use the same working method whether you intend to produce 2D plans or a completely rendered scene.

### To draft wall sections

1. Select the Single Polyline tool.
2. Click at the Centerpoint of the drawing (X=0, Y=0).
3. Move the pointer up; the drawing vector appears. Click at ΔX=0' ΔY=30'.
4. Move the pointer to the right and click at ΔX=38' and ΔY=0', and then press Return to complete the polyline.

This object will become two walls of a building.

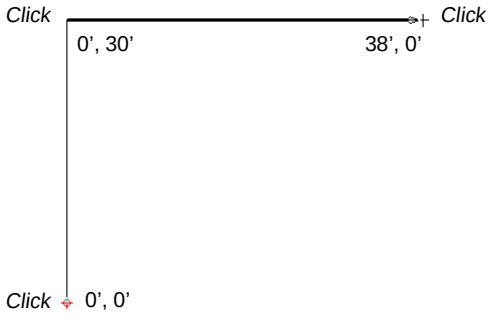



*Fig. 74 Drawing a polyline wall section*

5. Choose Offset in the Path submenu in the Object menu. In the Offset dialog box, type 8" in the Width box and 0" in the Offset box, and then click OK.


### To draw a ground plane

1. Select the Rectangle Diagonal tool. Click at X=-1' (1 foot left of the Centerpoint) and Y=31' to set the first point.
2. Click at ΔX=40' and ΔY= -32' to complete the rectangle.

### To insert a door from the library

Using a library object lets you quickly add a door and windows to the building's walls.

1. Double-click the Library tool in the toolbox. The Library Manager appears. The Library Manager contains the Library Objects stored in this document.


Library tool

2. Click the Object tab to bring it to the front, if necessary. Be sure "Display by Name" is selected. Then, select the name "Door" in the scrolling list (*fig. 75*).
3. Move the pointer to X=8' and Y=30' and click to insert the door into the drawing.
4. Press Shift and drag to the right until the drawing vector appears. This aligns the door within the wall.

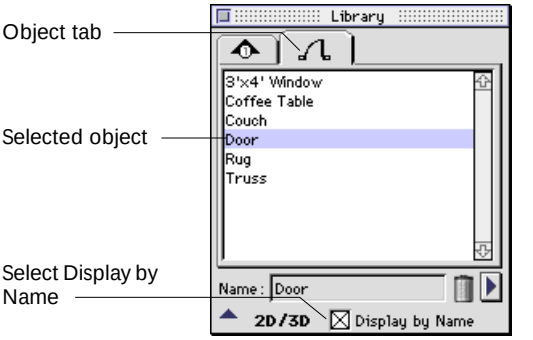



*Fig. 75 Library Object selection*

### To insert windows from the library

1. In the Library Manager scrolling list, select the "3'x 4' Window" object.
2. Move the pointer to X=35' and Y=30' and click to insert the window into the drawing.
3. Press Shift and drag to the right until the vector line appears. This aligns the window within the wall.
4. Click the close box to close the Library Manager.

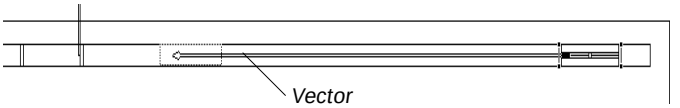
### To duplicate windows in an array

With one window inserted, you can use the Linear Array command to evenly space copies of the window along the wall.

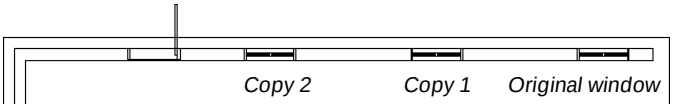
1. Select the window object.
2. Choose Linear Array in the Position submenu in the Object menu.
3. Move the pointer to the center of the window at X=35' and Y=30'. Click to define the starting point of the linear array.

4. Move the pointer to ΔX= -20' and ΔY=0'. Click to set the end point of the array. The Linear Array dialog box appears. Type 2 in the "# Copies" box.


5. Click the Inclusive button. This will evenly distribute the number of copies you specify within the distance defined by the drawing vector. The result is three windows spaced evenly along the wall.




The drawing vector indicates the direction and distance for copying with the Linear Array command




The copies appear evenly distributed within the distance defined by the drawing vector

*Fig. 76 Copying window objects*

## COMBINING OBJECTS AND WALLS

The commands in the Combine submenu let you create Boolean groups by adding and subtracting objects from each other.

### To combine objects into the wall

1. Hold down the Shift key and click each window object, the door object, and the wall objects to select them.

   ▲ You can press Command+A to select all the objects in the drawing, and then press Shift and click the ground plane object to deselect it.

2. Choose Unite in the Combine submenu in the Object menu, or click the Unite action button.

Unite action button

   ▲ The Unite command makes a new Boolean object by combining the door and windows with the wall.

   ▲ The door and window Library Objects include invisible rectangles. The rectangles define the area to be cut out when you apply a Combine submenu command to the Library Objects and a wall object.

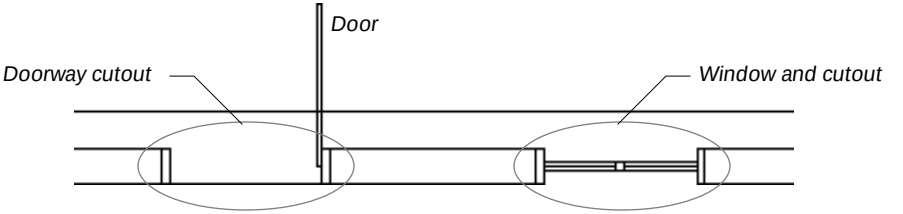



*Fig. 77 Boolean objects in a wall*

## PLACING ROOF TRUSSES

By inserting a roof truss from the Library Manager, and using the Linear Array command to quickly distribute trusses over the structure, we can construct a roof framework in only a few steps.

### To insert a roof truss library object

1. Double-click the Library tool in the toolbox to open the Library Manager.
2. Select the “Truss” item in the Library Manager scrolling list.
3. Move the pointer to X=0' and Y=30' and click the mouse to place the roof truss.
4. Press Shift and drag straight up until the drawing vector appears to align the truss with the left wall.
5. Click the close box to close the Library Manager.

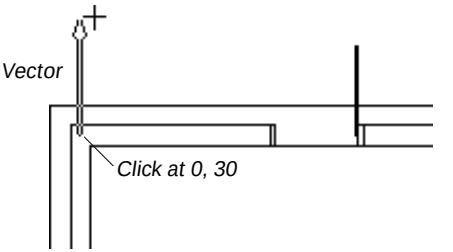



*Fig. 78 Inserting a roof truss*

### To array roof trusses

1. Select the roof truss inserted in the previous steps.
2. Choose Linear Array in the Position submenu in the Object menu.
3. Move the pointer to X=0' and Y=30' over the center of the wall. Click to set the starting point for the array of trusses.
4. Move the pointer to the right to ΔX=2' and ΔY=0'. The drawing vector indicates the distance and direction for the array operation.
5. Click to define the offset distance of the array. The Linear Array dialog box appears.
6. Type 19 in the “# Copies” box and click Offset. This creates an offset array, placing the specified number of copies the same distance apart. There are now 20 trusses above the walls.

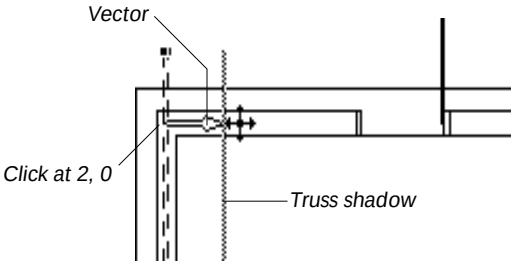



*Fig. 79 Arraying roof trusses*

## INSERTING FURNITURE FROM THE LIBRARY

While Library Objects can speed the design of elements such as walls and roofs, they can also be used to place interior design elements, such as couches and other furnishings.

**To insert a couch library object**

1. Double-click the Library tool in the toolbox to open the Library Manager.
2. Select the item named “Couch” in the Library Manager scrolling list.
3. Move the pointer to X=28' and Y= 20' and click to place the library object. Drag to the right in a straight line until the drawing vector appears; this sets the object’s orientation.
4. To insert another couch, select the Library tool again; the “Couch” should still be selected in the Library Manager. Click at X=34' and Y=14' to place the object. This time, drag downward in a straight line until the drawing vector appears.
5. Select “Rug” in the Library Manager. Click at X=28' and Y=13' to place the object in the drawing. Drag in a straight line to the right to set the orientation.
6. Select “Coffee Table” in the Library Manager. Click at X=28' and Y=13' to place the coffee table in the drawing. Drag straight to the right to set the table’s orientation.
7. Close the Library Manager.

