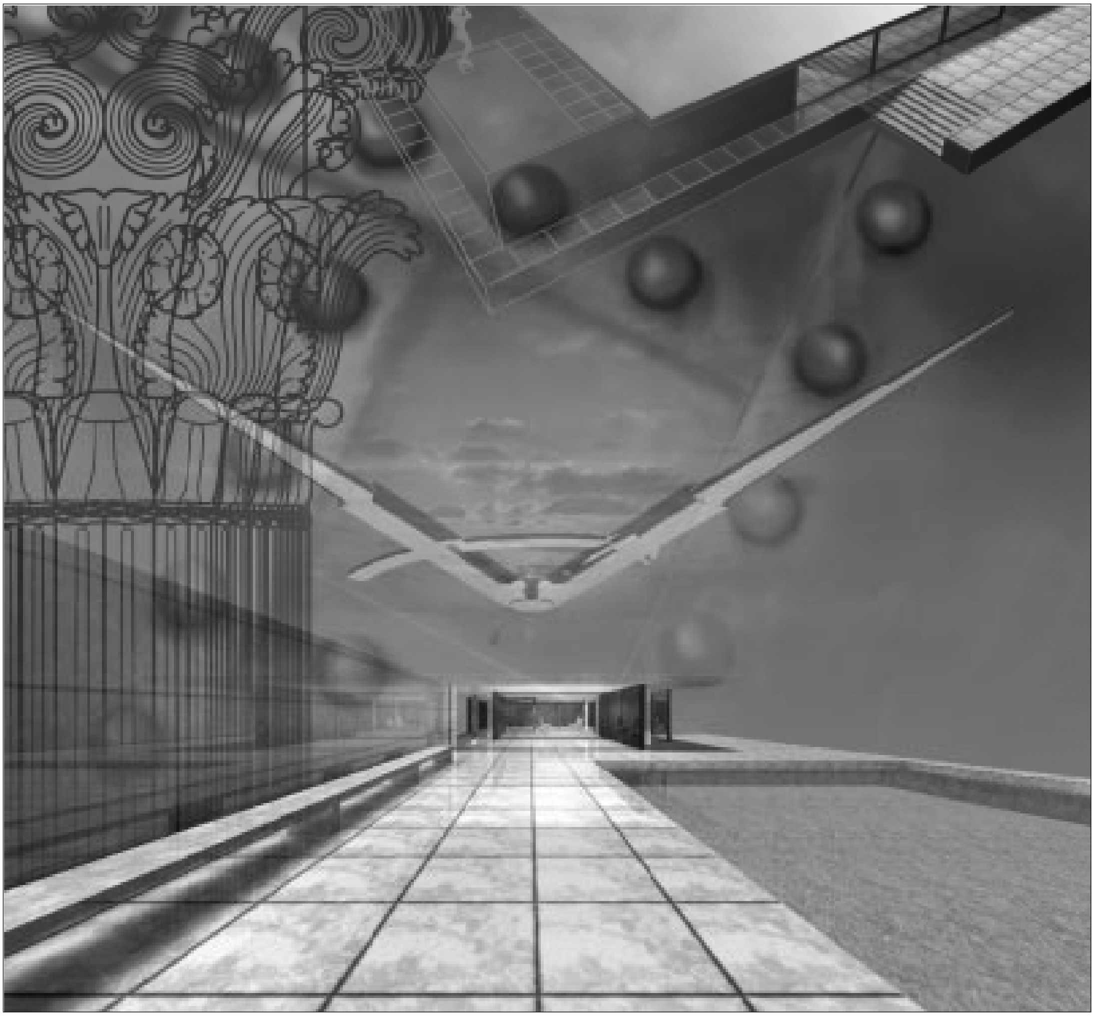
DenebaCAD™ 2



DENEBA

User's and Reference Guide
for Windows® 98/NT® and Mac™ OS

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DenebaCAD 2 User's & Reference Guide

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REFERENCE GUIDE

INTRODUCTION

Welcome to DENEBACAD, a new perspective for computer aided drafting and design for Windows 98/NT and Mac OS computers.

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 designs and production drawings, 3D models, and photo-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 lightning-fast rendering engine offers natural and artificial lighting, shading, reflectivity, transparency, and surface-mapping controls.

This chapter provides information on system requirements for running DENEBACAD, basic skills you should know, and how to use the documentation.

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:

Windows

- A Pentium-compatible processor
- 48 MB of RAM
- 40 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

- A PowerPC™ processor
- 32 MB of RAM
- 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

HOW TO USE THIS MANUAL

The DENEBACAD *User's & Reference Guide* describes the commands, tools and features of DENEBACAD.

- For an overview of the manual and to find topics of interest, see the Table of Contents.

- To familiarize yourself with DENEBACAD's basic concepts and tools, see the overview and tutorial exercises in the User's Guide section, which follows this introduction.

- To read in-depth descriptions of commands and tools, see the Reference section. This section describes every tool, beginning on page 77, and all menu commands, beginning on page 195.

- To find a specific command, tool or concept, refer to the alphabetical Index at the end of the book.

- To learn the definition of an unfamiliar term, refer to the Glossary, which begins on page 431.

Conventions used in the documentation

The DENEBACAD documentation uses certain terms and syntax when describing mouse actions, commands, and keyboard keys. In most cases, the program works exactly the same under Mac OS and Windows. The documentation points out functional differences when necessary.

Mac OS

Mac OS is the operating system for Macintosh and compatible computers. In the documentation, Mac OS refers to System 8 and later versions. When a procedure applies to Mac OS only, we write "(Mac OS)" in the instructions.

Windows

"Windows" refers to 32-bit Windows versions, including Windows 95, Windows 98, and Windows NT 4 and later versions. When a procedure applies to Windows only, we write "(Windows)" in the instructions.

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 (if it overlaps other windows). Windows that are not active appear gray or dimmed.

Keyboard keys and shortcuts

Standard names and abbreviations are used to refer to keyboard keys on each platform (your keyboard might use different labels).

Alt The Alternate key, usually labeled "Alt" on Windows keyboards.

Command This key labeled "Command," or displayed with a ⌘ on Mac OS keyboards.

Control or Ctrl The key labeled "Control" or "Ctrl" on Mac OS and Windows keyboards.

Option The key labeled "Option" or displayed with a ⌥ on Mac OS keyboards.

Shift The key labeled "Shift" or displayed with a ⇧ symbol.

INSTALLING DENEBACAD

A Windows setup or Mac OS installer program guides you throughout the DENEBACAD installation process. The applicable program transfers files from the compact disc to a folder that it creates on your hard disk. The program also creates a menu item on the Windows 98 or Windows NT Start menu.

Note: You must have a valid DENEBACAD serial number to run DENEBACAD.

To install DENEBACAD (Windows)

1. Insert the DENEBACAD compact disc into your CD-ROM drive.
2. Choose Start > Run.
3. In the Open field, type *d:\setup*, where *d* is the letter assigned to your CD_ROM drive.
4. Click OK and follow the instructions on your screen.

To install DENEBACAD (Mac OS)

1. Insert the DENEBACAD compact disc into your CD-ROM drive.
2. Double-click the Installer icon on the DENEBACAD CD-ROM.
3. Click Continue and follow the instructions on your screen.

LAUNCHING DENEBACAD

To start DENEBACAD, do either:

Windows Choose Start > Programs > DENEBACAD > DENEBACAD 2.

Mac OS Double-click the DENEBACAD icon in the DENEBACAD folder.

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

Note: If DENEBACAD can't locate the Materials folder, a directory dialog box appears asking you to locate the Materials folder. This might happen if you move 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.

ENDING A WORK SESSION

When you finish working and want to stop running DENEBACAD, use the Exit command (Windows) or the Quit command (Mac OS).

◆ **To stop running DENEBACAD:** Choose File > Exit or press Alt+F4 (Windows) or choose Quit in the File menu or press Command+Q (Mac OS).

Saving documents before quitting

If any unsaved documents are open when you choose the Exit/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 No (Windows) or Don’t Save (Mac OS).

◆ **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.

USER'S GUIDE

OVERVIEW AND TUTORIALS

CONCEPTS OVERVIEW 1

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

DENEBACAD is programmed to increase your productivity. In DENEBACAD, you can create and print 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 extract plans, sections, and elevations for working drawings from 3D models. 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 sheets.

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

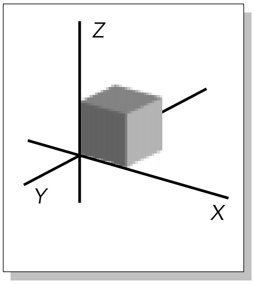
Fundamental to the operation of DENEBACAD is the use of Relative Views. In Sculpt mode, a Relative View lets you change the coordinate system to create a new drawing plane aligned with any surface. This allows precise drawing control in any environment and situation.

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.

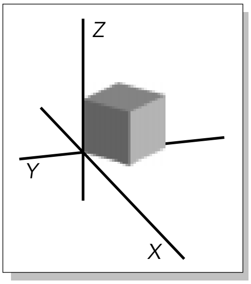
For example, you can shift the Relative View so that your point of reference — the computer screen — is parallel to the slope of a roof. Then, drawing a skylight in the roof is as simple as drawing a rectangle. You can work in 3D the same way you would work in a 2D drawing.

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. 1 Default and relative views

EXTRUSION METHODS FOR 3D MODELING

To create 3D objects in DENEBACAD, you can extrude 2D shapes using one of three extrusion methods.

Linear Extrudes an object by extending it along a single axis between two defined planes.

Spin Extrudes an object by revolving it around a defined axis.

Sweep Extrudes an object by extending it along an open or closed path.

DENEBACAD creates 3D objects automatically when you draw in Sculpt mode. You can also select a 2D object in Draft mode and extrude it.

EXTRUSION PLANES

Extrusion planes are used in Draft and Sculpt modes to control Linear extrusion of objects into 3D. For example, with extrusion planes at ground level and 10 feet, you can draw a double polyline to create 3D walls 10 feet high. If you set extrusion planes at 4 meters and 7 meters, and draw a circle, you generate a cylinder 3 meters high placed 4 meters above the floor.

Extrusion planes can create flat objects contained in them. Select End Cap format, draw a circle, and the result is a disk floating in space.

You define extrusion planes for a view from either of two other views. For example, to set the height of objects drawn in Top view, you set extrusion planes in Front view or Left view.

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

If you are using Top or Bottom view and define extrusion planes from Front or Back, the planes can be angled with respect to the horizontal plane and perpendicular to the front plane (X-Z plane). If you define extrusion planes from Left or Right view, the planes can be angled relative to the horizontal plane and are perpendicular to the side plane (Y-Z plane). These planes will determine the upper and lower limits of extruded objects.

If you are using Front or Back view and define extrusion planes Top or Bottom view, the planes can be angled with respect to the frontal plane and are perpendicular to the horizontal plane. These extrusion planes determine the front and back limits of extruded objects.

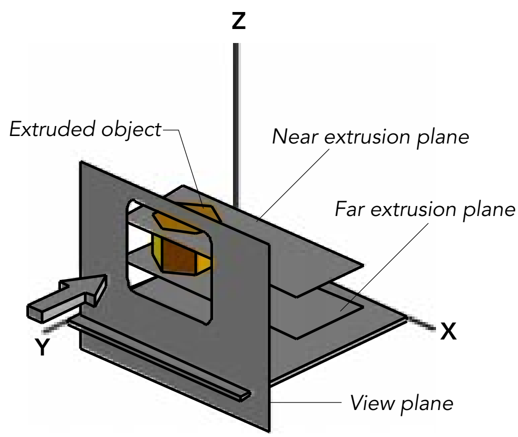


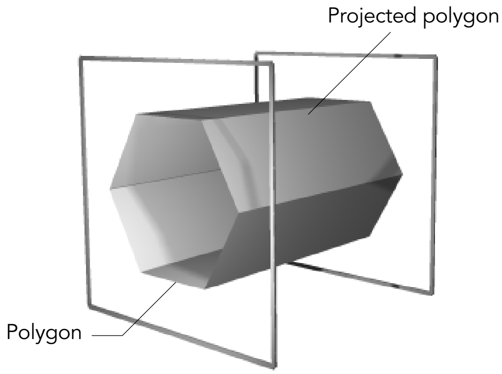
Fig. 2 Orientation of extrusion planes

If you are using Left or Right view and define extrusion planes from Top or Bottom view, the planes can be angled relative to the lateral plane and are perpendicular to the horizontal plane. These planes determine the left and right limits of extruded objects.

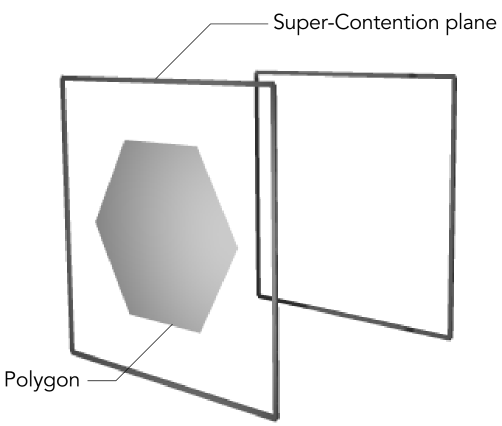
EXTRUSION FORMATS

DENEBACAD lets you generate several types of extrusions without altering the extrusion planes.

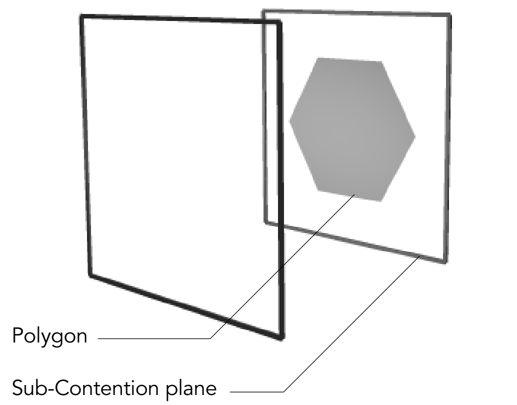
Projected Extruded objects are projected between the two extrusion planes. This is the result of selecting Sides extrusion format.



Super-contention Objects are projected as contained in the extrusion plane nearer to the observer. This is the result of selecting Front Cap extrusion format.



Sub-contention Objects are projected as contained in the extrusion plane farther from the observer. This is the result of selecting Back Cap extrusion format.



Extrusion formats are additive, so you generate solid objects by activating all three extrusion formats. The illustration below shows all possible combinations of extrusion formats.

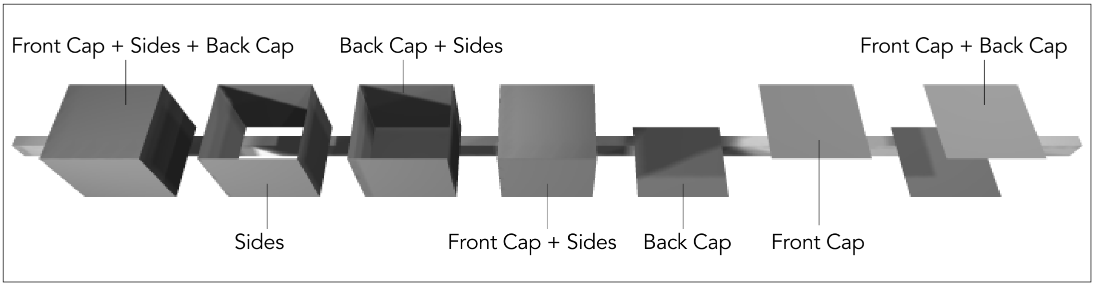


Fig. 3 Extrusion formats

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 7*), including the display of toolbars, information fields, and multiple windows.

By default, DENEBACAD displays 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 describes 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 D

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 P

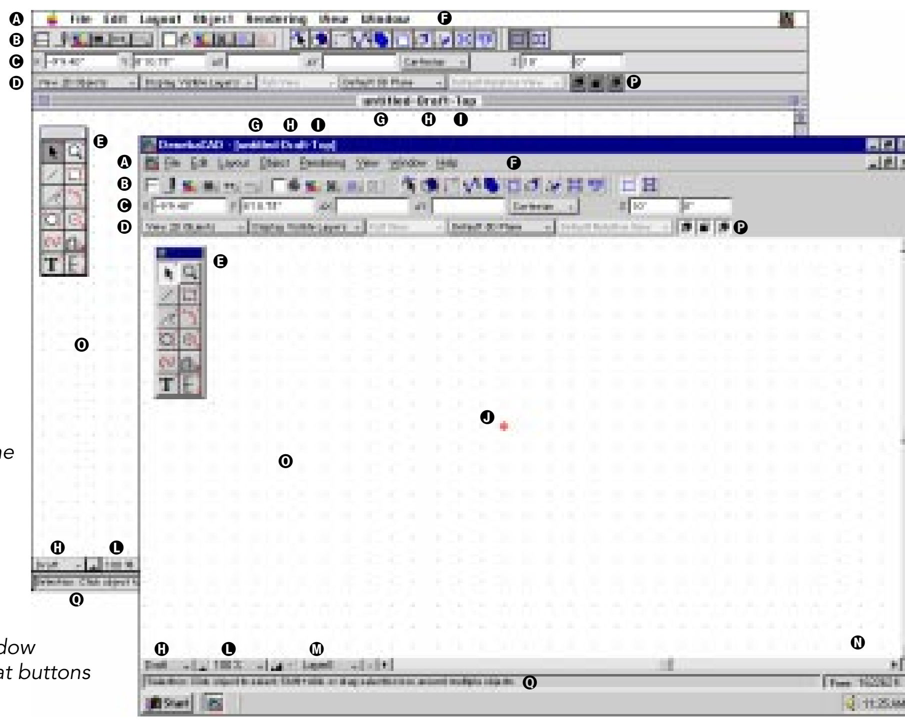
The Extrusion Format buttons set constraints for extruding objects as solids, or as sides, faces, or any combination.

Toolbox E

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

Help bar Q

The Help bar 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.



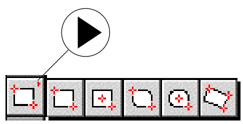
A *Menu bar*
B *Attributes bar*
C *Info bar*
D *Status bar*
E *Toolbox*
F *Action buttons*
G *Document name*
H *Mode*
I *View*
J *Centerpoint*
K *Help bar*
L *Zoom*
M *Layers*
N *Free memory*
O *Document window*
P *Extrusion format buttons*
Q *Help bar*

Fig. 7 Main elements of the DENEBACAD interface

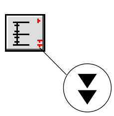
DRAFT, SCULPT, AND RENDER TOOLS

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.



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, the toolbox contains tools for 2D drafting, dimensioning, and placing text.
- In Sculpt, 3D drawing tools, the Light Source tools, and the Camera Path tool appear in the toolbox.
- In Render, 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.

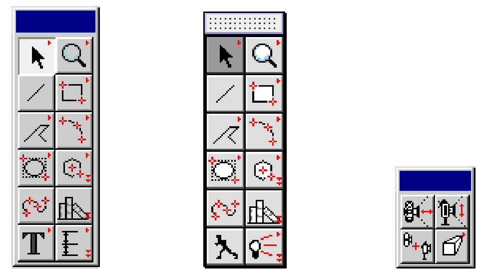
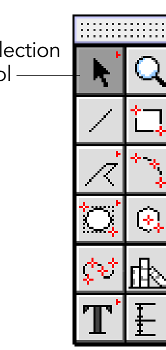


Fig. 8 Tools in Draft, Sculpt, and Render modes

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.

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.

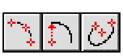
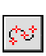
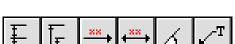
With most drawing tools, you can click the mouse to set an object's creation points. With the Circle Radius tool, for example, you click to set the center of the circle and click again to set the circle's radius. Or, you can drag the mouse from the first point to the second one.

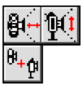
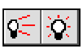
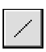
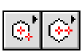
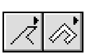
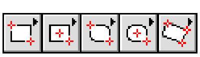
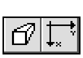
When the Info bar is displayed, you can enter numeric data when creating objects. Press the Tab key to enter X, Y, or Z coordinates (in the Cartesian coordinate system); click the check mark button or press Return to set the first point. Or, press Enter and then enter relative coordinates for subsequent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

Tools are listed in alphabetical order according to the toolbars in which they are grouped in the toolbox.

Toolbar		Tool names	Functions	Modes
	Arc	Arc Radius Arc 3 Points 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 Baseline Dimension Leader Dimension Constrained Dimension Angle Dimension 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 Ellipse Center to Corner Circle Radius Circle 3 Points 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 Vertical Focal Point 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 Omnidirectional Light	Place light sources for illumination in solid renderings	Sculpt
	Line	Line	Draw lines and planes	Draft, Sculpt
	Polygon	Polygon Vertex Polygon Midpoint	Draw closed, multiple-sided objects from the center to a side or vertex	Draft, Sculpt
	Polyline	Single Polyline Double Polyline	Draw open polygons with or without offset edges	Draft, Sculpt
	Rectangle	Rectangle Diagonal Rectangle Center to Corner Rounded Rectangle Diagonal Rounded Rectangle Center to Corner 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 Isometric Render	Select standard or isometric perspective for renderings	Render
	Selection	Selection Point Selection	Select objects and text Select points within objects	Draft, Sculpt
	Text	Text Text Rotated	Place text with horizontal or angled baselines	Draft
	View	Zoom 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.

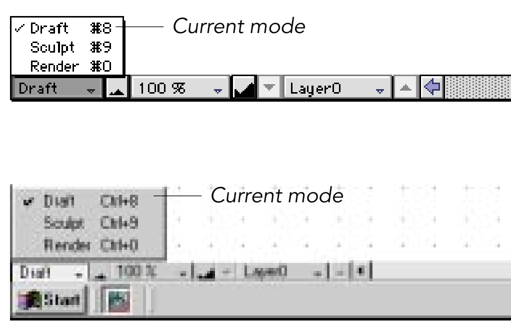
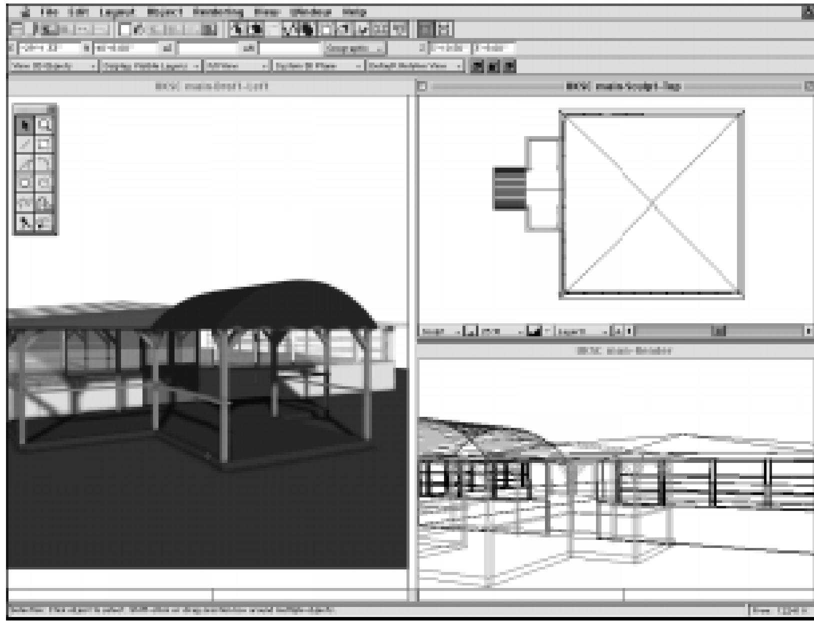


Fig. 9 Mode pop-up menu

When you use multiple windows, DENEBACAD arranges the windows so they don't overlap.

Choose the Tile command or press Ctrl+T (Windows) or Cmd+T (Mac OS) to arrange windows neatly.



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.

Attributes icons let you set Pen and Fill properties

Action buttons are shortcuts for common commands

Snap buttons set snap options

Text controls replace the Attributes icons when you use text tools

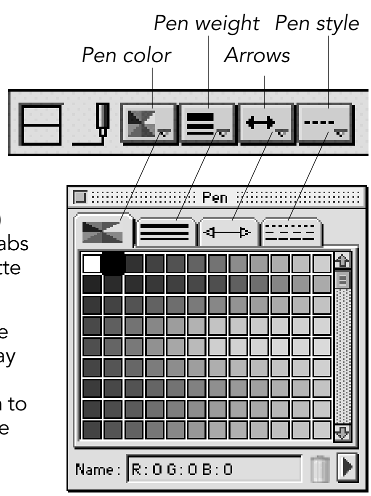
Fig. 10 Attributes bar and Text toolbar

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)

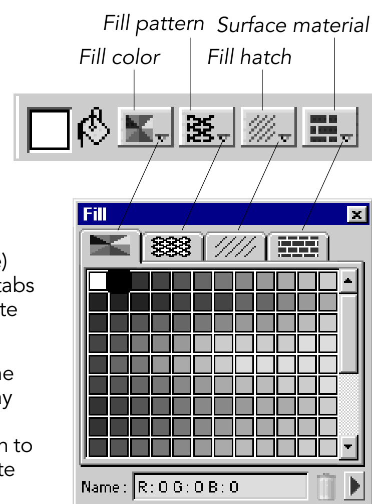
You can tear the Pen palette away from any Pen attribute button to keep the palette open

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 tools and commands in DENEBACAD. The first section introduces commands you use to set up documents, drawing scales and drawing grids. The following sections describe coordinate systems, and how to draw objects. This chapter also introduces the Snaps menu, a versatile constraint system that helps you control tools and object operations in DENEBACAD.

SETTING UP THE DRAWING ENVIRONMENT

To start DENEBACAD, do the following:

- Windows: Choose Start > Programs > DENEBACAD > DENEBACAD 2
- Mac OS: Double-click the DENEBACAD icon in the DENEBACAD folder.

Once DENEBACAD launches, it creates a blank document and displays a Draft mode window, with *"Untitled-Draft-Top"* in the window's title bar.

To be sure that DENEBACAD is set up in the best way for exploring the program, follow the instructions in the next section to customize the drawing environment.

Displaying toolbars

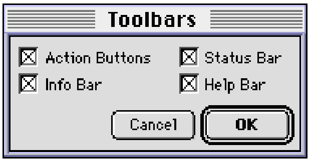
By default, DENEBACAD presents an uncluttered drawing interface. When you start the program, the toolbox appears on the left. The Attributes bar and the Action buttons appear in a row under the menus.

Several other toolbars are available. 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 tools, commands, and procedures.

When you begin using DENEBACAD, it's a good idea to display all the toolbars. Once you become familiar with the shortcuts they provide, you can decide whether to display or hide 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 has a check mark, and then click OK.

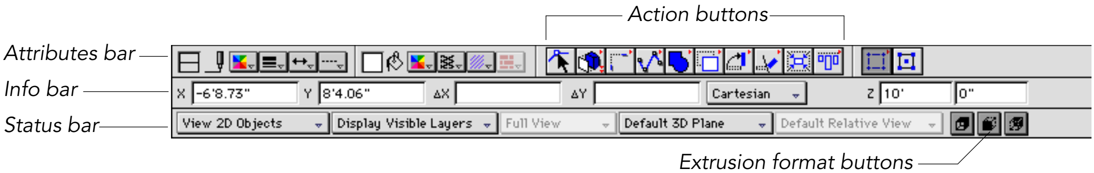


Fig. 11 Toolbars

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. 11*).
3. Click OK to close the Preferences dialog box.

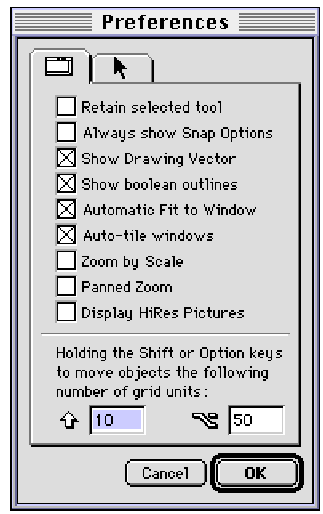


Fig. 11 General tab, Preferences dialog box

DRAWING SCALE AND GRID SETTINGS

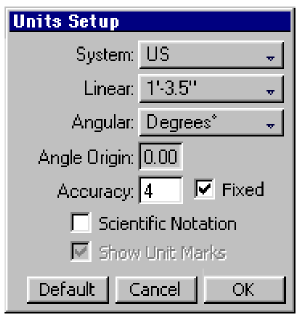
Before you begin any new project, you'll want to check the settings for measurement units, drawing scale, and the drawing grid.

Once you create a document with the settings you want to use often, save the empty document to use as a template. When you start a project, open the template document and use the Save As command to save a new document with the project name.

To set up measurement units

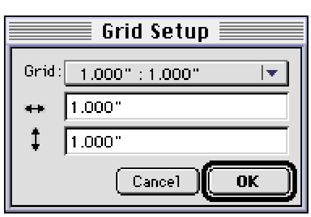
1. Choose Units Setup in the Layout menu. The Units Setup dialog box appears.
2. Set the dialog box to match the following illustration and click OK.

   ▲ 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: This grid setting is appropriate for drawing the doors described in the following tutorial lesson. For larger-scale drawings, you can set a larger grid spacing.

COORDINATE SYSTEMS

You can use several coordinate systems in DENEBACAD.

Coordinate Systems menu in the Info bar

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 are based on the current coordinate system. Also, you work in the current coordinate system if you type coordinates in the Info bar to create objects.

You can use six coordinate systems in DENEBACAD. The coordinate system are briefly described here. You can learn more by selecting each coordinate system in the Info bar and looking at how the text boxes in the Info bar change, and the coordinates that appear when you 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 set the location of a point, you can type its X,Y coordinates in the Info bar. The coordinates can be absolute distances along the X and Y axes from the Centerpoint, or relative coordinates, 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 (ΔD) and its angle (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.

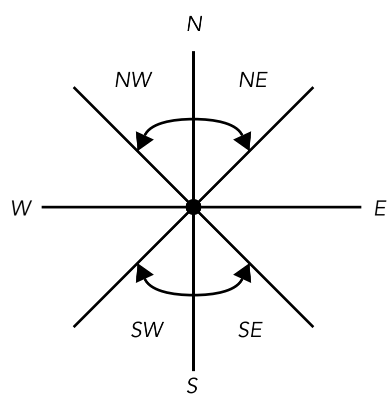


Fig. 12 Bearing coordinate system

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.

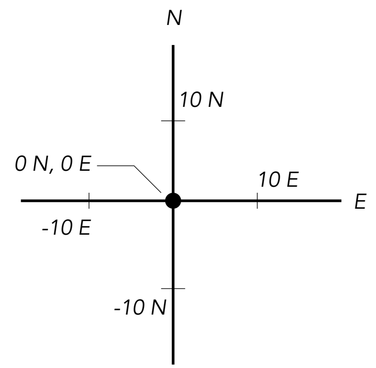


Fig. 13 Geographic coordinate system

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.

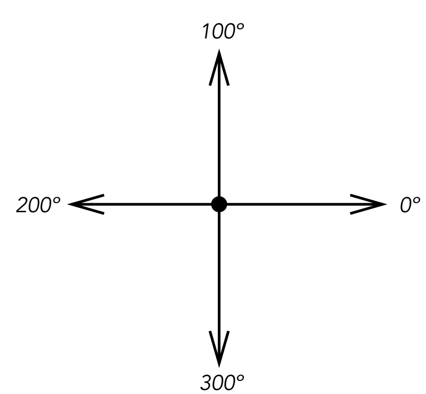


Fig. 14 Gradient coordinate system

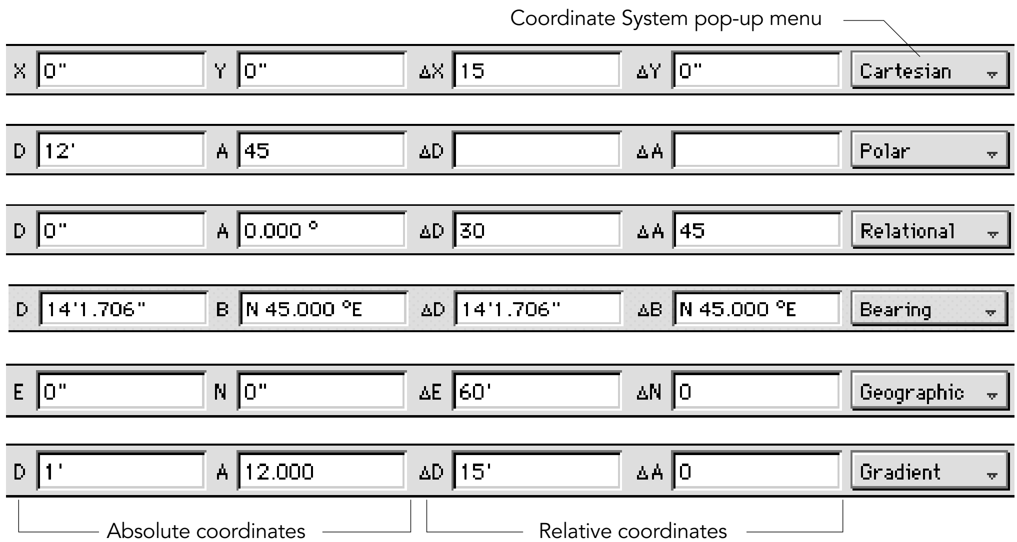


Fig. 15 Info bar data and coordinate systems

BASICS OF DRAWING OBJECTS

Objects are geometric shapes which form the basis of 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.

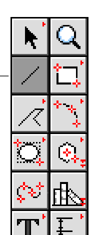
Using existing objects You can also create objects by using other objects. To do this, select an existing object and choose Duplicate in the Edit menu, or Mirror, Rotate, Copy, Array, or Combine in the Object menu.

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 two options:

Line tool



- 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 3 Points 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 click to set an object's first point, you can type coordinates or distances to set subsequent points.

To draw a line by clicking endpoints

1. Select the Line tool from the toolbox.
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

When you want to specify an object's location and size numerically, you can type coordinates and other data in the Info bar to set the object's creation points.

You can see the creation point values when you draw using the mouse. Notice that the numbers in the Info bar change as you move the pointer. When using the Info bar to create objects, you type in the same values that would appear if you were moving the mouse and clicking to set creation points.

In the Cartesian coordinate system, for example, the X and Y fields show the pointer's X,Y coordinates, relative to the Centerpoint at 0,0. After you set an object's first creation point, the ΔX and ΔY fields show the pointer's coordinates relative to 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.

To draw a line by entering coordinates

You 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, and select the Line tool in the toolbox.
2. Press Tab to highlight the X box in the Info bar and type the X coordinate of the first endpoint. This is the distance horizontally from the Centerpoint.
3. Press Tab to highlight the Y box. Type the Y coordinate of the first endpoint. This is the distance vertically from the Centerpoint.
4. Press Return or Enter to set the first endpoint at the X,Y coordinates. Then, press Tab to highlight the ΔX box in the Info bar

   ▲ *Note:* On Mac OS, pressing Enter (rather than Return) sets the first point and highlights the next text box in one step. In Windows, this is the same as pressing Enter on the numeric keypad on some keyboards.

5. Type the line's length on the X axis (the horizontal distance) in the ΔX box.
6. Press Tab to highlight the ΔY box, and type the line length on the Y axis (the vertical distance).
7. Press Return or Enter to complete the line. The line appears with the current attributes and is selected.

X,Y values set the first point relative to the Centerpoint.

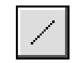
ΔX,ΔY values set the second point relative to the first point

Fig. 16 Creation point coordinates

To draw a horizontal line from the Centerpoint

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

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



Line tool

2. Type 0 in the X field. Press Tab, type 0 in the Y field, and then press Enter.
3. Press Tab, type 50' in the ΔX box, press Tab, type 0 in the ΔY box, and then press Enter.

MODIFYING PROPERTIES OF OBJECTS

The Properties Manager lets you view and edit the properties of objects. The Properties Manager is a floating palette that you can keep open while you work.

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

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

The Properties Manager has four tabs. By default, the Coordinates tab is in front. This tab displays information and position data for the selected object.

- The Object menu contains a list of objects in the current mode.
- The Name box displays the object's name, which you can edit.
- The pop-up menus under the name box let you use relative or absolute position data, and select the coordinate system.
- Under the pop-up menus, text boxes show data on the current creation point.
- If Show Vector 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.

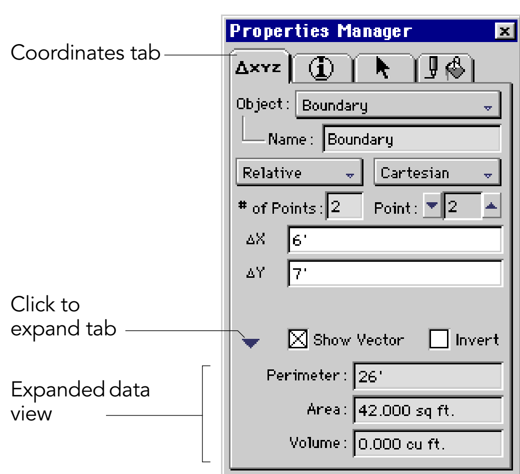


Fig. 17 Coordinates tab

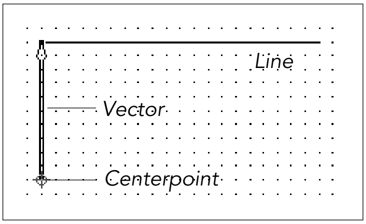
To edit a line in the Properties Manager

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

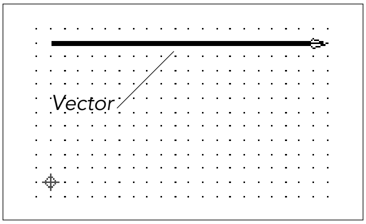
1. With a line selected, choose Properties Manager in the Window menu. The Properties Manager appears.
2. Click the Coordinates tab to bring it to the front, if necessary.

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 distance from the Centerpoint



When you select point 2, the Properties Manager reports the length from the first point.

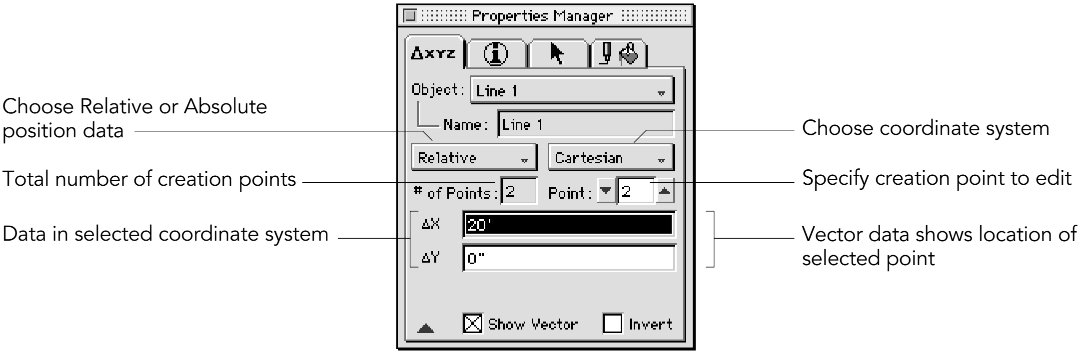


Fig. 18 With a line selected, the Coordinates tab displays data about the line's creation points

USING THE SNAPS MENU TO ALIGN OBJECTS

The Snaps menu helps you precisely align objects as you draw by constraining the pointer and the drawing vector, so you don't have to specify exact angles or distances.

The Snaps menu pops up in the drawing area when you press the right mouse button (Windows) or the Control key and mouse button (Mac OS).

To use the Snaps menu, you point to an object as a reference for the constraint, and then display the Snaps menu. In this section, the object you point to when using the Snaps menu is called the *reference* object.

When drawing objects, you can use Snaps constraints in combination, and use different constraints to set each creation point.

Constraints can be used for a variety of operations in addition to drawing. You can choose Snaps constrains for positioning extrusion planes, clipping planes, and relative view planes. For example, with the Define Frontal 3D Plane command, choose Included in the Snaps menu to align the plane with the slope of a roof gable.

Note: If you want the Snaps menu appear whenever you draw an object (without using the right button or Control key), select "Always Show Snap Options" on the General tab in the Preferences dialog box.

SNAPS MENU OPTIONS

The Snaps menu contains the following options (a keyboard shortcut, if available, appears after the option name).

Free Mouse

This option frees the pointer and drawing vector from any constraint that is in effect. Free Mouse restores the pointer to free movement.

Click

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

Snap Points (Spacebar)

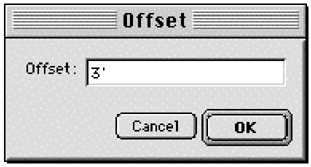
This option displays the Snap points on the reference object. When snap points are displayed, the vector snaps to the closest point. Refer to "# of Snap Pts." on page 228 for more information.

Included (I)

The Included option constrains the pointer or vector to the path of the reference object. If you move the pointer beyond the reference object, the constraint continues as if the object or segment extended through the entire drawing.

Parallel (P)

The Parallel option constrains the vector to a path parallel to the reference object. If you choose Parallel before setting an object's first creation point, a line appears at the pointer, parallel to the reference object. Move the pointer and then click to set an offset from the object. The Offset dialog box appears; confirm the offset value 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 option constrains the pointer or drawing vector to a path that is perpendicular to the segments of the reference object.

Tangent (T)

The Tangent option constrains the drawing vector so it remains tangent to a reference circle or circular arc. After setting the first endpoint of a line, for example, choose Tangent so the vector snaps to the perimeter of the reference circle. This constraint is available only when the pointer is on a circle or circular arc reference object.

Direction (D)

The Direction option constrains the drawing vector to the angular direction that you establish before choosing this option. For example, as you move the pointer away from a creation point of a polyline, you establish a direction; choose Direction to constrain the vector the established angle.

Center (C)

This option snaps the pointer and drawing vector to the center of the reference object or the segment you point to.

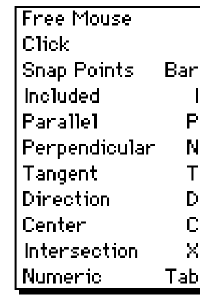
Intersection (X)

This option aligns the pointer and drawing vector to the intersection of two separate objects, such as the point where two lines meet or cross.