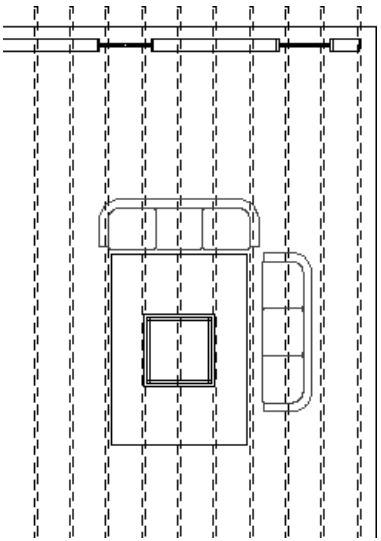


*Fig. 80 Placed furniture Library Objects*

# EXTRUDING THE PLAN INTO 3D

So far, this project has been designed in Draft mode with all objects created in 2D. Now you can extrude the drawing into 3D Sculpt mode. To be able to see the objects in different modes, you can open additional windows.

### To view Sculpt and Render windows

1. Choose Show Sculpt in the Window menu. A window appears in Sculpt mode.
2. Choose Show Render in the Window menu. A new window in Render mode appears.
3. Choose Tile in the Window menu. DENEBACAD arranges the three windows on screen so all are visible. This step isn't necessary if the preference "Auto-tile windows" is selected.
4. Click the Draft window to make it active. Press Command-Option+W (shortcut for the Fit to Window command) so you can see your entire drawing in the window.

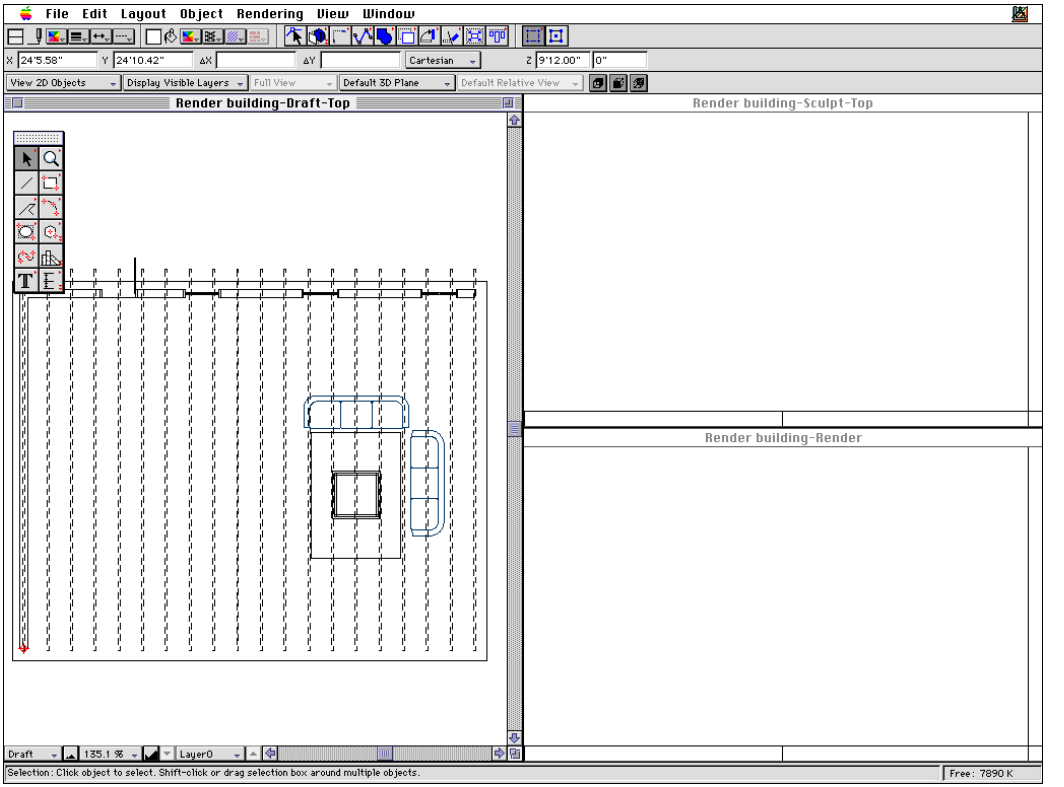



*Fig. 81 The Draft mode window active*

## EXTRUDING WITH PRESET EXTRUSION PLANES

Before extruding the objects in the building into 3D, you can define extrusion planes to set the height of a building's walls. You can save various sets of extrusion planes in a document, and then select one of the saved sets as needed.

To extrude parts of the building in this document, you can use several preset extrusion planes.

#### To extrude the building's slab

1. Choose the pre-defined set of extrusion planes named "1' Thick Slab" in the 3D Plane submenu in the Layout menu. You can also select the pre-defined extrusion planes from the 3D Plane pop-up menu in the Status Bar.
2. Select the rectangle representing the slab under the building.
3. Choose Linear in the Extrude submenu in the Object menu. DENEBACAD extrudes the rectangle into a 1-foot-thick slab, which appears in the Sculpt mode and Render mode windows.

#### To extrude the building's walls

1. Choose the pre-defined extrusion plane named "10' Tall Exterior Wall" in the 3D Plane submenu in the Layout menu.
2. Select the wall object.
3. Press Command+Shift+E or choose Linear in the Extrude submenu in the Object menu. The 3D wall appears in elevation in the Sculpt mode window and in perspective in the Render mode window.

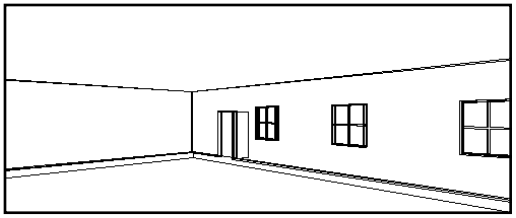


*Fig. 82 Extruded walls, door, and windows*

## EXTRUDING 2D/3D LIBRARY OBJECTS

You have already extruded the objects that needed to be extruded using preset extrusion planes. The next step is to extrude Library Objects that already have 3D information in them, such as the 2D/3D Library Objects in the drawing.

#### To extrude the doors, windows, and roof trusses

1. Click the Draft mode document window to make it active.
2. Press Command+A to select all the objects in the scene.
3. Press Shift and click the wall object and then the slab object to deselect them, because these have already been extruded.
4. Press Command+Shift+E or choose Linear in the Extrude submenu in the Object menu. The Library Objects now appear as 3D objects in the Sculpt and Render mode windows.

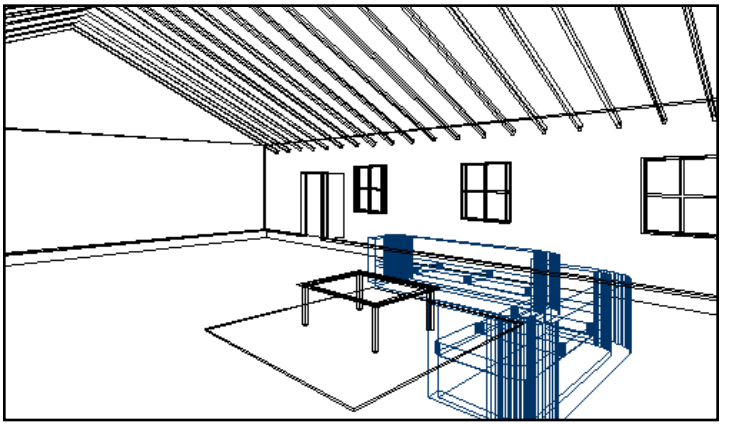


*Fig. 83 Wireframe rendering of 3D objects*

## USING 3D LIBRARY OBJECTS

The last items to be included in the structure are 3D-only Library Objects. These are Library Objects that contain only 3D information, so these objects are used in Sculpt mode.

### To insert 3D Library Objects

1. Click in the document window set to Sculpt mode to make it active. If the window does not show Top view (the title bar doesn't say "*Name*-Sculpt-Top"), choose Top in the View menu.
2. To make the model fit in the window, press Command+Option+W, the shortcut for the Fit To Window command in the Zoom pop-up menu.
3. Double-click the Library tool in the toolbox. The Library Manager appears. Several items which weren't listed when the Draft mode window was active now appear in the Library Manager's scrolling list.
4. Select the "Balcony" Library Object. Click at X=0' and Y=12' to place the balcony. Then press Shift and drag in a straight line to the right until the vector arrow appears. This sets the orientation of the balcony. The balcony appears in the Sculpt window and in the rendering of the room shown in the Render mode window.
5. To insert a set of lights, select the "Lights" object in the Library Manager. Click the pointer to X=20' and Y=15', click to insert the object, then drag straight to the right to set the orientation.
6. To insert a set of spot lights, select the "Spot Lights" Library Object. Click at X= 27' and Y=8' to place the object, and then drag straight to the right to set its orientation.

7. Finally, to insert the roof and finish the model, select the “Roof” Library Object. Click at X=0' and Y=9' to place the object, and then drag straight to the right to set its orientation.

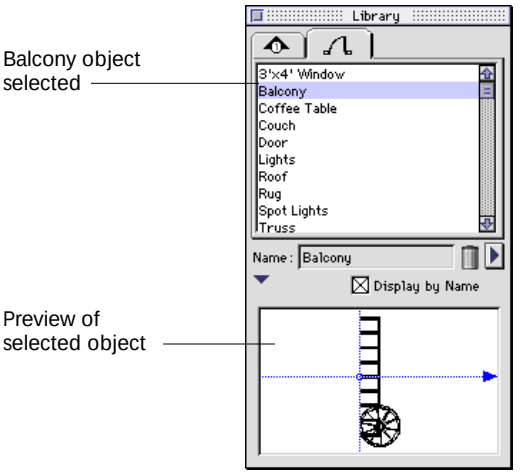



*Fig. 84 Preview in Library Manager*

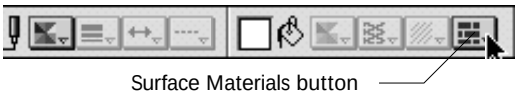
## APPLYING 3D SURFACE MATERIALS

Before producing a final rendering of the model, you can apply materials to the surfaces of the 3D objects.

All of the Library Objects that have been inserted into the model already have surface materials. You can apply other surface materials to the walls and the floor.

### To create a tiled floor

1. Be sure the Sculpt mode window is active and is set to Top view.
2. In the Attributes Bar, press the Surface Materials button and drag into the drawing area to float the Fill palette.




3. In the Sculpt window, select the floor object.
4. In the Fill palette, click the Materials tab. Scroll through the surface material swatches and notice the names of the surface materials in the Name text box as you select different surface materials. Click the “Checkers” swatch to apply the surface material to the floor.
5. In the pop-up menu in the Fill palette, choose Cubic. Select the Set Direction checkbox in the Fill palette.
6. Move the pointer to the upper-left corner of the floor, press Shift, and drag to the right of the object. This sets the direction of the surface material (*fig. 86*).

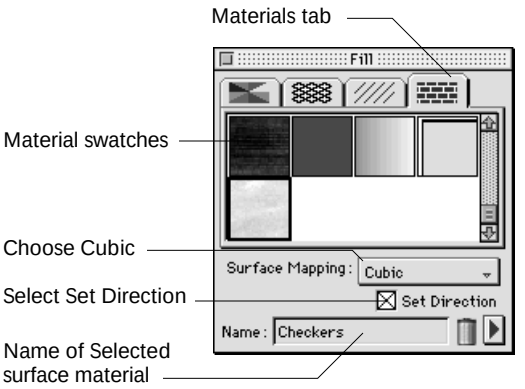



*Fig. 85 Applying a surface material*

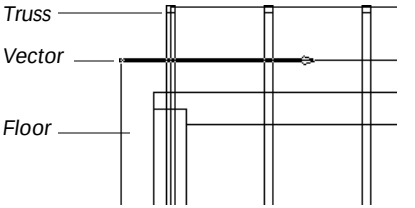



*Fig. 86 Setting the surface material direction*

**To apply a wall material**

1. Select the wall object.
2. On the Material tab, scroll through the materials and select the material named "Stucco Wall."
3. From the Surface Mapping pop-up menu, choose Cubic. Select the Set Direction checkbox.
4. Move the pointer to the upper-right corner of the wall. Press Shift and drag in a straight line to the right.

## CREATING A FINAL RENDERING

When a model is complete and you have placed the objects, lights, and surface materials that you want to use, you can produce a Solid rendering. DENEBACAD's rendering engine generates photo-realistic renderings, complete with transparency, reflectivity, gloss, and shadows.

**To render the model**

1. Click the Render mode document window to make it active.
2. Choose Solid in the Rendering menu. The program will calculate the scene, which can take several minutes, depending on the speed of the system. When complete, the final rendered image appears in the Render window.
3. To save a picture of the rendering, choose Copy in the Edit menu. Switch to the Draft mode window. Choose Paste in the Edit menu. A copy of the rendering is now pasted into the Draft drawing.

## VIEWING FINISHED RENDERINGS

If you weren't able to complete this tutorial or didn't get the results you expect, you can still view a complete rendering of the final result.

**To view a fully rendered scene:**

1. In DENEBACAD, choose Open in the file menu.
2. In the directory dialog box, open the Tutorial folder. Select the file named "Tutorial Rendering" and click Open. A document opens in Draft mode, with a "picture" of the fully rendered scene.
3. To view other types of rendering from the same perspective, choose Wireframe or Hidden Line in the Rendering menu.

*Fig. 87 A Solid rendering of the building*

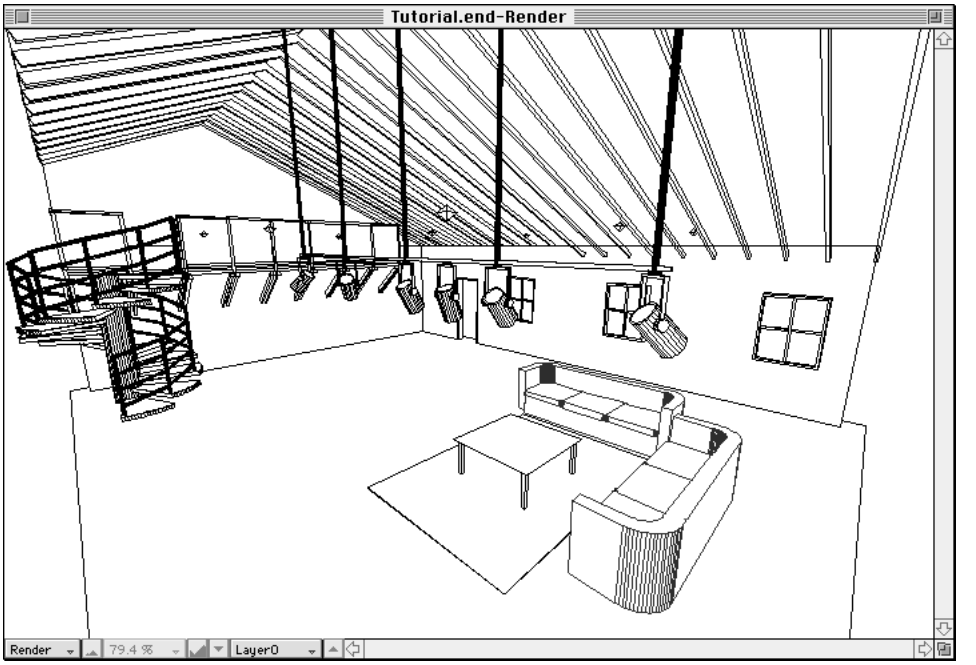



*Fig. 88 A Hidden Line rendering of the building*

**To view a QuickTime VR scene**

1. Locate the file named “Tutorial QTVR” in the Tutorial folder.
2. Double-click the file icon to launch the QuickTime VR application and display the stereoscopic rendering of the scene.

The QuickTime VR application is included in the DENEBACAD folder so you can view the QuickTime VR rendering. If you view the scene with stereoscopic glasses, you see true a true 3D rendering, and QuickTime VR lets you tilt and pan to see a 360-degree view of the rendered model.

## FINISHING UP

Congratulations on finishing this introduction to the basics of DENEBACAD. You can use the Save As command to save your project with a new name. Do not use the Save command unless you want to replace the tutorial file with your version.