Numeric (Tab)

Choosing the Numeric option is the same as pressing Tab during an operation, because it lets you type in the Info bar. For example, you can click to set an object's first creation point, choose Numeric, and then type coordinates to complete the object.

In the Snaps menu, the available constraints depend on the position of the pointer and the operation in progress.



DRAWING WITH SNAP CONSTRAINTS

The powerful Snaps menu options help you draw by constraining the drawing vector. As you draw, Snaps options can keep the pointer parallel, perpendicular, or tangent to an object, or maintain a constant angle. You can snap the pointer to an object's center, sides, or snap points, or to the intersection of separate objects.

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.

To use the Snaps menu

1. Select a drawing tool.
2. To display the Snaps menu:
   - ▲ Mac OS: Press the Control key and mouse button as you point to a reference object.

▲ Windows: Press the right (secondary) mouse button as you point to a reference object.

3. In the Snaps menu, choose an option.

▲ The available constraints depend on the reference object you point to in the drawing.

▲ To release an active constraint, choose Free Mouse. You can then choose another constraint.

4. Click or drag (depending on the tool and operation) to set a creation point. The active constraint controls the drawing vector and pointer movement.

To snap to a line's endpoint

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

1. Draw a line with the Line tool. Click away from the line to deselect it.

2. Select the Line tool again. Point to the first line, display the Snaps menu (see "To use the Snaps menu," page 25) and choose Snap Points. The line's snap points appear as small handles at the center and each end.

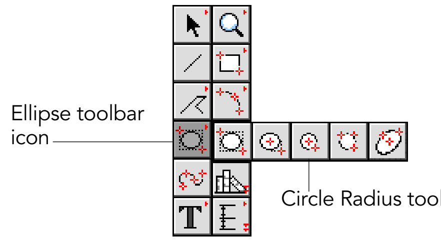
Snap points

3. Click the snap point at one end of the line. The first point of the new line snaps to the point you clicked.

4. Move the mouse and click anywhere to set the second endpoint of the new line.

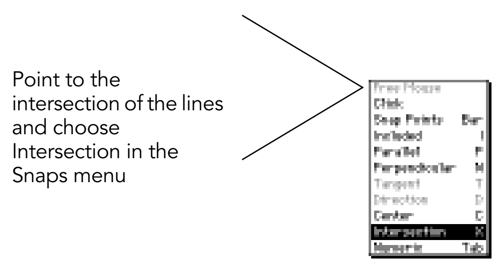
To draw using two Snaps constraints

1. Select the Circle Radius tool. (If the tool isn't visible, press the toolbar icon and then select the tool from the toolbar.)



2. Point to the intersection of the lines drawn in the previous procedure. Display the Snaps menu (see "To use the Snaps menu," page 25) and choose Intersection. (If Intersection is dimmed, be sure the pointer is on the intersection of the two lines.)

3. Point to either line, display the Snaps menu again and choose Center. The circle snaps to the center of the line.

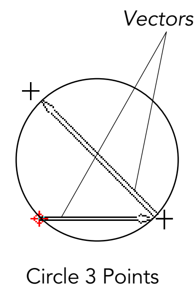


THE DRAWING VECTOR AND SNAPS

The drawing vector is a hollow 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.



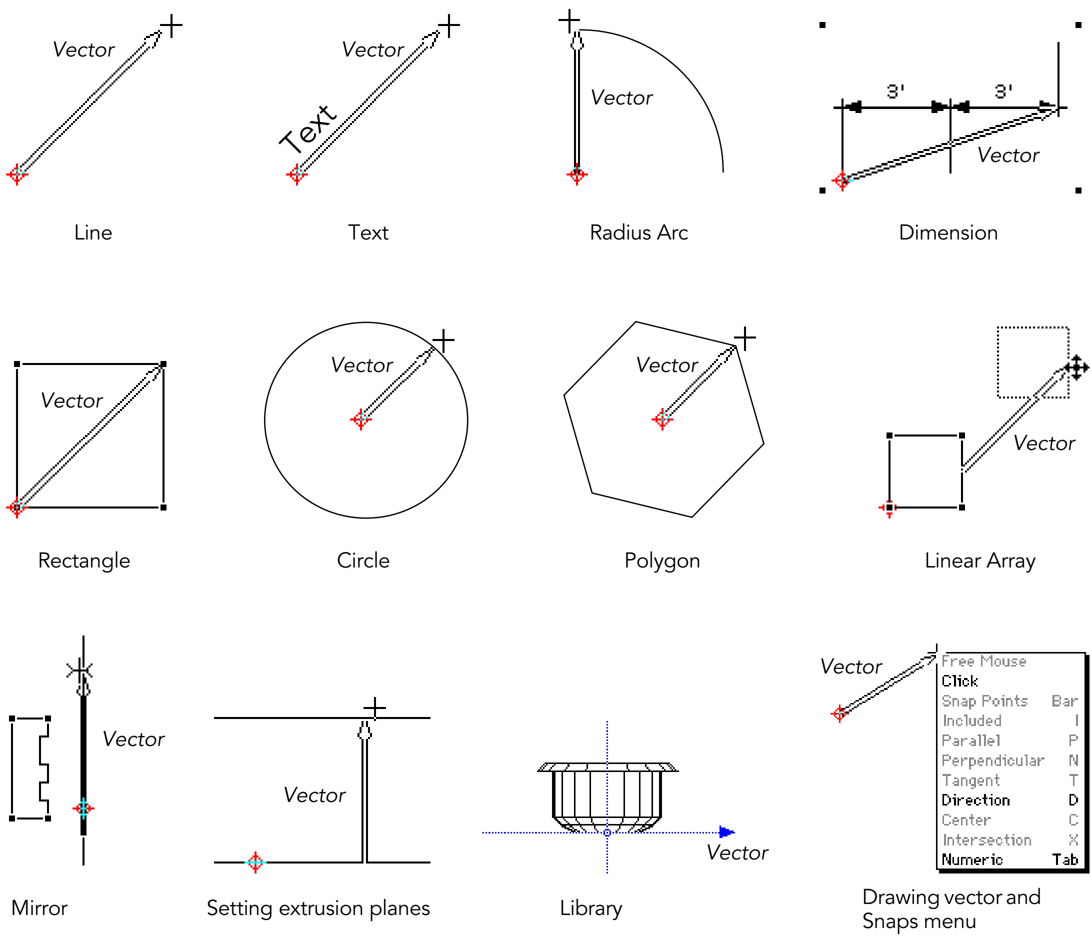


Fig. 19 The drawing vector in common operations

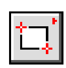
Clearing the drawing window

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 Ctrl+A (Windows) or Cmd+A (Mac OS) to select all objects in the drawing. Then press the Delete or Backspace key to remove the selected objects.

MAKING COPIES OF OBJECTS

Another method you can use to create objects is to duplicate existing objects. Several commands let you duplicate an object and rotate, flip, or reposition the copy in one operation.

To duplicate a rectangle



Rectangle Diagonal tool

1. Select the Rectangle Diagonal tool and drag the pointer to draw a rectangle.
2. With the rectangle selected (selection handles should be visible), select Duplicate in the Edit menu.
3. A copy of the rectangle appears in front of the original. Point to one edge of the rectangle and drag it away. You can see the original rectangle in position behind the copy.

DUPLICATING BY PASTING

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

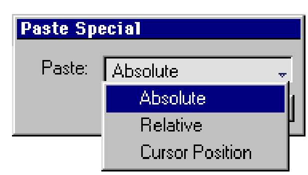
To position a pasted object with Paste Special

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.

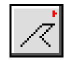


4. Select an option and click OK to paste the object. If you chose Cursor Position, click in the drawing to set the upper-left corner of the object's bounding box.

COPYING AND MIRRORING OBJECTS

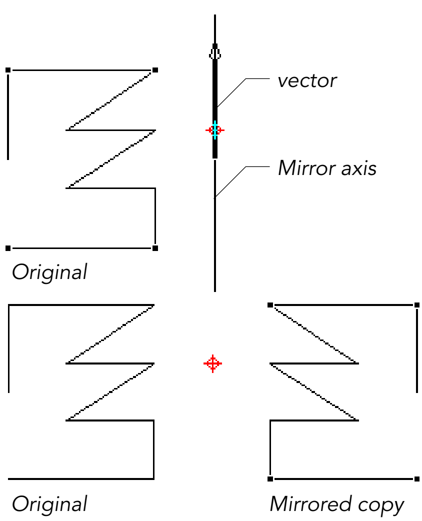
Mirroring creates a mirror-image copy of a selected object.

To copy an object by mirroring

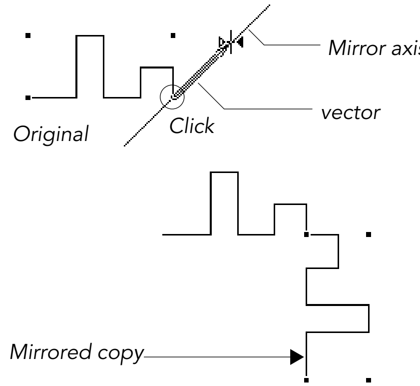


Single Polyline tool

1. Select the Single Polyline tool in the Polyline toolbar.
2. Click at several points, then double-click the last point to draw a polyline object. The polyline is selected.
3. Choose Position > Mirror a Copy in the Object menu. A vertical line, the mirror axis, appears at the pointer.
4. Move the pointer to position the axis horizontally, then click to set the position of the axis over which the object will be reflected.
5. The drawing vector appears. Move the pointer and then click to set angle of the reflection axis. A reflected copy of the original object appears. The new object is selected.



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

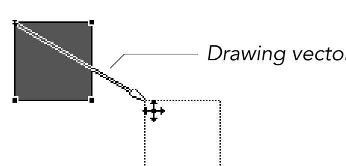


Setting the mirror axis at a 45-degree angle results in a copy rotated 90 degrees from the original

COPYING AND MOVING OBJECTS

The Move a Copy command lets you copy an object and set the position of the copy relative to the original.

1. Draw a rectangle or select an existing object.
2. Choose Position > Move a Copy in the Object menu.
3. Click to set a reference point for moving the copy. The drawing vector shows extends from the reference point. An outline shows where the copy will appear.
4. Click to set the position of the copied object.



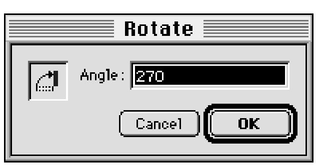
The drawing vector shows the relative movement of the reference point

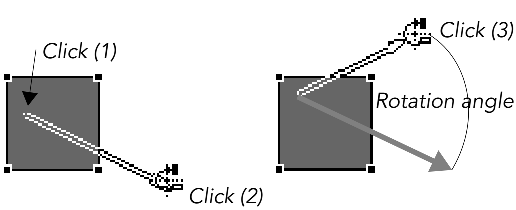
COPYING AND ROTATING OBJECTS

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 object.
2. Choose Position > Rotate a Copy in the Object menu.
3. Click in the drawing to set a rotation point. Move the pointer and the drawing vector extends from the rotation point. Click again to set a reference point. Click a third time to input the rotation angle.
4. In the dialog box, 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. 20 Using the Rotate a Copy command

DUPLICATING OBJECTS IN ARRAYS

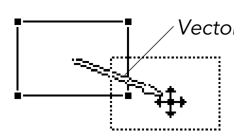
You can duplicate and create new objects using the Linear Array and Polar Array commands. These commands let you create copies and position the copies a set distance from one another.

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 copying the objects by offset distance or inclusively.

To use the Linear Array command

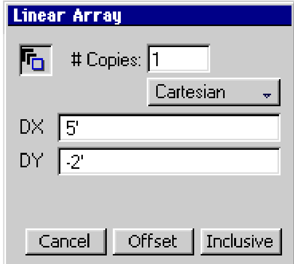
1. Select an object to duplicate and Choose Position > Linear Array in the Object menu.
2. The pointer becomes a four-way arrow. Click to set the reference point and move to set the array distance and angle, as shown by the drawing vector. You can also drag from one point to another.



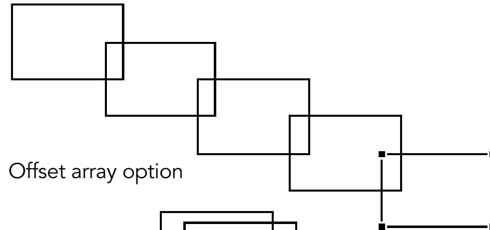
Setting the array distance

3. In the Linear Array dialog box, type the number of copies in the “# Copies” text box.

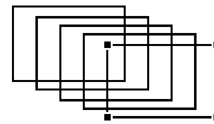
▲ You can type new coordinate values in the text boxes and select a coordinate system from the pop-up menu.



4. Click a button to choose the Array method. Click Offset to use the drawing vector as an offset distance and direction. Click Inclusive to distribute the copies in the distance specified by the drawing vector.



Offset array option



Inclusive array option

COMBINING OBJECTS WITH BOOLEAN OPERATIONS

Boolean operations let you combine objects to form complex 2D and 3D shapes. You can choose from six combine operations: Unite, Intersect, Punch from Front, Punch from Front and Trim, Punch from Back, Punch from Back and Trim. The resulting objects are called combined, or “Boolean” objects.

UNITING OBJECTS

The Unite command unifies selected objects into one new object.

To combine objects with Unite

1. Draw several objects, such as circles, squares, and polygons.

2. Drag the objects into position so that they overlap each other.

3. You can apply a fill color to one or more of the objects by selecting them and choosing colors from the Fill Color icon in the Attributes bar.

Fill color icon

Note: Fill colors apply to 2D objects only.

4. Press Ctrl+A (Windows) or Cmd+A (Mac OS) to select all the objects.

5. Choose Combine > Unite in the Object menu. The selected objects are combined, with the resulting single object made from the union of the original objects. The combined object takes on the attributes of the original backmost object.

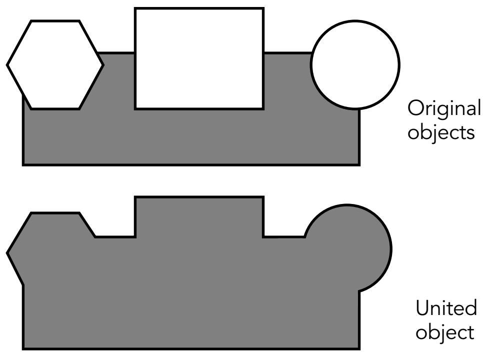


Fig. 21 Objects combined with Unite

RESHAPING OBJECTS

After you combine objects, you can reshape the resulting object. In reshape mode, you can edit the original shapes that make up a combined object. When you finish editing, DENEBACAD reapplies the command that created the combined object.

To reshape a Boolean object

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

Reshape button

▲ Because the preference “Show Boolean outlines” is selected, the original objects have black outlines, and the Boolean object has a red outline.

3. Drag one of the component objects to a new position (still overlapping the other objects).
4. Click outside of the objects. The Boolean object is reshaped based on the change to its component objects.

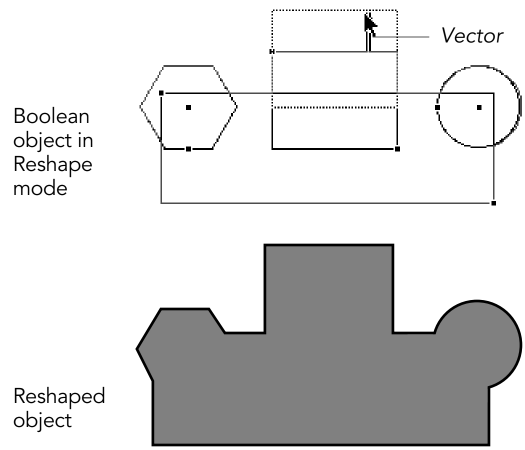


Fig. 22 Reshaping a Boolean object

ADJUSTING THE DRAWING VIEW

You can adjust your view of drawings by zooming in to magnify an area, or zooming out to view a larger area.

Using the Zoom tool

You can use the Zoom tool to change the magnification level of your drawing in Draft and Sculpt windows.

Zoom tool

- To zoom in, use the Zoom tool to drag a box around an area you want to magnify. DENEBACAD zooms in until the area fills the drawing window.
- To Zoom out, press Shift and drag the Zoom tool in the drawing.

Using the Zoom bar

You can also use the Zoom bar to change the magnification level of your drawing. The Zoom bar is located at the bottom of the active window.

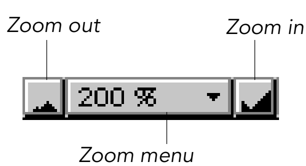


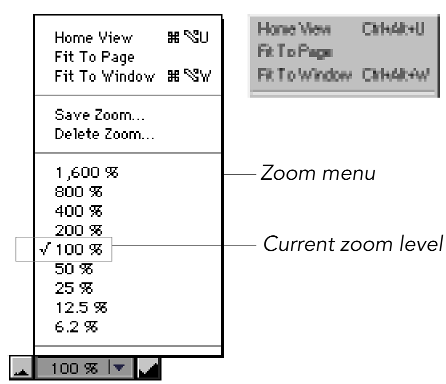
Fig. 23 The Zoom bar

Press the center of the bar to use the Zoom pop-up menu.

- Choose Fit to Window to zoom so the entire drawing fits in the window.
- Choose Home View to zoom to 100% with the Centerpoint centered in the window.
- Choose Fit to Page to zoom so the printable page area fits in the window.
- You can also choose from several preset zoom levels.

Zooming quickly

You can quickly zoom in or out by clicking the Zoom In and Zoom Out buttons.



USING LIBRARY OBJECTS 1

Library objects and *library symbols* are special objects you can use for common items in your drawings, such as doors, windows, roof sections, walls, and wiring symbols. Creating a library object is as simple as drawing and saving it. You use the Library palette to select and insert library objects into DENEBACAD drawings.

This chapter shows you the basic procedures for creating 2D and 3D library objects. By following this tutorial, you'll also learn how to use some additional DENEBACAD tools and commands.

CREATING A 2D LIBRARY OBJECT

There are two types of library items: *symbols* and *objects*.

- Library *symbols* are drawn in 2D, because they appear only in Draft (2D) drawings.
- Library *objects* can be created in 2D, 3D, or both 2D and 3D. You can use combined 2D/3D library objects in both Draft (2D) and Sculpt (3D) drawings.

To create a combined 2D/3D library object, you usually draw the 2D object first. In the first part of this tutorial, you will learn how to draw a door unit in 2D plan and elevation.

The second part of the tutorial shows you how to extrude the 2D door unit into a 3D unit, without additional drawing. The result is a 2D/3D library object, which you can insert into Draft (2D) and Sculpt (3D) drawings.

DRAWING THE DOOR UNIT PLAN

Follow the steps in this section to create a door unit that includes a door, frame, and door jambs. You will work first in Draft mode and Top view to draw the 2D door plan.

To begin the tutorial, choose File > Open. In the directory dialog box, select the file named "Library.start" in the Tutorial folder, and click OK.

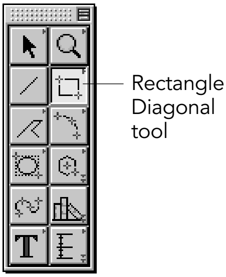
When the file opens, you will have a blank document in Draft mode. The scale and options have been set up for this project. As you work, you can use the zoom controls (at the lower-left of the drawing window) to adjust the magnification level.

Drawing the door frame

From the top, the door frame is a rectangle 3 feet wide by 6 inches deep.

1. Select the Rectangle Diagonal tool.

2. To set the first corner of the rectangle, click anywhere in the drawing window.



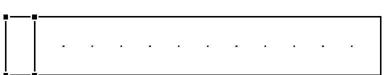
3. Press the Tab key. This highlights the ΔX box in the Info bar. Type 3' in the ΔX box.
4. Press Tab. This highlights the ΔY Box. Type 6" in the ΔY Box.

5. Press Return or Enter. This draws the rectangle in the Draft window.

Drawing the door jamb

1. Select the Rectangle Diagonal tool. 
2. To use snap points as a drawing aid, point to the outline of the door frame and press the Spacebar. This is the shortcut for choosing Snap Points in the Snaps menu.
3. Start at the left outside corner of the frame, and drag to draw a rectangle 2 inches wide by 6 inches high.

   ▲ The Info bar displays 2" (ΔX) and 6" (ΔY) as you drag the mouse. (The values are positive or negative, depending on the direction you drag.)



Jamb rectangle

Applying a fill color

To apply a fill color to the door jamb, press the Fill Color icon in the Attributes bar. This opens the Fill color palette. Click a color tile to apply the fill color to the jamb.

Fill color icon

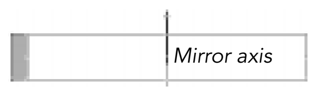
Jamb with fill color

Tip: If you click the Fill color icon, you apply the last color selected, without opening the palette. However, if you press the icon instead of clicking quickly, the palette opens.

Making a mirror copy

You can make a mirror copy of the left jamb to place the jamb on the right side of the frame.

1. With the jamb still selected, choose Object > Position > Mirror a Copy. A vertical line indicating the mirror axis appears.
2. Point to the door frame outline and press the Spacebar to display snap points. Move the mirror axis to the snap point at the center of the frame.



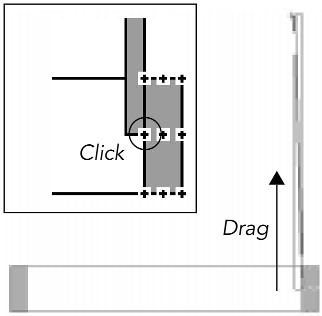
3. Click to set the mirror axis. A mirror copy of the door jamb appears on the right side of the door frame.

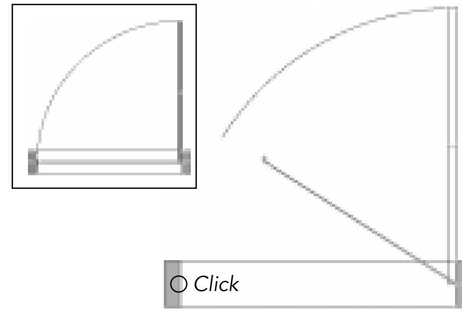
Mirrored rectangle

Drawing the door

The open door is a rectangle measuring 3 feet by 1 inch.

1. Select the Rectangle Diagonal tool. 
2. Point to the right door jamb and press the Spacebar to display snap points.
3. Click at the midpoint of the inside edge of the jamb. Drag upward and slightly to the left, to form a rectangle 1 inch wide (ΔX) by 3 feet high (ΔY).
4. Click to finish the rectangle.

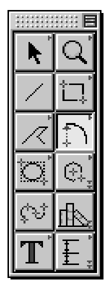




5. With the door still selected, open the Fill Color palette. Select the same fill color for the door that you selected for the jamb.

Drawing the door swing

The final element to draw is the door swing arc.



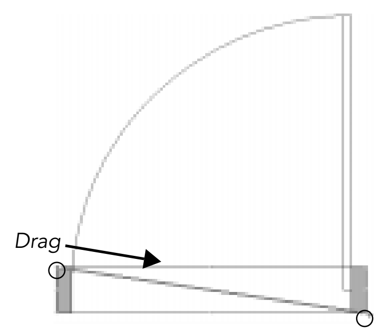
Arc Radius tool

1. Select the Arc Radius tool in the Arcs toolbar.
2. Point to the right door jamb, and press the Spacebar to display snap points.
3. At the midpoint of the door jamb (the point where the door is attached), click to set the center point of the arc.
4. Move up to the top-right corner of the door, then click to set one endpoint of the arc.
5. Move to the midpoint of the inside of the left jamb, and then click to set the arc's second endpoint. This completes the door swing.

Drawing the Boolean object

The last object to draw for the door unit is a rectangle. This rectangle will define the perimeter of the door frame for Boolean operations.

1. Select the Rectangle Diagonal tool. 
2. Draw a rectangle from the upper-left corner of the left jamb to the lower-right corner of the right jamb.



CONVERTING THE 2D DOOR UNIT TO A LIBRARY OBJECT

When you finish drawing the 2D door unit, you can convert it into a library object using the Properties Manager.

1. Use the Selection tool to drag a box around the entire door unit. This selects all the objects (door, frame, and jambs).

2. Choose Object > Group. One set of handles appears around the door unit.

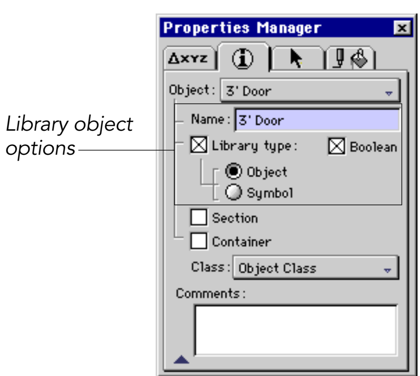
3. Choose Window > Properties Manager to display the Properties Manager.

4. Click the Info tab. This tab lets you view information about the door unit.

▲ Type **3' Door** in the Name box, and then press Tab.

▲ To convert the door to a library object, select the Library Type box and the Object option.

▲ Check the Boolean box. This sets the front object in the group (the final rectangle) as a *subtractive* shape. The rectangle will cut openings in walls where you place the door unit.

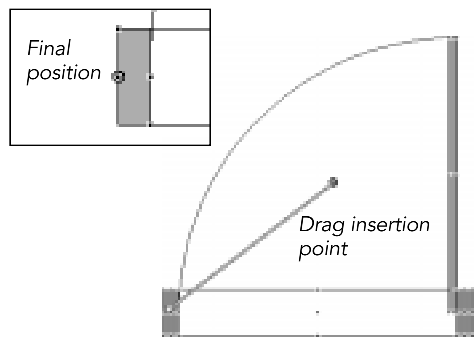


Setting the 2D insertion point

The *insertion point* is the point where a library object snaps to other objects. The default insertion point position is at the center a library object. For this door unit, an insertion point at the center isn't useful. You need to move the insertion point to the midpoint of the door frame. When you place the door unit in a wall, the insertion point will center the door frame within the wall.

1. With the door library object still selected, choose Edit > Reshape (or click the Reshape button on the Attributes bar). The insertion point appears at the center of the library object.

2. Display snap points on the left jamb, and then drag the insertion point to the midpoint of the outside edge of the jamb.



You have now completed the 2D door plan and library object.

DRAWING THE DOOR UNIT ELEVATION

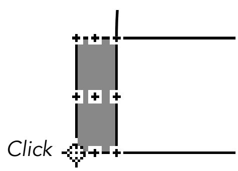
This steps in this section shows you how to draw a door unit in Draft (2D) Front (elevation) view. Although this drawing isn't saved as a library object, it is the basis for the 3D library object that you'll create in the final section of the tutorial.

Moving the Centerpoint

Coordinates shown in the Info bar are based on the position of the Centerpoint in the drawing. To make it easy to work in other views, you can set the Centerpoint as a reference point on the door.

Note: If the Centerpoint is at the lower-left corner of the left jamb, you don't have to move it.

1. Choose Layout > Centerpoint > Set Position.
2. Point to the left jamb and press the Spacebar to display snap points. Then, click the lower-left corner of the jamb. This sets the Centerpoint.



Changing to Front view

To draw the door elevation, you continue to work in Draft mode, but switch to Front view in the Draft window.

1. Choose View > Front. The window displays the Front view of the Draft workspace.

   ▲ No objects appear because were drawn in Front view. Draft (2D) objects appear only in the view where you draw them.

2. Make sure the Centerpoint is visible. If necessary, zoom or scroll so the Centerpoint is near the window's lower-left corner.

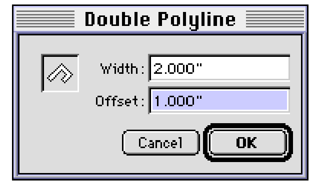
Drawing the door frame elevation

The Double Polyline tool lets you draw walls and framing at any thickness. You will set up the tool and use it to draw the door frame.

1. Double-click the Double Polyline tool icon. The Double Polyline dialog box appears.

Double Polyline tool

2. Type 2" in the Width box, type 1" in the Offset box, and then choose OK. This lets you draw the frame to match the jambs drawn in Top view.



3. To draw the frame, you will click at the frame corners. Use the Info bar to position the pointer as you set the following points:

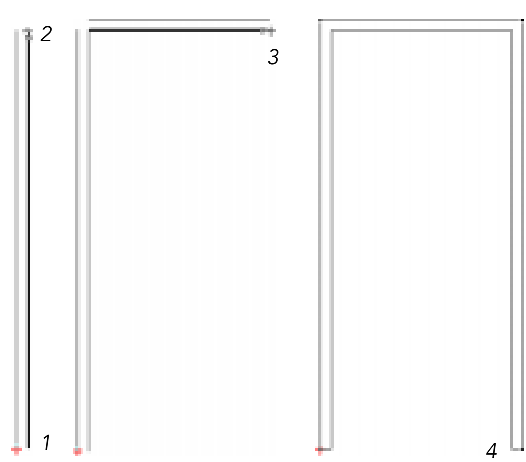
   ▲ Click 2 inches to the right of the Centerpoint (X: 2.0" and Z: 0").

   ▲ Click 6 feet 8 inches up from the first point. The Info bar "Δ" boxes show the distance from the previous point (ΔX: 0" and ΔZ: 6' 8").

   ▲ Click 3 feet to the right (ΔX: 3' and ΔZ: 0").

   ▲ Click 6 feet 8 inches down from the previous point (ΔX: 0', ΔZ: -6' 8").

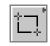
4. Press Return or Enter to complete the frame.



Drawing the door elevation

To complete the door unit elevation, you will draw the open door inside the frame, so the elevation matches the door plan drawn before.

1. Select the Rectangle Diagonal tool.



2. Point to the door frame and press the Spacebar to display snap points.

▲ Click at the upper-right inside corner of the frame (X: 3.0′ and Z: 6' 8").

▲ Move 1 inch to the left and 6 feet 8 inches down from the first point (ΔX: -1.0" and ΔZ: -6' 8") and click to finish the rectangle.

CREATING A 3D LIBRARY OBJECT

After drawing the door unit elevation, you can extrude the 2D objects to create 3D objects, and then set up and save the 3D library object.

First, you will set up DENEBACAD so you can work in multiple windows and view 2D and 3D objects.

Using multiple windows

1. To open a new drawing window in Sculpt mode, choose Show Sculpt in the Window menu. A Sculpt mode window in Top view appears.

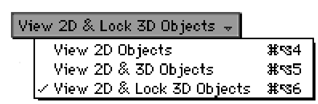
▲ DENEBACAD arranges the windows so you can see both. (If this doesn't happen, choose Preferences in the Edit menu. In the Preferences dialog box, be sure "Auto-tile windows" is selected.)

2. Click the Draft window to make it active.

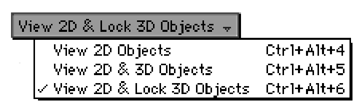
3. Choose Fit to Window in the Zoom pop-up menu, or press Ctrl+Alt+W (Windows) or Cmd+Shift+W (Mac OS). This adjusts the zoom level so you can see the entire drawing.

Displaying 2D and 3D objects

In the View Options pop-up menu at the left end of the Status bar, choose View 2D & Lock 3D Objects. This mode lets you see 3D information in the Draft window.



Mac OS



Windows

▲ When you change the display mode, a message tell you that 2D objects will be displayed in red and 3D objects will be displayed in blue. Click OK in the message box.

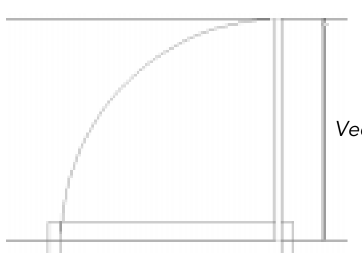
Select to prevent message from appearing again

EXTRUDING THE 2D DOOR UNIT INTO 3D

The first step in extruding 2D objects is defining (or selecting pre-defined) 3D extrusion planes.

Defining extrusion planes in Front view

1. Choose Layout > 3D Plane > Define Vertical 3D Plane. The Draft window changes to Top View and you should see the top view of the door.
2. A horizontal line extending from the pointer represents the first extrusion plane. Align the extrusion plane with the edge of the door at the right jamb, and click to set the extrusion plane.
3. Move the pointer up and align the horizontal line with the outside edge of the door, and click to set the second extrusion plane.



The Draft windows changes back to the original view (Front) after you set the two extrusion planes.

Extruding the door

The previous procedure defined extrusion planes corresponding to the 3D width of the door (3 feet).

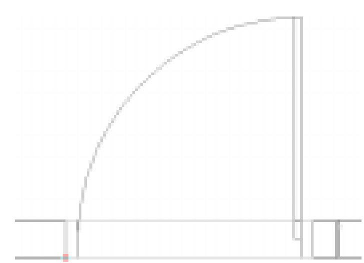
In Front view, you see the inner edge of the door at the jamb. The 1-inch dimension is the thickness of the door, and the 6-foot 8-inch dimension is its height. Extruding in Front view will extend the edge through 3 feet of space (between the extrusion planes you defined in Top view). This will add the third dimension to the door.

1. Click the door object to select it.
2. Choose Object > Extrude > Linear. A rectangle appears in the Sculpt window. This is the 3D door created from extrusion of the 2D door.

Extruding the door frame

Extruding the door frame is similar to the previous procedure. However, the frame extrusion requires a different set of extrusion planes to define the frame depth.

1. In Draft mode Front view, choose Layout > 3D Plane > Define Vertical 3D Plane. The Draft window switches to Top view.
2. Align the first extrusion plane with the front face of the door jamb, and click to set the extrusion plane.
3. Align the second extrusion plane with the back face of the door jamb, and click to set the second extrusion plane.



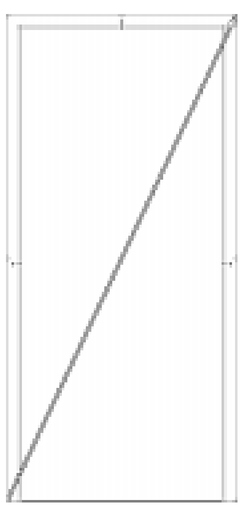
4. Select the door frame object.
5. Choose Object > Extrude > Linear. The extruded door frame appears in the Sculpt window. If you can’t see the frame, choose

Fit to Window from the pop-up menu in the Zoom bar (at the bottom of the Sculpt window).

Adding the Boolean object

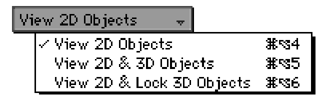
The last object needed for the 3D door is a Boolean object. Like the 2D door, the 3D door needs a rectangle that will cut though walls where you place the door library object.

1. Click the Sculpt window to activate it.
2. Choose View > Front.
3. Select the Rectangle Diagonal tool. 
4. Snap to the outside of the bottom-left corner of the frame, then drag up to the outside of the upper-right corner. This creates a rectangle corresponding to the perimeter of the door unit.

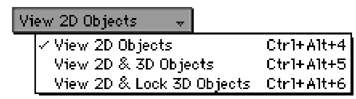


Drawing the rectangle in Sculpt mode causes it to be extruded between the active extrusion planes. In this case, the extrusion planes match the depth of the door frame, so the extruded Boolean object is the same width, height, and depth as the door unit.

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



Mac OS

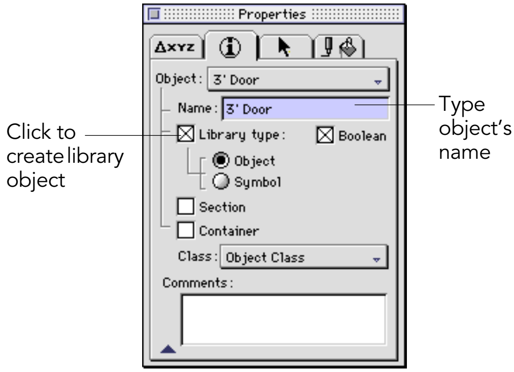


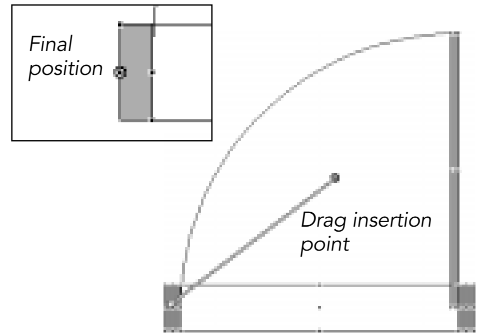
Windows

CONVERTING THE 3D DOOR UNIT TO A LIBRARY OBJECT

With the 3D door unit completed, you can convert it into a library object and link it to the 2D door unit library object.

1. Select the Sculpt window, then press Ctrl+1 (Windows) or Cmd+1 (Mac OS) to switch to Top view.
2. Press Ctrl+A (Windows) or Cmd+A (Mac OS) to select all the door unit objects.
3. Choose Object > Group, or press Ctrl+G (Windows) or Cmd+G (Mac OS) to group the door objects.
4. Choose Window > Properties Manager to display the Properties Manager.
5. Click the Info tab. To convert the group into a library object, select the Library Type box, then select the Object button. To give the door a Boolean feature, select the Boolean box.
6. Type **3' Door** in the Name text box, then press Tab. Making the name of the 3D door exactly the same as the 2D door links the two library objects, making a combined library object with 2D and 3D characteristics.





As before, the frontmost object in the group, the final rectangle you drew, is a subtractive shape. The rectangle will cut through the walls where you insert the door unit.

Setting the 3D insertion point

The final step in creating the 3D door library object is setting the insertion point. The insertion point will align the door unit when you insert it into a wall.

1. In the Sculpt mode window in Top view, select the door library object.
2. Choose Edit > Reshape (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 outside edge of the left door jamb.

SAVING THE LIBRARY OBJECTS

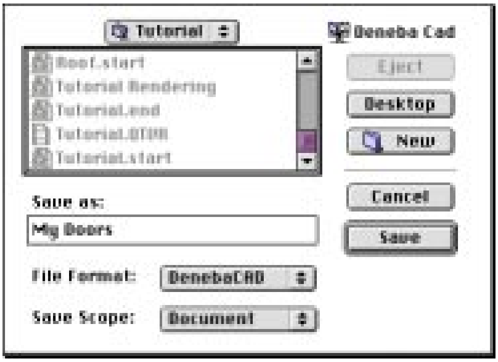
You should now save the DENEBACAD document to preserve the 2D and 3D library objects you created. The library objects can then be inserted into other documents, including the document described in the next tutorial.

To save the DENEBACAD document

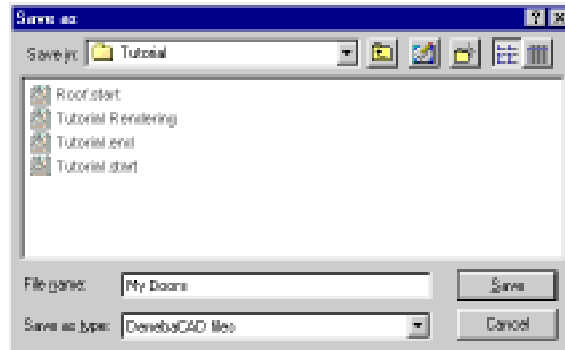
1. Choose File > Save As. In the directory dialog box, select the Tutorial folder in the DENEBACAD folder.
2. Type **My Doors** in the Save As (Mac) or File Name (Windows) text box.

   ▲ Other doors you want to use as library objects can be created in the same document using similar procedures.

3. Click Save to save the DENEBACAD document.



Mac OS



Windows

Fig. 24   Saving the library object document

DRAFTING A MODEL 5

This chapter demonstrates how to create a simple 3D structure from a 2D drawing.

By completing this tutorial, you will learn how to place a library object, such as the door unit created in the previous chapter, into a drawing. You'll extrude a 2D drawing and see how the 2D/3D library object appears in the 3D model.

Finally, you'll learn how to set a focal point in Render mode, and to view a hidden-line rendering of your model.

To open the tutorial document

1. Launch DENEBACAD, if it isn't running.
2. Choose File > Open.
3. In the directory dialog box, open the Tutorial folder in the DenebaCAD folder. Select the file "Library inserts.dcd" and click Open.

The document appears in a Draft window, Top view, with a blank drawing area.

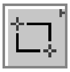
DRAWING A STRUCTURE IN 2D

To draw a basic structure, you can draw simple shapes, such as rectangles and circles, and then combine the shapes to form an outline of the structure.

You can use the Info bar to see the dimensions of objects as you draw. If the Info bar is not displayed, choose Layout > Toolbars. In the dialog box, select the Info Bar box and click OK.

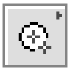
To draw shapes for the outline

1. Select the Rectangle Diagonal tool.



Rectangle Diagonal tool

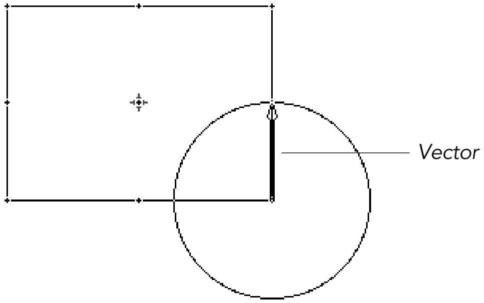
2. Draw a rectangle 20 feet wide by 14 feet high. The Info bar displays ΔX:20' and ΔY:14' when the rectangle is the correct size.



Circle Radius tool

3. Select the Circle Radius tool.
4. Point to the lower-right corner of the rectangle, and drag to draw a circle centered on the corner. Extend the circle until its edge meets the midpoint of the rectangle's side.

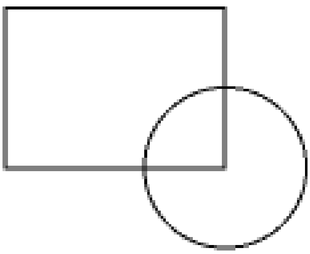
   ▲ You can use snap points to align the circle. Before dragging, point to the rectangle and press the Spacebar. The circle's center will snap to the corner and its edge will snap to the midpoint of the rectangle.



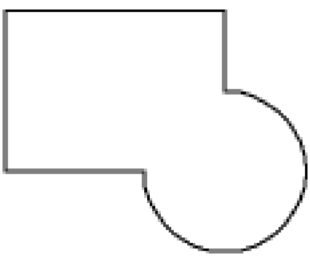
To combine objects

1. Select both objects.
2. Choose Object > Combine > Unite (or, click the Unite action button). This combines the selected objects into one outline.