# SETTING UP A PROJECT 8

This chapter presents an overview of options you can configure in DENEBACAD. Using the working drawings of a house as an example, this chapter explains how to specify a document's measurement system, output settings, grids, layer structure, and classes, as well as how to define extrusion planes for creating 3D models. It also describes situations in which you can use Libraries, layers, and classes to organize a project.

## SETUP OVERVIEW

To start using DENEBACAD, double-click the DENEBACAD application icon.

When DENEBACAD begins running, the program opens a Draft mode window. The window's title bar displays "*Untitled-Draft-Top*." You can configure all of the following settings with this window active.

The following are brief descriptions of the types of settings you probably want to configure when you start a new project. Later sections in this chapter go into greater detail on these topics.

**Units setup** Choose the Units Setup command in the Layout menu to select the measurement system and the units of measure you want to use in the document. For details, see "Drawing scale and measurements" on page 85.

When you launch DENEBACAD the first time, the US measurement system is selected, and the program uses feet and decimal inches in dimensions and all other measurements. If you normally work with this configuration, you can skip the Units Setup step.

**Output setup** Choose the Output Setup command in the Layout menu to select the output scale and set the number of pages for printing. For details, see "Output setup" on page 86.

**Grid setup** If the Show Grid command is active, DENEBACAD displays a grid of dots based on the current units and measurement system. Choose the Grid Setup command in the Layout menu to specify the grid spacing. For details, see "Grid setup" on page 88.

**Layers** Choose Layer Manager in the Window menu to set up layers. For details, see "Library items" on page 88.

### Other configuration steps

After you set up measurement units, output, the grid, and layers, you can begin to draw. There are, however, other things you might want to set up that will save time: You can import Library Objects and Symbols, establish classes, and define 3D planes before you start to draw.

**Library items** If you want to use existing Library Objects and Symbols in your document, you can double-click the Library tool to open the

Library Manager, and use the Add command in the pop-up menu to import the Library items. For details, see “Library items” on page 88.

**Classes** You can choose the Class Manager command in the Window menu to set up classes. If you do not plan to categorize objects in your drawings so you can create reports, you can skip this step. For details, see “Setting up classes” on page 90.

**Extrusion planes** If you want to model 3D objects, use the 3D Plane submenu commands to define and save extrusion planes. For example, you can set up extrusion planes for the height of walls, a porch floor and porch railings.

You can choose saved extrusion planes from a pop-up menu on the Status bar. The current extrusion planes set the depth of objects that you extrude in draft mode and model in Sculpt mode. For details, see “Setting up 3D extrusions” on page 91.

## USING DOCUMENT TEMPLATES

After you configure drawing and document settings you can create a template, which is a special kind of document that incorporates all the current DENEBACAD settings.

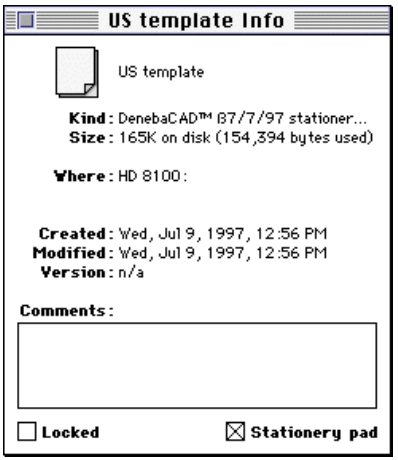
Templates can help you save time. By using a template to start a project, you avoid having to configure the same settings again. For example, you can save the drawing scale, layer structure, grid settings, and other options for a house drawing, and use the template again when you start the working drawings of another house.

### To create a document template

1. After establishing the document settings that you want to save, choose the Save As command in the File menu.
2. In the directory dialog box, be sure to select the DENEBACAD file format. Type a name for the document and click Save to store the document on disk.

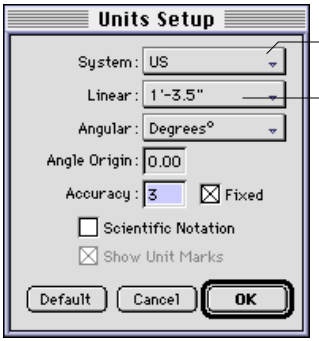
After you save a template, make the saved document a stationary pad. When you open a stationary pad, an untitled document appears to prevent you from overwriting the template.

1. Select the template document’s icon and choose Get Info in the File menu in the Finder. The Get Info dialog box appears.
2. Select the Stationary pad check box and then choose Close in the File menu to close the Get Info dialog box.



# DRAWING SCALE AND MEASUREMENTS

You can use the Units Setup command in the Layout menu to open the Units Setup dialog box. This dialog box lets you choose the U.S. or metric measurement system, the units and format for linear and angular measurements, and specify the accuracy to be used for measurements in the current document.




In U.S. versions of DENEBACAD, the U.S. measurement system is the default system, and the program displays measurements in feet and decimal inches, in the form 15'-3.5".

**Measurement system** If you use feet and decimal inches in your work, you do not need to change the settings in the Units Setup dialog box. If you use metric measurements, choose Metric in the System pop-up menu.

**Linear measurement formats** If you prefer a format for measurements other than the default feet and decimal inches, choose the format you want to use from the Linear pop-up menu. You can choose feet and fractional inches; decimal inches only; or fractional inches only.

For example, to create the plan and interior elevations of a house, you might want to change the format for measurements from feet and decimal inches to feet and fractional inches. To do this, choose 1'3-1/2" in the Linear pop-up menu.

To create cabinet and millwork drawings for a house, you might want to choose inches as the units. In this case, choose 15 1/2" or 15.5" in the Linear pop-up menu.

**Displaying unit marks** When you select only inch or foot measurements, select the Show Unit Marks check box if you want to display tick marks for feet and inches in measurements and dimension text. This option is not available when you choose either of the combined foot and inch formats in the Linear pop-up menu.

For a millwork drawing in inches, you might not want tick marks in dimension text. In this case, you would deselect the Show Unit Marks option.

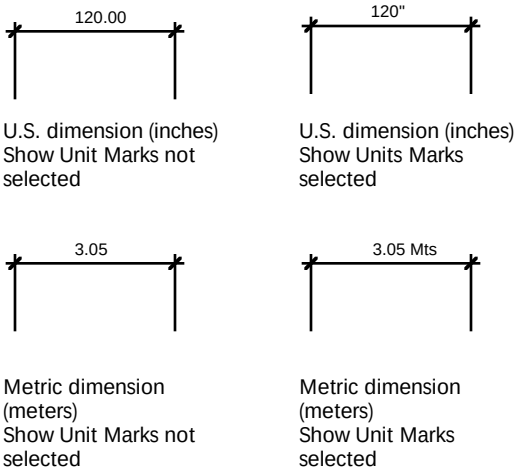



*Fig. 89 Show Unit Marks option*

You might find that you don't need to change many settings in the Units Setup dialog box to create working drawings of a house, for example. The default measurement system, linear and angular units, and accuracy are appropriate for many projects. You probably won't need to select the Scientific Notation option, which displays numerical values as a base value and base 10 exponent, except in special situations.

### Setting default measurement options

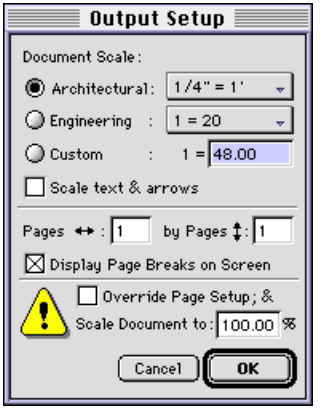
If you want the settings in the Units Setup dialog box to be the default settings for every new document, click the Default button.

Refer to the Units Setup command in the Layout menu chapter for more information about the options and settings in the Units Setup dialog box.

## OUTPUT SETUP

You can use the Output Setup command to set up a project for printing. In DENEBACAD, you print or plot in Draft mode.

The Output Setup dialog box lets you select a scale for printing the document. You can also specify the size of the printable area and the arrangement of pages (if the document must be divided into pages for printing).




### Setting the scale for a document

You can select the most common architectural and engineering scales from the Architectural or Engineering pop-up menus in the Output Setup dialog box. Or, you can enter a custom scale in the Custom text box.

If you are setting up a document for plans and interior elevations of a house, you probably want to select the 1/4"=1'-0" scale. To develop details of the house, you might select the 1-1/2"=1'-0" scale.

The document scale you select in the Output Setup dialog box affects an entire document. If you are accustomed to working in more than one scale, you can set up a document for each scale.

### Working within the printable area

It is helpful to set a document's output scale and specify the number of pages in the document so you can work within the printable area in the Draft window.

Two factors affect the size and configuration of the printable area in Draft mode: The settings for Document Scale and Pages in the Output Setup dialog box, and the page size and orientation specified in the Page Setup dialog box.

To see the printable area in Draft mode, select "Display Page Breaks on Screen" in the Output Setup dialog box. With this option selected, DENEBACAD displays one or more gray boxes in the Draft window to represent page boundries.

### Setting page boundaries

You can control how DENEBACAD divides the document into pages for printing, and how the pages are arranged when you output them.

You can set up a document so that the plan prints on one page, elevations print on another, and renderings print on a third page, for example (*figure 90*).

To set up the page arrangement, type the number of pages you want to print in the "Pages" and "by Pages" text boxes in the Output Setup dialog box. The value in the first "Pages" text box sets the number of pages across; the value in the second text box sets the number of pages down. To set up two pages side-by-side, for example, type 2 in the first box and 1 in the second box.

When you create a new document, the values in the "Pages" and "by Pages" boxes are set to 1. This means DENEBACAD prints one page, with the center at the Centerpoint of the current view.

To adjust the location of the printable area in the Draft window, choose Set Print Area in the File menu. Move the pointer to position the gray border that represents page boundaries, and then click to set the printable area.

When the printable area is displayed on screen (with the "Display Page Breaks on Screen" option selected in the Output Setup dialog box), you can drag parts of the project, such as plans, elevations, and renderings, to position them within the page boundaries.

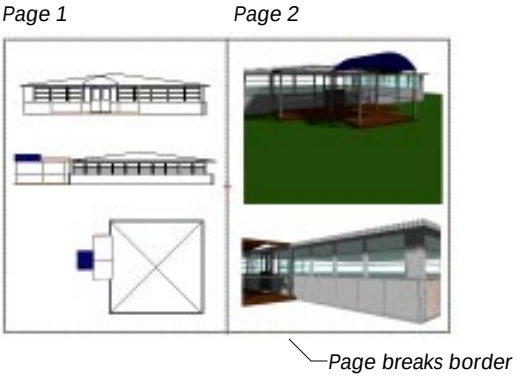


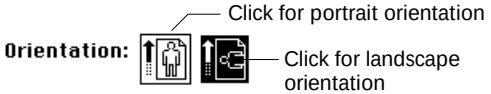

*Fig. 90 Printable area in Draft mode*

Keep in mind that the scale specified in the Output Setup dialog box applies to an entire document. Though you might print parts of the document on separate sheets of paper, the scale is the same for all pages.

When you open a new DenebaCAD document, the printable area is not visible on screen. If you create a drawing and print it without first ensuring that all of the objects are in the printable area, some objects might appear cut off in the printed output.

### Page setup

You can use the Page Setup command in the File menu to set the orientation of printed pages. For example, it's common to use landscape orientation for printing drawings. Click the landscape or portrait button to set the paper orientation. The setting you choose affects the orientation of the printable area in the Draft mode window.



## GRID SETUP

The grid is a pattern of evenly spaced dots in horizontal and vertical rows. When you draw and move objects, they can be made to snap to the grid, whether or not the grid is visible. Also, you can display the grid whether snap is active or not.

The Snap Grid button lets you turn grid snap on and off, and select from five preset grid spacing settings. Press the button to display a pop-up menu of grid spacing options.

You can use the Grid Setup command in the Layout menu to change the settings for grid spacing. When you specify a custom grid spacing, the new setting appears in the pop-up menu under the Snap Grid button. If the pop-up menu already contains five grid spacing settings, the new setting replaces one setting in the pop-up menu.

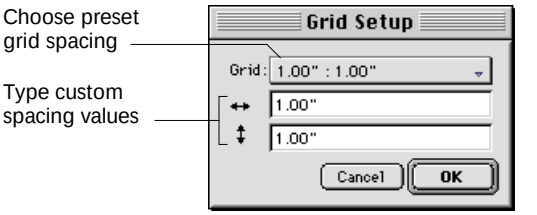
The grid spacing is affected by the units and measurement system specified in the Units Setup dialog box. If you are working in U.S. units, you specify grid spacing in U.S. units.

**To specify new grid spacing options**

1. Choose Grid Setup in the Layout menu.
2. In the Grid Setup dialog box, choose the grid setting you want to modify from the Grid pop-up menu.
3. The the horizontal spacing value in the first text box, and type the vertical spacing value in the second text box.
4. Click OK to implement the new Grid setting.

◆ **To activate grid snap:** Choose the Snap to Grid command in the Layout menu, or click the Snap Grid button.

◆ **To display the grid:** Choose the Show Grid command in the Layout menu.




# LIBRARY ITEMS

Before you begin to draw, you might want to add Library items to the document. You can import saved Library Objects and Symbols to use in your drawing. You can create and save your own Library items, and you can use sample Library items distributed on the DENEBACAD CD-ROM.

Library items let you place common elements such as elevation symbols in a document with the Library tool. This is helpful if you use the same drawing symbols and objects in all of your drawings. For example, you will probably use the same elevation symbol in every plan you draw.

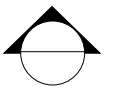


*Fig. 91 Elevation symbol*

You might also use the same interior door in every house you design. If this is the case, using the Library eliminates the repetitive task of creating the same door.

Doors, windows, plumbing fixtures, electrical symbols, and drawing symbols are examples of Library Objects and Symbols that you might want to add to a new document.

### Adding Library items

To add Library items to a document so they can be placed in a drawing, use the following procedure:

1. Click the Library tool icon to open the Library palettes.
2. Choose the Add command in the pop-up menu under the right triangle button.
3. In the directory dialog box, select the document that contains the Library items you want to add.
4. Use the Library Picker to add items one at a time to the current document (*figure 92*).

You can refer to the Library tool in the 2D and 3D Drawing tools chapter for more information on adding and working with Library Objects and Symbols.

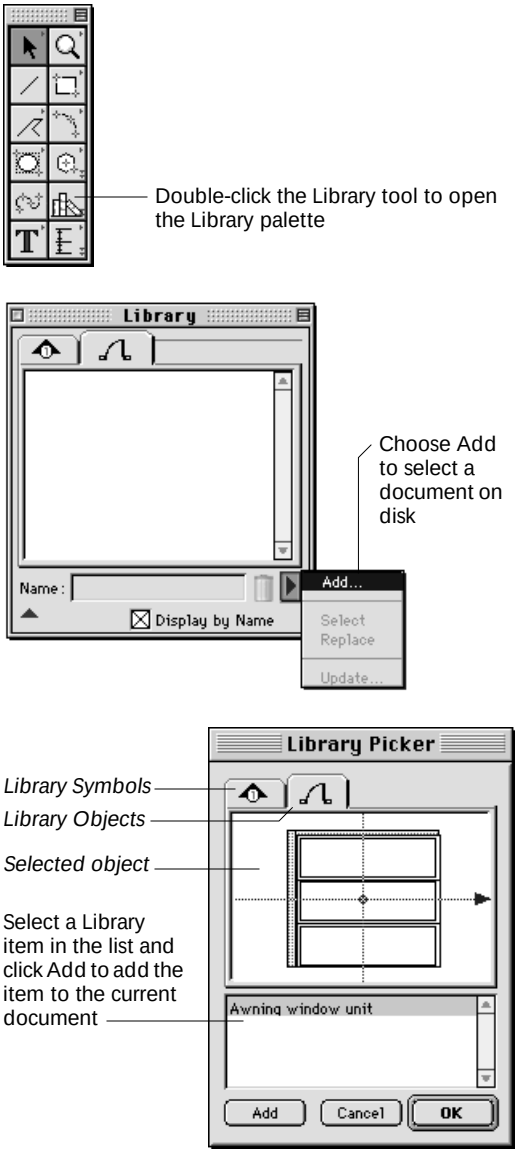



*Fig. 92 Adding Library items to a document*

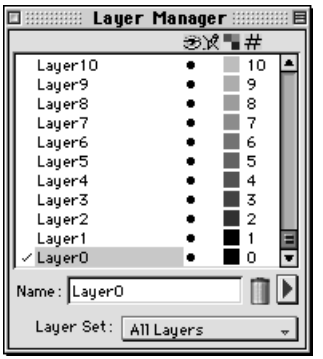
# USING LAYERS AND CLASSES

Creating a layer structure is one effective way you can use to organize objects and structural components in a drawing.