Unite action button



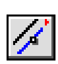
Original objects



United objects

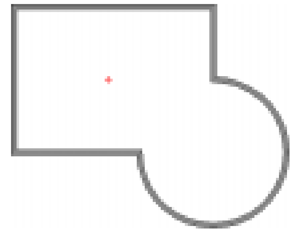
To set the wall thickness

1. With the combined object still selected, choose Object > Path > Offset (or, click the Offset action button). The Offset dialog box appears.



Offset action button

2. Type 6" in the Width box and 0" in the Offset box click OK.



Object offset 6"

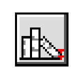
3. To apply a fill color to the walls, press the Fill Color icon in the Attributes bar and select a color.

INSERTING A LIBRARY OBJECT

After completing the structure’s exterior walls, you can insert a door by using a library object.

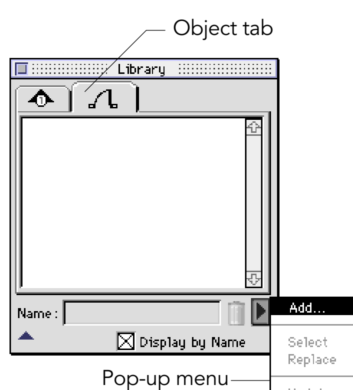
To load the library object

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



Library tool

2. Click the Object tab. Choose Add in the palette’s pop-up menu.

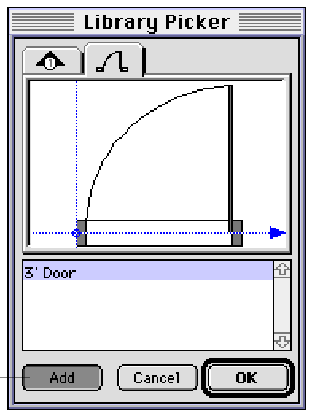


3. A directory dialog box appears. Select the file named “My Doors” that you created earlier and click Open.

▲ If you didn’t create this file, open the Tutorial folder in the DenebaCAD folder. Select the file named “Doors.dcd” and click Open.

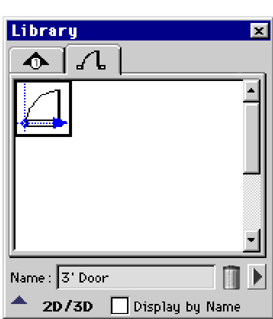
4. The Library Picker appears. The Library Picker displays and lists the library objects contained in the DENEBACAD document that you selected.

Click to add the selected item



5. Select the object named “3' Door” and click Add. This adds the selected library object to the Library palette in the current document.

6. Click OK to close the Library Picker.

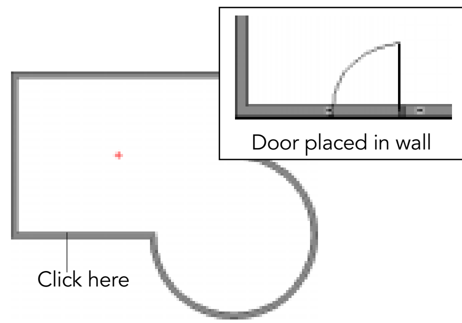


Library palette displays the loaded library objects

To insert the door library object

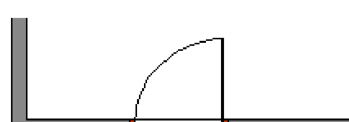
1. The Library palette displays a sketch of the door object (if Display by Name is not selected). Click the door in the Library palette to select it.

2. Move the pointer into the drawing; an outline of the door’s bounding box follows the pointer. Click at the center of the exterior wall at the bottom of the drawing, then press Shift and drag to the right in a straight line. This sets the orientation of the door.



3. Press Shift and click the building to select it and the door object. Choose Object > Combine > Unite (or, click the Unite action button). This cuts the door opening in the wall. One set of handles appears around the entire object, because the door and the wall are one Boolean object.

Unite action button



Door opening in wall

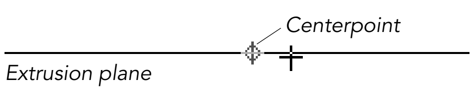
EXTRUDING THE 2D STRUCTURE INTO 3D

When you draw 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.

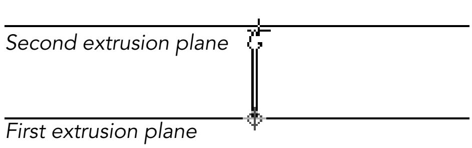
A 2D/3D library object, such as the door you inserted, will appear correctly in the 3D model when you extrude the wall that contains it.

To define the extrusion planes

1. Choose Layout > 3D Plane > Define Frontal 3D Plane. The view changes to Front.
2. A horizontal line indicates the extrusion plane. Align this with the Centerpoint, and click to set the first extrusion plane.



3. Move the pointer up 10 feet (ΔZ:10' appears in the Info bar), and click to set the second extrusion plane. The planes are set to extrude a 10-foot-high wall. The view changes back to Top.



To extrude the walls

1. Click the building to select it.
2. Choose Object > Extrude > Linear.

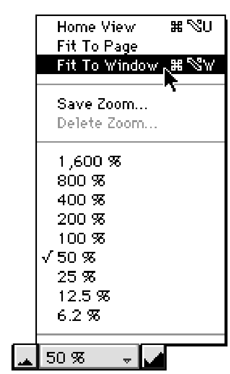
DENEBACAD extrudes the 2D objects to create a 3D model. You can view the model in Sculpt and Render windows.

To display Sculpt and Render modes

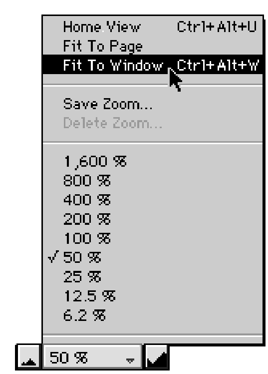
1. Choose Window > Show Sculpt. A window set to Sculpt mode appears.
2. Choose Window > Show Render. A window set to Render mode appears.

DENEBACAD arranges the Draft window on the left, the Sculpt window at the upper right, and the Render window at the lower right. (If the "Auto-tile windows" preference is not selected, choose Window > Tile to arrange the windows).

If the drawing is no longer visible in the Draft window, click the Draft window, and choose Fit to Window from the Zoom bar.



Mac OS



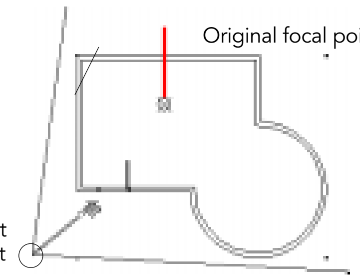
Windows

To view the model in Sculpt mode, click the Sculpt window and choose View > Front. Then, Choose Fit to Window from the Sculpt zoom bar. The Sculpt window shows the wall elevation, including the door library object.

VIEWING A RENDERING OF THE MODEL

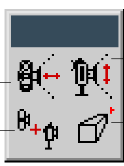
To view a rendering of the model, you will set the focal point in the Render window.

The focal point represents an observer (or camera) viewing the rendered scene. Whenever you change the focal point, DENEBACAD renders the scene from the new focal point.



To set the horizontal focal point

1. Click the Render window. The toolbox displays the Render mode tools.



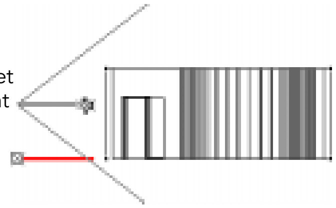
2. Select the Horizontal Focal Point tool. The Render window displays the model in from the top.
3. If necessary, zoom out so the entire model and some of the area around it is visible.
4. Move the pointer outside a corner of the building to change the horizontal view point (indicated by a cross). Click to set the horizontal view point.
5. Two lines angled from the view point represent the field of view. Move the pointer to adjust the view, then click to set it.

The window displays a rendered scene based on the new focal point.

To set the vertical focal point

1. Select the Vertical Focal Point tool. The Render window displays the model from the front.
2. If necessary, zoom out so the entire model and some of the area around it is visible.
3. Move the pointer up or down to change the vertical view point (indicated by a cross). Click to set the vertical view point.
4. Two lines angled from the view point indicate the field of view. Move the pointer to adjust the view angle, and then click to set it.

The window displays a rendered scene based on the new focal point.



To view a Hidden Line rendering

A final step in the visualization process is to view a hidden line rendering of a project.

After you set the focal point and view the model in wireframe, choose Rendering > Hidden Line.

DENEBACAD removes hidden lines, which can require several seconds, depending on the complexity of the drawing, and then displays the rendered scene.

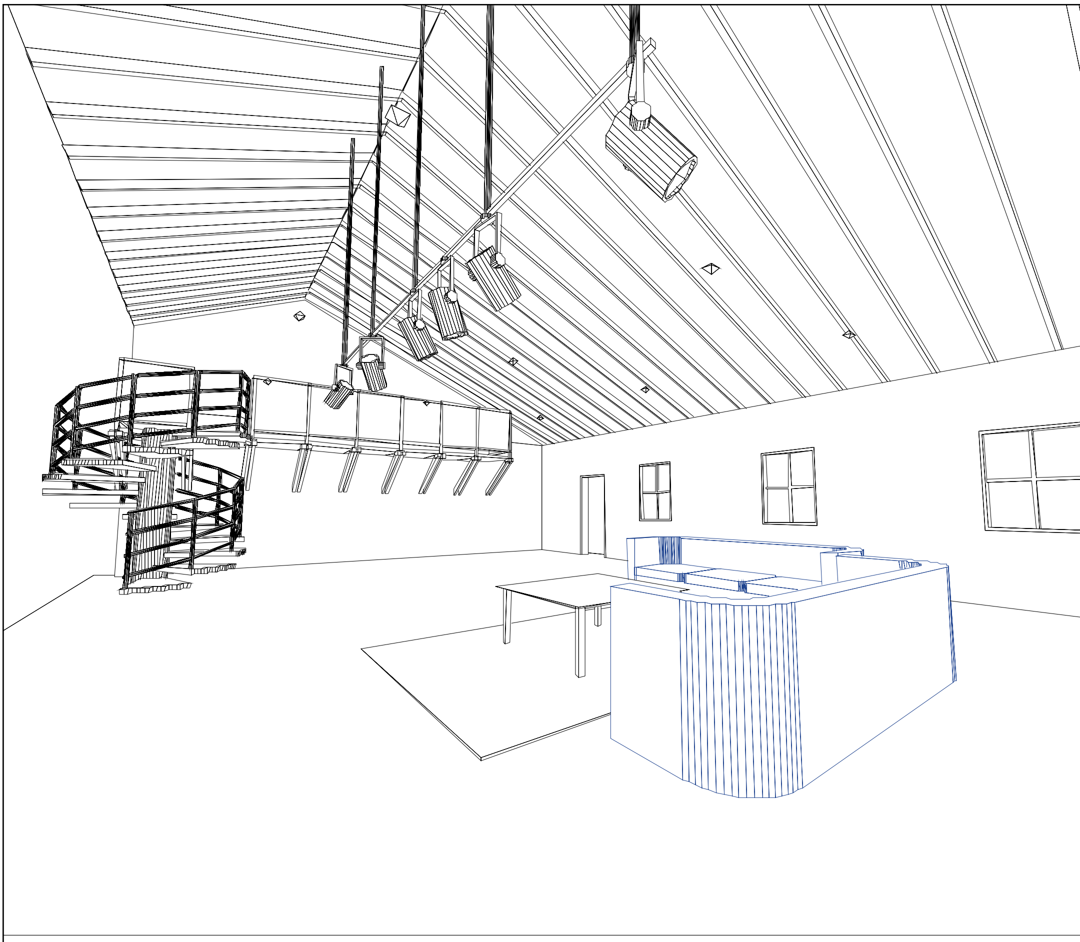


Fig. 25 A hidden line rendering

USING SWEEP EXTRUSION 6

The tutorial in this chapter shows you how to create a hip roof using the Sweep extrusion method.

The procedures demonstrate how to create a sweep section, and how to extrude the sweep section along a path.

To start the tutorial

1. Launch DENEBACAD if it isn't running.
2. Choose File > Open. In the directory dialog box, select the file "Roof Start.dcd" in the Tutorial folder and click Open. DENEBACAD opens the file.
3. Choose Layout > Toolbars. Select all the options, and click OK. (This step isn't necessary if the Action buttons, Status bar, Info bar, and Help bar are already displayed.)

CREATING A HIP ROOF SWEEP EXTRUSION

For a Sweep extrusion, you use an object as a *sweep section*. The sweep section is constructed as a cross-section of the final extruded object.

In the first part of the tutorial, you'll learn how to designate a sweep section.

CREATING A SWEEP SECTION

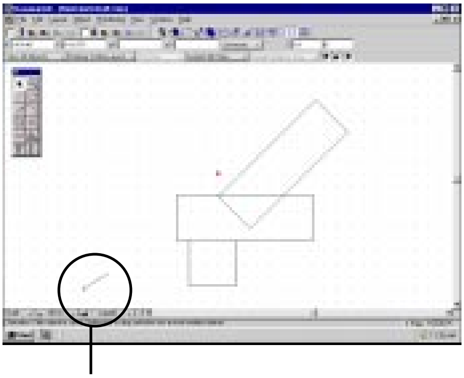
You can make a sweep section from any objects, including a single object or multiple objects that have been grouped.

In the tutorial document, the sweep section object has been drawn in Draft mode. The following procedure shows you how to designate the object as a sweep section.

To create a sweep section

1. The Draft mode window contains three rectangles, and a smaller object. To make the smaller object into a sweep section, you will use the Properties Manager.

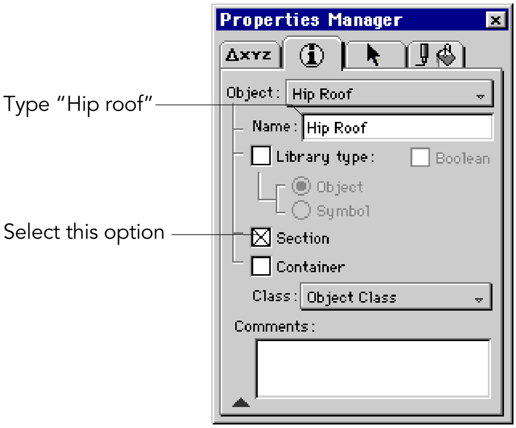
▲ If you do not see all the objects in the Draft window, choose Fit to Window in the Zoom bar pop-up menu, or press Ctrl+Alt+W (Windows) or Cmd+Opt+W (Mac OS).



Sweep section object

▲ To magnify the sweep section object, select the Zoom tool, and drag a box around the small object to the left of the rectangles.

2. Double-click the object. This selects the object and displays the Properties Manager. If necessary, move the Properties Manager so you can see the object.
3. In the Properties Manager, click the Info tab to bring it to the front. To designate the object as a sweep section, select the Section box.
4. In the Name text box, type “Hip roof” and press Enter
5. To close the Properties Manager, click the Close box in the Title bar.



Setting the insertion point

Before you use a sweep section, you might need to adjust its insertion point.

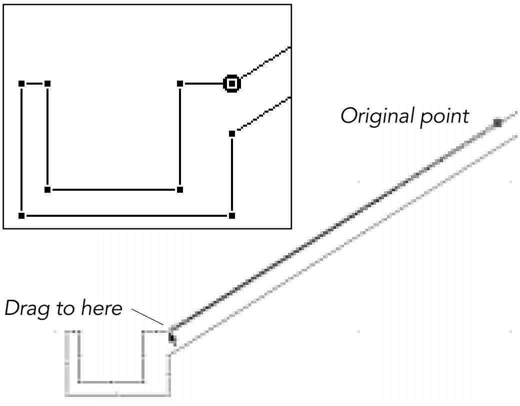
The insertion point of a sweep section affects the orientation of the sweep section along the extrusion path. Think of the insertion point as the point that runs along the sweep path. Changing the location of the insertion point can significantly change the resulting extrusion.

To move the insertion point

1. With the sweep section still selected, click the Reshape action button on the Attributes bar. This puts the sweep section into reshape mode, with the object’s creation points and the insertion point displayed.

   ▲ The insertion point is a highlighted handle on the sloping line.

2. Point to the object and press the Spacebar to display the object’s snap points.
3. Drag the insertion point to the junction of the sloping roof section and the gutter. The drawing vector snaps to this point as you drag near it.

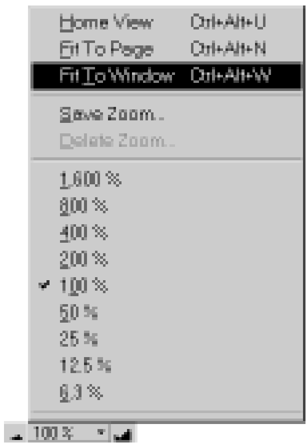


CREATING A BUILDING OUTLINE

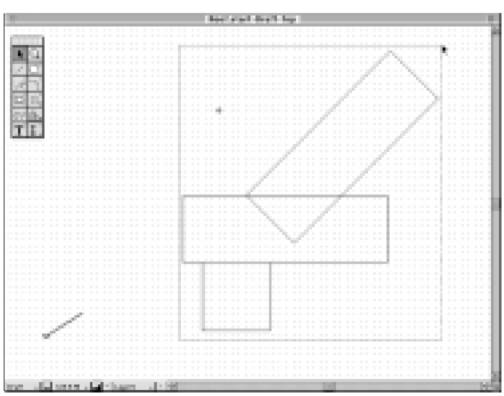
You can extrude a roof directly on a structure, using the perimeter of the building as a sweep path.

In the next procedure, you will use the Unite command to combine the three rectangles in the Draft window into the outline of a building.

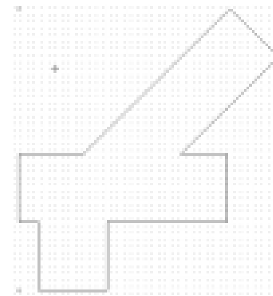
1. Choose Fit to Window in the Zoom pop-up menu so you can see the rectangles and the sweep section.



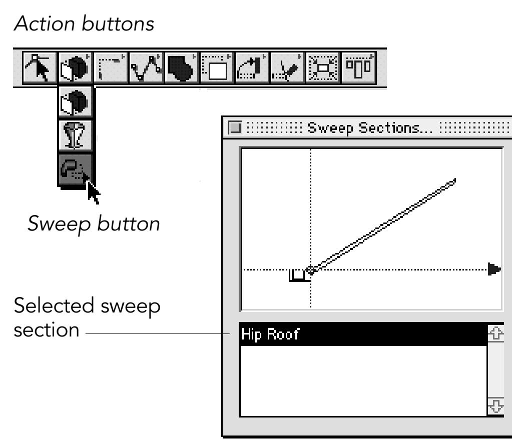
2. Drag a box around the three rectangles to select them.



3. Press Ctrl+U (Windows) or Cmd+U (Mac OS), the shortcut for the Unite command. 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.

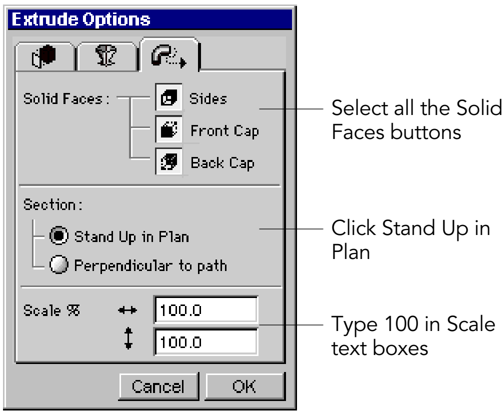


4. Deselect the united object by clicking in an empty area of the document.
5. Choose the Sweep extrusion method by selecting the Sweep action button in the Attributes bar.



▲ When you select the Sweep extrusion method, the Sweep Sections palette appears. This palette displays the sweep section designated earlier as "Hip roof."

6. Double-click the Sweep action button to open the Extrude Options dialog box. Make the settings in the dialog box match the following illustration, and then click OK.



7. Select the building outline object.
8. To extrude the sweep section, click the Sweep action button. DENEBACAD sweeps the section around the selected object.

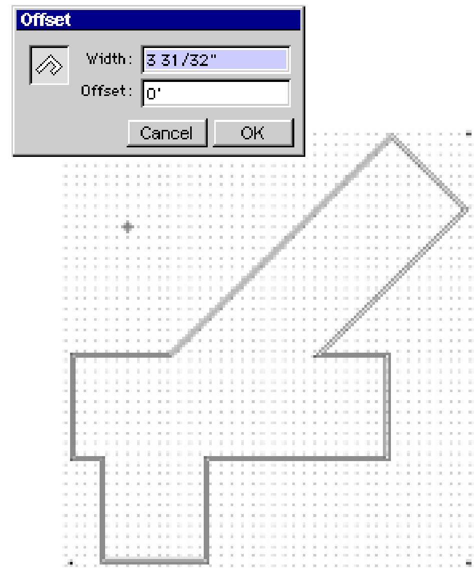
Note: You won't see the result of the sweep extrusion until you switch to Sculpt and Render modes as described later.

OFFSETTING AND EXTRUDING WALLS

Extruding the roof with the Sweep method creates a 3D object. To make the building match the hip roof, this section explains how to create walls from the building outline, and then extrude the walls into 3D to meet the roof.

To offset an outline for walls

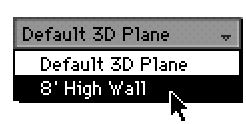
1. Select the building outline, which became deselected by the Sweep procedure.
2. Press Ctrl+Shift+O (Windows), Cmd+Shift+O (Mac OS), or choose Object > Path > Offset. The Offset dialog box appears.
3. Type 3 31/32" in the Width text box, and type 8" in the Offset text box. Click OK.



DENEBACAD offsets the building outline 8 inches.

To extrude the walls

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

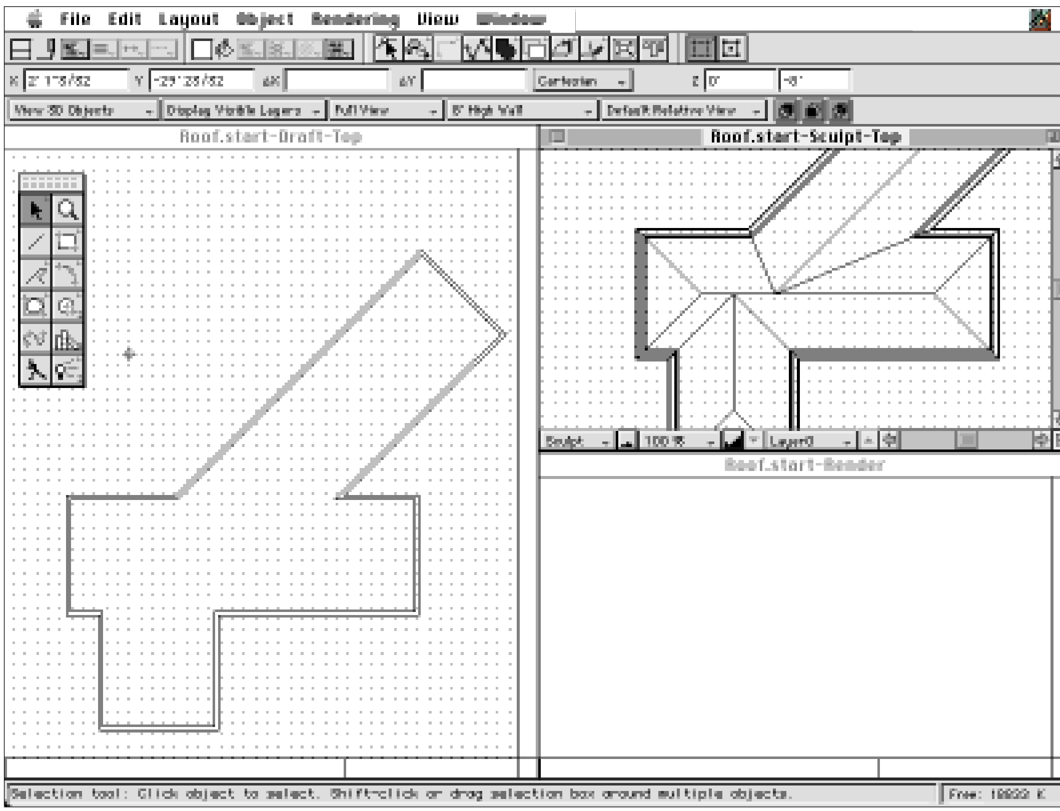


2. Select the building, then press Ctrl+Shift+E (Windows) or Cmd+Shift+E (Mac OS), the shortcut for the Linear command. DENEBACAD extrudes the walls from to 8 feet high in 3D space.

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 Window > Show Sculpt. To open a Render mode window, choose Window > Show Render. DENEBACAD arranges the windows when “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.



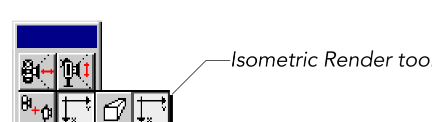
VIEWING RENDERINGS

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

In Render mode, you can view the building from any perspective. You can create your own perspectives by setting focal points wherever you want using the focal point tools.

To generate an isometric perspective

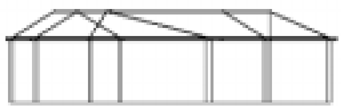
1. Click the Render window to make it active.
2. Choose the Isometric Render tool. DENEBACAD creates an isometric Wireframe rendering in the Render window.



3. To center the rendering in the window, click the maximize button (Windows) or the zoom box (Mac OS) in the Render window. This expands the window to fill the screen.

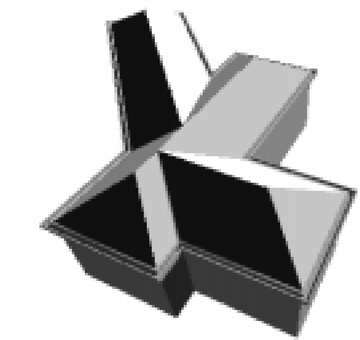
Mac OS

Windows



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.



Solid rendering with Sunlight

This tutorial describes how to create a 3D model from the ground up using drawing tools, extrusions, and library objects.

In addition to basic drawing techniques, you'll learn how to place lights and furniture, and apply surface materials to the model before creating a final solid rendering.

To begin the tutorial

1. Launch DENEBACAD if the program isn't already running.
2. To open the file for this tutorial, press Ctrl+O (Windows) or Cmd+O (Mac). In the directory dialog box, open 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.

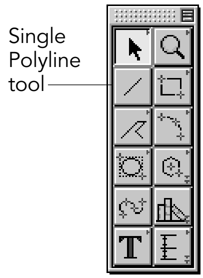
◆ **To display all toolbars:** If the Info bar and Status bar aren't displayed on screen, choose Toolbars in the Layout menu. In the Toolbars dialog box, select the four options, and then click OK.

DRAFTING A DESIGN FOR MODELING

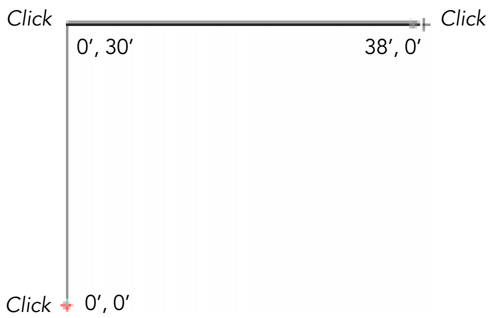
In DENEBACAD, you can use the same working method whether you intend to produce 2D plans or a completely rendered scene.

To draft wall sections

1. Select the Single Polyline tool.
2. Click at the Centerpoint of the drawing (X:0' and Y:0').
3. Move the pointer up; the drawing vector appears. Click at ΔX:0' ΔY:30'.

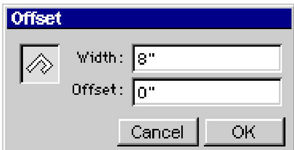


4. Move the pointer to the right and click at ΔX:38' and ΔY:0', and then press Return to complete the polyline.



This object will become two walls of a building.

5. Choose Object > Path > Offset. 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 Library Objects let you quickly add a door and windows to the building's walls.

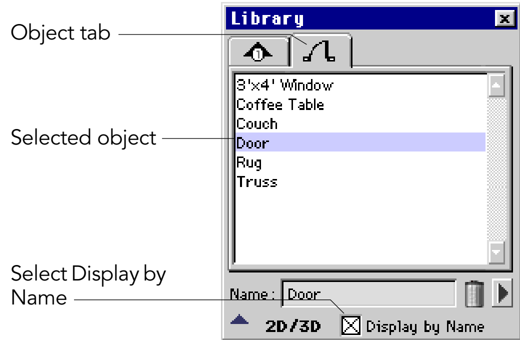
1. Double-click the Library tool in the toolbox. The Library palette appears. The palette displays the library objects stored in this document.

Library tool

2. Click the Object tab to bring it to the front, if necessary. Be sure "Display by Name" is selected. Then, select "Door" in the list.

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.



To insert windows from the library

1. In the Library palette 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 drawing vector appears. This aligns the window within the wall.

4. To close the Library palette, click the close box or button in the title bar.

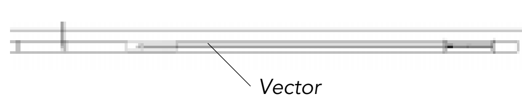
To duplicate windows in an array

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

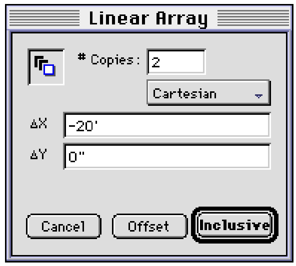
1. Select the window object, then choose Object > Position > Linear Array.

2. 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.

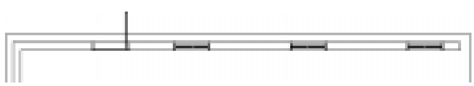
3. Move the pointer to the right, to ΔX:-20' ΔY:0'. Click to set the end point of the array.



4. 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.



COMBINING OBJECTS AND WALLS

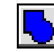
The commands in the Combine submenu let you create Boolean groups by adding and subtracting objects from each other.

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 Ctrl+A (Windows) or Cmd+A (Mac) to select all the objects in the drawing, and then press Shift and click the ground plane object to deselect it.

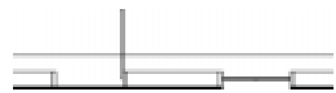
2. Choose Unite in the Combine submenu in the Object menu, or click the Unite action button.



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.

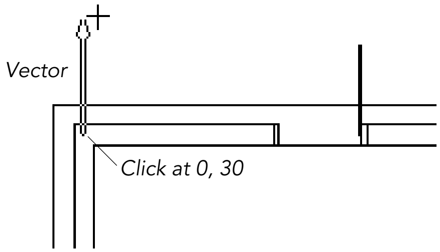


PLACING ROOF TRUSSES

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

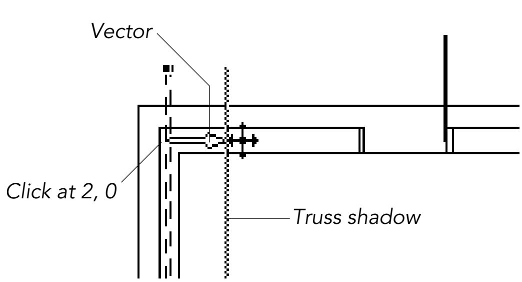
To insert a roof truss library object

1. Double-click the Library tool in the toolbox to open the Library palette.
2. Select the "Truss" item in the Library palette scrolling list.
3. Move the pointer to X:0' Y:30' and click the mouse to place the roof truss.
4. Press Shift and drag straight up until the drawing vector appears, which aligns the truss with the left wall.



To array roof trusses

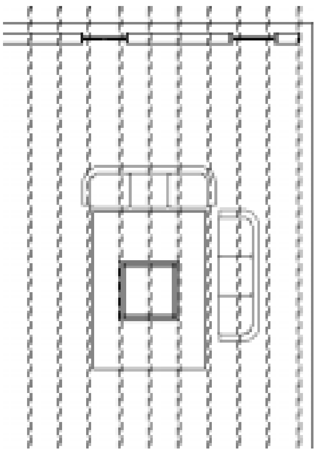
1. Select the roof truss inserted in the previous steps.
2. Choose Object > Position > Linear Array.
3. Move the pointer to X:0' Y:30' over the center of the wall. Click to set the starting point for the array of trusses.
4. Move the pointer to the right to ΔX:2' ΔY:0'. The drawing vector indicates the distance and direction for the array operation. Click to define the offset distance of the array. The Linear Array dialog box appears.
5. 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.



INSERTING FURNITURE

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.

1. If necessary, double-click the Library tool in the toolbox to open the Library palette. Select "Couch" in the Library palette list.
2. Move the pointer to X:28' 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.
3. To insert another couch, select the Library tool again; "Couch" should still be selected in the Library palette. Click at X:34' Y:14'. This time, drag downward in a straight line until the drawing vector appears.
4. Select "Rug" in the Library palette. Click at X:28' Y:13'. Drag in a straight line to the right to set the orientation.
5. Select "Coffee Table" in the Library palette. Click at X:28' Y:13'. Drag to the right to set the table's orientation.
6. Close the Library palette.

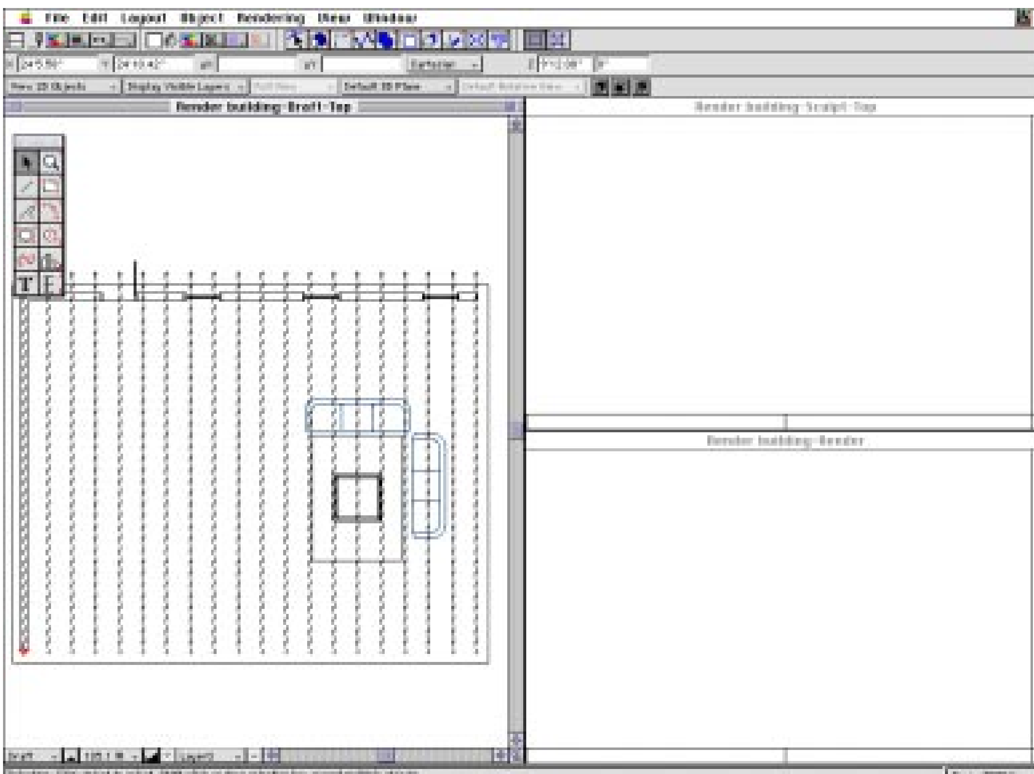


EXTRUDING THE PLAN INTO 3D

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 Window > Show Sculpt. A window appears in Sculpt mode.
2. Choose Window > Show Render. A new window in Render mode appears.
3. Click the Draft window to make it active. Select Fit to Window from the Zoom pop-up menu, or press Ctrl+Alt+W (Windows) or Cmd+Opt+W (Mac) so you can see your entire drawing in the window.



SELECTING EXTRUSION PLANES

Before extruding objects, you can define extrusion planes. You can save sets of extrusion planes in a document, and then select one of the saved sets as needed.

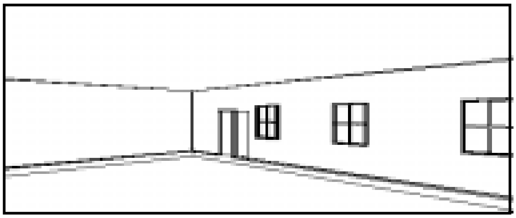
To extrude parts of the building in this document, you can use preset extrusion planes.

To extrude the building's slab

1. Choose Layout > 3D Plane > "1' Thick Slab." You can also select this set of 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 Object > Extrude > Linear. 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 Layout > 3D Plane > "10' Tall Exterior Wall." This activates another set of saved extrusion planes.
2. Select the wall object.
3. Press Ctrl+Shift+E (Windows), Cmd+Shift+E (Mac) to extrude the walls. The 3D walls appear in elevation in the Sculpt mode window, and in perspective in the Render mode window.



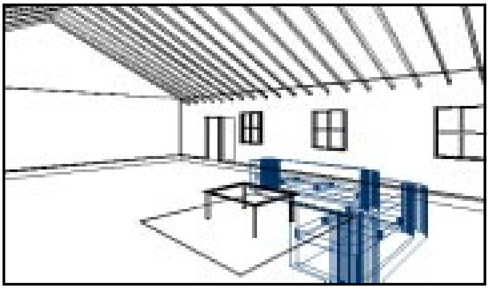
Render mode perspective

EXTRUDING 2D/3D LIBRARY OBJECTS

After using saved extrusion planes to extrude the ground plane and walls, the next step is to extrude the library objects in the drawing, which have 2D and 3D components.

To extrude the doors, windows, and roof trusses

1. Click the Draft mode document window to make it active.
2. Press Ctrl+A (Windows) or Cmd+A (Mac) 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 Ctrl+Shift+E (Windows), Cmd+Shift+E (Mac) to extrude the objects. The extruded library objects appear as 3D objects in the Sculpt and Render windows.



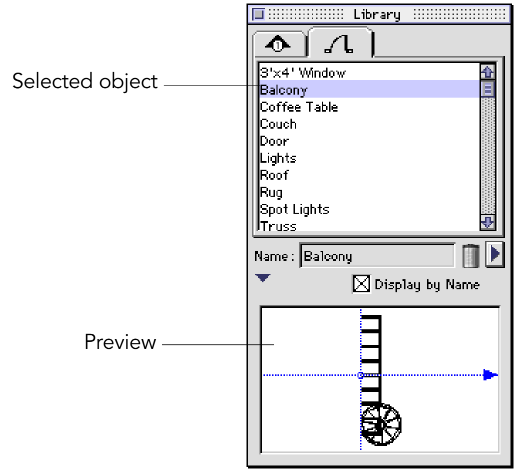
Wireframe rendering

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 Sculpt window to make it active. If the window does not show Top view (the title bar doesn't say "*Name*-Sculpt-Top"), choose View > Top.
2. Select Fit to Window from the Zoom pop-up menu, or press Ctrl+Alt+W (Windows) or Cmd+Opt+W (Mac) so you can see the entire drawing in the window.
3. Double-click the Library tool in the toolbox. The Library palette appears. Several items which weren't listed when the Draft mode window was active now appear in the Library palette's scrolling list.
4. Select the "Balcony" library object. Click at X:0', Y:12' to place the balcony. Then press Shift and drag in a straight line to the right until the drawing vector appears. This sets the orientation of the balcony. The balcony appears in the Sculpt window and in the Render window.
5. To insert a set of lights, select the "Lights" object in the Library palette. Move the pointer to X:20', Y:15' and click to insert the object. Then, drag straight to the right to set the orientation of the lights.
6. To insert a set of spot lights, select the "Spot Lights" library object. Click at X:27', Y:8' to place the object. Then, drag straight to the right to set the orientation.
7. Finally, to insert the roof and finish the model, select the "Roof" Library Object. Click at X:0', Y:9' to place the object. Then, drag straight to the right to set the orientation.



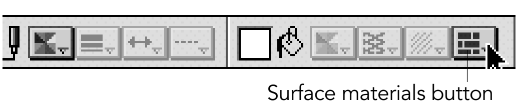
APPLYING 3D SURFACE MATERIALS

The last step before producing a final rendering of your model is to apply materials to the 3D objects.

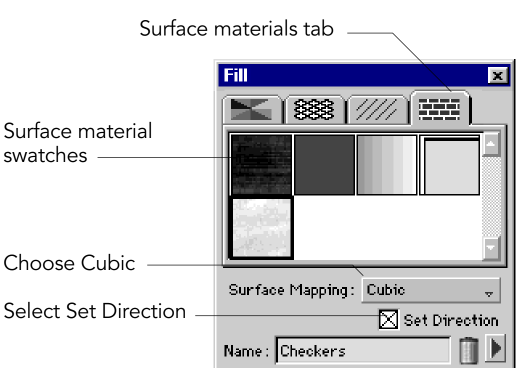
All of the library objects that have been inserted into the model already have surface materials. You can apply other surface materials to the walls and the floor.

To create a tiled floor

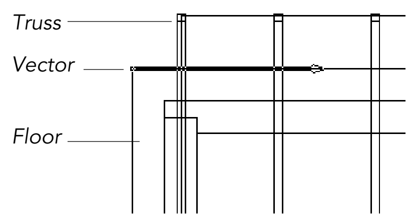
1. Click the Sculpt mode window, which should be set to Top view.
2. Select the floor object.
3. In the Attributes bar, press the Surface Materials button. When the Fill palette appears, drag into the drawing area to float the palette.



4. In the palette's pop-up menu, choose Cubic.
5. Select the Set Direction checkbox.



6. Click some of the surface material swatches in the Fill palette. Notice that the name of each surface material appears in the Name text box. Click the "Checkers" swatch to apply this surface material to the floor.
7. 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.



To apply a wall material

1. Select the wall object.
2. From the Surface pop-up menu, choose Cubic. Select the Set Direction checkbox.
3. Scroll through the surface materials and select the material named "Stucco Wall."
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 window to make it active.
2. Choose Rendering > Solid. DENEBACAD will calculate the scene, which might require several minutes, depending on the speed of the system. The rendered image will appear in the Render window.
3. To save the rendering, choose Edit > Copy.
4. Click the Draft window to make it active.
5. Choose Edit > Paste. A copy of the rendering appears in the Draft drawing.

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. Choose File > Open.
2. In the directory dialog box, open the Tutorial folder. Select the file named "Tutorial Rendering" and click Open.

The document opens in Draft mode, which contains an image of the fully rendered scene.

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. QuickTime VR lets you tilt and pan to see a 360-degree view of the rendered model.

FINISHING UP

Congratulations! You've finished the introduction to DENEBACAD. You can use the Save As command to save your project with a new name.

Be sure to use Save As instead of Save, unless you want to replace the original tutorial file with your version.

A solid rendering

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 library objects, layers, and classes to organize a project.

SETUP OVERVIEW

When you start DENEBACAD, the program opens a Draft mode window. The window's title bar displays *"Untitled-Draft-Top*." You can configure all the settings described in this section with the Draft 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 Layout > Units Setup to select the measurement system and the units for the document. For details, see "Drawing scale and measurements" on page 69.

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 Layout > Output Setup to select the output scale and the number of pages for printing. For details, see "Output setup" on page 70.

Grid setup If the Show Grid command is active, DENEBACAD displays a grid of dots based on the current units and measurement system. Choose Layout > Grid Setup to change the grid spacing. For details, see "Grid setup" on page 71.

Layers Choose Window > Layer Manager to set up layers. For details, see "Using layers and classes" on page 74.

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 extrusion planes before you start to draw.

Library items If you want to use existing library objects and symbols in your document, double-click the Library tool to open the Library Manager. Use the Add command in the manager's pop-up menu to import library items. For details, see "Library items" on page 72.

Classes You can choose Window > Class Manager 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 74.

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 75.

USING DOCUMENT TEMPLATES

After you set up a document, you can create a template to save your settings. A template 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 (Windows)

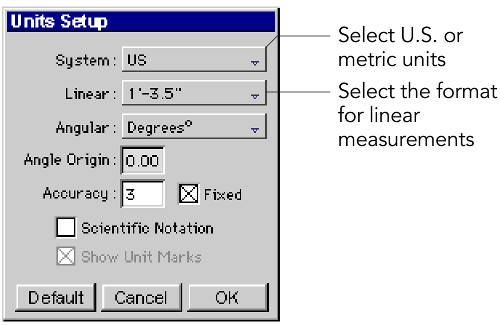
1. After establishing the document settings that you want to save, choose File > Save As.
2. In the directory dialog box, select the DENEBACAD Template file format. Type a name for the document and click Save to store the document on disk.

To create a document template (Windows)

1. After establishing the document settings that you want to save, choose File > Save As.
2. In the directory dialog box, select the DENEBACAD file format. Type a name for the document and click Save to store the document on disk.
3. After you save the document, make it a stationary pad. Select the file and choose Get Info in the File menu in the Finder. The Get Info dialog box appears. Select the Stationary pad check box and then close the Get Info window.

DRAWING SCALE AND MEASUREMENTS

You can use the Units Setup command in the Layout menu to choose U.S. or metric measurements, specify the format for linear and angular measurements, and specify the accuracy for measurements.



In U.S. versions of DENEBACAD, the U.S. measurement system is the default system, and measurements are displayed in feet and decimal inches, in the form 1'-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 15.50" in the Linear pop-up menu.

Displaying unit marks When you select measurements in inches or feet only, select the Show Unit Marks check box if you want to display tick marks in measurements and dimension text. This option is not available when you choose a combined feet-and-inches format.

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

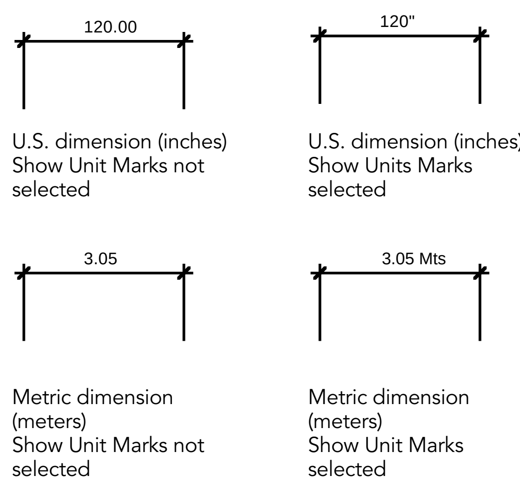


Fig. 81 Show Unit Marks option

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. The default measurement system, linear and angular units, and accuracy are appropriate for many projects.

Unless you use extremely small or large values, you won't need to select Scientific Notation to display numerical values as a base value and base 10 exponent.

Setting default measurement options

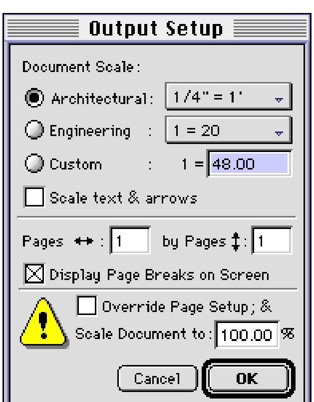
If you want the settings in the Units Setup dialog box to be the default settings for every new document, click the Default button.

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

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. 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 common 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' scale. To develop details of the house, you might select the 1-1/2"=1' 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 the output scale and the number of pages before drawing so you can work within the printable area in the Draft window.

Two factors control the printable area. First, the document scale and number of pages specified in the Output Setup dialog box. Second, the page size and orientation specified in the Page Setup dialog box (in Mac OS) or the Printer Setup dialog box (in Windows).

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

Setting page boundaries

You can control how DENEBACAD divides a 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 82*).

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 page values are 1. This means DENEBACAD will print 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 File > Set Print Area. Moving the mouse will move the gray border that represents page boundaries. 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.

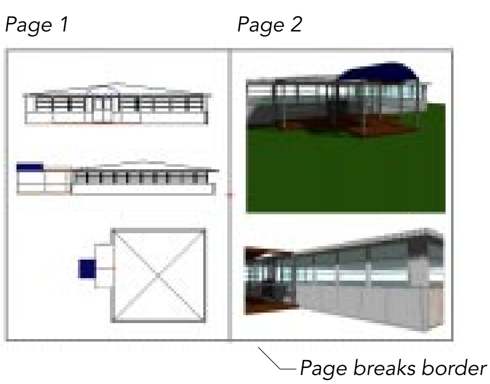


Fig. 82   Printable area in Draft mode

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 be cut off.

For more information about options in the Output Setup dialog box, see “Output Setup” on page 247.

Page setup

You can use the Page Setup command (Mac OS) or Printer Setup command (Windows) in the File menu to set the orientation of printed pages. It’s common to select landscape orientation for printing drawings. The setting you choose affects the printable area in the Draft mode window.

For more information, see “Page Setup (Mac OS)” on page 201, or “Printer Setup (Windows)” on page 202.

GRID SETUP

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

Click the Snap Grid button to turn on grid snap or turn it off. Press the button to select from five settings for grid spacing.

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.

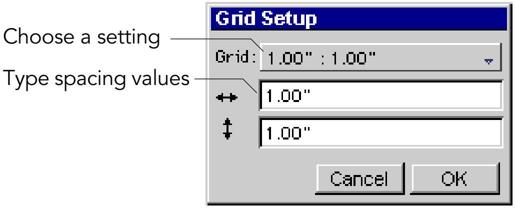
The grid spacing is affected by the units and measurement system specified in the Units Setup dialog box. If you are working in U.S. units, you specify grid spacing in U.S. units.

To specify new grid spacing options

1. Choose Layout > Grid Setup.
2. Choose the grid setting to modify from the Grid pop-up menu.
3. Type the horizontal spacing value in the first text box. Type the vertical spacing value in the second text box.
4. Click OK to implement the Grid setting.

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

◆ **To display the grid:** Choose Layout > Show Grid.



LIBRARY ITEMS

Before you begin to draw, you might want to add saved library objects and symbols to use in a document. You can create and save your own library items, and you can use sample library items distributed on the DENEBACAD CD-ROM.

The Library palette lets 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 many drawings. For example, you will probably use the same elevation symbol in every plan you draw.

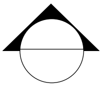


Fig. 83 Elevation symbol

You might also use the same interior door in many house drawings. If this is the case, using the Library palette eliminates the repetitive task of drawing 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

Use this procedure to add Library items to the Library palette so they can be placed in a drawing.

1. Double-click the Library tool icon to open the Library palette.
2. Choose Add in the palette’s pop-up menu under the right triangle button.

3. In the directory dialog box, select a document that contains Library items you want to add to the palette.
4. Use the Library Picker to add items one at a time to the current document (*figure 84*).

Refer to the Library tool section in the 2D and 3D Drawing tools chapter for more information on library objects and symbols.

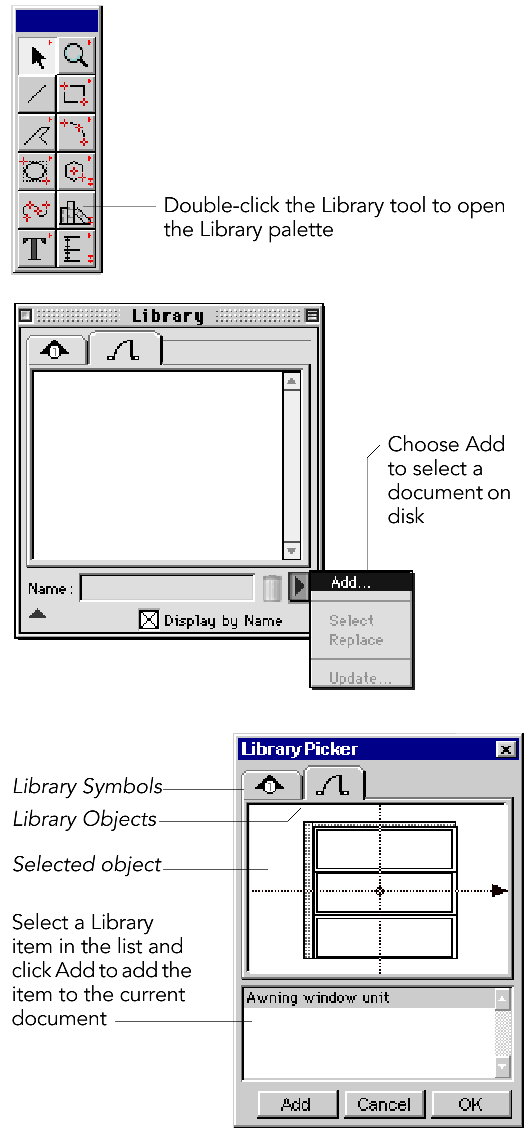


Fig. 84 Adding Library items to a document

USING LAYERS AND CLASSES

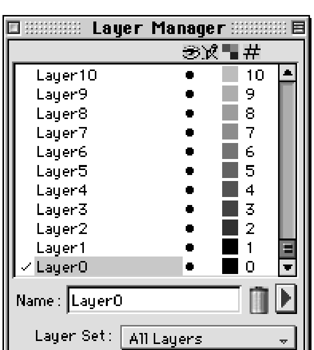
Layers and classes are effective tools for organizing objects and structural components in DENEBACAD drawings.

SETTING UP LAYERS

Layers divide a document into a series of overlays that can be independently displayed and printed.

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

If you do not need to use a standard layer structure, you can name layers according to the 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, for exmaple, you can hide the other layers so they don’t interfere with editing. 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, which is a palette for working with layers. You can define as many as 256 layers before you begin to draw, so you don’t have to go back and add layers later.

For information on using the Layer Manager, see “Layer Manager” on page 363.

SETTING UP CLASSES

DENEBACAD lets you set up *classes*, which are classifications for objects. Using classes lets you generate reports using information that you enter into the class text boxes.

To use classes, you first classify the objects in a document, and then generate reports using the classes you have set up.

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 required quantity of the door knobs.

To use classes, you set up a class system using the Class Manager. You can then assign an object to a class in the Properties Manager, and generate reports using the Analysis Manager.

For more information, see “Class Manager” on page 360, “Analysis Manager” on page 354, and “Properties Manager” on page 375.

SETTING UP 3D EXTRUSIONS

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

You will use extrusion planes if you model a 3D structure in Sculpt mode, or draw 2D plans in Draft mode and then extrude objects to make a 3D model.

Extrusion planes are a set of two planes that define the boundries of a linear extrusion. The planes can be parallel to each other or independently angled.

Note: You can use three extrusion methods in DENEBACAD: Linear, Spin, and Sweep. For Linear extrusion, you define a set of extrusion planes, as 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; setting 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.

The extrusion planes you might set up for a house project would define the height of the walls, the length of the roof and overhangs, the height and depth of countertops and built-ins.

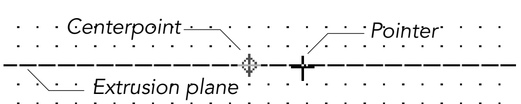
Extrusion planes are also used to set the position of lights for rendering, and the height of camera paths for walkthroughs of 3D models.

The following procedure illustrates how to define extrusion planes for walls by setting planes at floor and ceiling level. With the wall height defined, you can then create 3D walls by drawing the plan of the walls in Top view.

To define extrusion planes for walls

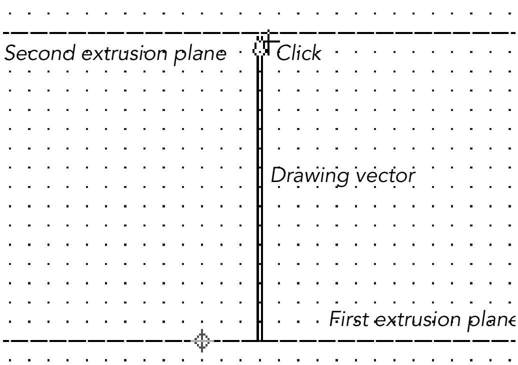
This procedure uses Top view in Draft mode. This is the view and mode of the drawing window when you create a new document. It's best if you can see the Centerpoint (⌖) in the window so you know you are near the origin (0,0) of the document when you begin.

1. Choose Layout > 3D Plane > Define Frontal 3D Plane. 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. Position the first extrusion plane at floor level by moving the the line so it passes through the Centerpoint (the Y coordinate in the Info bar will be `0"`). Click to set the first extrusion plane.
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 plan of the walls in Top view. You can draw the walls in Draft mode and then 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 a name that will make it easy to select the extrusion planes.

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 Layout > 3D Plane > Save 3D Plane. The Save 3D Plane dialog box appears.

2. Type a name for the extrusion planes in the text box.

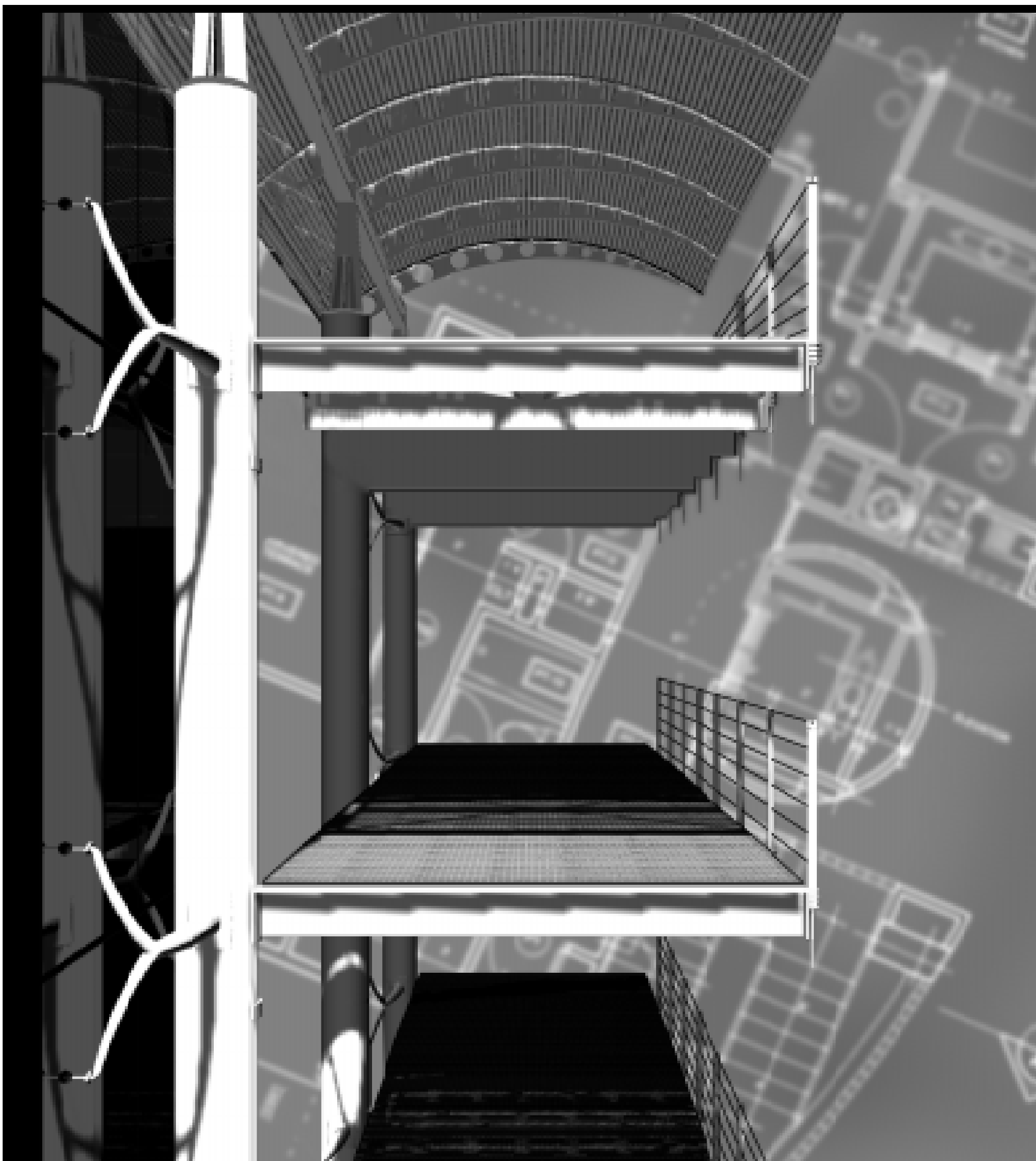
3. Click OK to save the extrusion planes.

Selecting saved extrusion planes

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

REFERENCE GUIDE

TOOLS AND COMMANDS



2D AND 3D DRAWING TOOLS 9

This chapter describes how to use drawing tools to create objects in Draft and Sculpt modes. Information on general 2D and 3D drawing principles is included with instructions for using each tool. The sections on each type of object also include procedures for applying attributes to objects, and methods for editing the shapes of 2D and 3D objects created with each tool.

DRAWING IN DRAFT AND SCULPT MODES

In general, you use the same tools and procedures to create objects in Draft mode and Sculpt mode. However, you draw 2D objects in Draft mode and draw 3D objects in Sculpt mode.

For example, you can use the Rectangle Diagonal tool to draw a 2D rectangle in a Draft mode window. In a Sculpt mode window, the same tool draws a 3D box, one face of which is visible when you draw it.

Therefore, each drawing tool can create two kinds of objects. The object names depend on the mode you use. For example, using an arc tool in Draft mode produces a 2D arc. In Sculpt mode, the same tool produces a 3D curved plane.

Objects created in different modes can often appear to be the same because you see them in only two dimensions (*figure 85*).

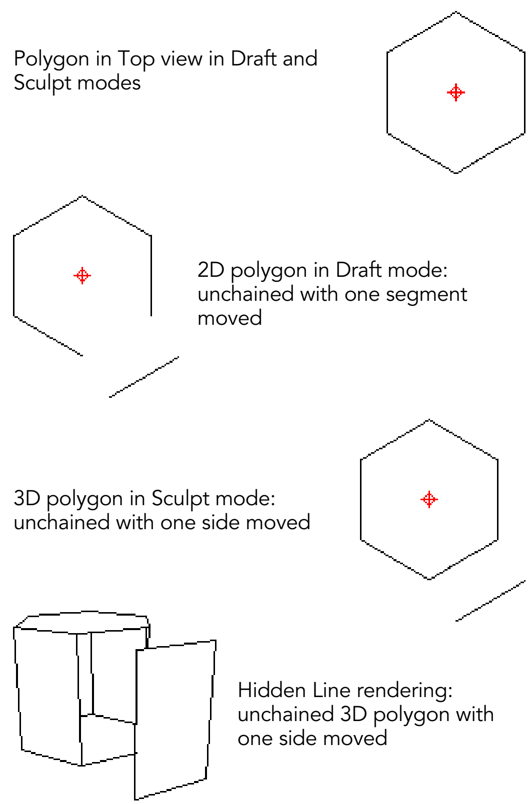


Fig. 85 Objects in Draft and Sculpt modes

Note: You do not create objects in the Render mode window, although you can view 3D objects in Render mode. You create 3D objects in Sculpt mode before rendering them.

Draft mode objects

An object created in Draft mode is a two-dimensional object. A 2D object has a width and a height. You can see a 2D object only when the Draft mode window is set to the same view — Top, Bottom, Left, Right, Front, or Back — in which you created the object. You can't see a Draft mode object in other views because each view is a separate drawing.

Sculpt mode objects

An object created in Sculpt mode is a three-dimensional object. A 3D object is made of chained polygons, which correspond to the faces of the 3D solid. For example, a cube is a 3D object made up of 6 rectangles, which are chained to form the faces of the cube.

You can see a 3D object in any view — Top, Bottom, Left, Right, Front, or Back —in the Sculpt mode window. This is because a conventional 3D object has depth in addition to its width and height. In Top view, when you draw a rectangle, you draw the top face of a cube. If you switch to Bottom view, you see the bottom face, which matches the top face of the cube.

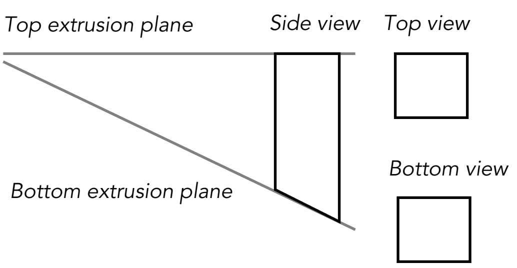


Fig. 86 3D object drawn with inclined extrusion plane

Extrusion methods

When you draw objects in Sculpt mode, or extrude objects in Draft mode, DENEBACAD uses an extrusion method that you specify. The default type of extrusion is a linear extrusion.

Extrusion formats

When you draw objects in Sculpt mode, or extrude objects in Draft mode, DENEBACAD uses an extrusion format that you specify.

There are three extrusion formats: Sides, Front Cap, and Back Cap.

With these formats, you can create 3D objects that are complete solids, or objects that have one or more sides missing.

The default extrusion formats create 3D solid objects with solid sides and both end caps. In this chapter, the discussions of drawing tools and 3D objects generally refer to these types of objects.

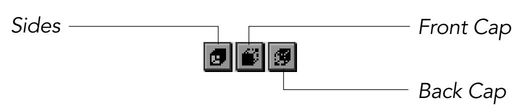


Fig. 87 Extrusion format buttons on the Status bar

When you open a new drawing, all three extrusion formats are selected. Any object you create with all three buttons on will be a complete solid. See "Extrusion Format buttons" on page 186 for more information.

Sculpt mode pointer

The pointer you use in Sculpt mode gives valuable information as to which extrusion formats are selected.

This pointer can be a useful visual reminder, if the Status bar is off, of what extrusion formats are active.

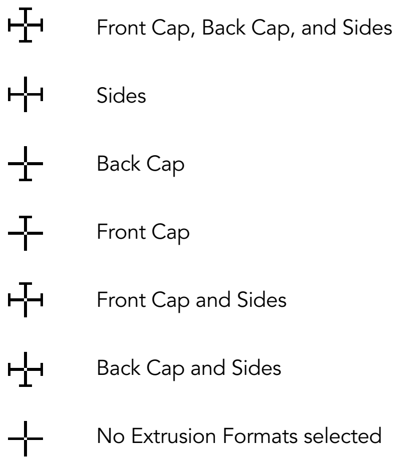


Fig. 88 Sculpt mode pointers

Extrusion planes

The depth to which objects in Sculpt mode are projected in 3D space is specified by the range between two extrusion planes.

The default set of extrusion planes is in effect for new DENEBACAD documents. Using the 3D Plane commands in the Layout menu, you can set the extrusion depth before drawing 3D objects. You can set up new extrusion planes at any time, and also save a set of extrusion planes.

As you create 3D objects, you can change the active set of extrusion planes using the 3D Plane pop-up menu on the Status bar. Additionally, this pop-up menu shows you which set of extrusion planes is active. You can open the menu to select a set of extrusion planes that you have saved, a temporary set of extrusion planes created with the commands in the 3D Plane submenu, or the default set of extrusion planes. See “3D Plane submenu” on page 259 for more information.

Unchaining objects

In Draft mode, if you unchain a 2D polygon withg the Unchain command, the object separates into line segments, which are separate object s.

In Sculpt mode, if you unchain a 3D polygon, it becomes separate polygons, one for each side and face of the original object. You can edit the unchained polygons individually (*figure 85*).

USING DRAWING TOOLS

Objects can be created in Draft and Sculpt modes with the Arc tools, Ellipse tools, Curve tool, Line tool, Polygon tools, Polyline tools, and Rectangle tools. In addition, you can also use the Dimension and Text tools in Draft mode.

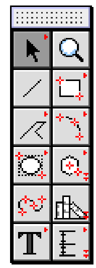
You’ll notice that the tool names refer to the 2D objects created by drawing in Draft mode. Although a few toolbox items change when you switch from Draft mode to Sculpt mode, the drawing tools remain in the same locations in the toolbox.

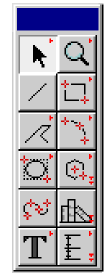
To create an object in either Draft or Sculpt mode, activate the window you want to work in, select a tool, and follow the drawing procedure for the selected tool. When you select a tool, the Help bar displays the basic procedure for drawing objects with the tool.

THE TOOLBOX

The tools you can use for drawing are located in the DENEBACAD toolbox. The toolbox is a small window that floats in front of DENEBACAD drawing windows.

Tools appear in the toolbox as small square icons. You select a tool by clicking its icon. When a tool is selected, it becomes active and its icon is shaded. Only one tool at a time is active.


Mac OS


Windows

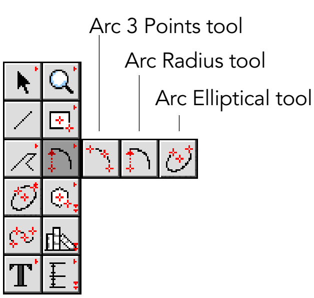
ARC TOOLS

Three tools for drawing arcs and curved surfaces are located in the Arc toolbar in the toolbox.

Arc 3 Points tool Draws a 2D arc or a 3D curved surface through three points you specify.

Arc Radius tool Draws a 2D arc or a 3D curved surface with the radius and length you specify.

Arc Elliptical tool Draws a 2D arc or a 3D curved surface by defining five points.



Arcs are sections of ellipses or circles. In Draft mode, the arc tools draw perfect circular arcs. In Sculpt mode, the arc tools draw curved planes that are made of chained polygons.

You can set the number of segments for 3D arcs in the Preferences dialog box by choosing Edit > Preferences.

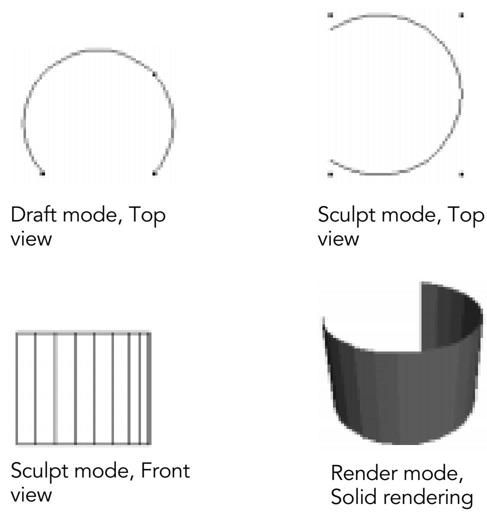


Fig. 89 Arcs in Draft, Sculpt, and Render modes

Arc data types

DENEBACAD uses a data type abbreviation for the name it assigns to objects created by a tool. Data type abbreviations also appear in place of tool names in DENEBACAD's PTF language.

The arc tools have the following data types:

Arc 3 Points	"3PointsArc"
Arc Radius	"CenterRadiusArc"
Arc Elliptical	"EllipticArc"

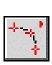
DRAWING ARCS AND CURVED PLANES

Arcs created in Draft mode are two-dimensional objects. Like all 2D objects, arcs appear only in the Draft mode view in which they are created.

When you draw them in Sculpt mode, curved planes look the same as arcs. When you change to other views in Sculpt mode, curved planes have depth produced by the projection of the arc between the active set of extrusion planes.

The "arc" referred to in the following procedures is a 2D arc in Draft mode, or the arc component of a 3D curved plane in Sculpt mode.

To use the Arc 3 Points tool



The Arc 3 Points tool lets you draw an arc through two endpoints and a third point on the arc's perimeter.

1. Select the Arc 3 Points tool in the Arc toolbar.
2. Click to set one endpoint of the arc.
3. Move to the location of the second endpoint. The drawing vector follows the pointer and corresponds to the open part of the arc.
4. Click to set the second endpoint of the arc.
5. Move to a point on the perimeter of the arc. The arc expands or contracts depending on the direction you move the pointer. The drawing vector indicates the chord from the second endpoint.
6. Click to set the perimeter point and finish the arc. The new arc is selected. To deselect the object, click an empty area of the drawing.

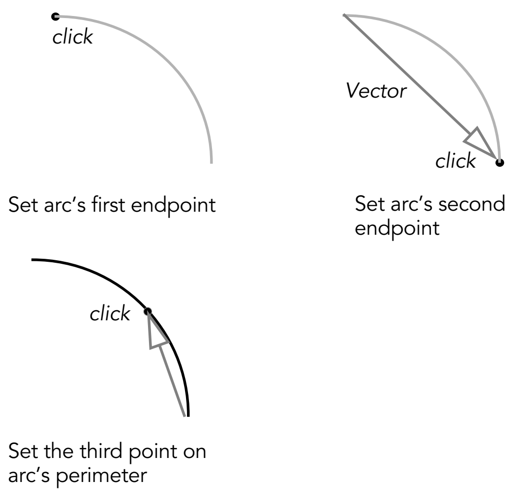


Fig. 90 Using the Arc 3 Points tool

To use the Arc Radius tool

The Arc Radius tool lets you specify the center point, radius, and length of an arc.

1. Select the Arc Radius tool in the Arc toolbar.
2. Click to set the center point. The arc will be a section of a circle centered on this point.
3. Move to the first endpoint of the arc. The outline of a circle and the drawing vector follow the pointer, indicating the arc's radius.

4. Click to set the first endpoint of the arc and establish the radius of the arc. The circle outline disappears.

5. Move to the other endpoint. The drawing vector follows the pointer, rotating around the arc's center. An arc section appears to indicate the arc's length.

6. Click to set the second endpoint and finish the arc. The new arc is selected. To deselect the object, click an empty area of the drawing.

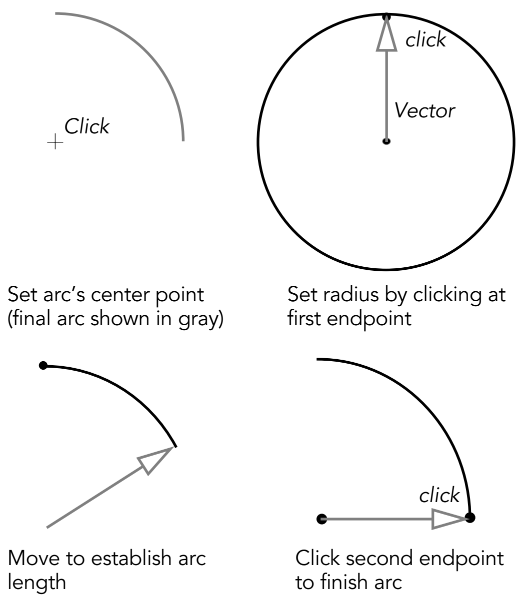


Fig. 91 Using the Arc Radius tool

To use the Arc Elliptical tool

The Arc Elliptical tool lets you specify the center point and endpoints of an arc section of an ellipse.

1. Click the Arc Elliptical tool in the drawing to set the center point. The arc will be a section of an ellipse centered on this point.

2. Move the pointer and click to set the length of an ellipse that contains the arc.

   ▲ Press the Shift key before clicking to snap the vector to 15 degree intervals.

3. Move the pointer and click to set the ellipse width. The drawing vector is perpendicular to the length of the ellipse.

4. Move the pointer and click to set the first endpoint.

   ▲ The length of the drawing vector does not affect the dimension or shape of the arc. The arc's endpoints will be set where the drawing vector crosses the circumference of the established ellipse.

5. Move the pointer and click to set the second endpoint of the arc. The new arc is selected.

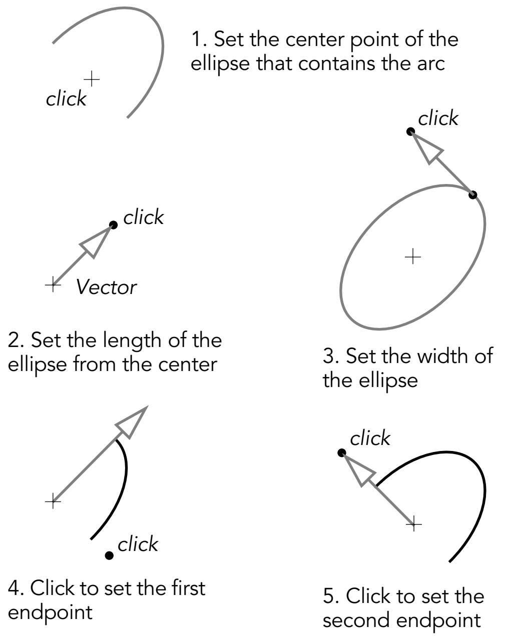


Fig. 92 Using the Arc Elliptical tool

EDITING ARCS

Procedures for editing a 2D arc differ from procedures for editing most other objects. An arc doesn't display a bounding box that can be reshaped, and its creation points are arc-specific.

Editing arcs in Draft mode

To edit the shape or angular length of a 2D arc, you can move the arc's creation points. The position of the creation points depends on the tool used to draw the arc in Draft mode.

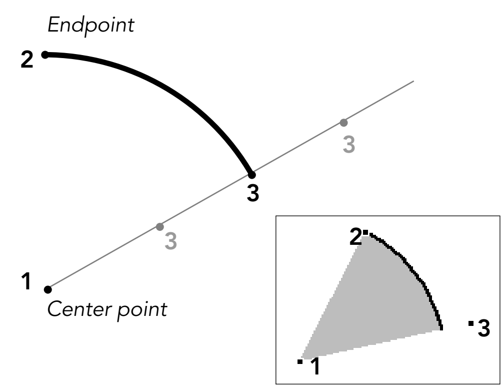
Arc Radius tool An arc drawn with the Arc Radius tool has a creation point at the center, one at the first endpoint, and one at a point indicating the arc's angular length (*figure 93*). This point can be inside or outside the arc; the angle it defines from the first endpoint is significant, but its distance from the arc is not.

Arc 3 Points tool An arc drawn with the Arc 3 Points tool has a creation point at each endpoint, and a third point on the perimeter of the arc.

Arc Elliptical tool An arc drawn with the Arc Elliptical tool has a creation point at the center point of the arc and each point you set to establish the width and length of the arc.

To edit an arc's creation points

1. In the Draft window, select the arc. The arc's creation points appear.
2. Drag a creation point to move it. The drawing vector follows from the point's original location.
3. After you move a creation point, the arc remains selected. You can continue moving points to reshape the arc.
4. When you finish editing the arc, click an empty area of the drawing to deselect it.



The third creation point defines the arc's length. It can appear on the arc, or inside or outside the arc.

Fig. 93 Creation points placed by Arc Radius tool

Editing curved planes in Sculpt mode

Like most other objects in Sculpt mode, curved planes are 3D objects made of chained polygons. The original arc's creation points do not exist in Sculpt mode. However, you can use the Reshape command to display handles of the chained polygons, which you can drag to reshape the curved plane object.

A curved plane can be edited in any view in Sculpt mode.

To edit a curved plane

1. In the Sculpt window, click the curved plane with the Selection tool to select it. The curved plane's bounding box appears.
2. Choose Edit > Reshape, or click the Reshape button. Handles appear at the corners of each polygon in the curved plane.
3. Drag a handle to a new location. As you drag, the drawing vector follows from the handle's original location.
4. After you move a handle, the curved plane remains selected. You can continue moving handles to reshape the curved plane.
5. When you finish editing, click an empty area of the drawing to deselect the object.

Applying commands to arcs and curved planes

You can perform operations on an arc or a curved plane by selecting it and applying commands in the Edit and Object menus.

Many of the commands that you can use in the Object menu and submenu are available only when you select more than one object. For example, to use the Align command, at least two objects must be selected.

Refer to a specific command for more information on how to use it to revise an object.

Edit menu commands The following commands in the Edit menu can be applied to selected arcs or curved planes: Cut, Copy, Clear, Select, Reshape, Duplicate, Create Publisher.

Object menu commands Commands in the Object menu you can apply to selected arcs or curved planes are: Align, Scale, Chain, Unchain, Group, Ungroup.

Commands in the following submenus in the Object menu can be applied to selected arcs or curved planes: Arrange, Copy to Layer, Send to Layer, Combine, Extrude (Draft mode only), Position, Path, Trim.

Unchaining arcs, converting arcs to polygons, and unchaining curved planes

Unchain arcs If you apply the Unchain command to an arc, it becomes a series of separate polyline objects, each made of one straight line segment.

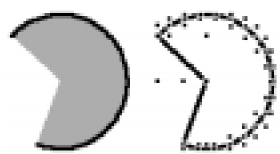
Convert to Polygon Applying the Convert to Polygon command changes an arc into a polygon object. Polygon objects are made of multiple straight segments.

Unchain curved planes If you apply the Unchain command to a curved plane, it becomes a series of separate polygon objects.

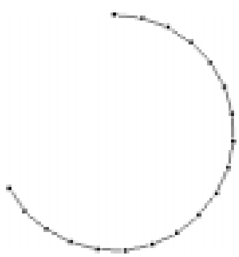
The number of line segments or polygon objects created when an arc or curved plane is unchained or converted to a polygon is determined by a Preference setting in effect when you draw the object. The smaller the line segments, the smoother the arc or curved plane appears after you apply the Unchain or Convert to Polygon commands.

To set the number of segments in arcs and curved planes

1. Choose Edit > Preferences.
2. In the Preferences dialog box, change the Arc step setting. A larger number results in more line segments in unchained arcs or curved planes.



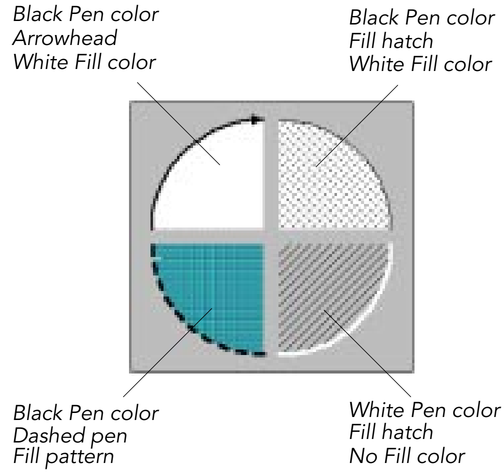
An original arc becomes multiple polylines when the Unchain command is applied.



An arc converted to a polygon is made of a pre-defined number of segments.

Applying attributes to arcs and curved planes

When you draw an arc or curved plane, DENEBACAD applies the current Draft or Sculpt mode attributes to the object. You can modify the object's attributes in the appropriate mode.



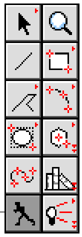
Draft mode Arcs accept the following attributes: Pen color, Pen weight, Pen style; Arrowheads; Fill color, Fill pattern, and Fill hatch.

You can apply a Fill color, a combination of a Fill color and a Fill pattern, or a combination of a Fill color and a Fill hatch to an arc. With fill attributes applied, arcs appear as pie-shaped wedges or pies with slices removed.

Sculpt mode Curved planes accept Pen colors and Surface materials.

CAMERA PATH TOOL

The Camera Path tool creates camera paths. This tool is located in the toolbox.



The Camera Path tool draws a camera path by setting a series of points. The camera path is also referred to as a walkthrough path.

There are three parts to a walkthrough path. They are the camera location, the camera sightline and the path line.

- The handle at the base of the arrow indicates the location of the camera.
- The arrow indicates the camera's sightline. The sightline is set to the direction and angle the camera is pointed. The length of the sightline determines the duration of the view.
- The path line represents the path the camera follows through the model.

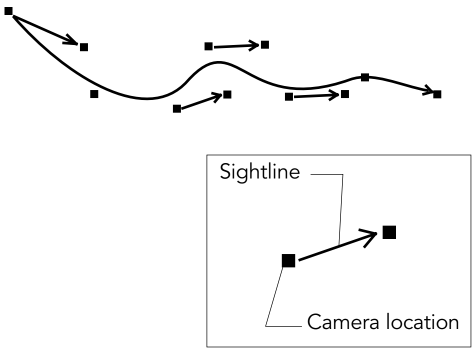


Fig. 94 Parts of a walkthrough path

You create a walkthrough path in Sculpt mode, but you use the walkthough path in Render mode.

You view a walkthrough of a model in the Render mode using the Walkthrough command.

You also use the walkthrough path as the path the camera follows when you generate a QuickTime movie in Render mode.

Data type

DENEBACAD uses a data type abbreviation for the name it assigns to objects created by a tool. Data type abbreviations also appear in place of tool names in DENEBACAD's PTF language.

The Camera Path tool has the data type "Walkthrough" in DENEBACAD.

To use the Camera Path tool

The Camera Path tool lets you establish the walkthrough path the camera follows by setting a series of points. The following procedure is for creating a simple camera path through a plan view of a model. This procedure can be used with any view.

1. Activate or open a Sculpt mode window and set the view to Top.

   ▲ Because you are creating a 3D path in only two dimensions, you need to create the walkthrough path in one view and specify the vertical angles of the camera's sightlines in a different view.