Standard layer structures let architects and engineers share common information between documents. By using the same layer structure in all documents, they can share CAD files easily and improve consistency within documents.

If you do not need to use an industry standard layer structure, you can name layers according to what kind of plans you are creating. For a house, you might create layers named “walls,” “roof,” “electrical,” and “lighting.” When you want to edit walls, you can hide the other layers so they don’t interfere with editing. If you want to print a lighting plan, you can hide all other layers. When a layer is not visible, it does not print.

You use the Layer Manager command in the Window menu to open the Layer Manager, a palette for working with layers. You can then define as many as 256 layers. You can add 255 layers to the layer structure at one time before you begin to draw, so you don’t have to go back and add layers later.




## SETTING UP CLASSES

DENEBACAD lets you set up classifications of objects so that you can generate reports using information that you enter into the class text boxes.

In other words, you can classify objects and then generate reports using the classifications.

Using a house as an example, you can create hardware, plumbing, and electrical classes. You can then organize the information by categories such as model, price, and color.

You can then generate reports based on the specific information you need. For example, if you need to order door knobs for the house, you can categorize door knobs under the hardware class. You can then generate a report based on the hardware class, and more specifically the model, price and quantity of the door knobs.

To do this, you set up a class system using the Class Manager. You can then assign an object a class using the Properties Manager, and generate reports based on the object’s categorizations using the Analysis Manager.

# SETTING UP 3D EXTRUSIONS

After you configure your document's drawing environment, you might want to set up 3D extrusion planes.

Extrusion planes are a set of two planes (which can be parallel to each other or not) that define the boundries of a linear extrusion. You will use extrusion planes if you model your structure in 3D in Sculpt mode, or draw plans in 2D in draft mode and then extrude objects to make a 3D model.

*Note:* You can use three extrusion methods in DENEBACAD: linear, spin, and sweep. For linear extrusion, you define a set of extrusion planes, which is described in this section. For spin extrusion, you define an extrusion axis. For sweep extrusion, you select a sweep section and a path.

## Configuring extrusion planes

Defining and saving extrusion planes at the start of a project is a very effective way to work. You can save the extrusion planes with useful names, and then select them from the 3D Plane pop-up menu in the Status bar. As you work on various aspects of your project, you will probably switch among several sets of extrusion planes; settings these planes up before you begin lets you continue drafting and modeling without stopping to set up new extrusion planes each time you want to extrude an object in either Draft of Sculpt mode.

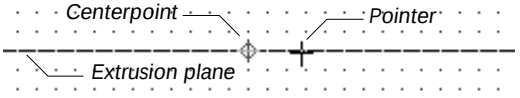
The extrusion planes you might set up for a house project would define things such as the height of the walls, the length of the roof and overhangs, the height and depth of countertops and built-ins. Extrusion planes are also used to set the position of lights used for rendering, and the height of camera paths used to generate walk-throughs of 3D models.

The following procedure illustrates how to define and save extrusion planes for the walls of a house. You establish the height of the walls by defining planes at floor and ceiling level. With the wall height defined, you can then create 3D walls by drawing the tops of the walls in Top view.

### To define extrusion planes for walls

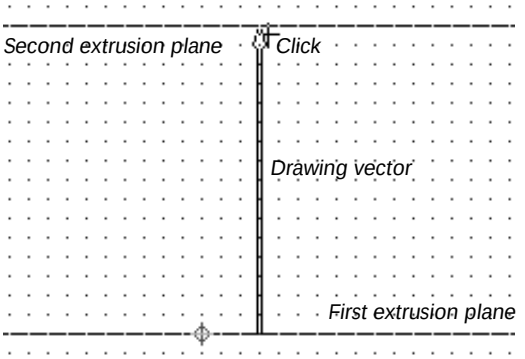
This procedure uses Top view in Draft mode. This is the view and mode of the drawing window when you create a new document. It's best if you can see the Centerpoint ( ⌖ ) in the window so you know you are near the origin (0,0) of the document when you begin.

1. Choose Define Frontal 3D Plane in the 3D Plane submenu in the Layout menu. The window changes to Front view and the pointer becomes a small cross with a horizontal line that extends to the window edges. The line represents the first extrusion plane.

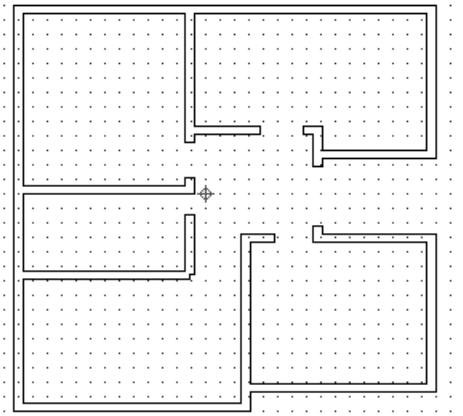



2. To position the first extrusion plane at floor level, move the pointer up or down until the line passes through the Centerpoint (the Y coordinate in the Info bar will be 0"). Click to set the first extrusion plane in place.

3. Move the pointer upward to position the second extrusion plane at ceiling height. A second horizontal line (the second extrusion plane) follows the pointer. For this example, move the pointer to Z=8' to establish an 8-foot-high wall extrusion. Click to set the second extrusion plane in place. The Draft window switches back to top view.




After you define the extrusion planes, you can create the walls of the house by drawing the tops of the walls in Top view. You can draw the walls in Draft mode and use the Linear command to extrude 3D objects, or draw 3D objects directly in Sculpt mode when the Linear extrusion method is selected.



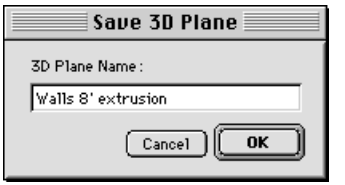
## Saving extrusion planes

After you define a set of extrusion planes, you can save them using any name you want — preferably one that will make it easy to select the correct extrusion planes later.

*Note:* If you do not save extrusion planes, they will be replaced by the next set of extrusion planes you define.

### To save the extrusion planes

1. Choose the Save 3D Plane command in the 3D Plane submenu in the Layout menu. The Save 3D Plane dialog box appears.




2. Type the name you want to use for the extrusion planes in the text box.
3. Click OK to save the extrusion planes in the document.

## Selecting saved extrusion planes

After you save extrusion planes, you can select them using the 3D Plane pop-up menu in the Status bar.

# GLOSSARY

**Absolute Origin** — The coordinates x=0, y=0, z=0 in the Cartesian coordinate system. This is the Default Centerpoint for a new document and for the DENEBACAD workspace. The Absolute Origin can never be changed. *See also* Custom Centerpoint.

**Action buttons** — The group of icons in a row at the right on the Attributes bar. Action buttons are shortcuts for choosing commands. You click an action button to execute the associated command, or press the button to open a toolbar of related action buttons.

**active document window** — The Draft, Sculpt, or Render mode window with DENEBACAD's focus. The Info bar and Status bar display information related to the active document window.

**alignment** — To horizontally or vertically line up objects with the Align command or action button.

**Analysis Manager** — A palette that lets you display summary reports of objects and other information in a DENEBACAD document. The Analysis Manager compiles information from preset object data and Classes defined in the Class Manager. Analysis Manager reports can be exported in tab-delimited form.

**array** — Copies of an object positioned a set distance from each other or within a specified distance from the original object.

**Arrowhead** — A line terminator shape such as an arrow, circle, or witness line that can be applied to lines from the Arrow tab in the Pen palette.

**attributes** — Graphical modifiers such as color and line width that can be applied to an object. *See* Pen color, Pen weight, Pen style, Arrowhead, Fill color, Fill pattern, Fill hatch, and surface material.

**Attributes bar** — A bar near the top of the screen containing buttons that display the Pen palette and Fill palette. The Action buttons appear on the right end of the Attributes bar. When the Text tool is selected, text controls appear in the Attributes bar.

**Attributes cluster** — The group of buttons at the left end of the Attributes bar, which let you apply attributes from the Pen and Fill palettes.

**axis** A reference line from which distances or angles are measured. DENEBACAD utilizes X, Y, and Z axes to draw 3D objects.

**Back view** One of six orthogonal views in Draft and Sculpt modes. Back view shows the DENEBACAD workspace from the back. This is an elevation view.

**Boolean object** Two or more objects combined into a single object using a command in the Combine submenu.

**Bottom view** One of six orthogonal views in Draft and Sculpt modes. Bottom view shows the DENEBACAD workspace from the bottom.