2. Open or activate a second Sculpt mode window and set the view to Front.

   ▲ This lets you see height of the cameras. You can edit the vertical angle of the sightlines in this window, once you've completed your walkthrough.

3. Select a saved set of extrusion planes, or define a new set. The walkthrough path appears on the lower extrusion plane of the active set.

   ▲ The Info bar shows the location of the lower extrusion plane. The lower number indicates the lower extrusion plane.

Lower extrusion plane

4. Select the Camera Path tool in the toolbox.

5. In the Top view window, click to set the first camera point at the start of the path.

6. Move and click to set the second camera point. The drawing vector follows from the first camera point.

7. Move and click to set the next camera point. Continue to move and click to set other camera locations.

8. Double-click to set the end point of the walkthrough path. Or, press Return or Enter to finish the walkthrough path. A dialog box appears.

9. Type a name for the walkthrough path and click OK. The new walkthrough path is selected.

EDITING A WALKTHROUGH PATH

You can move the entire walkthrough path by dragging it to another location.

To edit a walkthrough path, you can move the points that represent the cameras' locations, or the points that represent the cameras' sightlines.

The points that represent the cameras' locations are the creation points.

Moving a camera handle also alters the pathline because you are moving a point that created the walkthrough path.

You can also move the points that represent the cameras' sightlines. The handle at the tip of the arrow is the point you move to change the direction of the sightline.

Moving a sightline handle won't alter the pathline because you are editing the direction the camera is pointing.

To edit a camera path

1. Activate a Sculpt window set to Top view to change the camera location, the horizontal direction of a sightline, or the shape of the walkthrough path.

   ▲ To change the height of the camera or the vertical angle of a sightline, use a Sculpt mode window set to Front view.

2. Select the walkthrough path. Handles appear at the creation points and the ends of the camera sightlines.
3. Drag a handle to a new location. The drawing vector follows from the original handle location.
4. After you move a creation point, the walkthrough path remains selected. You can continue moving points to reshape the walkthrough path.
5. Click an empty area of the drawing when you finish editing the walkthrough path. The creation points are no longer visible.

Applying commands to walkthrough paths

You can perform operations on a walkthrough path by selecting it and applying commands in the Edit and Object menus.

Many of the commands that you can use in the Object menu depend on selecting additional objects, or establishing certain attributes to perform the operations properly.

For example, to apply the Align command to a selected walkthrough path, at least one other object must be selected.

Refer to a specific command for more information on how to use it to revise an object.

Edit menu commands The following commands in the Edit menu can be applied to selected walkthrough paths: Cut, Copy, Clear, Select, Reshape, Duplicate, Create Publisher.

Object menu commands The following Object menu commands can be applied to selected walkthrough paths: Align, Scale, Group, and Ungroup.

The commands found in the following submenus in the Object menu can be used to edit walkthrough paths: Arrange, Copy to Layer, and Send to Layer.

Applying attributes to walkthrough paths

When you draw a walkthrough path, DENEBACAD applies the current Pen color. You can change the Pen color of a walkthrough path in Sculpt mode.

CURVE TOOL

The Curve tool draws 2D curves or 3D curved planes defined by creation points. The Curve tool is the fifth from the top in the first column of the toolbox.

In Draft mode, the Curve tool draws curved paths. In Sculpt mode, the Curve tools draws curved planes, which are 3D objects composed of chained polygons.

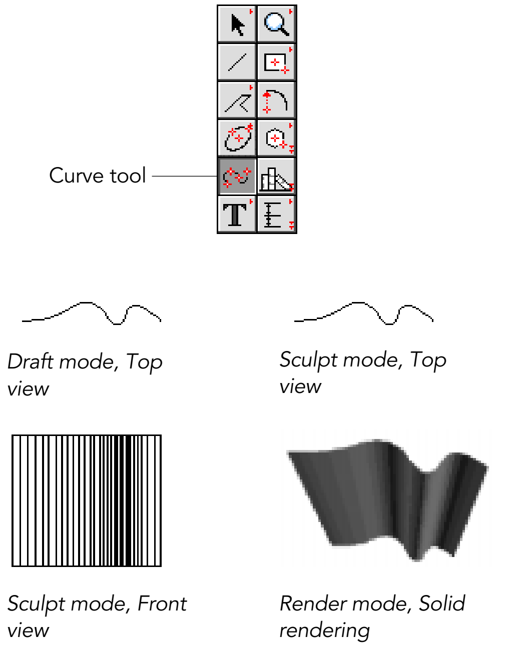


Fig. 95 Curves drawn in Draft, Sculpt, Render modes

Data type

DENEBACAD uses a data type abbreviation for the name it assigns to objects created by a tool. Data type abbreviations also appear in place of tool names in DENEBACAD's PTF language.

The Curve tool has the data type "QB_Spline."

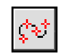
DRAWING CURVES AND CURVED PLANES

Curves drawn in Draft mode are two dimensional objects that can be seen in only the view they were created in.

Curved planes drawn in Sculpt mode look the same as two-dimensional curves from the view they were created in. Curved planes can be seen in other views because they have depth.

The curve referred to in the following procedures is the curve object in Draft mode, and the curved path of the curved plane in Sculpt mode.

To use the Curve tool



The Curve tool lets you create single lines with curved or straight segments. When you use the tool, you set successive points to define the curved line.

1. Click the Curve tool in the drawing to set the first point of the curve.
2. Move and click to set the second point. As you move the pointer, the drawing vector follows from the first point.
3. Move and click to set the third point. Continue to move and click to set successive points on the curve.
4. Double-click to set the last point and finish the curve. Or, press Return or Enter to finish the curve after clicking the last point. The new curve is selected.

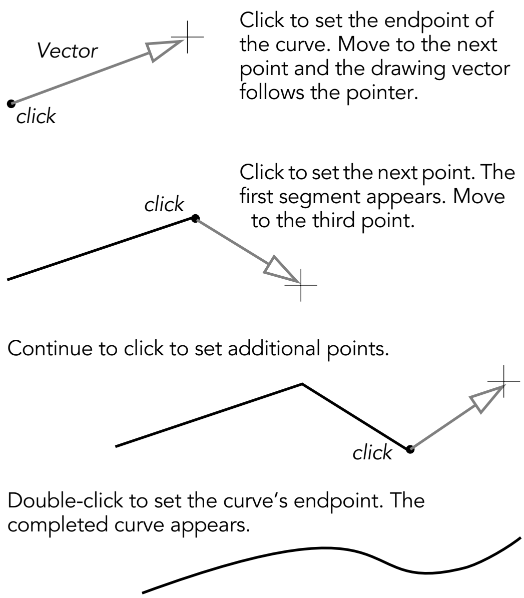
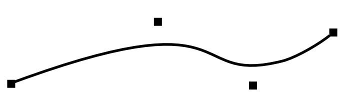


Fig. 96 Drawing with the Curve tool

EDITING CURVES AND CURVED PLANES

After drawing a 2D curve, you can move the creation points to change the shape and dimension of the curve.



To edit a curve's creation points

1. Select the curve with the Selection tool. The curve's creation points appear.
2. Drag a creation point to a new location. As you drag, the drawing vector follows from the original location.
3. After you move a creation point, the curve remains selected. You can continue moving points to reshape the curve.
4. Click an empty area of the drawing to deselect the object when you finish editing.

Editing curved planes in Sculpt mode

Like most other objects in Sculpt mode, curved planes are 3D objects composed of chained polygons. A curved plane's creation points do not exist in Sculpt mode. You can, however, use the Reshape command to display and drag the handles of the chained polygons to reshape the curved plane.

A curved plane can be edited in any view. Refer to "Orthogonal View commands" on page 337 for more information on changing views.

To edit a curved plane

1. Select the curved plane with the Selection tool. The curved plane's handles appear.
2. Choose Edit > Reshape, or click the Reshape button. The handles of each chained polygon are visible.
3. Drag a handle to a new location. As you drag, the drawing vector follows from the original location.
4. After you move a handle, the curved plane remains selected. You can continue moving handles to reshape the curved plane.
5. Click an empty area of the drawing when you finish editing the curved plane.

Applying commands to curves

You can perform operations on a curve or a curved plane by selecting it and applying commands in the Edit and Object menus.

Many of the commands that you can use in the Object menu and submenu are dependent on selecting additional objects, or establishing certain attributes to perform the operations properly.

For example, to apply the Align command to a selected curve or curved plane, at least one other object must be selected.

Refer to a specific command for more information on how to use it to revise an object.

Edit menu commands The following commands in the Edit menu can be applied to selected curves or curved planes: Cut, Copy, Clear, Select, Reshape, Duplicate, Create Publisher.

Object menu commands The following Object menu commands can be applied to selected curves or curved planes: Align, Scale, Chain, Unchain, Group and Ungroup.

The commands found in the following submenus in the Object menu can be used to edit curves or curved planes: Arrange, Copy to Layer, Send to Layer, Combine, Extrude (Draft mode only), Position, Path, and Trim.

Unchaining curves and curved planes

If you apply the Unchain command to a curve, it becomes a series of separate polyline objects, each made of one straight line segment.

If you apply the Unchain command to a curved plane, it becomes a series of separate polygon objects.

The number of line segments in an unchained curve or curved plane is determined by a Preference setting in effect when you draw the object. The smaller the line segments comprising the object, which is determined by the size of the object and the number of line segments, the smoother the object's outline appears.

To set the segment preference for curves

1. Choose Edit > Preferences.
2. In the Preferences dialog box, change the Curve Step setting. A larger number results in more line segments in unchained curves or curved planes.

Applying attributes to curves and curved planes

When a new curve or curved plane is created, it is assigned the current attributes. You can modify an existing object by changing its attributes.

Draft mode Curves accept the following attributes: Pen color, Pen weight, Pen style; Arrowheads; Fill color, Fill pattern, and Fill hatch.

You can apply a Fill color, a combination of a Fill color and a Fill pattern, or a combination of a Fill color and a Fill hatch to a curve.

Sculpt mode Curved planes accept the following attributes: Pen color and Surface materials.

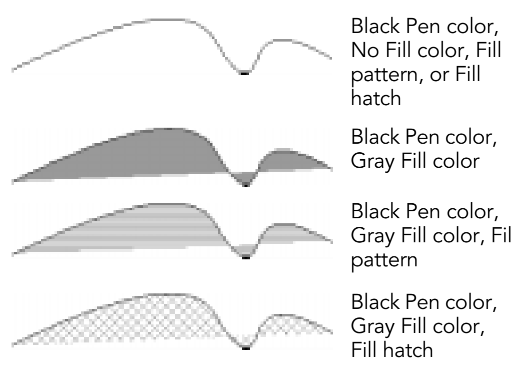


Fig. 97 Attributes applied to curve objects

DIMENSION TOOLS

DENEBACAD provides five tools to dimension objects and a tool to annotate objects. These tools are located in the Dimensions toolbar.

Chain Dimension tool Draws a chain of dimensions from point to point.

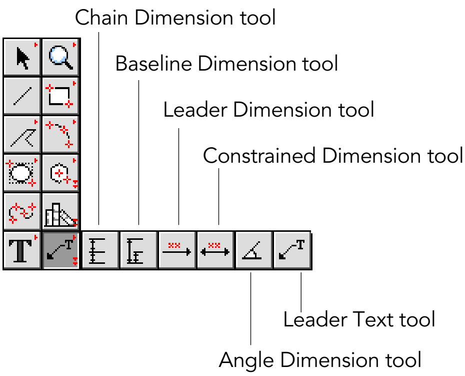
Baseline Dimension tool Draws a series of measurements made from a common starting point.

Leader Dimension tool Lets you draw dimensions by setting two points.

Constrained Dimension tool Lets you draw dimensions by setting two points.

Angle Dimension tool Draws dimensions by defining a radius and an angle.

Leader Text Lets you draw a leader line and place text to annotate objects.



Dimension lines have settings such as the size of lines, gaps, text, and tolerances that can be customized by double-clicking a dimension tool icon and using the tool's dialog box, or by selecting an existing dimension line and using the Tools tab in the Properties Manager.

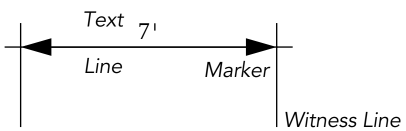


Fig. 98 Parts of a dimension object

Data types

DENEBACAD uses a data type abbreviation for the name it assigns to objects created by a tool. Data type abbreviations also appear in place of tool names in DENEBACAD's PTF language.

The Dimension tools have the data type listed below.

Chain Dimension	"ChainDimm"
Baseline Dimension	"BaseLineDimm"
Leader Dimension	"LeaderDimm"
Constrained Dimension	"DoubleLeaderDimm"
Angle Dimension	"AngleDimm"
Leader Text (line)	"Polyline"
Leader Text (text)	"Text"

To use the Chain Dimension tool

The Chain Dimension tool draws a chain of dimensions from point to point.

1. Select the Chain Dimension tool from the Dimensions toolbar.

   ▲ To change the current properties of the tool, double-click the Chain Dimension icon to open the Chain Dimension dialog box.

2. Click to set the first point of the first dimension in the chain.

3. Move and click to set the second point. The drawing vector follows from the first point as you move.

   ▲ The dimensioned length is between the two points, and the points are the endpoints of the witness lines, if Long is selected in the Witness Line pop-up menu in the tool's dialog box; otherwise, the witness lines are the same length).

4. Move and click to set the third point, which sets the offset of the dimension line.

5. Move and click to set the fourth point. As you move the pointer, the drawing vector follows. The fourth point defines the second dimension in the chain and is the endpoint of the third witness line.

6. Continue to move the pointer and click to set successive dimensions in the chain.

7. To finish the chain of dimensions, double-click to set the last point. Or, click to set the last point and then press Return or Enter. The new dimension chain is selected. To deselect the object, click an empty area of the drawing.

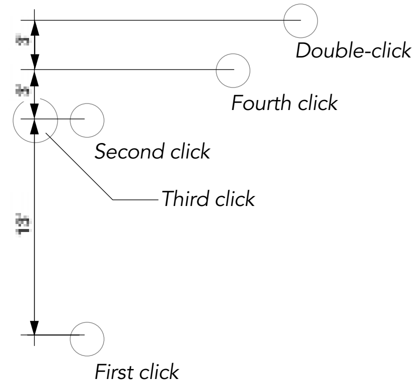


Fig. 99 Drawing chain dimensions

To use the Baseline Dimension tool

The Baseline Dimension tool draws a series of measurements from a common starting point.

1. Select the Baseline Dimension tool from the Dimensions toolbar.

   ▲ To change the current properties of the tool, double-click the icon to open the Baseline Dimension dialog box.

2. Click the first point to set the starting point of the dimensions.

3. Move and click the second point. The drawing vector follows from the first point.

   ▲ The dimensioned length is between these two points, and the points are the endpoints of the witness lines, if Long is selected in the Witness Line pop-up menu in the tool's dialog box; otherwise, the witness lines are the same length.

4. Move and click the third point, which sets the offset of the first dimension line.

5. Move and click the fourth point. This defines the second dimension and is the endpoint of the second witness line of the second dimension.
6. Continue to move the pointer and click to set successive points.
7. To finish the dimensions, double-click to set the last point. Or, click to set the last point and then press Return or Enter. The new dimension object is selected. To deselect the object, click an empty area of the drawing.

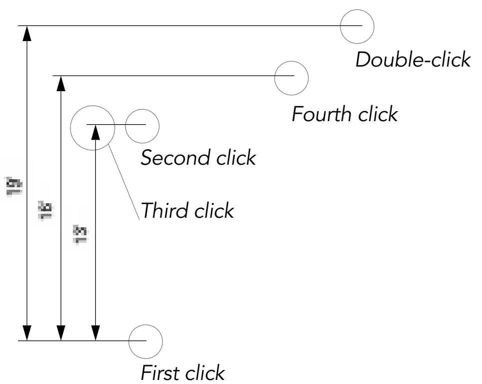
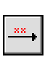


Fig. 100 Drawing baseline dimensions

To use the Leader Dimension tool



The Leader Dimension tool lets you draw dimensions by setting two points.

1. Select the Leader Dimensions tool from the Dimensions toolbar.

   ▲ To change the current properties of the dimension line, double-click the Leader Dimension icon to open the Leader Dimension dialog box.

2. Click to set the first endpoint of the dimension line.
3. Point to the location of the second point. As you move the pointer, the drawing vector follows to the location of the second endpoint, which is also the location of the dimension marker.
4. Click to set the second endpoint. The new dimension line is selected.To deselect the object, click an empty area of the drawing.

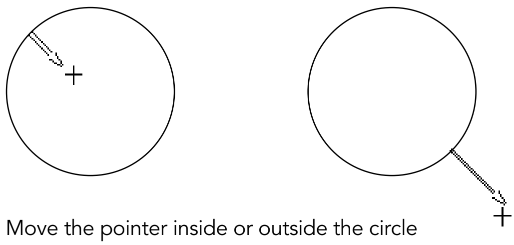
Fig. 101 Drawing leader dimensions

To use the Leader Dimension tool with circles and arcs

The Leader Dimension tool lets you dimension the radius of a circle or an arc by selecting the object and setting a point for the location of the marker of the dimension line.

1. Select the Leader Dimension tool from the Dimensions toolbar.
2. Click the perimeter of the circle or arc to snap the drawing vector to the perimeter.
3. Move the pointer and click to set the angle and length of the dimension line.

   ▲ You can move the pointer inside or outside the circle or arc. If you move the pointer inside, the dimension line will appear inside the object. If you move the pointer outside, the dimension line will appear outside the object.



Move the pointer inside or outside the circle

4. The dimension line, with its marker on the perimeter of the circle or arc, is selected. To deselect the object, click an empty area of the drawing.

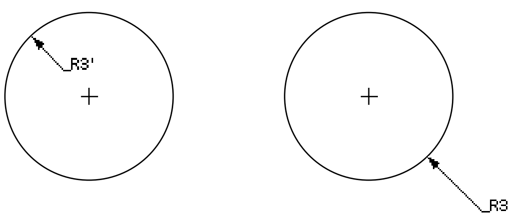
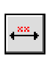


Fig. 102 Circle with leader dimensions

To use the Constrained Dimension tool



The Constrained Dimension tool lets you draw dimensions by setting two points.

1. Select the Constrained Dimension tool from the Dimensions toolbar.

   ▲ If you need to change the current properties of the dimension line, double-click the Constrained Dimension icon to open the Constrained Dimension dialog box.

2. Click to set the first endpoint of the dimension line. Move the pointer to set the second point. As you move the pointer, the drawing vector follows to the location of the second endpoint.

3. Click to set the second endpoint. The new dimension line is selected. To deselect the object, click an empty area of the drawing.

Fig. 103 Constrained dimension

To use the Constrained Dimension tool with circles and arcs

The Constrained Dimension tool lets you dimension the diameter of a circle or an arc.

1. With the Constrained Dimension tool, click the perimeter of the circle or arc to snap the drawing vector to the perimeter.

2. Move the pointer and click to set the angle and length of the dimension line. The drawing vector follows the pointer from the perimeter of the object.

   ▲ You can move the pointer inside or outside the circle or arc. If you move the pointer inside, the dimension line will appear inside the object. If you move the pointer outside, the dimension line will appear outside the object.

3. The dimension line with its marker on the perimeter of the circle or arc is selected. To deselect it, click an empty area of the drawing.

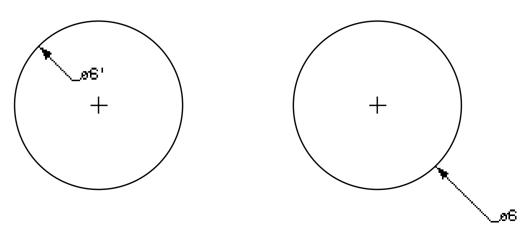


Fig. 104 Constrained dimension of a circle

To use the Angle Dimension tool

The Angle Dimension tool lets you draw dimensions by defining a radius and an angle.

1. Select the Angle Dimension tool from the Dimensions toolbar.

   ▲ If you need to change the current properties of the dimension line, double-click the Angle Dimension icon to open the Angle Dimension dialog box.

2. Click to set the center point of the dimensioning arc.
3. Point to the location of the first endpoint of the arc. As you move the pointer, a circle expands from the center and the drawing vector indicates the arc's radius.
4. Click to set the first endpoint of the dimensioning arc.
5. Point to the location of the second endpoint. As you move the pointer, the drawing vector rotates around the arc's center. The position of the pointer establishes the angle and length of the radius.
6. Click to set the angle and length. The dimension line is selected. To deselect the object, click an empty area of the drawing.

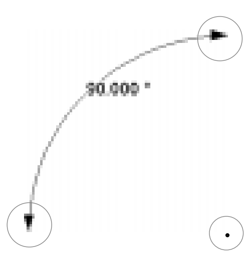


Fig. 105 Drawing angle dimensions

To dimension non-parallel lines

The Angle Dimension tool lets you dimension the angle of two non-parallel lines. The lines do not have to touch and can intersect.

1. Select the Angle Dimension tool from the Dimensions toolbar.
2. Click the first line of the angle you want to dimension.
3. Move and click the second line. As you move the pointer, the drawing vector follows from the first point.

   ▲ Where you click does not affect the dimensioned angle. DENEBACAD calculates the angle based on the relationship of one line to the other. The radius of the dimensioning arc is the intersection of the two lines.

4. Move to position the dimensioning arc. As you move the pointer, the drawing vector follows from the point you set on the second line. The dimensioning arc expands from the intersection point of the two lines.
5. Click to set the dimensioning arc. The dimension line is selected. To deselect it, click an empty area of the drawing.

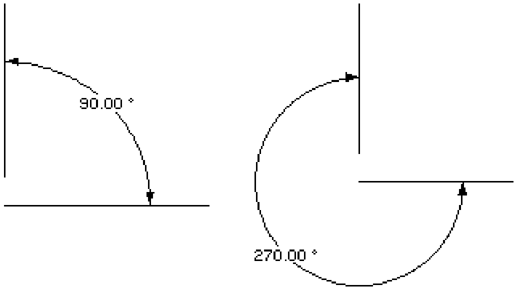


Fig. 106 Angle dimensions of two lines

To use the Leader Text tool

The Leader Text tool lets you draw a leader line by setting points along a path. You can then type text at the end of the leader line.

1. Select the Leader Text tool from the Dimensions toolbar.

   ▲ To change the tool settings, double-click the Leader Text icon to open the Leader dialog box.

2. Click to set the first point of the leader line. This point is the location of the marker.
3. Move and click to set the second point. As you move the pointer, the drawing vector follows from the first point.

   ▲ To draw constrained line segments, press Shift while moving the mouse. The line will snap at 15 degree angles.

4. Move and click to set the third point of the leader line.
5. Continue to move the pointer and click to set each successive point on the leader line.
6. Double-click to set the last point and finish the leader line. Or, click to set the last point and press Return or Enter to finish the line. The new leader line is selected. An additional line segment and insertion point appear at the endpoint of the leader line.

   ▲ DENEBACAD adds a horizontal line segment at the endpoint of the leader line to indicate the final placement of the text. You can change the length of this line in the Leader dialog box.

7. Type to enter text in the document. When you finish typing, press Enter. The text appears with the current attributes and is centered at the endpoint of the horizontal line segment. The leader line and the text, which are separate objects, are selected.
8. To deselect the objects, click an empty area of the drawing.

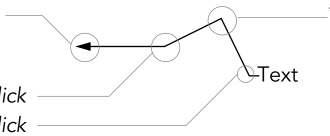


Fig. 107 Drawing leader text

APPLYING COMMANDS TO DIMENSIONS

You can perform operations on dimensions by selecting them and applying commands in the Edit and Object menus.

Some commands, such as align, require that you select more than one object. Refer to a specific command for more information on how to use it.

Edit menu commands The following commands in the Edit menu can be applied to selected dimensions: Cut, Copy, Clear, Select, Reshape, Duplicate, Create Publisher.

Object menu commands The following Object menu commands can be applied to selected dimensions: Align, Scale, and Group.

The commands found in the following submenus in the Object menu can be used to edit dimensions: Arrange, Copy to Layer, Send to Layer, and Position.

RESHAPING DIMENSIONS

In Reshape mode, you can change the length of dimensions, move dimension text, change the angle of dimension lines, or change the length of witness lines.

You can reshape dimension objects in Draft or Sculpt windows. To reshape a dimension object in a Sculpt window, select “View 3D & 2D Objects” in the View Options pop-up menu.

You can use the Reshape command to reshape dimension objects created with the Chain or Baseline Dimension tools.

You do not need to use the Reshape command to reshape dimension objects created with the Constrained, Leader, or Angle Dimension tools. These objects appear in Reshape mode when you select them with the Selection tool.

To reshape Chain or Baseline dimensions

Select the dimension object and choose Edit > Reshape, or click the Reshape action button. The dimension object’s creation points appear.

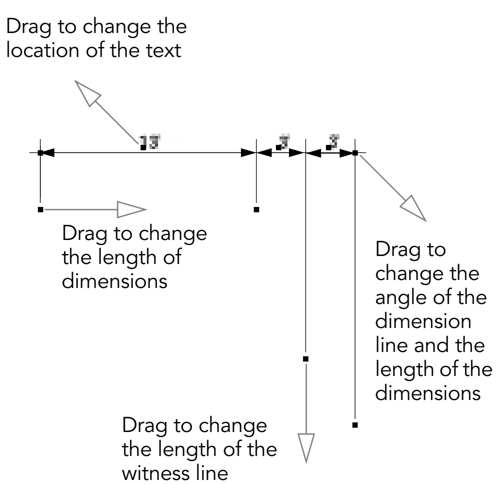
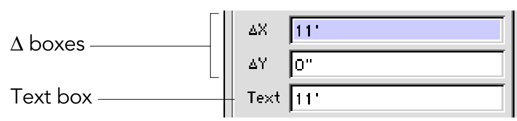


Fig. 108 Chain Dimension in Reshape mode

EDITING DIMENSIONS

Editing the text of a dimension can affect the dimension line. If you add to the measurement of a dimension in the Properties Manager, using the “Δ” boxes, you lengthen the dimension line. If you subtract from a measurement, you shorten the dimension line.

You can change the dimension text without changing the length of the dimension line by entering a new value in the “Text” box in the Properties Manager. You can also type words in this box.



You can edit the text in dimensions in Draft and Sculpt mode. Refer to “Coordinates tab” on page 380 for more information on how to edit dimensions and dimension text using the Properties Manager.

Applying attributes to dimensions

A dimension is assigned the current attributes when you draw it. You can then modify its attributes.

Dimensions accept Pen color, Pen weight, Pen style, and Arrowheads in Draft mode.

SETTING UP DIMENSION TOOLS

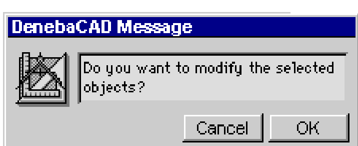
Each Dimension tool has a dialog box you can use to set up dimension objects with arrowheads, witness lines, and text formatting.

To set up a Dimension tool, double-click its icon in the toolbox. The dialog box for the tool appears. You can set up a dimension tool with or without objects selected.

You can choose standard settings for dimension objects, or you can specify custom settings. You can save custom settings to use later.

Changing a tool with a dimension selected

If you double-click a Dimension tool when a dimension object is selected, the settings in the dialog box do not reflect the settings of the dimension object. If you change the dialog box settings, DENEBACAD asks you if you want to modify the selected object.



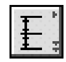
Click Yes to modify the selected object using the new settings for the Dimension tool.

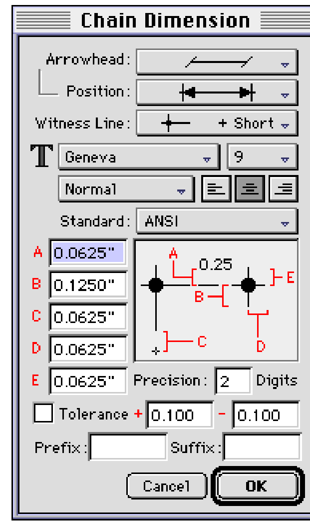
To change the attributes of a dimension object without affecting the settings for a Dimension tool, double-click the dimension object, then use the Properties Manager to edit the object's attributes.

To see the settings of dimension object, double-click the object. You can review its settings in the Properties Manager.

To set up a Dimension tool

1. Double-click a Dimension tool icon in the toolbox. The dialog box for that Dimension tool appears.





2. To use standard settings, choose a standard (ANSI, DIN, ISO, BS-380, or JIS) in the Standard pop-up menu. To customize the tool settings, use the options described next.

   ▲ If you change a setting, "custom setting" appears in the Standard pop-up menu, and the Save Settings command becomes available in the pop-up menu.

3. Click OK to apply the settings to the Dimension tool.

To save custom settings

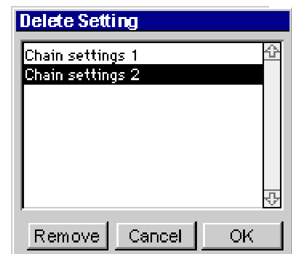
You can save settings using the Save Settings command in the Standard pop-up menu.

1. Set up the Dimension tool options.
2. Choose Save Settings in the Standard pop-up menu. In the Save Setting dialog box, type a name for the settings.

3. Click OK to save the custom settings. DENEBACAD returns to the Dimension tool dialog box. The name of the new saved settings appears in the Standard pop-up menu. If you want to use other settings, choose the name in the Standard pop-up menu.
4. Click OK to apply the current settings in the Dimension tool dialog box to the Dimension tool.

To delete custom settings

1. Double-click the Dimension tool icon.
2. Choose the name of the custom settings to delete in the Standard pop-up menu. The Delete Setting dialog box appears.



3. In the Delete Setting dialog box, select the settings to remove. The Remove button becomes available.
4. To delete the selected settings, click Remove, and then click OK.

DIMENSION TOOL OPTIONS

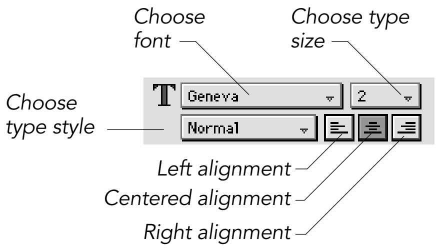
Each Dimension tool has its own dialog box of options. Some dimension tools don't have all of the options described here.

Arrowhead Choose an arrowhead from the pop-up menu. The current arrowhead appears in the menu box. You can choose from arrowheads that appear on the Arrowhead tab in the Pen palette.

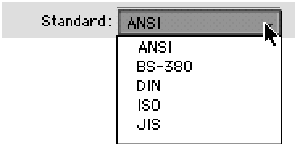
Position Choose placement of arrowheads by choosing one of the graphics in the Position pop-up menu. This option is available for the Baseline and Chain Dimension tools.

Witness line Choose Long or Short in the pop-up menu to set the length of witness lines. This option is available for the Baseline and Chain Dimension tools.

Text options Use the pop-up menus to specify font, type size, and type style for the text in dimension objects. Use the buttons to specify text alignment within dimension objects.



Standard If you want to use standard settings for all options in a Dimension dialog box, choose the name of the standard in the pop-up menu.



The name appears in the Standard box, and the standard's settings appear throughout the Dimension tool's dialog box. The Standard menu contains the following dimension object standards:

ANSI American National Standards Institute

BS-380 British Standards Institute

DIN Deutches Institut fur Normung

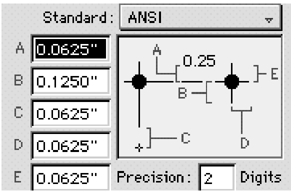
ISO International Organization for Standardization

JIS Japanese Industrial Standard

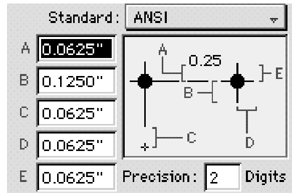
The Standard pop-up menu also contains commands to save and delete custom settings that you create.

Position and spacing values The values in the text boxes labeled A, B, C, D, and E control spacing and positioning of parts of a dimension object. The adjacent diagram in the dialog box identifies the items for each Dimension tool. You can type new values in the text boxes to set custom spacing and positioning.

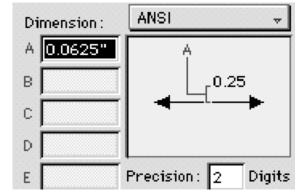
Certain position and spacing options are not available for all Dimension tools. No position and spacing options are available for the Angle Dimension tool.



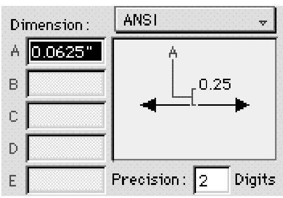
Chain Dimension tool



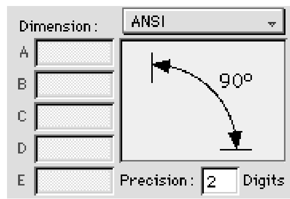
Baseline Dimension tool



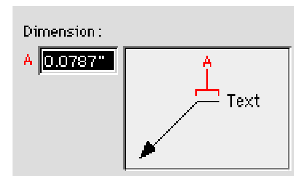
Leader Dimension tool



Constrained Dimension tool



Angle Dimension tool

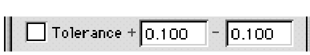


Leader Text tool

Fig. 109 Dimension tools position and spacing settings

Precision Type the number of digits (0 to 16) to include after the decimal point in measurements.

Tolerance Select the Tolerance check box to display the tolerance values in the (+) and (-) text boxes in the dimension text. To set Tolerance values, type numbers in the (+) or (-) text boxes.



Prefix Type the text in the Prefix text box that you want placed in front of the dimension text.

Suffix Type the text in the Suffix text box that you want placed behind the dimension text.

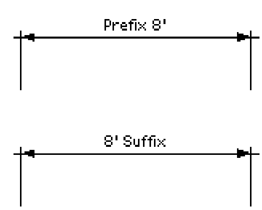


Fig. 110 Examples of Prefix and Suffix

ELLIPSE TOOLS

DENEBACAD provides five tools to draw circles, ellipses and cylinders. These tools are located together in the Ellipse toolbar in the toolbox.

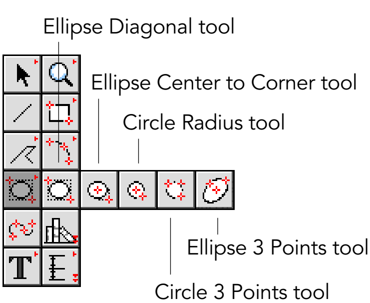
Ellipse Diagonal tool Draws a circle, ellipse, or cylinder from two points defining the corners of a bounding box.

Ellipse Center to Corner tool Draws a circle, ellipse or cylinder from the center to the corner of a bounding box.

Circle Radius tool Draws a circle or cylinder from the center with the radius that you specify.

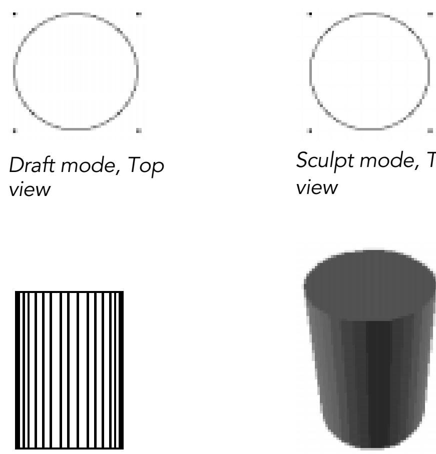
Circle 3 Points tool Draws a circle or cylinder whose circumference passes through three points you specify.

Ellipse 3 Points tool Draws an ellipse or cylinder by defining the center point, length and width of the object.



Circles are closed 2D shapes in which all points on the object's outline are equidistant from one point, the circle's center. Ellipses are a deviation of a circle.

In Draft mode, the ellipse tools draw 2D ellipses or circles. The circle tools draw only perfect circles. In Sculpt mode, the ellipse and circle tools create 3D cylinder-shaped objects, which are made of chained polygons.



Draft mode, Top view

Sculpt mode, Top view

Sculpt mode, Front view

Render mode, Solid rendering

Fig. 111 Ellipses in Draft, Sculpt, Render modes

Data types

DENEBACAD uses a data type abbreviation for the names of objects and in DENEBACAD's PTF language. The ellipse tools have the following data types:

Ellipse Diagonal	"DiagonalEllipse"
Ellipse Center to Corner	"CenterEllipse"

Circle Radius	“CenterRadiusCircle”
Circle 3 Points	“3PointsCircle”
Ellipse 3 Points	“FreeEllipse”

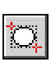
DRAWING CIRCLES, ELLIPSES AND CYLINDERS

Circles and ellipses created in Draft mode are two-dimensional objects. Like all 2D objects, ellipses and circles appear only in the Draft mode view in which they are created.

When you draw them in Sculpt mode, cylinders look the same as ellipses or circles. When you change to other views in Sculpt mode, cylinders have depth produced by the projection of the ellipse or circle between extrusion planes.

The circle or ellipse referred to in the following procedures is a 2D circle or ellipse in Draft mode, or the circle or ellipse component of a cylinder in Sculpt mode.

To use the Ellipse Diagonal tool



The Ellipse Diagonal tool lets you specify opposite corners of a bounding box to draw an ellipse or circle. If the bounding box is a square, the tool draws a circle.

1. Select the Ellipse Diagonal tool in the Ellipse toolbar.
2. Click to set one corner of the object’s bounding box.
3. Move to the opposite corner of the bounding box. The drawing vector follows from the first point, indicating a diagonal of the bounding box, and the object appears.

   ▲ To draw a circle press Shift and move the pointer from the first point at a 45-degree angle. You can also snap to other angles in 15-degree increments while pressing Shift.
4. Click to set the opposite corner of the bounding box. The ellipse or circle appears.

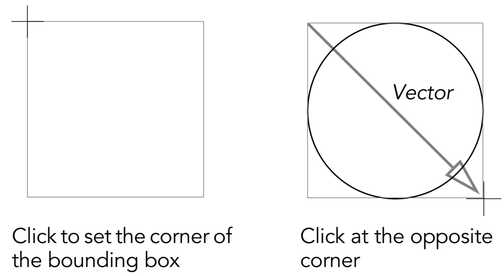


Fig. 112 Drawing with the Ellipse Diagonal tool

To use the Ellipse Center to Corner tool

The Ellipse Center to Corner tool lets you set a center point and a bounding box corner point to draw an ellipse or circle. If the bounding box is square, the tool draws a circle.

1. Select the Ellipse Center to Corner tool in the Ellipse toolbar.
2. Click to set the center point of the ellipse or circle.
3. Move to the corner of the object’s bounding box. The drawing vector follows the pointer from the first point, indicating half of the bounding box diagonal. The object appears.

   ▲ To draw a circle press Shift and move the pointer from the first point at a 45-degree angle. You can also snap to other angles in 15-degree increments while pressing Shift.
4. Click to set the corner of the bounding box. The ellipse or circle appears and is selected.

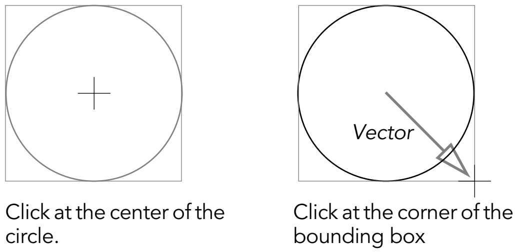
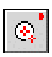


Fig. 113 Drawing with the Ellipse Center to Corner tool

To use the Circle Radius tool



The Circle Radius tool lets you set a center point and radius to draw a circle.

1. Select the Circle Radius tool in the Ellipse toolbar.
2. Click to set the center point of the circle.
3. Move to expand the circle from the center point. The drawing vector follows the pointer and indicates the circle's radius.
4. Click to set the circle's radius. The circle appears and is selected.

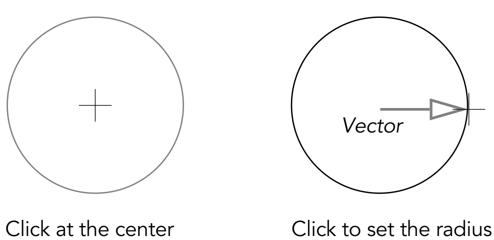
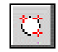


Fig. 114 Drawing with the Circle Radius tool

To use the Circle 3 Points tool



The Circle 3 Points tool lets you draw a circle by setting three points on the circle's circumference.

1. Select the Circle 3 Points tool in the Ellipse toolbar.
2. Click to set the first point on the circle's circumference.
3. Move to a second point on the circle. The drawing vector follows from the first point. Click to set the second point.
4. Move to a third point on the circle's circumference. The circle follows the pointer, and the drawing vector indicates a chord of the circle from the second point to the third point.
5. Click to set the third point on the circle. The circle appears and is selected.

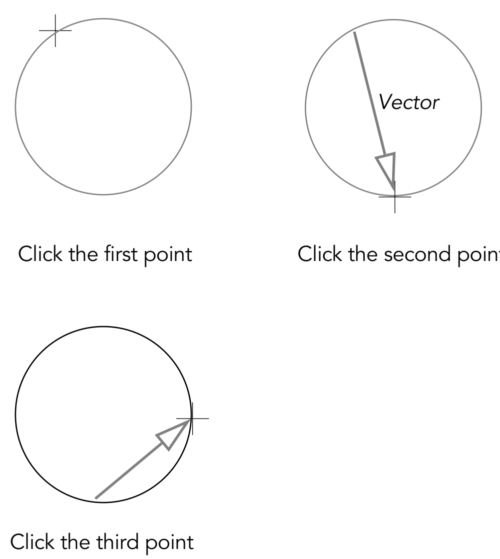


Fig. 115 Drawing with the Circle 3 Points tool

To use the Ellipse 3 Points tool

The Ellipse 3 points tool lets you set the center point, length, and width of an ellipse.

1. Select the Ellipse 3 Points tool in the Ellipse toolbar.
2. Click to set the center point of the ellipse.
3. Move the pointer to set the length of the ellipse. The drawing vector follows from the center point.

   ▲ Press the Shift key while dragging the drawing vector to constrain the vector to 15 degree intervals.
4. Click to set the length of the ellipse.
5. Move the pointer to set the width of the ellipse. The drawing vector moves perpendicular to length of the ellipse. The ellipse expands from the center point.
6. Click to set the width of the ellipse. The ellipse appears and is selected.