**bounding box** An invisible rectangle that defines the boundaries of a selected object in Draft and Sculpt modes. The bounding box is shown on screen by tiny black handles located at the corners of a selected object

**Centerpoint** A location in a document at the Cartesian coordinates x=0, y=0, z=0. You can create a Custom Centerpoint and save Custom Centerpoints so you can assign the 0, 0, 0 coordinates to various locations in a document as you work.

**Class** A classification that you can define and then assign to objects as a way of categorizing information about a project. Classes are defined in the Class Manager with up to eight data fields for each Class.

**Class Manager** A dialog used to define classes of objects. In the dialog box, you can enter a Class name and up to eight fields of information for that Class. Classes appear in the Properties Manager and can also be used in the Analysis Manager.

**Clipboard** A part of the Mac OS operating system that stores objects cut or copied in any program. The Show Clipboard command in DENEBACAD displays the Clipboard contents.

**clipping planes** Planes defined in Sculpt mode or Render mode that limit the view of a document so objects between the planes are visible and other objects are not visible.

**combined object** Two or more objects combined into a single shape using the commands in the Combine submenu.

**constrain** To restrict the movement of the pointer and the drawing vector to a certain direction or type of movement. For example, to snap the pointer to 15-degree intervals by pressing the Shift key when moving an object with the Selection tool. When you constrain the drawing vector, you can also restrict an object to a shape, such as a perfect circle or square, when drawing or editing.

**container** An object defined as a container in the Properties Manager. Containers can be used to display information in the Analysis Manager about objects within their boundaries.

**coordinate system** A method of specifying the positions of points in space by reference to fixed points, lines, or planes. DENEBACAD provides six coordinate systems: Cartesian, Polar, Relational, Bearing, Geographic, and Gradient.

**coordinates** Two numbers that identify the location of a point in space in a given dimension by its distance from a set point. The current coordinate system determines whether coordinates are expressed as linear or angular measurements.

**Copy** A command that copies selected objects or text and places the data on the Clipboard.

**creation points** The points that define an object, such as the endpoints of a line or the opposite corners of a rectangle. Creation points are visible in reshape mode and when they are selected with the Point Selection tool.

**custom Centerpoint** A location in a drawing designated as coordinates x=0, y=0, z=0 by the user.

**custom Relative View** The last Relative View defined by a user in Sculpt mode, regardless of whether the view was saved.

**custom 3D Axis** The last 3D Axis defined by a user in the current view in Draft mode or Sculpt mode.

**custom 3D Plane** The last 3D Plane defined by a user in the current view in Draft mode or Sculpt mode.

**current attributes** The attributes applied to a new object that is created with a drawing tool.

**current layer** In Draft or Sculpt mode, the layer on which a newly created object appears. Because Draft layers and Sculpt layers are separate, each mode has its own current layer when the mode is active.

**default** — A setting for object attributes, tool functions, window options, or operations that are in effect when you start a DENEBACAD work session.

**default Centerpoint** — The Centerpoint that is in effect when you create a new document or choose the Default Centerpoint command.

**default 3D Axis** — The 3D Axis that is in effect when you create a new document or choose the Default 3D Axis command.

**default 3D Plane** — The set of 3D planes that are in effect when you create a new document or choose the Default 3D Plane command.

**default Relative View** — The Relative View that is in effect when you create a new document or choose the Default Relative View command.

**DENEBACAD workspace** — An infinite 3D space for construction of 3D models, and a related 2D space for construction of 2D objects and working drawings. These spaces can be viewed from various perspectives in Draft and Sculpt mode windows.

**ΔX** — Delta X; the change in the value represented by X, such as the horizontal coordinate of a point.

**ΔY** — Delta Y; the change in the value represented by Y, such as the vertical coordinate of a point.

**ΔZ** — Delta Z; the change in the value represented by Z, such as the coordinate of a point.

**dialog box** — A window displayed to report or request information. You use dialog boxes to choose options, specify values, and start and cancel operations.

**dimension object** — An object containing dimensions created with a Dimension tool.

**document scale** — A ratio of the size of objects in a drawing to the actual size in the physical world represented by the drawing.

**drawing vector** — A vector represented by a gray arrow on screen in DENEBACAD. The drawing vector indicates the vector from the absolute origin to a point, or from one creation point to the next, when you draw, move, reshape, rotate, extrude, resize or otherwise manipulate an object's position.

**Draft mode** — The mode of DENEBACAD when a Draft mode window is active. In Draft mode you can create 2D objects and assemble working drawings based on the 2D and 3D objects in a document.

**Draft mode window** Any window showing your work in Draft mode.

**duplicate** To make an exact copy of an object with the Duplicate command. Also, the object resulting from duplication.

**endpoint** Either of two points that define the ends of a line segment, a polyline, or an arc.

**extrude** The process of adding a third dimension to a 2D object using the Linear, Spin, or Sweep extrusion method.

**extrusion axis** The axis along which Linear, Spin, and Sweep extrusion methods add depth to a 2D object. The Extrusion axis changes based on the current view, current Relative View, and current coordinate system.

**extrusion format** A setting that controls which parts of a 2D object are extruded into 3D. The extrusion format can be set using three buttons in the Status bar.

**extrusion method** The procedure used to extrude 2D objects, either Linear, Spin, or Sweep, which affects the resulting 3D shape. You specify an extrusion method in Sculpt mode by selecting an Extrusion action button. In Draft mode, you choose an extrusion method when you extrude objects with an action button or Extrude command in the Object menu.

**extrusion planes** A set of two planes, parallel or not, that define the boundaries of linear extrusions. The planes are identified by creation order as first and second, or by position in a particular view as lower and higher. Some objects, such as walkthrough paths and light sources, are always drawn on the lower or higher extrusion plane.

**Fill color** A color on the Fill Color tab of the Fill palette, which can be applied to the interior of a 2D object.

**Fill hatch** A hatch pattern on the Fill Hatch tab of the Fill palette, which can be applied to the interior of a 2D object.

**Fill palette** A floating window containing four types of attributes — colors, patterns, hatches, and surface materials — that can be applied to objects.

**Fill pattern** A pixel-based pattern on the Fill Pattern tab of the Fill palette, which can be applied to the interior of a 2D object.

**Fit to Window** — A command that changes the magnification of a document so that all objects appear within the active Draft or Sculpt window.

**Front view** — One of six orthogonal views in Draft and Sculpt modes. Front view shows the DENEBACAD workspace from the front. This is an elevation view.

**first plane** — The plane you set first, regardless of position, when you define a set of extrusion planes.

**grid** — A pattern of evenly spaced dots in horizontal and vertical rows. The pointer and objects can be made to snap to the grid dots, whether or not they are visible. The Grid Setup command can be used to set the grid spacing.

**group** — (*v.*) To associate two or more objects so they behave as a unified object; (*n.*) the resulting object.

**handles** — Small black squares displayed at the corners of an object's bounding box when the object is selected. Also, similar squares that define parts of walkthrough paths and control points on curve objects.

**Help bar** — The area at the bottom of the screen that displays messages about tools, commands, and procedures based on the position of the pointer and the currently selected tool.

**Hidden Line rendering** — A type of rendering displayed in the Render mode window in which 3D objects appear as solids without any shading.

**Horizontal Focal Point** — A point set with the Horizontal Focal Point tool in the Render mode window. The point defines the distance and the horizontal direction of the sight line from the viewer to the objects to be rendered.

**Info bar** — An area at the top of the screen containing the coordinate system pop-up menu and a set of text boxes. In the text boxes, you can view and enter coordinates for creating and modifying objects, planes, and axes.

**insertion point** — (1.) A point on a library object that can snap to other objects. (2.) The vertical blinking line that indicates where text you type will appear in a document or dialog box. *See also* Sweep insertion point.

**layer** A virtual drawing overlay used to segregate objects in Draft and Sculpt modes. Draft mode can contain up to 256 layers per view, or 1,536 total. Sculpt mode can contain up to 256 layers that are part of all views.

**Layer Manager** A floating palette in which you can create, arrange, select, and configure options for drawing layers.

**layer structure** The overall organization of layers in a DENEBACAD document. You can name layers, reorder layers, and assign colors to layers. Each layer has an absolute number which designates the order in which it was created.