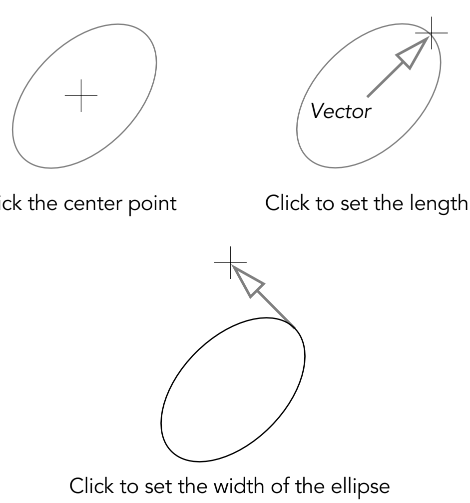


Fig. 116 Drawing with the Ellipse 3 Points tool

EDITING CIRCLES, ELLIPSES, AND CYLINDERS

After drawing a 2D circle or an ellipse, you can move creation points to change the shape. The location of an object's creation points depends on the tool used to create the object.

To edit creation points of circles and ellipses

1. Select the circle or ellipse with the Point Selection tool. The object's creation points appear.
2. Drag a creation point to a new location. As you drag, the drawing vector points from the original location.
3. After you move a creation point, the circle or ellipse remains selected. You can continue reshaping the object by moving creation points.
4. Click an empty area of the drawing when you finish editing the circle or ellipse.

Editing cylinders in Sculpt mode

Like most other objects in Sculpt mode, cylinders are 3D objects composed of chained polygons. A cylinder's creation points do not exist in Sculpt mode. You can, however, use the Reshape command and drag the handles of the chained polygons to reshape the cylinder.

To edit a cylinder

1. Select the cylinder with the Selection tool. The cylinder's handles appear.
2. Choose Edit > Reshape, or click the Reshape button. The handles of each chained polygon are visible.
3. Drag a handle to a new location. As you drag, the drawing vector follows from the handle's original location.

4. After you move a handle, the cylinder remains selected. You can continue moving handles to reshape the cylinder.
5. Click an empty area of the drawing when you finish editing the cylinder.

Applying commands to circles, ellipses, and cylinders

You can perform operations on a circle, an ellipse, or a cylinder by selecting it and applying commands in the Edit and Object menus.

Some commands, such as Align, require that you select additional objects. Refer to a specific command for more information.

Edit menu commands The following commands in the Edit menu can be applied to selected circles, ellipses, or cylinders: Cut, Copy, Clear, Select, Reshape, Duplicate, Create Publisher.

Object menu commands The following Object menu commands can be applied to selected circles, ellipses, or cylinders: Align, Scale, Chain, Unchain, Group and Ungroup.

The commands found in the following submenus in the Object menu can be used to edit circles, ellipses, or cylinders: Arrange, Copy to Layer, Send to Layer, Combine, Extrude (Draft mode only), Position, Path, and Trim.

Unchaining circles, ellipses, and cylinders

If you apply the Unchain command to a circle or an ellipse, it becomes a series of separate polyline objects, each made of one straight line segment.

If you apply the Unchain command to a cylinder, it becomes a series of separate polygon objects.

The number of line segments in an unchained circle, ellipse, or cylinder is determined by a Preference setting when you draw the object. The smaller the line segments comprising the object, which is determined by the size of the object and the number of line segments, the smoother the object's outline appears.

To set the segment preference for ellipse objects

1. Choose Edit > Preferences.
2. In the Preferences dialog box, change the Circle Step setting. A larger number results in more line segments in circles, ellipses, and cylinders when these objects are unchained or converted to polygons.

Applying attributes to circles, ellipses and cylinders

When a new circle, ellipse, or cylinder is created, DENEBACAD applies the current Draft or Sculpt mode attributes to the object. You can modify the object's attributes in the appropriate mode.

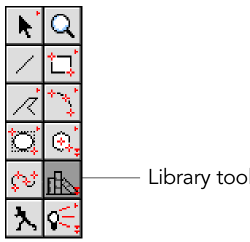
Draft mode Circles and ellipses accept the following attributes: Pen color, Pen weight, Pen style; Arrowheads; Fill color, Fill pattern, and Fill hatch.

You can apply a Fill color, a combination of a Fill color and a Fill pattern, or a combination of a Fill color and a Fill hatch to a circle or an ellipse.

Sculpt mode Cylinders accept Pen color and Surface materials.

LIBRARY TOOL

The Library tool lets you place saved objects in DENEBACAD documents.



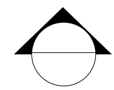
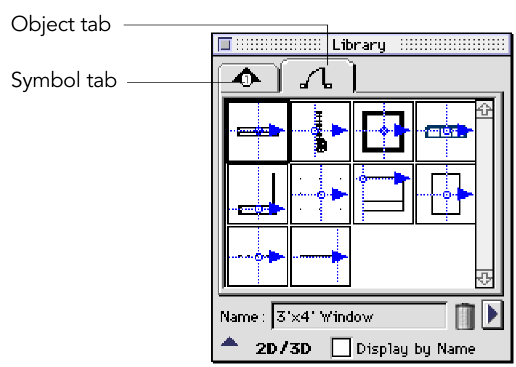


Fig. 117 A library symbol

Library palette

The Library palette organizes library items on two tabs. To bring a tab to the front, click the tab. The tab with a symbol icon contains library symbols. The tab with the door icon contains library objects.



Library objects

When you place a library object in a drawing, DENEBACAD scales it to the drawing.

For example, you can create and save a door in a drawing with a scale set to 1/2"=1'-0". You can than place the door, using the Library tool, into a drawing with a scale of 1/4"=1'-0", and DENEBACAD scales the door to fit the drawing.

A library object can be a 2D or 3D object, and you can create linked instances of 2D/3D library objects.

Library symbols

Library symbols are designed to be used for drawing elements such as electrical, HVAC, drawing, and mechanical symbols. Library symbols are 2D objects.

A library symbol is the same size in every document, no matter what a document's output scale is set to.

To use the Library tool

The Library tool opens the Library palette. You can select items from the Object or Symbol tabs.

1. Double-click the Library tool in the toolbox. The Library palette opens. The Library palette shows the available objects or symbols.

   ▲ If Display by Name is checked, the names of objects and symbols appear in the palette.

   ▲ 2D/3D appears at the bottom left of the palette if the Library has a 2D and 3D version of the same library object.

2. If you need to, click the Object or Symbol tab to bring it to the front.
3. Click the item you want to place in the drawing. The name of a selected item is highlighted.
4. Without pressing the mouse button, move the pointer from the Library palette to the drawing, to place the object's or symbol's bounding box.
5. Drag the drawing vector to set the item's orientation. The object or symbol appears and is selected.
6. To deselect the object or symbol, click an empty area of the drawing.

Library palette commands

Commands in the Library pop-up menu let you manage the objects and symbols in the active document.

Add Lets you add objects and symbols to the active document's library. This command is not available for symbols in Sculpt mode.

Select Lets you select the objects and symbols you have placed in the drawing using the Library tool. This command is not available for symbols in Sculpt mode.

Replace Lets you replace an existing object or symbol with another from the Library palette. This command is not available for symbols in Sculpt mode.

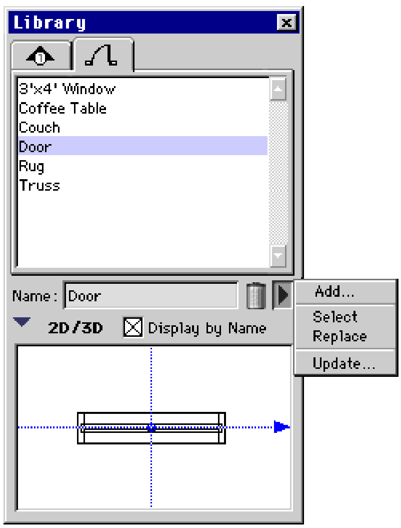
Update Lets you update the currently placed object or symbol with one that has been revised in the original document in which it was created and saved. This command is not available for symbols in Sculpt mode.

To use the Add command

1. Double-click the Library tool in the toolbox. The Library palette opens. The palette is empty if this is a new document.
2. Choose Add from the pop-up menu under the right triangle button. A directory dialog box opens.
3. Go to the folder containing DENEBACAD documents and select a document that contains saved library items.
4. Click Open. The Library Picker appears.
5. Click the Object or Symbol tab.
6. Click the name of the item to add to the Library. The selected item appears highlighted in the scrolling list. A picture of the selected item appears in the palette.

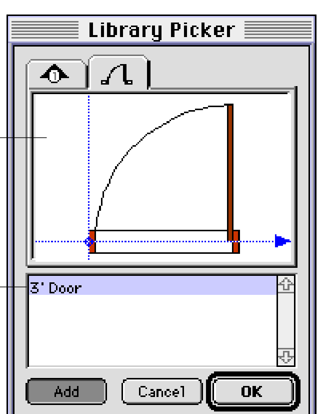
7. Click the Add button to import the selected item into the library of the current document.

8. When you finish using the Library Picker to add library items, click OK. The Library Picker closes. The items you added appear in the Library Palette in the active document.



Sketch of selected item

Selected item



To use the Select command

1. Double-click the Library tool in the toolbox. The Library palette opens. The Library palette shows the available objects or symbols.

2. If you need to, click the Object or Symbol tab to bring it to the front.

3. Click to highlight the name of the item you want to select.

   ▲ If Display by Name is not checked, you can click the picture of the object or symbol to select it.

4. Choose the Select command in the pop-up menu under the right triangle button. DENEBACAD selects all the objects or symbols with the selected name throughout the active document.

To use the Replace command

1. Select the objects or symbols in the active document to be replaced.

2. Double-click the Library tool in the toolbox. The Library palette opens. The Library palette shows the available objects or symbols.

3. If you need to, click the Object or Symbol tab to bring it to the front.

4. Click to highlight the name of the new object or symbol to replace the selected ones in the document.

   ▲ If Display by Name is not checked, click the picture of the object or symbol.

5. Choose Replace in the pop-up menu under the right triangle button. A message asks you to confirm the operation.

6. Click Yes to replace the selected objects or symbols. DENEBACAD replaces the items and the new items are selected.

To use the Update command

1. Double-click the Library tool in the toolbox. The Library palette opens. The Library palette shows the available objects or symbols.
2. If you need to, click the Object or Symbol tab to bring it to the front.
3. Click to highlight the name of the object or symbol you want to update.

   ▲ If you do not have Display by Name checked, you can click the picture of the object or symbol.
4. Choose Update in the pop-up menu under the right triangle button.
5. The objects or symbols are updated throughout the active document, and are selected.

Applying attributes to library objects and symbols

When you place a library object or symbol, it does not take on the current attributes.

If you want to change the attributes of a library object or symbol, group the item using the Group command. Then, ungroup the object using the Ungroup command. (You might have to use the Ungroup command twice to ungroup all of the objects). The result is separate objects which you can apply new attributes to.

However, if you ungroup a library object or symbol, it no longer exists as a library object or symbol. You can't select it, update it or replace it using the Library tool.

You can change the attributes of library objects and symbols after ungrouping by selecting an object and using the Fill and Pen palettes.

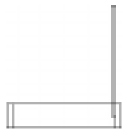
3D library objects and views

Aa 3D library object is view-dependent. In other words, a library object keeps its spatial relationship to the views in which you create it.

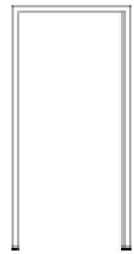
For example, if you create a door library object so that Top view is the plan of the door, and Front view is the front of the door, then Left view shows the left side of the door.

You can place the door library object in any view. In general, you should place 3D library objects in Top view, because Top view lets you orient the plan of the object to any other view.

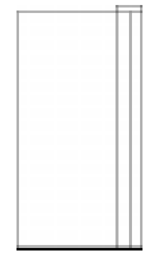
When you change views, the Library palette changes the way it displays 3D library objects to reflect the change in views. The Library palette displays 3D library objects as they will appear if placed in the current view.



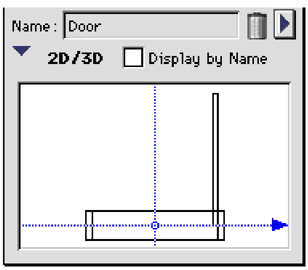
3D door in Top View



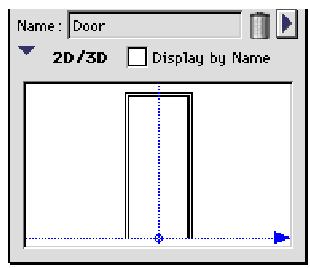
3D door in Front View



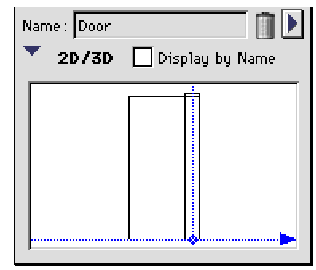
3D door in Left View



Library palette in Top View



Library palette in Front View



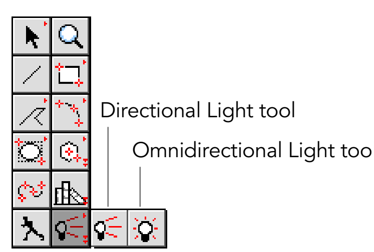
Library palette in Left View

LIGHT SOURCE TOOLS

DENEBACAD provides two tools to place lights in a model. These tools are located in the Light Source toolbar.

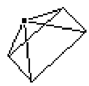
Directional Light tool Lets you place a light source by specifying two points: one point at the origin of the light, and the second point indicating the length and direction of the cone of illumination.

Omnidirectional Light tool Lets you place light sources by placing a point at the origin of the light.



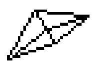
The Directional Light tool places a light source that projects a a cone of illumination on all modeled objects. The tool draws lights as vector objects whose origin defines the light source position and whose direction coincides with the direction and angle of the cone of illumination.

A directional light has three properties: power, size and angle.



Angled directional light, Sculpt mode

▪ *Handle*

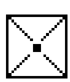


Directional light displayed as a Wireframe rendering in Render mode

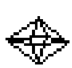
Fig. 118 Directional Light

The Omnidirectional Light tool places a light source that projects a spherical illumination field on all modeled objects.

An Omnidirectional Light has two properties: power and size.



Omnidirectional Light, Sculpt mode, Top view



Omnidirectional Light displayed as a Wireframe rendering in Render mode

Fig. 119 Omnidirectional Light

DENEBACAD uses a data type abbreviation for the name it assigns to objects and in DENEBACAD's PTF language.

Light tools have the following data types:

Directional Light	"LightSource"
Omnidirectional Light	"DirectLightSource"

To use the Directional Light tool

The Directional Light tool places the origin of the light and the light's cone of illumination.

1. Activate a Sculpt mode window and choose View > Top. In Top view, you can place the light in your plan.

▲ Because you are placing a 3D light in two dimensions only, you need to place the light in one view, and then specify the vertical height and the direction of the light in a different view.

2. Open or activate a second Sculpt mode window, and choose View > Front.

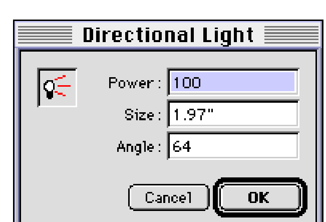
▲ In Front view, you can see the height of the light. This lets you edit the vertical direction of the light, and the length of the light's cone of illumination.

3. Activate a saved set of extrusion planes or define a new set of extrusion planes. DENEBACAD places the light on the upper plane of the active set of extrusion planes.

▲ The Info bar shows the location of the upper extrusion plane. The higher number indicates the upper extrusion plane.

4. Select the Directional Light tool in the Light Sources toolbar.

▲ You can double-click the Directional Light tool icon to change the current settings in the Directional Light tool dialog box.



5. In the Top view window, click to set the location of the light source.
6. Click to set a second point to indicate the direction of the cone of illumination. The light appears and is selected.

   ▲ Because you are working in the Top view, the second point is on the same plane as the first point.

   ▲ You can see the height of the light in the Sculpt window in Front view.
7. Click anywhere in the Sculpt mode window, set to Front view, to activate it.
8. Reselect the light.
9. Drag the second point, the point that represents the direction of the cone of illumination, to the desired position. The drawing vector indicates the direction and location of the point.

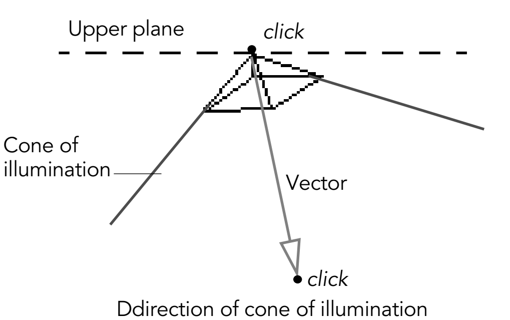
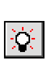


Fig. 120   Parts of the Directional light

To use the Omnidirectional Light tool

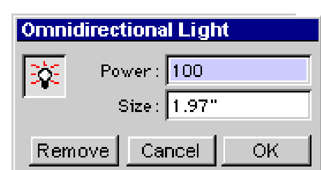


The Omnidirectional Light tool lets you place light sources that shine in all directions.

1. Activate a Sculpt window and choose View > Top. In Top view, you can place the light in your plan.
2. Select saved extrusion planes or define extrusion planes with the upper plane in the position you want.

   ▲ When you set a light source, DENEBACAD places it on the upper extrusion plane. The boxes at the right end of the Info bar show the location of the extrusion planes. The higher number is the location of the upper extrusion plane.
3. Select the Omnidirectional Light tool in the Light Sources toolbar.

   ▲ You can double-click the Omnidirectional Light tool icon to change the current settings in the Omnidirectional Light tool dialog box.



4. Click in the Sculpt window to place the light source. The light object appears and is selected.

Applying commands to lights

You can perform operations on lights by selecting them and applying commands in the Edit and Object menus.

Some commands require that you select additional objects. For example, to apply the Align command to a selected light, at least one other object must be selected. Refer to a specific command for more information.

Edit menu commands The following commands in the Edit menu can be applied to selected lights: Cut, Copy, Clear, Select, Reshape (Directional), Duplicate, Create Publisher.

Object menu commands The following Object menu commands can be applied to selected lights: Align, Scale, Chain, Group and Ungroup.

The commands found in the following submenus in the Object menu can be used to edit lights: Arrange, Copy to Layer, Send to Layer, and Position.

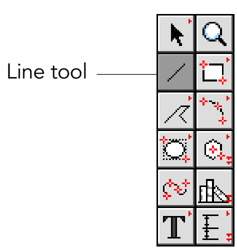
Applying attributes to lights

When a new light is placed, it is assigned the current Pen color. You can change the pen color after its creation. The Pen color assigned to a light object does not affect the color of the light shining from the object when a 3D scene is rendered.

LINE TOOL

The Line tool draws a straight single line in Draft mode or a flat plane in Sculpt mode. The tool draws a straight segment between two endpoints that you define.

The Line tool is located in the toolbox in the second row, under the Selection tools.



Lines are a set of two points that satisfy a linear equation.

In Draft mode, the Line tool draws single straight lines. In Sculpt mode, the Line tool draws flat planes, which are single polygon objects.

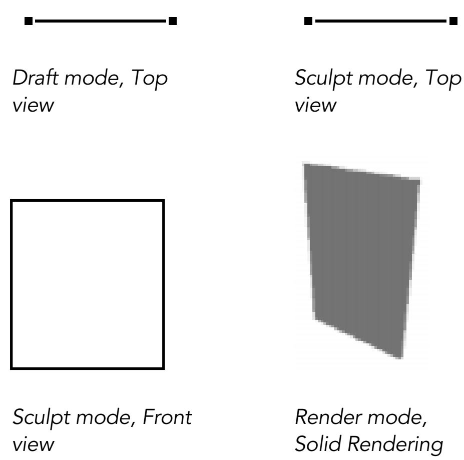


Fig. 121 Lines drawn in Draft, Sculpt, Render modes

DenebaCAD uses a data type abbreviation for the name it assigns to objects created by a tool. Data type abbreviations also appear in DenebaCAD's PTF language.

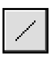
The Line tool is assigned the data type "line."

DRAWING LINES AND PLANES

Lines drawn in Draft mode are two-dimensional objects. A line appears in the same view where it is drawn only.

Planes drawn in Sculpt mode (or extruded from Draft mode) look the same as two-dimensional lines from the view they were created in. They can also be seen in other views because they have depth.

To use the Line tool



The Line tool lets you specify the two endpoints of a line.

1. Select the Line tool from the Toolbox.
2. Click to set the first endpoint of the line. Move to the second endpoint of the line. The drawing vector follows from the first point.

   ▲ To draw a constrained line, press Shift while dragging the drawing vector. The line will snap at 15 degree intervals.
3. Click to set the second endpoint of the line. The new line is selected.

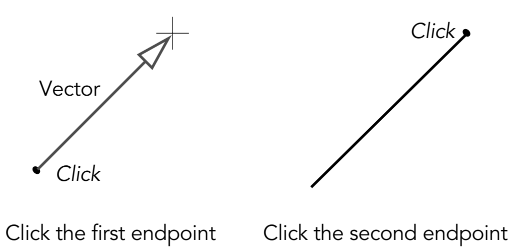


Fig. 122 Drawing with the Line tool

EDITING LINES AND PLANES

After drawing a line, you can move the creation points to change the line.

To edit a line's creation points

1. Select the line with the Point Selection tool. The line's creation points appear.
2. Drag a creation point to a new location. As you drag, the drawing vector points from the original location.
3. After you move a creation point, the line remains selected. You can drag creation points to continue resizing the line.
4. Click an empty area of the drawing when you finish editing the line. The creation points are no longer visible.

Editing planes in Sculpt mode

Like most other objects in Sculpt mode, planes are 3D objects composed of a polygon. A plane doesn't have creation points. You can, however, use the Reshape command to display the handles of the polygons which comprise the plane. You can then drag the handles to reshape the plane. Because planes are 3D objects, you can edit them in any view.

To edit a plane

1. Select the plane with the Selection tool. The plane's handles appear.
2. Choose Edit > Reshape, or click the Reshape button. The handles of the polygon are visible.
3. Drag a handle to a new location. As you drag, the drawing vector follows from the handle's original location.
4. After you move a handle, the plane remains selected. You can continue moving handles to reshape the plane.
5. Click an empty area of the drawing when you finish editing the plane. The handles are no longer visible.

Applying commands to lines and planes

You can perform operations on a line or plane by selecting it and applying commands in the Edit and Object menus.

Some commands, such as Align, require at least two objects be selected. Refer to a specific command for more information.

Edit menu commands The following commands in the Edit menu can be applied to selected lines or planes: Cut, Copy, Clear, Select, Reshape, Duplicate, Create Publisher.

Object menu commands The following Object menu commands can be applied to selected lines or planes: Align, Scale, Chain, Unchain, Group and Ungroup.

The commands found in the following submenus in the Object menu can be used to edit lines or planes: Arrange, Copy to Layer, Send to Layer, Combine, Extrude (Draft mode only), Position, Path, and Trim.

Applying attributes to lines and planes

When a line or plane is created, it is assigned the current attributes. You can modify an existing object by changing its attributes.

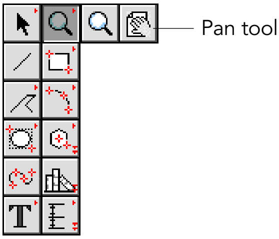
Draft mode Lines accept the following attributes: Pen color, Pen weight, Pen style and Arrowheads.

Sculpt mode Planes accept Pen color and Surface materials in Sculpt mode.

PAN TOOL

The Pan tool lets you change your view of a drawing by moving the drawing around in the drawing window.

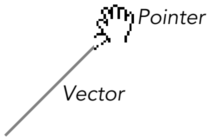
The Pan tool appears in the View toolbar with the Zoom tool. This toolbar is at the top right of the toolbox in Draft and Sculpt modes.



The Pan tool moves a drawing in the active window, like the scroll bars do, but the Pan tool is more flexible.

To use the Pan tool

1. Select the Pan tool in the toolbox. The pointer changes to a hand.
2. Drag in the drawing to move the document. The drawing vector defines the distance and direction that the drawing moves.



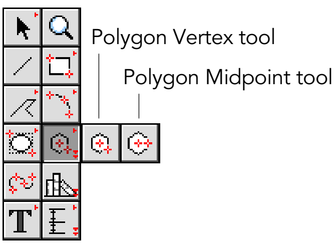
Using the Pan tool (or the scroll bars) does not actually reposition any objects in relation to the document. Scrolling or panning changes the view only.

POLYGON TOOLS

DENEBACAD tools that draw regular polygons and columns are located in the Polygon toolbar.

Polygon Vertex tool Draws a polygon from the center to the vertex of one side.

Polygon Midpoint tool Draws a polygon from the center to the midpoint of one side.



Polygons are closed objects bounded by three or more line segments. In Draft mode, polygon tools draw regular polygons with equal-length sides. In Sculpt mode, polygon tools draw columns made of chained polygons.

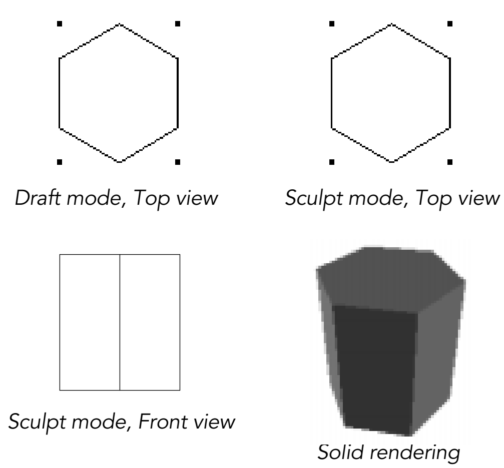


Fig. 123 Polygons drawn in Draft and Sculpt mode

DENEBACAD uses a data type abbreviation for the name it assigns to objects created by a tool. Data type abbreviations also appear in DENEBACAD's PTF language.

These tools have the data types listed below.

Polygon Vertex	"PolygonVertex"
Polygon Midpoint	"PolygonMidpoint"

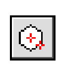
DRAWING POLYGONS AND COLUMNS

Polygons drawn in Draft mode are two-dimensional objects, which can be seen in the view they are drawn in only.

Columns in Sculpt mode appear as polygons from the view they are created in. They can also be seen in other views because they have depth.

The term "polygon" in the following procedures refers to a 2D polygon in Draft mode, or a 3D polygon-shaped object in Sculpt mode.

To use the Polygon Vertex tool



The Polygon Vertex tool draws a polygon from the center to a vertex.

1. Select the Polygon Vertex tool from the Polygon toolbar.

   ▲ To set the number of sides for a polygon, double-click the Polygon tool icon. In the Polygon Vertex dialog box, enter the number in the "# of sides" text box.

2. Click to set the center point of the polygon.

3. Move and click to set the vertex of the polygon. The drawing vector follows from the first point.

4. The new polygon is selected. Click an empty area of the drawing to deselect it.

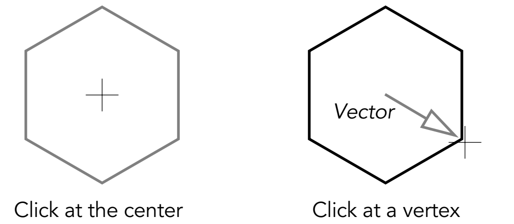


Fig. 124 Drawing with the Polygon Vertex tool

To use the Polygon Midpoint tool

The Polygon Midpoint tool draws a polygon from the center to one side midpoint.

1. Select the Polygon Midpoint tool from the Polygon toolbar.

   ▲ To set the number of sides for the polygon, double-click the Polygon tool icon. In the Polygon Vertex dialog box, enter the number in the “# of sides” text box.

2. Click to set the center point of the polygon.
3. Move and click to set the midpoint of a side of the polygon. The drawing vector follows from the first point.
4. The new polygon is selected. Click an empty area of the drawing to deselect it.

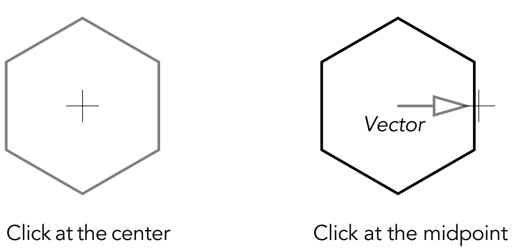


Fig. 125 Drawing with the Polygon Midpoint tool

EDITING POLYGONS AND COLUMNS

After drawing a polygon in Draft mode, you can move creation points to change the polygon’s shape. The location of the creation points depends on the tool used to draw the polygon.

To edit a polygon

1. Select the polygon with the Point Selection tool. The polygon’s creation points appear.
2. Drag a creation point to a new location. As you drag, the drawing vector points from the original location.
3. After you move a creation point, the polygon remains selected. You can continue reshaping the object by moving its creation points.
4. Click an empty area of the drawing when you finish editing the polygon.

Editing columns in Sculpt mode

Like most other objects in Sculpt mode, columns are 3D objects composed of chained polygons. A column does not have creation points in Sculpt mode.

You can use the Reshape command to display the handles of the chained polygons that make up the object, and drag the handles to reshape it.

A column can be edited in any view. Refer to the commands in the View menu for information on changing views.

To edit a column

1. Select the column with the Selection tool. The column’s handles appear.
2. Choose Edit > Reshape, or click the Reshape button. The handles of each chained polygon appear.

3. Drag a handle to a new location. As you drag, the drawing vector follows from the handle's original location.
4. After you move a handle, the column remains selected. You can continue moving handles to reshape the column.
5. Click an empty area of the drawing when you finish editing the column.

Applying commands to polygons and columns

You can perform operations on a polygon or a column by selecting it and applying commands in the Edit and Object menus.

Some commands, such as Align, require that you select additional objects. Refer to a specific command for more information.

Edit menu commands The following commands in the Edit menu can be applied to selected polygons or columns: Cut, Copy, Clear, Select, Reshape, Duplicate, Create Publisher.

Object menu commands The following Object menu commands can be applied to selected polygons or columns: Align, Scale, Chain, Unchain, Group and Ungroup.

The commands found in the following submenus in the Object menu can be used to edit polygons or columns: Arrange, Copy to Layer, Send to Layer, Combine, Extrude (Draft mode only), Position, Path, and Trim.

Unchaining polygons and columns

If you apply the Unchain command to a polygon, it becomes a series of separate polyline objects, each made of one straight line segment.

If you apply the Unchain command to a column, it becomes a series of separate polygon objects.

The number of line segments in an unchained polygon or column is the same as the object's number of sides. If you change the the "# Sides Poly" number in the Preferences dialog box, you set the number of sides for the Polygon tools.

Applying attributes to polygons and columns

When a new polygon or column is created, it is assigned the current attributes. You can modify a polygon's attributes after drawing it.

Draft mode Polygons accept the following attributes: Pen color, Pen weight, Pen style, Arrowheads, Fill color, Fill pattern, and Fill hatch.

You can apply a Fill color, a combination of a Fill color and a Fill pattern, or a combination of a Fill color and a Fill hatch to a polygon.

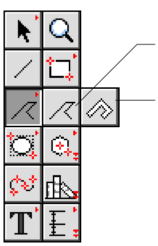
Sculpt mode Columns accept Pen color and Surface materials in Sculpt mode.

POLYLINE TOOLS

DENEBACAD tools that draw polylines and planes are located in the Polyline toolbar in the toolbox.

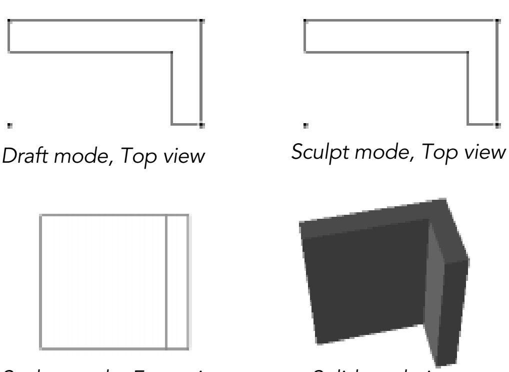
Single Polyline tool Draws a polyline or plane through the points set along a path.

Double Polyline tool Draws a double polyline or plane through the points set along a path.



Polylines are a series of segments chained to form an object. Double polylines are a series of parallel segments chained to form one object.

In Draft mode, Polyline tools draw single or double polylines. In Sculpt mode, Polyline tools create 3D planes composed of chained polygons.



Draft mode, Top view *Sculpt mode, Top view*

Sculpt mode, Front view *Solid rendering*

Fig. 126 Polylines in Draft, Sculpt, and Render modes

DENEBACAD uses a data type abbreviation for the name it assigns to objects created by a tool. Data type abbreviations also appear in DENEBACAD's PTF language.

These tools have the data types listed below.

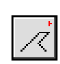
Single Polyline	"Polyline"
Double Polyline	"OffsetPoly"

DRAWING POLYLINES AND PLANES

Polylines and double polylines drawin in Draft mode are two-dimensional objects, which can be seen in the view they are drawn in only.

Planes created in Sculpt mode look the same as 2D polylines from the view they were created in. They can also be seen from other views because they have depth.

To use the Single Polyline tool



The Single Polyline tool lets you specify points to draw a single polyline.

1. Select the Polyline tool from the Polyline toolbar.
2. Click to set the first point of the polyline.
3. Move and click to set the second point of the polyline. The drawing vector follows from the first point.

   ▲ Press Shift to snap the drawing vector at 15 degree intervals.
4. Continue to move and click to set successive points on the polyline.

5. Double-click to set the last point and finish the polyline. Or, click to set the last point and press Return or Enter to finish the polyline. The new polyline is selected.

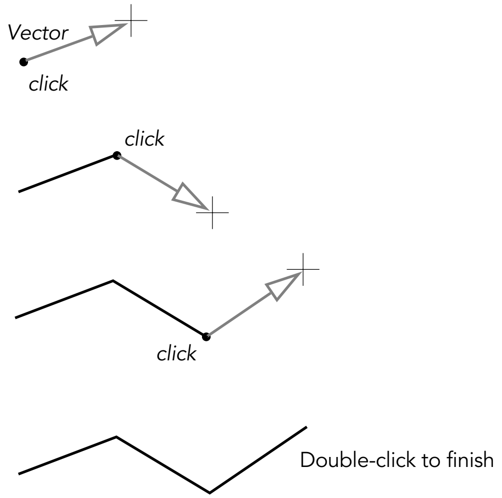
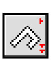


Fig. 127 Drawing with the Single Polyline tool

To use the Double Polyline tool



The Double Polyline tool lets you specify points to draw a double polyline.

1. Select the Double Polyline tool from the Polyline toolbar.

   ▲ To set the width and offset for double polylines, double-click the Double Polyline tool icon. In the Double Polyline dialog box, enter values in the Width and Offset text boxes.

2. Click to set the first point of the polyline.

3. Move and click to set the second point of the polyline. The drawing vector follows from the first point.

   ▲ Press Shift to snap the drawing vector at 15 degree intervals.

4. Continue to move and click to set successive points on the double polyline.

5. Double-click to set the last point and finish the polyline. Or, click to set the last point and press Return or Enter to finish the polyline. The new double polyline is selected.

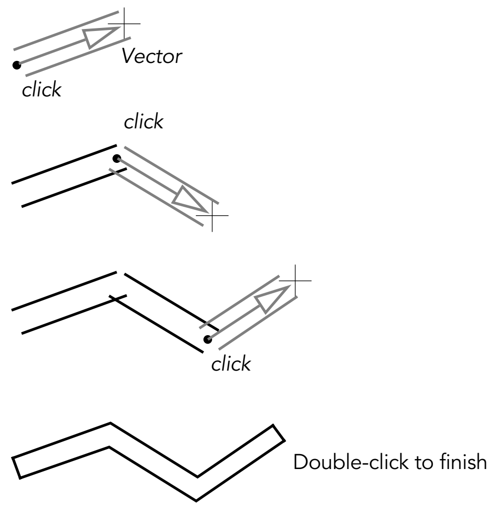


Fig. 128 Drawing with the Double Polyline tool

EDITING POLYLINES AND PLANES

You can move creation points of a 2D polyline (or double polyline) to change its shape.

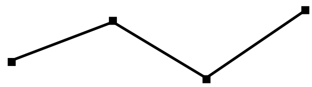
To edit a polyline or double polyline

1. Select the polyline with the Point Selection tool. The object's creation points appear.

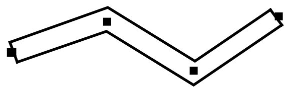
2. Drag a creation point to a new location. As you drag, the drawing vector points from the original location.

3. After you move a creation point, the object remains selected. You can drag creation points to continue reshaping the polyline.

4. Click an empty area of the drawing when you finish editing.



Creation points of a single polyline



Creation points of a double polyline

Fig. 129 Editing polylines

Editing planes in Sculpt mode

Like most other objects in Sculpt mode, planes are 3D objects composed of chained polygons. You can use the Reshape command to display the handles of chained polygons and drag handles to reshape a plane.

To edit a plane

5. Select the plane with the Selection tool. The plane's handles appear.
6. Choose Edit > Reshape, or click the Reshape button. The handles of each chained polygon appear.
7. Drag a handle to a new location. As you drag, the drawing vector follows from the handle's original location.
8. After you move a handle, the plane remains selected. You can continue moving handles to reshape the plane.
9. Click an empty area of the drawing when you finish editing the plane.

Applying commands to polylines

You can perform operations on a polyline, a double polyline, or a plane by selecting it and applying commands in the Edit and Object menus.

Some commands, such as Align, require that you select additional objects. Refer to a specific command for more information on how to use it.

Edit menu commands The following commands in the Edit menu can be applied to selected single or double polylines, or planes: Cut, Copy, Clear, Select, Reshape, Duplicate, Create Publisher.

Object menu commands The following Object menu commands can be applied to selected single or double polylines, or planes: Align, Scale, Chain, Unchain, Group and Ungroup.

The commands found in the following submenus in the Object menu can be used to edit single or double polylines, or planes: Arrange, Copy to Layer, Send to Layer, Combine, Extrude (Draft mode only), Position, Path, and Trim.

Unchaining polylines and planes

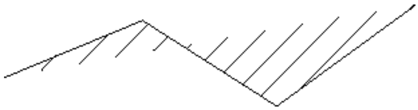
If you apply the Unchain command to a polyline, it becomes a series of separate polyline objects, each made of one straight line segment.

If you apply the Unchain command to a plane, it becomes a series of separate polygon objects.

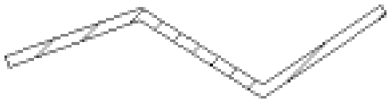
The number of line segments in an unchained polyline or plane is determined by the number of points you set when you draw the object.

Applying attributes to polylines and planes

When a new polyline or plane is created, it is assigned the current attributes. You can change the attributes of a polyline or plane object at any time.



Polyline: Black Pen color, Black Fill color, Fill pattern



Double polyline: Black Pen color, Black Fill color, Fill pattern

Draft mode Single and double polylines accept the following attributes: Pen color, Pen weight, Pen style; Arrowheads; Fill color, Fill pattern, and Fill hatch.

You can apply a Fill color, a combination of a Fill color and a Fill pattern, or a combination of a Fill color and a Fill hatch to a polyline.

Sculpt mode Planes accept Pen color and Surface materials in Sculpt mode.

RECTANGLE TOOLS

Tools to draw rectangles and cubes are located in the Rectangle toolbar in the toolbox.

Rectangle Diagonal tool Draws a rectangle by setting two points as the opposite corners of the rectangle.

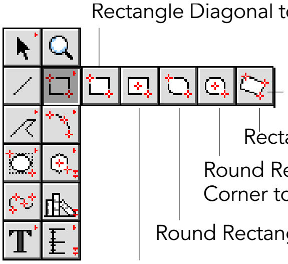
Rounded Rectangle Diagonal tool Draws a rectangle with rounded corners by setting two points as the opposite corners of an invisible bounding box.

Rectangle Center to Corner tool Draws a rectangle from the center to a corner.

Rounded Rectangle Center to Corner tool

Draws a rectangle with round corners from the center to a corner of the object's bounding box.

Rectangle 3 Points tool Draws a rectangle by setting three corner points.



Rectangles are parallelograms with right angles. Rounded rectangles are parallelograms with right angles and radius corners.

In Draft mode, Rectangle tools draw rectangles and rounded rectangles. In Sculpt mode, Rectangle tools create cube-shaped 3D objects composed of chained polygons.

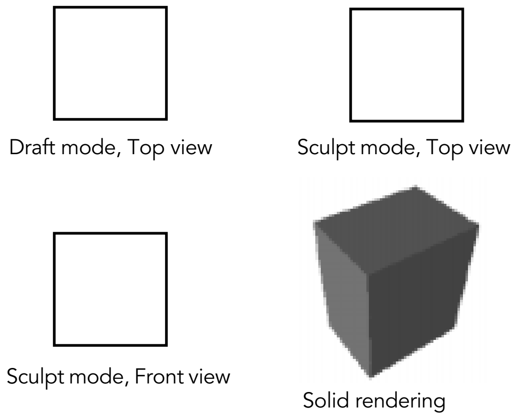


Fig. 130 Rectangles in Draft, Sculpt, and Render modes

DENEBACAD uses a data type abbreviation for the name it assigns to objects created by a tool. Data type abbreviations also appear in DENEBACAD's PTF language.

These tools have the data types listed below.

Rectangle Diagonal	"DiagonalRect"
Rounded Rectangle Diagonal	"DiagonalRoundRect"
Rectangle Center to Corner	"CenterRect"
Rounded Rectangle Center to Corner	"CenterRoundRect"
Rectangle 3 Points	"FreeRect"

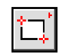
DRAWING RECTANGLES AND CUBES

Rectangles and rounded rectangles created in Draft mode, and cubes created in Sculpt mode are drawn with the same tools and by following the same procedures. However, the resulting objects are not the same.

Rectangles created in Draft mode are two-dimensional objects, which can be seen in the view they were drawn in only.

Cubes created in Sculpt mode look the same as 2D rectangles from the view they were created in. They can be seen from other views because they have depth.

To use the Rectangle Diagonal tool



The Rectangle Diagonal tool lets you set points at opposite corners of a rectangle.