**layer set** A subset of the total number of layers. Each view in Draft mode has separate layers and layer sets. Sculpt mode has layers and layer sets shared by all views. Only one layer set can be active. Layers in non-active layer sets do not appear in the Layer Manager. Objects on layers in non-active layer sets do not appear in a drawing. Layer sets can be defined and named in the Layer Manager.

**Left view** One of six orthogonal views in Draft and Sculpt modes. Left view shows the DENEBACAD workspace from the left. This is an elevation view.

**Library Object** An object that has been designated as a Library Object in the Properties Manager. Library Objects can contain a combination of 2D and 3D objects.

**linear array** Copies of an object arranged a set distance from each other or distributed over a specified distance.

**Linear extrusion** A method of adding a third dimension to a 2D object by extending the object along a single axis.

**message box** A window that displays information and asks for a response from the user before continuing.

**midpoint** The point on a line segment or curvilinear arc that divides the object into two parts of equal length.

**mode** *See* Draft mode, Sculpt mode, and Render mode.

**Mode pop-up menu** A pop-up menu that lets you set the active document window to Draft, Sculpt, or Render mode.

**object** An item that can be selected in a DENEBACAD Draft or Sculpt window, including 2D and 3D shapes such as lines, rectangles, Boolean objects, cubes, cylinders, dimensions, text, pasted renderings, and views copied to Draft mode.

**page breaks** The boundaries, which can be displayed on screen, between individual pages of a document when it is printed.

**palette** A type of dialog box that can remain open. For example, you apply attributes to objects from the Pen and Fill palettes.

**Panned Zoom** A method of changing the view that lets you position a rectangle on screen to select the view and zoom level.

**paste** To insert the Clipboard contents into the active window with the Paste command.

**Pen color** A color on the Pen Color tab of the Pen palette, which can be applied to the outlines of a 2D or 3D object.

**Pen palette** A floating window that contains Pen colors, Pen Weights, Pen styles, and Arrowheads, which can be applied to lines and the outlines of objects.

**Pen style** A pattern of dashes made of solid lines and blank spaces, which can be applied to the outlines of 2D objects.

**Pen weight** The thickness of a line or the outline of a 2D object, which can be set from the Pen Weight tab of the Pen palette.

**PICT** A standard Mac OS graphics file format that supports vector objects, raster images, and formatted text.

**polar array** Radial copies of an object that are separated from one another by a specific distance and angle

**polygon** A closed 2D shape bounded by three or more lines or segments.

**polyline** An open shape drawn by setting endpoints of connected line segments with the Single Polyline or Double Polyline tools.

**preferences** Settings that users can adjust with the Preferences command, which control the operation of DENEBACAD's tools and other features.

**properties** Aspects of an object's identity, including size, coordinates, the tool used to create it, its attributes, name, Class, and status as a container, library object, or symbol. The Properties Manager lets you view and edit information about a selected objects's properties.

**Properties Manager** A palette that displays a variety of information about the properties of a selected object. You can display information about an object's attributes, position, name, creation points and class in the Properties Manager, and use it designate a selected object as a Container and Library Object.

**QuickTime** A Mac OS technology, installed as a system extension, used for recording and reproducing digital movies. QuickTime is used with DENEBACAD to create animated walkthroughs of renderings.

**QuickTime movie** A digital animation file created in DENEBACAD from a walk-through path of a rendering.

**QuickTime VR** A Mac OS technology used with DENEBACAD to create rendered scenes that can be viewed in 360-degree perspective.

**refresh** The process of redisplaying the screen contents after choosing the Refresh command in the Window menu.

**Relative View** A plane set at any angle as a window through which the user views a drawing. A Relative View lets you work on any surface in a drawing as if the surface were parallel to the computer screen.

**render** To create a picture of the 3D objects in a document in a Render mode window with the Wireframe, Hidden Line, or Solid commands.

**Render mode** The mode in which DENEBACAD operates when a Render window is active. Render mode is used to create Wireframe, Hidden Line, and Solid renderings of the 3D objects in a document.

**Render mode window** Any window showing your work in Render mode.

**reshape mode** A mode initiated by the Reshape command which allows you to reposition creation points of selected objects.

**Right view** One of six orthogonal views in Draft and Sculpt modes. Right view shows the workspace from the right. This is an elevation view.

**rotate** To change an object's orientation by moving it around a specified point with the Rotate command.

**scale-dependent** An object in a DENEBACAD document whose size measurement is relative to the scale set for the document. Nearly all objects, except text and dimension objects, are scale-dependent.

**scale-independent** An object, such as text or a dimension, which maintains a particular size measurement regardless of the scale set for the document.

**Sculpt mode** The mode in which DENEBACAD operates when a Sculpt window is active. Sculpt mode can be used to create and view 3D objects and to set up lighting and walkthroughs.

**Sculpt mode window** Any window showing your work in Sculpt mode.

**second plane** The plane you set second, regardless of position, when you define a set of extrusion planes.

**snap** To align to an object or a point on the grid through constraints available in the Snaps menu.

**snap points** The points on the outline of an object that can attract the pointer so it snaps to the exact position of the point.

**Snaps menu** A menu that appears in the drawing window when you press the pointer on an object and press the Control key. The commands in the menu let you constrain the pointer and drawing vector to align objects when you draw them.

**Solid rendering** A rendering created when you choose Solid in the Rendering menu. A Solid rendering shows objects as solids shaded with either their pen colors or surface materials.

**Spin axis** A line around which a 2D object is rotated to create a 3D object using the Spin extrusion method.

**Spin extrusion** A method of extending a 2D object into three dimensions by rotating the object around a Spin axis.

**Status bar** An area at the top of the screen that displays five pop-up menus from which you choose object or layer display options, a Full View or Sectioned View, 3D Planes, and Relative Views.

**Surface material** A pixel-based image that can be applied to 3D objects and displayed in a Solid rendering to give the appearance of real-world surfaces, such as wood, glass, and metal.

**Sweep extrusion** A method of extruding a 2D object into three dimensions by extending the object along a specified path.

**Sweep insertion point** A point on a Sweep section that follows the Sweep path when using the Sweep extrusion method to create a 3D object from a 2D Sweep section.

**Sweep section** A 2D object designated for extrusion into three dimensions using the Sweep extrusion method.

**Sweep path** A path along which a Sweep section can be extruded.

**Symbols** Objects that have been grouped and saved as Symbols in the Properties Manager. You can access Symbols on the Symbols tab in the Library palette. Symbols can contain 2D and 3D objects. Unlike Library objects, symbols are scale independent.

**text objects** An object created with a text tool that contains text. Text objects are scale independent.

**3D Axis** *See* Spin axis.

**3D objects** Objects defined in three dimensions. 3D objects have width, height, and depth. 3D objects can be created by extruding 2D objects in Draft mode, or by drawing objects in Sculpt mode. Only 3D objects can be rendered.

**3D Plane** A linear extrusion space defined by a pair of extrusion planes. In Draft and Sculpt modes, a 3D Plane can be selected only from the view in which it was defined.

**tile** To arrange multiple windows of an active document so two or more windows do not overlap.

**tools** Items used to create objects and set points of view in DENEBACAD. Tools are represented by icons in the toolbox.

**toolbars** Optional drawing and information elements that can be displayed on screen, including the Action buttons, Info bar, Status bar, and Help bar, using the Toolbars command.

**toolbox** A floating palette from which you can select tools to draft, model, and render in DENEBACAD.

**Top view** One of six orthogonal views in Draft and Sculpt modes. Top view shows the DENEBACAD workspace from the top. This is the plan view.

**2D objects** Objects defined in two dimensions. They have width and height, but no depth.

**units** The smallest named unit of measurement from which all calculations are based in the document.

**view** One of the six orthogonal views of the DENEBACAD workspace, designated Top, Bottom, Front, Back, Left, or Right.

**View Options mode** The mode that determines if 2D and 3D objects are displayed in Draft and Sculpt modes.

**vertex** A point of intersection of two vectors, or a point on a polygon.

**Vertical Focal Point** A point of view specified by the height of a camera and the vertical angle of its sight lines, which determines the view of 3D objects seen in a rendering.

**walkthrough** A type of animated rendering made by recording the views along a specified path through the 3D space of a DENEBACAD document.

**Wireframe rendering** A rendering created in the Render mode window when you choose the Wireframe command in the Rendering menu. A Wireframe rendering shows 3D objects as transparent outlines.

**zoom** To change the view of a drawing by changing the view magnification. Changing magnification changes only the appearance on screen, not the actual size of the objects in the drawing.

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