1. Select the Rectangle Diagonal tool in the Rectangle toolbar.
2. Click to set the first corner of the rectangle.
3. Move and click at the opposite corner of the rectangle. The drawing vector follows from the first point.

   ▲ To constrain the rectangle to a square, press Shift and move the pointer from the first point at a 45-degree angle.
4. The new rectangle is selected. Click an empty area of the drawing to deselect it.

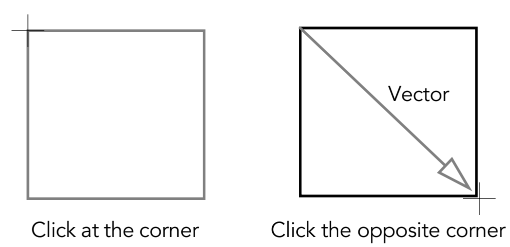
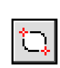


Fig. 131 Drawing with the Rectangle Diagonal tool

To use the Rounded Rectangle Diagonal tool



The Rounded Rectangle Diagonal tool lets you set points at opposite corners of a rectangle's invisible bounding box.

1. Select the Rounded Rectangle Diagonal tool in the Rectangle toolbar.

   ▲ To set the radius for the rectangle's corners, double-click the tool's icon. Enter a value in the Radius text box and click OK.

2. Click to set one corner of the bounding box.
3. Move and click at the opposite corner of the bounding box. The drawing vector follows from the first point, indicating the diagonal of the bounding box.

   ▲ To constrain the rounded rectangle to a square, press Shift and move the pointer from the first point at a 45-degree angle.

4. The new rounded rectangle is selected. Click an empty area of the drawing to deselect it.

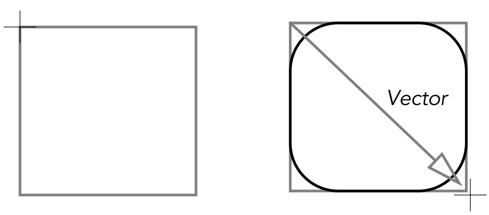
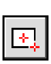


Fig. 132 Drawing with the Rounded Rectangle Diagonal tool

To use the Rectangle Center to Corner tool



The Rectangle Center to Corner tool lets you set the center and one corner of a rectangle.

1. Select the Rectangle Center to Corner tool in the Rectangle toolbar.
2. Click to set the center of the rectangle.
3. Move and click at a corner of the rectangle. The drawing vector follows from the first point, indicating the diagonal of the rectangle.

   ▲ To constrain the rectangle to a square, press Shift and move the pointer from the first point at a 45-degree angle.

4. The new rectangle is selected. Click an empty area of the drawing to deselect it.

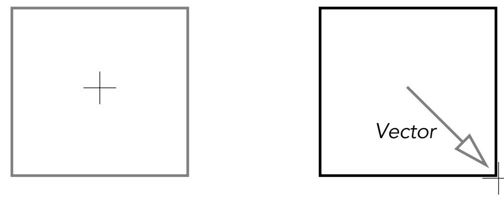
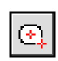


Fig. 133 Drawing with the Rectangle Center to Corner tool

To use the Rounded Rectangle Center to Corner tool



The Rounded Rectangle Center to Corner tool lets you set the center point and a corner of a rectangle's bounding box.

1. Select the Rounded Rectangle Center to Corner tool in the Rectangle toolbar.

   ▲ To change the radius for a rounded rectangle's corners before drawing, double-click the tool's icon. Enter a value in the Radius text box and click OK.

2. Click to set the center of the rounded rectangle.
3. Move and click at a corner of the bounding box. The drawing vector follows from the first point, indicating the diagonal of the bounding box.

   ▲ To constrain the rounded rectangle to a square, press Shift and move the pointer from the first point at a 45-degree angle.

4. The new rounded rectangle is selected. Click an empty area of the drawing to deselect it.

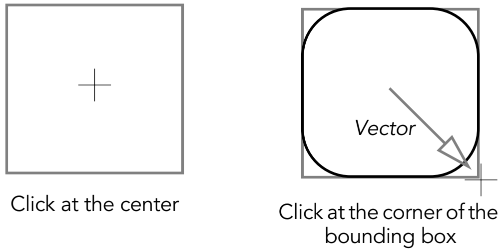


Fig. 134 Drawing with the Rounded Rectangle Center to Corner tool

To use the Rectangle 3 Points tool

The Rectangle 3 Points tool lets you set three corners of a rotated rectangle.

1. Select the Rectangle 3 Points tool from the Rectangle toolbar.
2. Click to set the first corner of the rectangle.
3. Move and click to set the first side of the rectangle. The drawing vector indicates the angle of the rectangle.
4. Move and click to set the side perpendicular to the first side, finishing the rectangle.

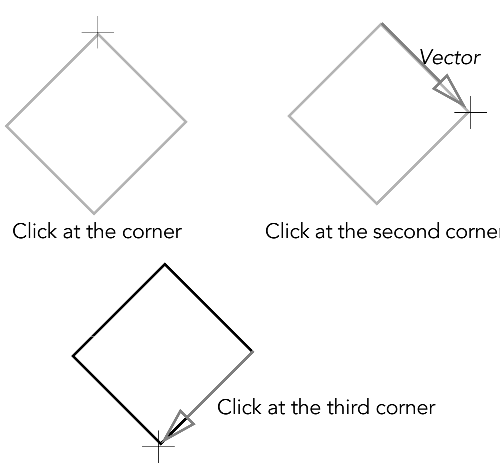


Fig. 135 Drawing with the Rectangle 3 Points tool

EDITING RECTANGLES AND CUBES

After drawing a rectangle, you can move creation points to change its shape. The location of the creation points depends on the tool used to create the rectangle.

To edit a rectangle

1. Select the rectangle with the Point Selection tool. The object's creation points appear.
2. Drag a creation point to a new location. As you drag, the drawing vector points from the original location.
3. After you move a creation point, the object remains selected. You can drag creation points to continue reshaping the rectangle.
4. Click an empty area of the drawing when you finish editing the rectangle or rounded rectangle.

Editing cubes in Sculpt mode

Like most other objects in Sculpt mode, cubes are 3D objects composed of chained polygons. You can use the Reshape command to display handles of the chained polygons and drag the handles to reshape the object.

To edit a cube

1. Select the cube with the Selection tool. The cube's handles appear.
2. Choose Edit > Reshape, or click the Reshape button. The handles of each chained polygon appear.
3. Drag a handle to a new location. As you drag, the drawing vector follows from the handle's original location.
4. You can continue moving handles to reshape the cube. Click an empty area of the drawing when you finish editing.

Applying commands to rectangles, rounded rectangles, and cubes

You can perform operations on a rectangle, a rounded rectangle, or a cube by selecting it and applying commands in the Edit and Object menus.

Refer to a specific command for more information on how to use it to revise an object.

Edit menu commands The following commands in the Edit menu can be applied to selected rectangles, rounded rectangle, or cubes: Cut, Copy, Clear, Select, Reshape, Duplicate, Create Publisher.

Object menu commands The following Object menu commands can be applied to selected rectangles, rounded rectangles, and cubes: Align, Scale, Chain, Unchain, Group and Ungroup.

The commands found in the following submenus in the Object menu can be used to edit rectangles, rounded rectangles, or cubes: Arrange, Copy to Layer, Send to Layer, Combine, Extrude (Draft mode only), Position, Path, and Trim.

Unchaining rectangles, rounded rectangles, and cubes

If you apply the Unchain command to a rectangle, it becomes four separate polyline objects; each one a straight line segment.

If you apply the unchain command to a rounded rectangle, it becomes a series of separate polyline objects, each made up of one straight line segment. The number of line segments of an unchained rounded rectangle is determined by the radius preference setting in effect when you draw the object.

If you apply the Unchain command to a cube, it becomes six separate polygon objects.

To set the radius preference

You can set the preferred radius for the corners of rounded rectangles. The radius also affects the number of segments or polygons created when a rounded rectangle or 3D rounded cube is unchained.

1. Choose Edit > Preferences.
2. In the Preferences dialog box, enter a Round Radius value. A larger number results in more line segments in unchained rounded rectangles.

   ▲ Changing the Round Radius preference also changes the current Round Radius setting.

Applying attributes to rectangles and cubes

When a new rectangle or cube is created, it is assigned the current attributes. You can modify an existing object by changing its attributes after its creation.

Draft mode Rectangles accept the following attributes: Pen color, Pen weight, Pen style; Arrowheads; Fill color, Fill pattern, and Fill hatch.

You can apply a Fill color, a combination of a Fill color and a Fill pattern, or a combination of a Fill color and a Fill hatch to a rectangle.

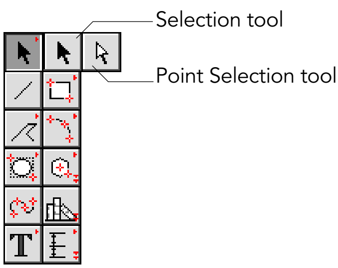
Sculpt mode Cubes accept Pen color and Surface materials in Sculpt mode.

SELECTION TOOLS

Two tools are used to select objects in DENEBACAD. These tools are located in the Selection toolbar in the toolbox.

Selection tool Selects entire objects, which distinguishes them from other, unselected objects.

The Point Selection tool Puts an object in reshape mode so its creation points or handles appear. See “Reshape” on page 232 for more information on Reshape mode.



In Draft and Sculpt modes, the Selection tool selects an object and highlights handles at each corner of its bounding box. These handles indicate the object can be resized, moved, or modified by various commands.

In Draft and Sculpt mode, an object can be selected only if it is on an active layer. Objects that are visible but not active can’t be selected. Refer to “Activating and Selecting Layers” on page 365 for information on working with layers.

To use the Selection tool

1. Select the Selection tool.
2. Click any edge of the object. Or, drag a selection box around the entire object.
3. Handles appear on the object, or on the object’s bounding box, to indicate the object is selected.

To select more than one object

1. Select the Selection tool.
2. Click each object you want to select while pressing the Shift key. Or, drag a selection box around the objects you want to select. The selection box must completely enclose all of the objects.
3. Handles appear on each object, or surrounding each object, to show that it is selected.

To deselect an object

1. Select the Selection tool.
2. Click an empty portion of the drawing window. The handles disappear from all selected objects.

To deselect one object among several

1. Select the Selection tool.
2. Press Shift and click objects that are selected. The handles of the deselected objects disappear.

To move an object

1. Select the Selection tool.
2. Point to the edge of the object, and drag the object to a new location. As you move the pointer, the drawing vector follows from the original location to the new location.

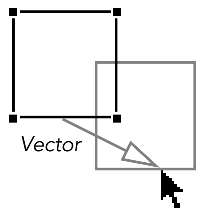
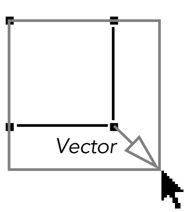


Fig. 136 Moving an object with the Selection tool

To resize an object with the Selection tool

1. Select the Selection tool.
2. Click an object to select it. The object's selection handles appear.
3. Point to a handle and drag it to the desired location. The drawing vector follows from the original location.

   ▲ You can press Shift while dragging at a 45 degree angle to constrain an object to its original proportions.
4. Release the mouse button. The object remains selected.



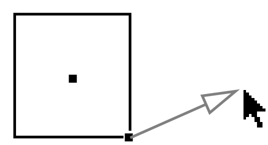
To use the Point Selection tool

The Point Selection tool lets you select creation points of an object in Draft mode. It lets you select vertices of polygons in Sculpt mode.

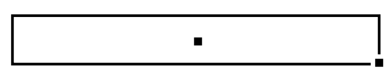
1. Select the Point Selection tool.
2. Drag a selection box around an object. The selection box doesn't need to completely enclose the object, but must enclose the points you want to edit.

   ▲ In Draft mode, handles appear at the creation points you select.

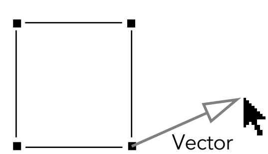
   ▲ In Sculpt mode, handles appear at the vertices you select.
3. Drag a handle to move it. The drawing vector follows from the original location.



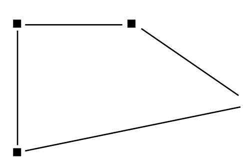
Rectangle drawn with Rectangle Diagonal tool



Reshaped rectangle



Reshaping cube in Sculpt mode



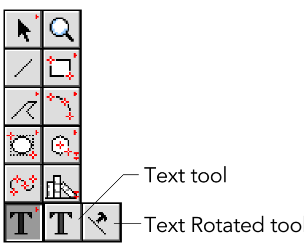
Reshaped cube

TEXT TOOLS

Tools to place text in a document are located in the Text toolbar in the toolbox.

Text tool Places text as a single horizontal line or as a multi-line block of text with margins that you can define.

Text Rotated tool Places a single line of text at any angle.



DENEBACAD uses a data type abbreviation for the name it assigns to objects created by a tool. Data type abbreviations also appear in DENEBACAD's PTF language.

These tools have the data types listed below.

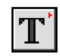
Text	"Text"
Text Rotated	"VectorText"

Creating text objects

Text objects exist only in Draft mode. You can view and edit text in Sculpt mode if you select View 3D & 2D Objects in the View Options submenu or the View Options pop-up menu on the Status bar.

Note: When working with text objects, you must use the Enter key (on the numeric keypad) to deselect a text object or end edit mode. The Enter/Return key on the keyboard inserts a carriage return into text you are editing.

To place a line of text in Draft mode



The Text tool lets you click and type to place a single line of text in Draft mode.

1. Select the Text tool from the Text toolbar. The pointer becomes an I-beam.
2. Click in the Draft window to set the location of the starting point of the text. An insertion point appears where you click.
3. Type a line of text in the document. When you finish typing, press Enter on the numeric keypad. The text appears with the current attributes and the text object is selected.

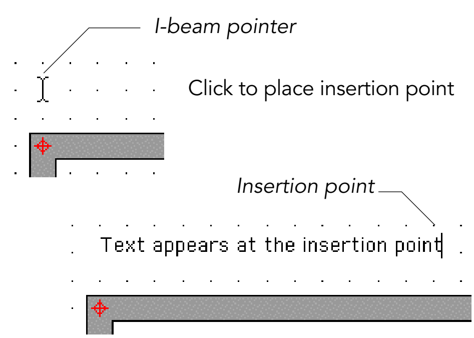
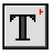


Fig. 137 Typing a line of text with the Text tool



To create a block of text

The Text tool lets you place a block of text in a document in Draft mode.

1. Select the Text tool from the Text toolbar. The pointer becomes an I-beam.
2. Drag to create a rectangular text box in the document. An insertion point appears at the upper-left corner of the invisible text box.
3. Begin typing. When text reaches the right side of the text box, it wraps to the next line. If the text exceeds the depth of the text box, the box expands from the bottom.
4. When you finish typing, press Enter. The text appears with the current attributes and the text object is selected.

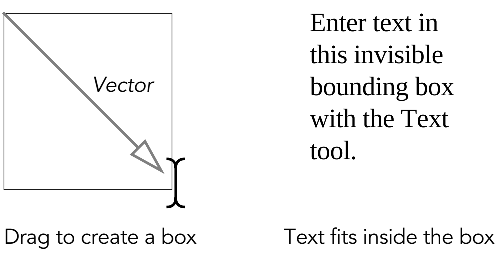


Fig. 138 Creating a text box with the Text tool

To use the Text Rotated tool

The Text Rotated tool lets you type a single line of text at any angle.

1. Select the Text Rotated tool from the Text toolbar. The pointer becomes an I-beam.
2. Click to set the starting point of the line of text, and then move the pointer to define the angle of the text baseline. The drawing vector follows the pointer and indicates the baseline angle (the length of the vector is not significant). Click again to set the text baseline angle.
3. An insertion point appears where you clicked. Type the text, which appears from the insertion point on a horizontal baseline.
4. When you finish typing, press Enter (use the Enter key on the numeric keypad) to leave text entry mode. The text baseline rotates to the specified angle, and the text object is selected.

Note: If you type more characters than DENEBACAD can accommodate, or while editing text, insert more characters than can be accommodated in the angled text object, DENEBACAD beeps and displays a message telling you that no more text characters can be inserted. If this message appears, click OK to continue.

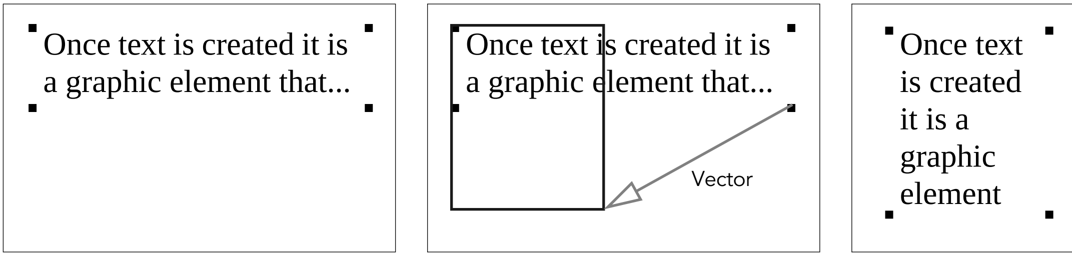
Adjusting text objects

The handles that surround selected text can be dragged to change the shape of the text box. This does not change the size of the type.

To resize a text box

1. Select the text object. Handles appear at the corners of the object.
2. Drag a handle to change the box shape. The drawing vector follows the pointer as you drag the handle. The revised text box remains selected. You can continue resizing the text box by dragging a handle.

3. When you finish, click an empty area to deselect the text object.



A selected text object has handles at the corners

Dragging a handle resizes the text box

Text conforms to new text box margins

Fig. 139 Resizing a text object

EDITING TEXT

After placing a text object in a drawing, you can edit the object and the text it contains using the Text tools, the text attributes controls, and various commands.

Selecting a text object

When you want to apply a command or a text attribute to an entire text object, you first select the text object.

For example, if you want to change the font of the text in a label, then apply a background color and rotate the text 90 degrees, you need to select the text object first.

You can select a text object using the same methods that you use to select other objects.

- With the Selection tool, click the text object, or to drag a selection box around the text object.
- Use the Select All command to select all objects in a document.
- Use the Select Palette. Choose Edit > Select to do this.

When a text object is selected, handles appear at the corners of the object's bounding box.

To deselect a text object, use the same methods you use to deselect other objects.

- Click in an empty area of the drawing
- Select another object

Applying commands to text objects

You can perform operations on text objects by selecting them and applying commands in the Edit and Object menus.

Edit menu commands The following commands in the Edit menu can be applied to selected text objects: Cut, Copy, Clear, Reshape, Duplicate, Create Publisher.

Object menu commands The following commands in the Object menu can be applied to selected text objects: Align, Scale, Group, Ungroup.

Commands in the following submenus in the Object menu can be applied to selected text: Arrange, Copy to Layer, Send to Layer, Combine, Position.

Note: You can copy text into the Material Editor in the Fill palette to use bitmapped text as a Surface material in Render mode.

WORKING IN TEXT EDIT MODE

To alter the text contained in a text object, you work in text edit mode. When you place a text object in edit mode, you can do the following:

- Select text characters so you can cut or copy them to the Clipboard.
- Edit the text by inserting, deleting, and replacing text characters.
- Change the color, font, type size, and type style of characters selected in the text.

◆ **To place a text object in edit mode:** Double-click the object with the Selection tool, or click the text with the Text tool.

When you place a text object in edit mode, either an insertion point appears or a previous text selection is restored:

- In a text object in which no text was selected the last time it was in edit mode, the insertion point (a vertical bar) appears in the text to indicate the point where characters can be inserted or deleted. The insertion point appears at the end of the text, or at the previous insertion point location.
- In a text object that had a text selection active the last time it was in edit mode, the same text selection appears highlighted.

The tool that you use to place a text object in edit mode remains active. The tool you use also affects the action of the pointer:

- If you double-clicked the object with the Selection tool, the pointer displays an I-beam when it is in the text object, and is a selection arrow when it is outside the object. If you click in the text, the insertion point appears where you click; if you click outside the object, you leave edit mode and deselect the text object.
- If you clicked the object with the Text tool, the pointer is an I-beam wherever you move it in the document. If you click in the text, the insertion point appears where you click; if you click outside the text object, you create a new text object where you click, and the previous text object leaves edit mode and is deselected.

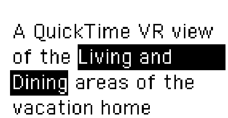
Editing rotated text

When you place a rotated text object in edit mode, the text baseline becomes horizontal for editing. When you exit edit mode, the text object rotates to its previous angle.

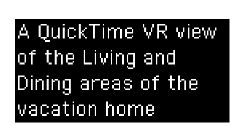
Selecting text characters

When a text object is in edit mode, you can make a selection of characters within the text.

When you select text, you can apply text attributes and commands to the text selection, rather than the entire object. A text selection appears highlighted.



Selected characters are highlighted in a text object



Entire text object is highlighted when all characters are selected

To select characters in a text object

1. Place a text object in edit mode (see "Working in text edit mode" on page 136).
2. Use the I-beam pointer to drag over the characters you want to select, or do one of the following:

   ▲ Click before the first character (or space) you want to select, then press Shift and click after the last character of the text selection.

   ▲ Press Shift and use the keyboard arrow keys to extend a selection one character left or right, or one line up or down, from the insertion point.

   ▲ Press Shift+Option (Mac), or Shift+Alt (Windows), and use the Left and Right arrow keys to extend a selection one word left or right from the insertion point.

   ▲ Double-click a word to select it.

   ▲ Choose Select All to select all the text characters in the object.

Deselecting a text selection

When text is highlighted in a text object, you can click the I-beam pointer anywhere in the text to deselect the selected text. Or, you can press Enter to leave text edit mode. If you click outside of the text object, you also deselect the text object.

Inserting and deleting text

When a text object is in text edit mode, you can insert and delete characters, and replace existing text. You can also apply the Cut, Copy, Clear, and Paste commands to a text selection.

To insert text

1. Place the object in text edit mode (see "Working in text edit mode" on page 136).
2. Click the I-beam pointer where you want to add text. An insertion point appears. Begin typing and the new text appears at the insertion point.
3. Press Enter when you finish editing to leave text edit mode.

To delete text in a text object

Use this procedure to delete a selection within a text object, or to replace a selection with new text that you type or paste from the Clipboard.

1. Place the object in text edit mode (see "Working in text edit mode" on page 136).
2. Do one of the following to delete text characters:

   ▲ Place the insertion point to in the text and press the Backspace or Delete key to erase characters one at a time.

   ▲ Make a text selection, and then press the Delete key, or choose Cut or Clear in the Edit menu to erase the selection. See "Selecting text characters" on page 136 for selection procedures.

   ▲ Begin typing to replace the text selection with the text that you type.
3. When you finish editing, press the Enter key on the numeric keypad to leave text edit mode.

APPLYING ATTRIBUTES TO TEXT

When you create a text object, DENEBACAD assigns the current text attributes to the text. Text uses the following attributes: Text color, font, type size, style, alignment, Fill color, Fill pattern, and Fill hatch.

You can use the current attributes or change them before you type text. You can modify existing text by changing the attributes of an entire text object or of a selection within the text.

Applying attributes to text objects

When you select a text object, you can change the attributes of all the text within the object at once. Select the attribute from the Text controls on the Attributes bar, or from the floating Fill palette.

You can apply a Fill color, a combination of a Fill color and a Fill pattern, or a combination of a Fill color and a Fill hatch to the text box background. However, the palette must be floating before you select a text object, because the Fill controls do not appear on the Attributes bar when a text object is selected or in edit mode.

Applying attributes to text selections

You can change some text attributes of individual characters within a text object by selecting the text you want to change and using the text controls on the Attributes bar.

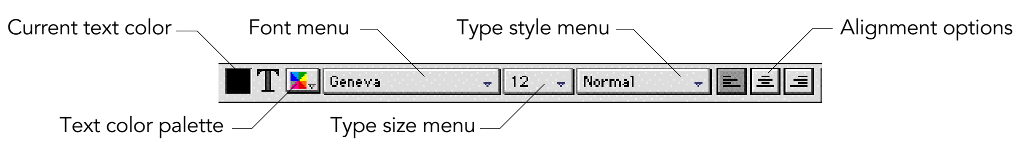
You can change the text color, font (typeface), type size, and style of selected text.

You cannot apply fill attributes (color, hatch, and pattern) to a text selection, only to selected text objects.

Note: If you change the alignment setting when text is selected, the new alignment applies to the entire text object.

To change the attributes of a text selection

1. Place the text object in text edit mode (see "Working in text edit mode" on page 136).
2. Select the text you want to modify. See "Selecting text characters" on page 136.
3. Select an attribute using the text controls on the Attributes bar. You can change the color, font, type size, and type style of the selection using the text color palette and the pop-up menus.
4. Press the Enter key to leave edit mode when you finish editing the text's attributes.

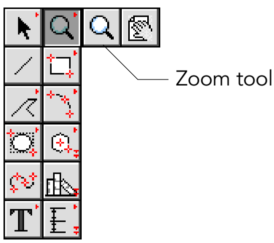


Text controls appear when a text object is in edit mode or the Text tool is selected

Fig. 140 Text controls on the Attributes bar

ZOOM TOOL

The Zoom tool lets you change the view magnification in a drawing. The Zoom tool is located in the toolbox.



You can use the Zoom tool to select an area of the drawing to enlarge or reduce the view.

The Zoom tool is available in Draft and Sculpt modes.

The zoom level should not be confused with the scale of a document. Though they both affect the way a drawing appears on screen, the zoom level doesn't affect the scale of the document.

Note: Changing the view magnification changes the appearance of objects on screen, but does not change the actual size of objects in the drawing.

To use the Zoom tool to magnify a view

1. Select the Zoom tool from the toolbox.

2. Drag a selection box around the area you want to magnify. The view of the objects within the selection box is magnified.

To use the Zoom tool to reduce a view

1. Select the Zoom tool from the toolbox.
2. Press Shift key while dragging a selection box around the area you want to reduce. The view of the objects within the selection box is reduced.

The Zoom bar

The Zoom bar in the lower-left corner of the screen displays the current magnification, and other magnification options. You can use the Zoom bar to step to the next preset magnification levels or to set a specific magnification level.

Refer to“Zoom bar” on page 193 for more information.

When you work in Render mode, you can use the Render mode tools to set focal points and angles. You can also view perspective or isometric renderings using tools that appear in Render mode.

When you open a Render mode window for the first time in a new document, the window displays a perspective Wireframe rendering of 3D objects. The focal point and its cone of vision are centered horizontally and vertically to all of the 3D objects in the document. The Render mode window displays the objects that fall inside the cone of vision.

If all the objects are not within the cone of vision, you might not see any objects when you first open the Render mode window. You can use the focal point tools to adjust the placement and angle of the focal point and its cone of vision so that the Render mode window displays the objects that you want to see.

Horizontal Focal Point tool Sets the viewer's position horizontally, and the view direction.

Vertical Focal Point tool Sets the viewer's position vertically, and the view angle.

Dual Focal Point tool Sets the viewer's position and view angle horizontally and vertically.

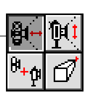
Isometric Render tool Sets the Render window to display isometric renderings.

Perspective Render tool Sets the Render window to display perspective renderings.

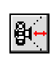
HORIZONTAL FOCAL POINT TOOL

The Horizontal Focal Point tool establishes the distance from the viewer to the rendering, and also sets the horizontal direction of the sight line as it relates to objects.

Horizontal Focal Point tool



To use the Horizontal Focal Point tool



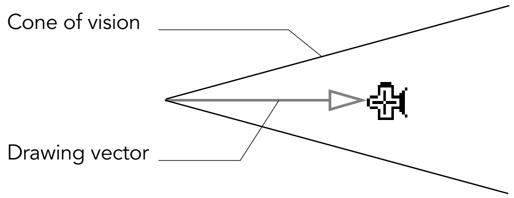
The Horizontal Focal Point tool lets you set a point to represent the distance from the viewer to objects, and a point to represent the horizontal direction of the sight line in respect to objects.

1. Select the Horizontal Focal Point tool. The Render mode window changes to Sculpt mode and the view changes to Top.

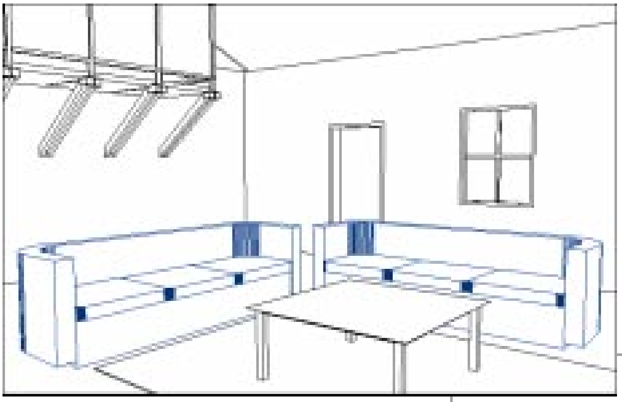
   ▲ The location of the viewer is indicated by a circle with an X. A red line indicates the horizontal direction of the sight line.

2. Move the pointer to set the new location of the viewer. Click to set the point.

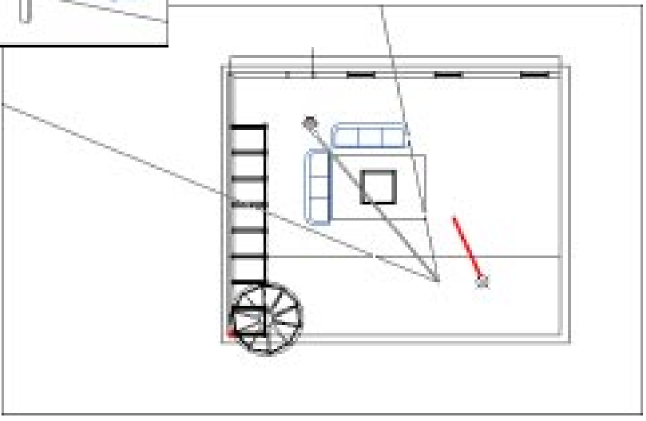
3. Move the pointer to set the horizontal direction of the viewer's sight line. The sight line is represented by the a vector extending from the set point. A cone of vision also extends from the set point. Objects within the cone will appear in the rendering.



4. Click to set the horizontal direction of the sight line. The window changes to Render mode and displays a Wireframe rendering.



In Render mode, a Hidden Line rendering displays the objects within the cone of vision



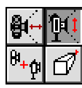
In Top view of a Sculpt mode window, set the location of the viewer and the horizontal direction of the sight line

Fig. 141 Setting the horizontal focal point

VERTICAL FOCAL POINT TOOL

DENEBACAD provides a tool to set the vertical focal point of a perspective. This tool is located in the toolbox.

The Vertical Focal Point tool Establishes the height of the viewer and vertical angle of the sight line as it relates to objects.



To use the Vertical Focal Point tool

The Vertical Focal Point tool lets you set a point to represent the height of the viewer, and the vertical angle of the sight line in.

1. Select the Vertical Focal Point tool in the toolbox.

   ▲ Notice in the Title bar that the window changes to Sculpt mode and the view changes to Front view.

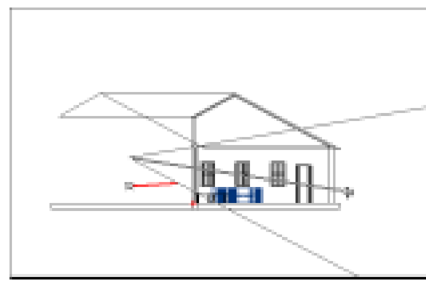
   ▲ The current location of the viewer is shown by a circle with an X. A red line indicates the direction of the sight line.

2. Move the pointer to set the new height of the viewer. Click to set the point.

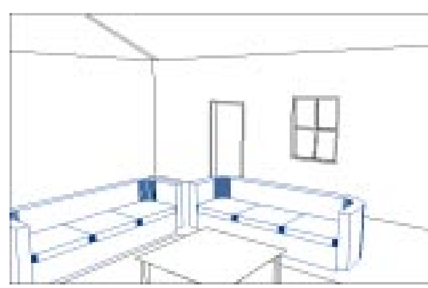
3. Move the pointer to set the vertical angle of the viewer's sight line.

   ▲ The viewer's sight line is represented by the drawing vector as it extends and revolves at a 180 degree angle from the set point. A cone of vision also extends from the set point. All objects that fall between the two lines of the cone of vision will be displayed in the selected view.

4. Click to set the vertical angle of the sight line. The window changes back to the original Render mode window, and displays the objects as a Wireframe rendering.



In the Front view of a Sculpt mode window, set the height of the viewer and the vertical angle of the sight line

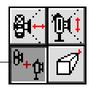


In Render mode, a Hidden Line rendering displays the objects within the cone of vision

Fig. 142 Setting the vertical focal point

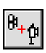
DUAL FOCAL POINT TOOL

The Dual Focal Point tool establishes the distance and height of the viewer as well as the horizontal direction and vertical angle of the sight line as it relates to objects in a rendering.



Dual Focal Point tool

To use the Dual Focal Point tool



The Dual Focal point tool lets you set the horizontal and vertical focal points of a perspective. Using this tool is the same as using the Horizontal Focal Point and Vertical Focal Point tools in succession.

First, you set the distance from the viewer to the objects, and then set the horizontal angle of the sight line.

Second, you set the height of the viewer, and then set the vertical angle of the sight line.

1. Select the Dual Focal Point tool in the Toolbox.

   ▲ Notice that the Render window changes to Sculpt mode and Top view (shown in the window's title bar).

   ▲ The location of the viewer is indicated by a circled "X". The red line indicates the horizontal direction of the sight line.

2. Move the pointer and click to set a new horizontal point of view.

3. After you set the location point, the sight line is represented by the drawing vector, which extends as you move the mouse. Two angled lines represent a "cone of vision." Click to set the sight line direction.

4. The window changes to Sculpt mode and Front view.

   ▲ The location of the viewer is indicated by a circled "X". The red line indicates the vertical direction of the sight line.

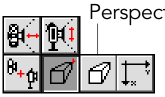
5. Move the pointer and click to set the height of the point of view.

6. After you set the view height, the sight line is represented by the drawing vector, which extends and changes angle as you move the mouse. Two angled lines represent the "cone of vision." Click to set the sight line direction.

7. The window changes back to Render mode, and displays a Wireframe rendering of the scene based on the new focal points you have set.

PERSPECTIVE RENDER TOOL

The Perspective Render tool lets you view renderings as perspectives.


Perspective Render tool

The Perspective Render tool creates a perspective view of objects. This view is the representation of three dimensional objects and depth relationships in a two dimensional environment.

A perspective has vanishing points on the horizon where all parallel lines drawn in perspective converge or terminate.

To use the Perspective render tool

1. Activate a Render mode window.


Perspective Render tool

2. Select the Perspective Render tool in the toolbox.
   The isometric Wireframe rendering changes to a perspective Wireframe rendering.

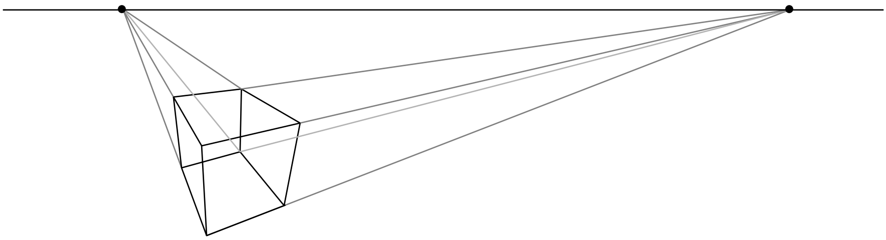
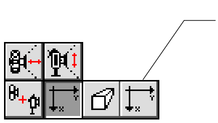


Fig. 143 Perspective view of a cube

ISOMETRIC RENDER TOOL

The Isometric Render tool lets you view renderings as isometrics.



The Isometric render tool creates paraline drawings. These drawings are projected representations of objects that give the objects a three dimensional quality.

These drawings differ from perspective drawings in that the projection lines remain parallel instead of converging to a point on the horizon.

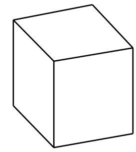


Fig. 144   Isometric view of a cube

To use the Isometric Render tool

1. Activate a Render mode window.
2. Select the Isometric Render tool in the toolbox. The perspective Wireframe rendering changes to an isometric Wireframe rendering.

OBJECT ATTRIBUTES 11

You can use colors, patterns, and Surface materials to distinguish objects in 2D drawings and give realism to renderings that you create in DENEBACAD.

This chapter describes the attributes that can be applied to objects: Pen color, Fill color, Pen weight, Pen style, Arrowhead, Fill pattern, Fill hatch, and Surface material.

The buttons on the Attributes bar make it convenient to apply attributes to objects, while the Pen and Fill palettes let you apply attributes from a floating window. You can also use the Pen and Fill palettes to modify or create new colors, patterns, hatches, arrowheads, and materials, as described in this chapter.

HOW ATTRIBUTES APPLY TO OBJECTS

You can select attributes from the buttons on the Attributes bar, or from the Pen palette and the Fill palette.

Current attributes

DENEBACAD applies some default attributes to objects when you create the objects. The term "current attributes" refers to the attributes that DENEBACAD applies to a new object. You can change the current attributes, and you can change the attributes of existing objects, using the Attributes bar, Pen palette, and Fill palette.

When you create a new object, DENEBACAD applies the current attributes that are shown in the small boxes at the left of the Pen and Fill buttons on the Attributes bar (*figure 176*).

You can change the current attributes at any time when no objects are selected.

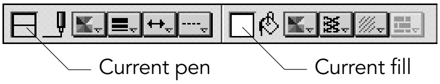


Fig. 176 Current attribute boxes on the Attributes bar

If you select an attribute when one or more objects are selected, the change applies to the selected objects; the current attributes do not change.

Available attributes

The attributes that you can select depend on the current mode (Draft, Sculpt, or Render). When a Draft mode window is active, Surface materials are not available, because these are attributes of 3D objects. In Sculpt mode, only Pen colors and Surface materials are available. In Render mode, no attributes can be applied to objects.

Changing attributes of objects

After you create an object, you can modify its attributes at any time.

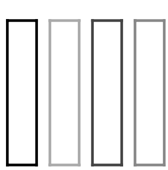
The Properties Manager is a useful for checking attributes of objects in Draft mode and Sculpt mode. And, while you can't display Surface materials in Sculpt mode, you can use the Properties Manager to check the name and appearance of the Surface material of a selected object.

ATTRIBUTES OF 2D OBJECTS

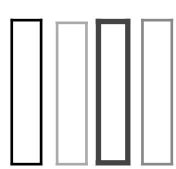
The attributes that can be applied to objects in Draft mode are Pen color, Pen weight, Pen style; Arrowhead; Fill color, Fill pattern, and Fill hatch. These attributes are described here in brief:

Pen color A solid color defined in the RGB color space, which appears on the outline of an object in Draft mode. All 2D objects have a Pen color.

Note: The display of Pen colors assigned to objects can be overridden by display settings, such as View 2D & 3D Objects (if you select the option that distinguishes object type by color), or other display options that affect object colors in Draft mode.



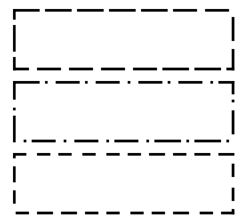
Examples of Pen colors



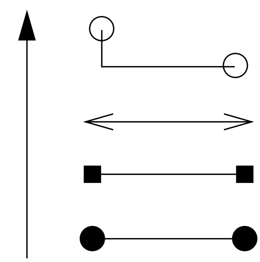
Examples of Pen weights

Pen weight The thickness of an object's outline, specified in millimeters. All 2D objects have a Pen weight.

Pen style A pattern of solid, dashed, and dotted segments, which appears on an object's outline. An object's Pen color is used on the solid segments of the Pen style; blank segments appear transparent. All 2D objects have a Pen style.



Examples of Pen styles

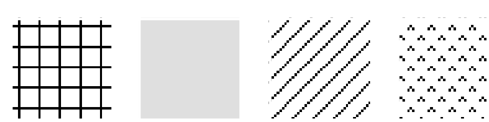


Examples of Arrowheads

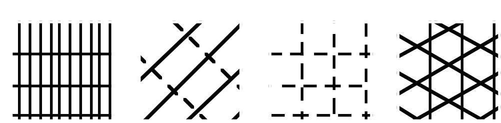
Arrowhead A terminator, such as an arrow, slash, or other shape, that appears at the end of a line segment.

Fill color A solid color that tints an object's Fill pattern or Fill hatch. Fill colors can be applied only to 2D objects in Draft mode.

Examples of Fill colors (printed as grays)



Examples of Fill patterns (with black Fill color)



Examples of Fill hatches (with black Fill color)

Fill pattern A raster pattern that appears in the interior of an object. If an object has a black-and-white Fill pattern, you can apply a Fill color to the object and the color will replace black pixels in the Fill pattern.

Fill hatch A pattern made of line segments, which appears in the interior of an object. Hatch patterns are typically used to represent building materials.

ATTRIBUTES OF 3D OBJECTS

In Sculpt mode, only two attributes apply to 3D objects: Pen color and Surface material.

In a Sculpt mode window, only Pen color is displayed on a 3D object. In a Render mode window, either an object's Pen color or its Surface material is displayed on the object's surfaces, depending on the type of rendering that you select.

When a Sculpt mode window is active, the Pen color and Surface material buttons are available on the Attributes bar, while other attribute controls are dimmed (*figure 177*).

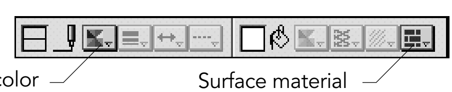


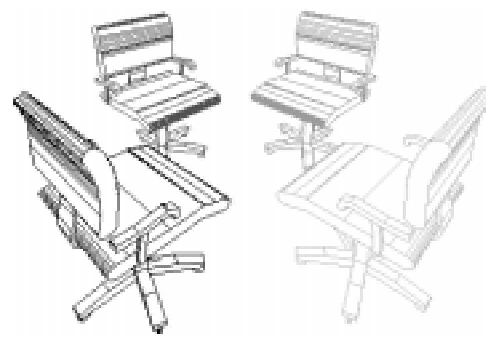
Fig. 177 Attributes available in Sculpt mode

Pen color In Sculpt mode, Pen color is similar to Pen color in Draft mode: the color appears on the outline of the 3D object.

In Render mode, the appearance of an object's Pen color depends on the type of rendering you select, and whether the object's Surface material is rendered.

- In Wireframe or Hidden Line renderings, an object's Pen color appears on the outlines of it surfaces. Surface materials do not appear in Wireframe or Hidden Line renderings.

- In renderings created with the Solid command, an object's Pen color appears as a solid surface color if the object has no Surface material (or if the object has a Surface material, but the Materials checkbox is not selected in the Rendering Options dialog box). Otherwise, the object's Surface material appears in the rendering, and the Pen color isn't displayed.



Example of Pen colors (printed as grays) in a Wireframe rendering

Example of Pen colors (printed as grays) in a Solid rendering

Surface material A Surface material is a color raster image, which DENEBACAD applies as a skin over the surfaces of 3D objects when you use the Solid command to create renderings.

Example of Surface materials in a Solid rendering

To apply Surface materials, you select 3D objects in Sculpt mode and choose Surface materials from the Fill palette.

You can create Surface materials from digitized photographic images or computer-generated designs. Surface materials can incorporate brightness, reflectivity, gloss, and transparency settings.

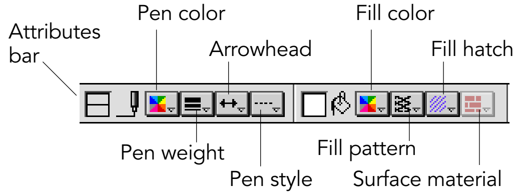
Attributes applied to extruded objects

When you create a 3D object by extruding a 2D object, DENEBACAD applies the current Pen color to the new 3D object; the attributes of the 2D object do not transfer to the 3D object.

USING THE ATTRIBUTES BAR

The Attributes bar is a toolbar that appears at the top of the screen (under the menu bar) whenever DENEBACAD is running. You can use the buttons on the Attributes bar to assign attributes to objects in Draft mode and Sculpt mode.

From the buttons on the Attributes bar, you can select each type of object attribute: Pen color, Pen weight, Arrowhead, Pen Style, Fill color, Fill pattern, Fill hatch, and Surface material.



The buttons on the Attributes bar act like pop-up menus. When you press a button, a palette of attributes pops open. Each pop-up palette corresponds to a tab in the Pen palette or the Fill palette. For example, when you press the Pen weight button, the Pen weight tab of the Pen palette pops open.

If you drag a palette away from any of the Pen attributes buttons, the Pen palette appears. If you drag a palette away from any of the Fill attributes buttons, the Fill palette appears. See “Using the Pen and Fill palettes” on page 151 for more information on how to use these floating palettes. For information on specific attributes, see the section on each attribute later in this chapter.

Text attributes Whenever the Text tool or a text object is selected, buttons and pop-up menus for applying attributes to text replace the Attributes buttons on the Attributes bar.

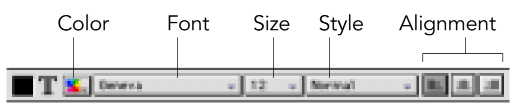


Fig. 178 The Attributes bar with text controls

USING THE PEN AND FILL PALETTES

You can open the Pen palette and the Fill palette by dragging a pop-up palette away from its button on the Attributes bar. The Pen palette appears when you drag from the Pen color, Pen weight, Arrowhead, or Pen Style button. The Fill palette appears when you drag from the Fill color, Fill hatch, Fill pattern, or Surface material button. After you "tear off" either palette, it becomes a floating palette that you can move to any position on screen by dragging the palette's title bar.

You can use the Pen and Fill palettes to apply attributes to objects, the same as you would use the pop-up palettes that open from the buttons on the Attributes bar.

However, the Pen and Fill palettes offer features that are not available from the buttons on the Attributes bar: the palettes let you create, edit, and delete attributes. The palettes also let you select all objects in the active window that share a particular attribute.

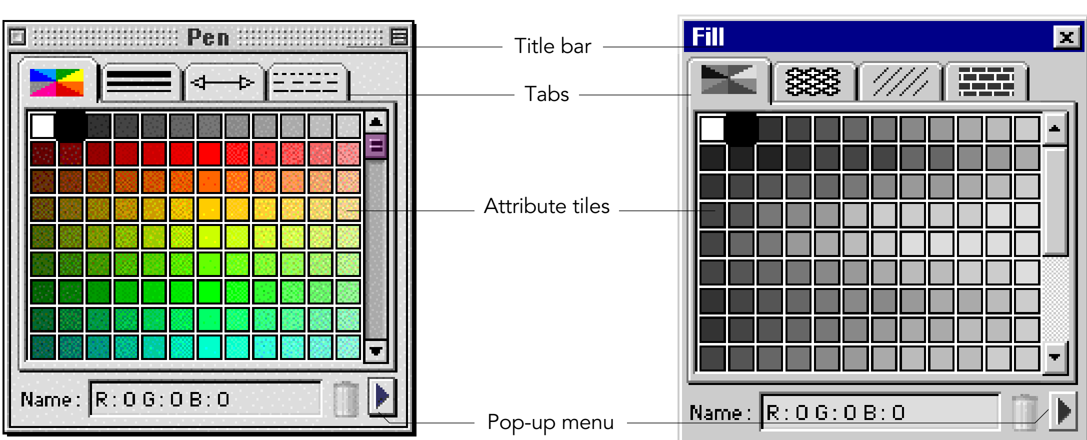
To display the Pen or Fill palette

1. Press a Pen attribute button (for the Pen palette) or a Fill attribute button (for the Fill palette). The button's palette pops up.
2. Drag the palette away from the Attributes bar. As you drag, the outline of the floating palette appears. When you release the mouse button, the palette remains open.

To apply attributes from the palettes

You might find that it's easier to apply attributes using the Pen and Fill palettes, rather than the buttons on the Attributes bar, when you apply multiple attributes. With the palettes, you just click an attribute to apply it to any selected objects. To apply a different type of attribute, just click the appropriate tab to bring it to the front.

For example, select an object in a plan. Set its Pen color with a click on the Pen color tab. Click the Pen weight tab and click the 0.5 mm setting. Then, click the Pen style tab and click a dashed setting, without having to move the mouse back and forth to the Attributes bar to select and apply each attribute.



The Pen palette contains the Pen color, Pen weight, Arrowhead, and Pen style tabs

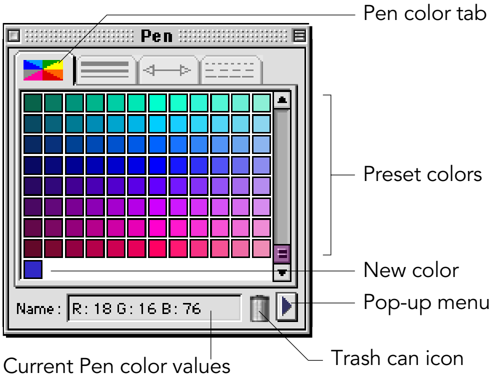
The Fill palette contains the Fill color, Fill pattern, Fill hatch, and Surface material tabs

PEN ATTRIBUTES

Pen attributes affect the outlines of objects. For general information on Pen attributes, see “How attributes apply to objects” on page 147.

This section describes how to apply, create, and modify Pen attributes. You can use the Pen palette or buttons on the Attributes bar to apply pen attributes to objects. The tabs in the Pen palette correspond to the Pen color, Pen weight, Arrowhead, and Pen style buttons on the Attributes bar.

For information on using the palette to apply attributes, see “Using the Pen and Fill palettes” on page 151.



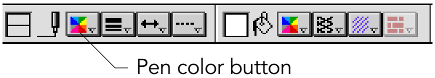
PEN COLOR

An object’s Pen color sets the color of the object’s outline in Draft and Sculpt modes. In Wireframe and Hidden Line renderings, Pen colors appear on the outlines of objects. In Solid renderings, Pen colors appear as solid surface colors on objects that do not have rendered Surface materials.

Pen color can be applied to 2D objects in Draft mode and to 3D objects in Sculpt mode.

All objects have a Pen color attribute.

You can use the Pen color button on the Attributes bar or the Pen color tab in the Pen palette to set the current Pen color and to change the Pen colors of selected objects.



To apply a Pen color

1. Select 2D or 3D objects whose Pen color you want to change.
2. Press the Pen color button in the Attributes bar and drag into the Pen color tab to select a Pen color. If necessary, use the scroll bar to view additional colors (if the palette is floating).

To create a Pen color

When you add a color to the Pen color tab, the new color appears after the preset colors on the tab. The color also appears on the Fill color tab.

1. Open the Pen palette by dragging the palette away from the Pen color button on the Attributes bar.
2. Press the right-arrow button and choose New from the pop-up menu. The Color Picker dialog box appears. See “Pen weight” on page 153.

To edit a Pen color

1. Open the Pen palette by dragging the palette away from the Pen color button on the Attributes bar.
2. Double-click a color you created that you want to edit. The applicable operating system Color Picker dialog box appears.

   ▲ You can also click a color and then choose Edit in the palette's pop-up menu to edit the selected color.

Note: You can't edit or delete the preset colors on the Pen color tab. If you double-click a preset color, nothing happens. If a preset color is selected, the Edit command is not available in the pop-up menu, and the trash can icon at the bottom of the palette is not available.

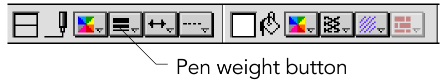
To delete a Pen color

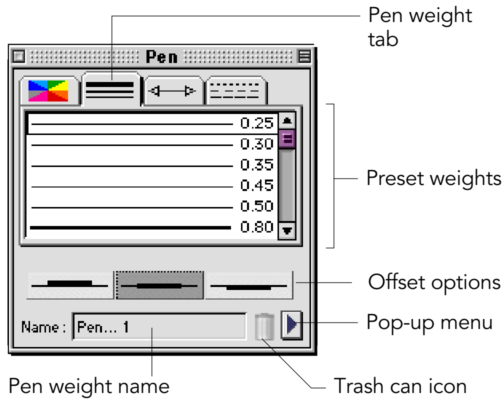
1. Open the Pen palette by dragging the palette away from the Pen color button on the Attributes bar.
2. Click the color on the Pen color tab that you want to delete. You can delete any color that you have added to the palette, but not the preset colors.
3. Click the trash can icon in the bottom of the Pen palette to delete the selected color.

PEN WEIGHT

Pen weight is a measure of the thickness of an object's outline. Pen weight is specified in millimeters. You can apply Pen weights in Draft mode only. All 2D objects have a Pen weight.

You can use the Pen weight button on the Attributes bar or the Pen weight tab in the Pen palette to set the current Pen weight, and to change the Pen weight of selected objects.

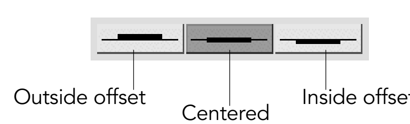




To apply a Pen weight

1. Select objects whose Pen weight you want to change.
2. Press the Pen weight button in the Attributes bar and drag into the Pen weight tab to select a Pen weight. If necessary, use the scroll bar to view additional settings (if the palette is floating).

   ▲ If you use the Pen palette, you can click one of the three buttons at the bottom of the palette to specify the position of the Pen weight on the outline of objects.



For information about the Pen palette, see "Using the Pen and Fill palettes" on page 151.

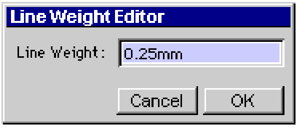
To create a Pen weight

1. Open the Pen palette by dragging the palette away from the Pen weight button in the Attributes bar.
2. Press the right-arrow button and choose New from the pop-up menu. The Line Weight Editor dialog box appears.
3. In the text box, type the width in millimeters of the Pen weight you want to create.
4. Click OK to create the Pen weight. The new Pen weight appears after the preset Pen weights on the Pen weight tab.

To edit a Pen weight

1. Open the Pen palette by dragging the palette away from the Pen weight button on the Attributes bar.
2. Double-click a Pen weight you created that you want to edit. The Line Weight Editor dialog box appears.

   ▲ You can also click a Pen weight to select it and choose Edit in the palette's pop-up menu to edit the selected Pen weight.
3. In the text box, type the new width in millimeters for the selected Pen weight.
4. Click OK to modify the selected Pen weight.



Note: You can't edit or delete the preset Pen weights. If you double-click a preset Pen weight, nothing happens. If a preset Pen weight is selected, the Edit command is not available in the pop-up menu, and the trash can icon at the bottom of the palette is not available.

To delete a Pen weight

1. Open the Pen palette by dragging the palette away from the Pen weight button on the Attributes bar.
2. Select a Pen weight to delete. You can delete any Pen weight you have created, but not the preset Pen weights.
3. Click the trash can icon at the bottom of the Pen palette to delete the selected Pen weight.

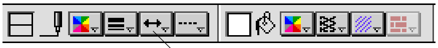
ARROWHEAD

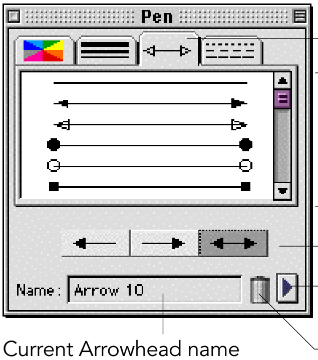
An Arrowhead is terminator, such as an arrow, slash, or other shape, that appears at the end of a line segment. You can apply Arrowheads in Draft mode only.

You can use the Arrowhead button on the Attributes bar or the Arrowhead tab in the Pen palette to change the current Arrowhead or apply Arrowheads to objects.

Only 2D lines, polylines, and dimension objects can display an Arrowhead. An Arrowhead can appear at one or both ends of a line or dimension line, and at one or both end segments of a single polyline.

Note: You can apply an Arrowhead to other objects, including rectangles and polygons, but Arrowheads do not appear unless you separate the objects into segments by applying the Unchain command to the objects.





To apply Arrowheads

1. Select the objects to which you want to apply an Arrowhead.
2. Press the Arrowhead button on the Attributes bar and drag into the palette to select an Arrowhead. If necessary, use the scroll bar to view additional settings (if the palette is floating).

   ▲ To change the position of the Arrowhead, drag to the Arrowheads Mode submenu and choose the left, right, or two-sided option. The Arrowhead examples in the palette are based on the current Arrowhead Mode.

   ▲ If you use the Pen palette, you can click one of the three buttons under the Arrowhead examples to set the position of the Arrowhead on line segments.

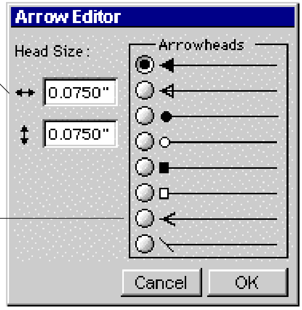
For information about the Pen palette, see “Using the Pen and Fill palettes” on page 151.

To create an Arrowhead

1. Open the Pen palette by dragging the palette away from the Arrowhead button on the Attributes bar.
2. Press the right-arrow button and Choose New from the pop-up menu. The Arrow Editor dialog box appears.
3. Click an Arrowhead style in the Arrowheads list.
4. Type the horizontal dimension of the Arrowhead in the first text box under Head Size. Type the vertical dimension of the Arrowhead in the second text box.
5. Click OK to create the Arrowhead. The new Arrowhead appears after the preset Arrowheads on the Arrowhead tab.

Type horizontal and vertical dimensions

Select an Arrowhead style



To delete an Arrowhead

1. Open the Pen palette by dragging the palette away from the Arrowhead button on the Attributes bar.
2. Select an Arrowhead to delete. You can delete any Arrowhead you have created, but not the preset Arrowheads.
3. Click the trash can icon at the bottom of the Pen palette to delete the selected Arrowhead.

To edit an Arrowhead

1. Open the Pen palette by dragging the palette away from the Arrowhead button on the Attributes bar.
2. Double-click an Arrowhead you created that you want to edit. The Arrow Editor dialog box appears .

   ▲ You can also click an Arrowhead to select it and choose Edit in the palette's pop-up menu to edit the selected Arrowhead.
3. Change any of the current settings for the Arrowhead: the Arrowhead style in the Arrowheads list, the horizontal dimension in the first text box, and the vertical dimension in the second text box.
4. Click OK to replace the selected Arrowhead on the Arrowhead tab with the new settings.

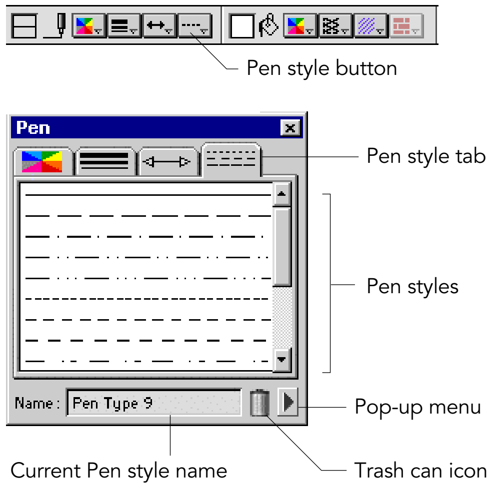
Note: You can't edit or delete the preset Arrowheads. If you double-click a preset Arrowhead, nothing happens. If a preset Arrowhead is selected, the Edit command is not available in the pop-up menu, and the trash can icon at the bottom of the palette is not available.

PEN STYLE

A Pen style defines a 2D object's outline as either an unbroken stroke, or a pattern of dashed or dotted segments. All 2D objects have a Pen style. Pen styles are available in Draft mode only.

Pen styles are represented in the Pen palette as black and white patterns. However, when applied to an object, a Pen style appears in the object's Pen color. Blank segments in a Pen style are transparent.

You can use the Pen style button on the Attributes bar or the Pen palette to apply a Pen style to selected objects, or to change the current Pen style.



To apply a Pen style

1. Select the objects to which you want to apply a Pen Style.
2. Press the Pen style button on the Attributes bar and drag into the pop-up palette to select a Pen style. If necessary, use the scroll bar to view additional settings (if the palette is floating).

   ▲ You can also use the Pen palette to select and apply Pen styles. For information about the Pen palette, see “Using the Pen and Fill palettes” on page 151.

To create a Pen style

1. Open the Pen palette by dragging the palette away from the Pen style button on the Attributes bar.
2. Press the right-arrow button and choose New from the pop-up menu. The Dash Editor dialog box appears.
3. Click in the Dash Editor window below the ruler where you want a line segment to begin or end. A segment handle appears where you click. Segment handles begin and end segments.
4. Continue to add and position segment handles until you have the dash pattern you want. You can reposition a segment handle by dragging it.
5. Click OK to create the Pen style. The new Pen style appears after the preset Pen styles on the Pen style tab.

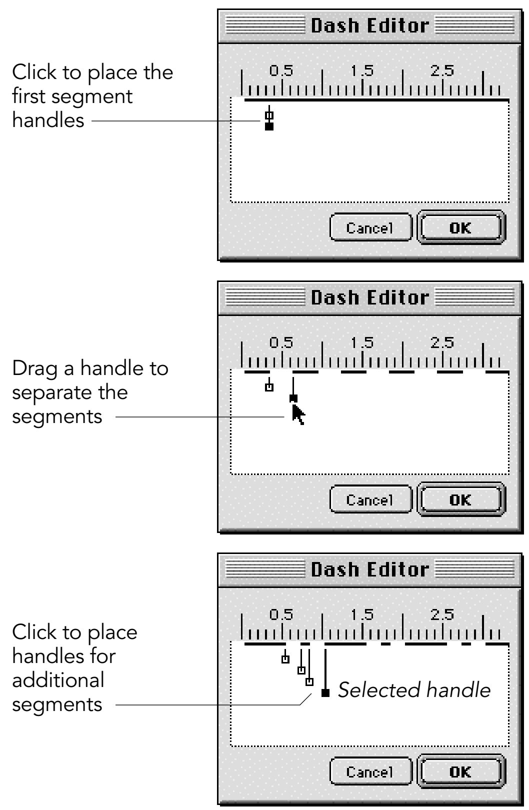


Fig. 179 Using the Dash Editor

To edit a Pen style

1. Open the Pen palette by dragging the palette away from the Pen Style button on the Attributes bar.
2. Double-click a Pen Style you created that you want to edit. The Dash Editor dialog box appears (*figure 179*).

   ▲ You can also click a Pen style to select it and choose Edit in the palette's pop-up menu to edit the selected Pen style.
3. Click in the Dash Editor window below the ruler were you want the first segment to end. Two segment handles appear where you click.
4. Use the handles in the dash preview window to adjust the segments in the selected Pen style.

   ▲ To adjust the segment length, drag the right handle on the segment to the left to shorten the segment, or drag it to the right to lengthen the segment.

   ▲ To move a segment closer to the preceding segment, drag the handle at the left end of the segment to the left.
5. Click OK to replace the selected Pen style on the Pen style tab with the new settings.

Note: You can't edit or delete the preset Pen styles. If you double-click a preset Pen style, nothing happens. If a preset Pen style is selected, the Edit command is not available in the pop-up menu, and the trash can icon at the bottom of the palette is not available.

To delete a Pen style

1. Open the Pen palette by dragging the palette away from the Pen style button on the Attributes bar.
2. Select a Pen style you created that you want to delete.
3. Click the Trash can icon at the bottom of the Pen palette.

FILL ATTRIBUTES

Fill attributes affect the area inside an object's outline. For general information on Fill attributes, see "How attributes apply to objects" on page 147.

This section describes how to apply, create, and modify Fill attributes. You can use the Fill palette or buttons on the Attributes bar to apply fill attributes to objects. The tabs in the Fill palette correspond to the Fill color, Fill pattern, Fill hatch, and Surface material buttons on the Attributes bar.

For information on using the Fill palette to apply attributes, see "Using the Pen and Fill palettes" on page 151.

FILL COLOR

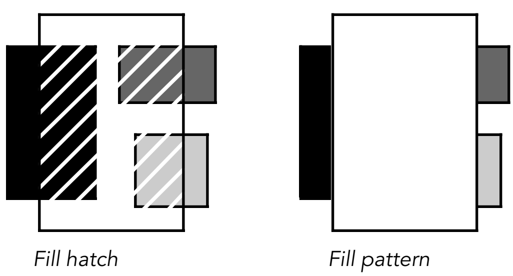
A Fill color is a hue that appears inside a 2D object. Fill colors can be applied to 2D objects in Draft mode. Fill colors are not available in Sculpt mode or Render mode, and can't be applied to 3D objects.

All 2D objects have a Fill color attribute, although the Fill color's appearance depends on the object's Fill pattern or Fill hatch setting. A Fill pattern or Fill hatch fills an object, while the object's Fill color tints the pattern or hatch.

If an object's Fill pattern is set to solid white, the object has an opaque white fill. If its Fill pattern is set to "no pattern" (symbolized by a white box with a diagonal slash in the first position of the Fill pattern tab), the object appears hollow, with no visible Fill color, pattern, or hatch.

Fill hatches are transparent, while Fill patterns are not. Therefore, the effect of a white Fill color is different when an object has a Fill pattern or a Fill hatch.

If you apply white Fill color to an object that has a Fill pattern, the object appears to be filled with solid white. If you do the same to an object that has a Fill hatch, only the hatch lines are white and the rest is transparent. On a white background this isn't visible, but it can be seen if darker objects are in the background.

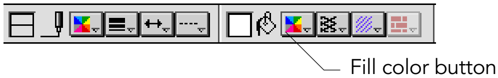


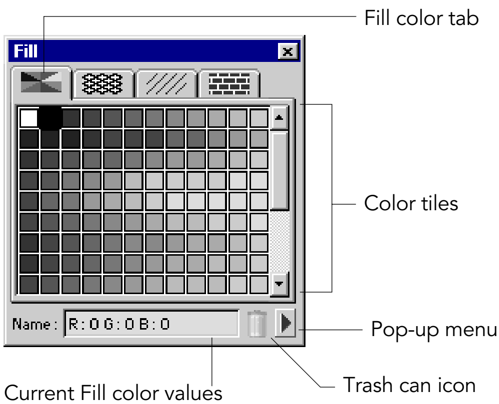
The large rectangle in the foreground has a Fill hatch (left) and a solid Fill pattern (right). When the object's Fill color is white, the lines in the Fill hatch are white, and the Fill pattern is an opaque white rectangle.

If an object has a Fill pattern other than solid white, the black areas of the Fill pattern are colored with the object's Fill color. For example, if you apply a checkerboard Fill pattern to an object, and the object's Fill color is red, the object's interior has a red-and-white checkerboard pattern.

Fill colors are defined in the RGB color space. The preset colors in DENEBACAD, and any colors you create, are available as both Pen colors and Fill colors; the same tab appears in the Pen palette and the Fill palette.

You can use the Fill color button on the Attributes bar, or the Fill color tab in the Fill palette, to set the current Fill color, and to change the Fill colors of selected objects. For more information, see "Current attributes" on page 147.





To apply a Fill color to objects

1. Select 2D objects whose Fill color you want to change.
2. Press the Fill color button in the Attributes bar and drag into the Fill color tab to select a color. If necessary, use the scroll bar to view additional colors (if the palette is floating).

To create a Fill color

When you add a color to the Fill color tab, the new color appears after the preset colors on the tab. The color also appears on the Pen color tab.

1. Open the Fill palette by dragging the palette away from the Fill color button on the Attributes bar.
2. Press the right-arrow button and choose New from the pop-up menu. The Color Picker dialog box appears.

To edit a Fill color

1. Open the Fill palette by dragging the palette away from the Fill color button on the Attributes bar.
2. Double-click a color you created that you want to edit. The Color Picker dialog box appears.

   ▲ You can also click a color and then choose Edit in the palette's pop-up menu to edit the selected color.

Note: You can't edit or delete the preset colors on the Fill color tab. If you double-click a preset color, nothing happens. If a preset color is selected, the Edit command is not available in the pop-up menu, and the trash can icon at the bottom of the palette is not available.

To delete a Fill color

1. Open the Fill palette by dragging the palette away from the Fill color button on the Attributes bar.
2. On the Fill color tab, click the color that you want to delete. You can delete any color that you have added to the palette, but not the preset colors.
3. Click the trash can icon at the bottom of the Fill palette to delete the selected color. DENEBACAD removes the color from the Pen color tab and the Fill color tab.

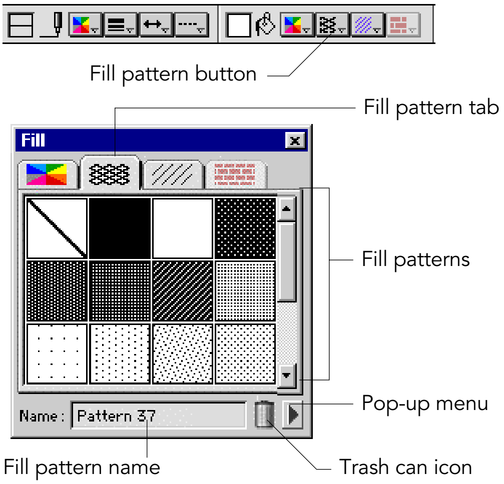
FILL PATTERN

A Fill pattern is a raster pattern that appears in the interior of a 2D object. If the pattern is composed of black-and-white pixels, the black pixels are colored by the object's Fill color.

Fill patterns can be applied to 2D objects in Draft mode only.

An object can have either a Fill pattern or a Fill hatch, but not both. If you apply a Fill pattern or Fill hatch to an object, the new attribute replaces either previous attribute.

You can use the Fill pattern button on the Attributes bar or the Fill pattern tab in the Fill palette to apply Fill patterns to 2D objects and to change the current Fill pattern. For more information, see "Using the Attributes Bar" on page 150, and "Using the Pen and Fill palettes" on page 151.



Fill pattern orientation

Each Fill pattern has a fixed orientation independent of the objects it fills. If you move the object, the Fill pattern doesn't reorient itself. The effect is the same as if you were to move a frame on a wallpaper pattern; different parts of the pattern would appear in the frame. The effect is usually insignificant with small, simple patterns, but the shift might be noticeable with a large pattern.

To apply a Fill pattern to objects

1. Select the 2D objects whose Fill patterns you want to change.
2. Press the Fill pattern button in the Attributes bar and drag into the tab to select a Fill pattern. If necessary, use the scroll bar to view additional Fill patterns (if the palette is floating).

   ▲ To set the current Fill pattern or the Fill pattern of selected objects to "none," choose the box with a diagonal line at the upper-left corner of the Fill pattern tab.

To create a Fill pattern

You can add patterns to the Fill pattern tab by pasting raster images from the Clipboard. You can create patterns using any image-editing or screen capture software.

1. Copy a raster image to the Clipboard.
2. Open the Fill palette by dragging the palette from the Fill pattern button on the Attributes bar.
3. Press the right-arrow button and choose New from the pop-up menu. The Pattern Editor dialog box appears.
4. Press Ctrl+V (Windows) or Cmd+V (Mac) to paste the pattern into the dialog box. Click OK to close the dialog box and add the pattern to the Fill pattern tab.

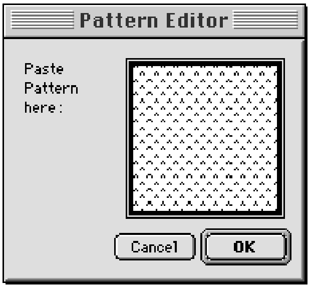
To edit a Fill pattern

1. Open the Fill palette by dragging the palette from the Fill pattern button on the Attributes bar.

   ▲ You can also click a Fill pattern and then choose Edit in the palette's pop-up menu to edit the selected pattern.

2. Double-click a Fill pattern you created that you want to edit. The Pattern Editor dialog box appears.
3. Press Ctrl+V (Windows) or Cmd+V (Mac) to paste the pattern into the dialog box. Click OK to close the dialog box and replace the selected pattern on the Fill pattern tab.

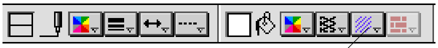
Note: You can't edit or delete the preset Fill patterns. If you double-click a preset Fill pattern, nothing happens. If a preset Fill pattern is selected, the Edit command is not available in the pop-up menu, and the trash can icon at the bottom of the palette is not available.



To delete a Fill pattern

1. Open the Fill palette by dragging the palette from the Fill pattern button on the Attributes bar.
2. Select a Fill pattern you created that you want to delete. You can delete any pattern that you have added to the palette, but not the preset patterns.

3. Click the trash can icon at the bottom of the palette to delete the selected Fill pattern.



FILL HATCH

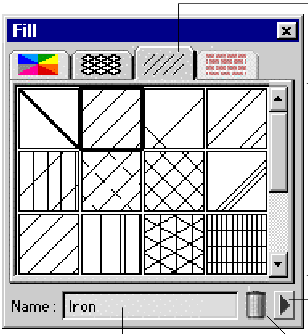
A Fill hatch is a pattern made of line segments, which appears in the interior of a 2D object. Hatch patterns are typically used to represent building materials.

Fill hatches can be applied to 2D objects in Draft mode only.

An object can have either a Fill pattern or a Fill hatch, but not both. If you apply a Fill pattern or Fill hatch to an object, the new attribute replaces either previous attribute.

You can use the Fill hatch button on the Attributes bar or the Fill hatch tab in the Fill palette to apply Fill hatches to 2D objects and to change the current Fill hatch. For more information, see “Using the Attributes Bar” on page 150, and “Using the Pen and Fill palettes” on page 151.

The lines in an object’s Fill hatch appear in the hue specified by the object’s Fill color. For more information, see “Fill Color” on page 158.

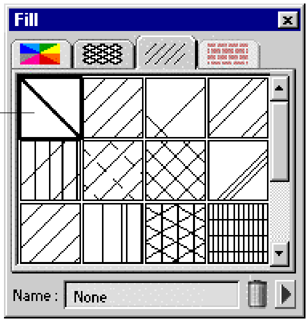


To apply a Fill hatch to objects

1. Select the 2D objects that you want to have Fill hatches.

2. Press the Fill hatch button in the Attributes bar and drag into the tab to select a Fill hatch. If necessary, use the scroll bar to view additional Fill hatches (if the palette is floating).

   ▲ To set the current Fill hatch, or the Fill hatches of selected objects, to “none,” choose the box with a diagonal line at the upper-left corner of the Fill hatch tab.



Note: Selecting “None” in the Fill hatch tab removes an object’s Fill hatch or its Fill pattern, if the object has either a Fill hatch or a Fill pattern.

To create a Fill hatch

1. Open the Fill palette by dragging from the Fill hatch button on the Attributes bar.
2. Press the right-arrow button in the Fill palette and choose New in the pop-up menu. The Hatch Editor dialog box opens.
3. Specify settings for the lines in each Hatch Set. See “Settings for Fill hatches,” next, for more information about the Hatch Editor.
4. Click OK to add the new Fill hatch to the Fill hatch tab in the Fill palette.

Settings for Fill hatches

The options in the Hatch Editor dialog box let you specify the angle, spacing, number of lines, and other settings for a Fill Hatch (*figure 180*).

Press Tab to move the insertion point from one text box to another in the Hatch Editor dialog box. When you change a setting, a new preview of the Fill hatch appears in the preview box.

Hatch Set Select Set 1, Set 2, Set 3, or Set 4 from the Hatch Set pop-up menu. The settings in the dialog box affect the lines in the selected Hatch Set. A Fill hatch can have up to four groups of lines.

 # Lines This value sets the number of lines in the current Hatch Set, from 1 to 255. If the “# Lines” value is zero, no lines appear in the current Hatch Set. To remove a set of lines, type 0 in the text box.

Angle Type a value from -360 to 360 degrees to specify the angle of the lines in the current Hatch Set. A value of 90 or 270 degrees places the lines vertically, while a value of 0 or 180 degrees places the lines horizontally.

Spacing and Stage Spacing Type the distance between lines in the current Hatch Set in the Spacing text box. This value and the Stage Spacing value together set the total distance between the lines in the current Hatch Set.

Origin Offset A value greater than zero places blank space at the origin of the Hatch Fill, which appears as a space at the upper or left edge of a filled object, or the lower or right edge, depending on the Angle setting.

Pen style and Pen weight Use the pop-up menus to set the Pen style (dash pattern) and Pen weight of the lines in the current Hatch Set.

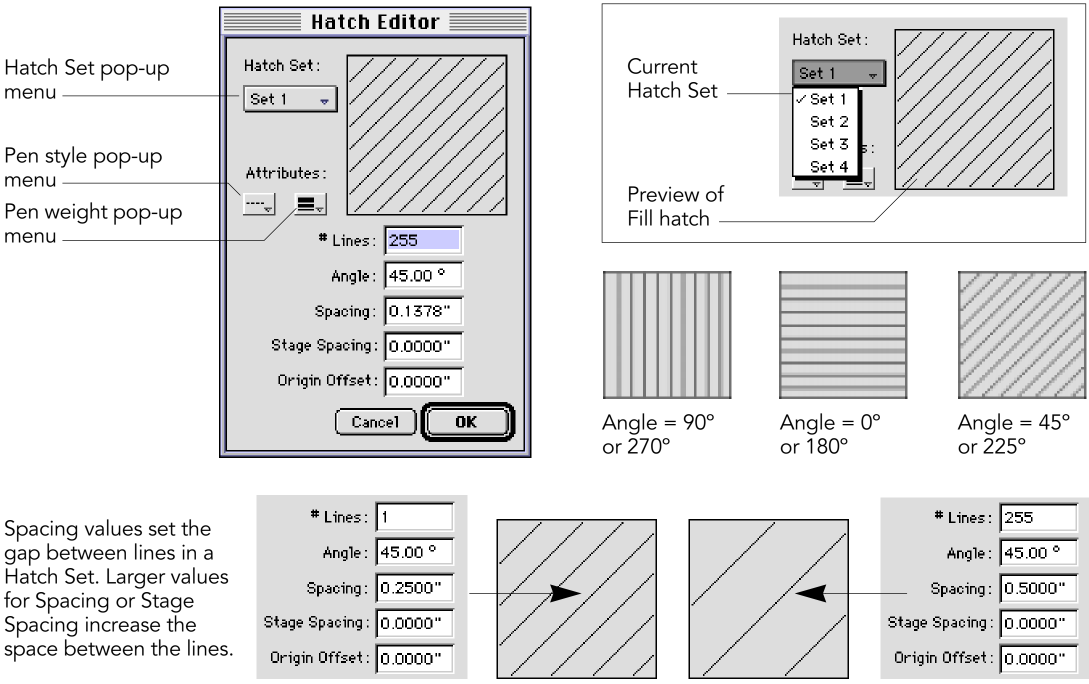


Fig. 180 The Hatch Editor dialog box

To edit a Fill hatch

1. Open the Fill palette by dragging from the Fill hatch button on the Attributes bar.
2. Double-click a Fill hatch you created that you want to edit. The Hatch Editor dialog box appears.

   ▲ You can also click a Fill hatch and then choose Edit in the palette's pop-up menu to edit the selected hatch.
3. Use the Hatch Editor dialog box to adjust the values of the Hatch Sets that make up the Fill hatch. See "Settings for Fill hatches" on page 163 for more information.
4. Click OK to replace the selected Fill hatch with the new settings in the Hatch Editor.

Note: You can't edit or delete the preset Fill hatches on the Fill hatch tab. If you double-click a preset Fill hatch, nothing happens. If a preset Fill hatch is selected, the Edit command is not available in the pop-up menu, and the trash can icon at the bottom of the palette is not available.

To delete a Fill hatch

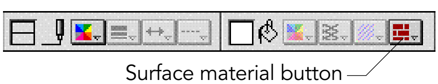
1. Open the Fill palette by dragging from the Fill hatch button on the Attributes bar.
2. Select a Fill hatch you created that you want to delete. You can delete any Fill hatch that you added to the palette, but not the preset Fill hatches.
3. Click the Trash can icon at the bottom of the palette to delete the selected Fill hatch.

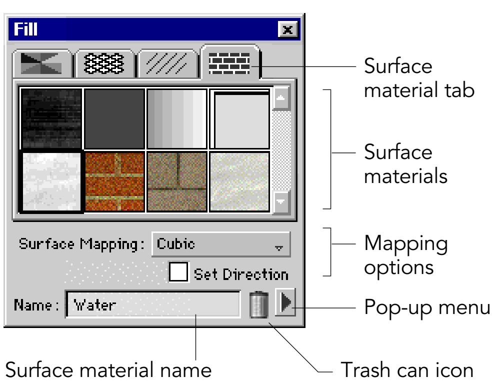
SURFACE MATERIAL

A Surface material is a color raster image. When you use the Solid command to create renderings, DENEBACAD renders a 3D object's Surface material over the object's surfaces.

You can apply Surface materials to 3D objects in Sculpt mode only. However, Surface materials do not appear in Sculpt mode, but appear in Render mode when you use the Solid command and the checkbox labeled Materials is selected in the Render Options dialog box.

You can use the Surface material button on the Attributes bar or the Surface material tab in the Fill palette to set the current Surface material and to change the Surface materials of selected objects. For more information, see "Current attributes" on page 147.





To apply a Surface material to objects

1. Select the 3D objects to which you want to apply a Surface material.
2. To pick a Surface Mapping option, drag into the Surface Mapping pop-up menu and choose Cubic, Brick, Cylindrical, or Planar. For more information, see "Surface Mapping options" on page 165.

   ▲ If you use the Surface material tab in the Fill palette, you can choose a Surface Mapping option from the Surface Mapping pop-up menu on the tab.
3. Press the Surface material button on the Attributes bar and drag into the pop-up palette to select a Surface material. As you drag in the palette, the name of the Surface material under the pointer appears in the Help bar at the bottom of the screen (if the Help bar is displayed).

   ▲ To set the current Surface material, or the Surface material of selected objects, to "None," choose the box with a diagonal line at the upper-left corner of the Surface material tab.

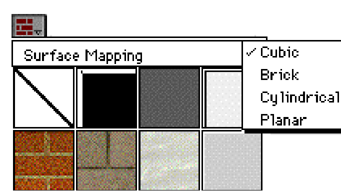


Fig. 181 Surface Mapping pop-up menu

Surface Mapping options

Cubic Tiles the Surface material from the center of each face of the object. This results in the same Surface material pattern on each face of the rendered object.

Brick Creates a seamless effect on objects by conforming the Surface material to the corners of each face of the object. For example, applying a "wall paper" material and choosing Brick maintains the seams of the wall paper by wrapping it around the corners of the walls.

Cylindrical Maps the Surface material to the object based on a radius you set. The radius you set for Cylindrical mapping depends on how you want the Surface material to map the object. In general, the larger the radius, the smaller the pattern will be after it maps to the object's faces. Cylindrical is ideal for mapping Surface materials to rounded objects.

Planar Maps the Surface material to the object's face in the current view, and then extends the edge pixels of the material over every other face of the object.

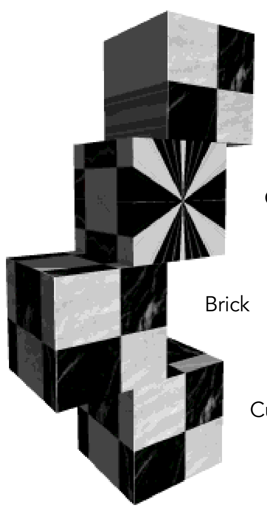


Fig. 182 Examples of the Surface Mapping options

Set Direction You can select the Set Direction checkbox on the Surface material tab to use the drawing vector to set the direction in which the selected Surface material begins mapping around the object. This option is not available from the Surface material button on the Attributes bar.

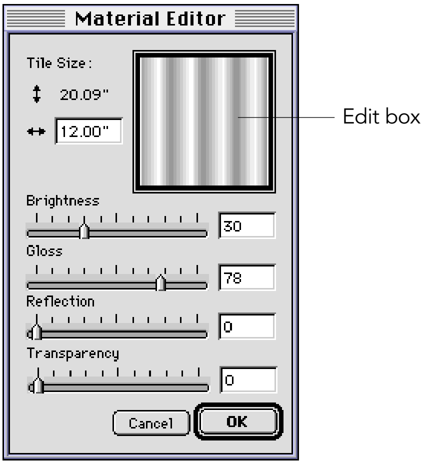
To create a Surface material

1. Copy to the Clipboard a raster image that you want to use as a Surface material. You can use any image-editing or screen capture program to prepare a raster image and copy it to the Clipboard.
2. Drag the Fill palette from the Surface material button on the Attributes bar.
3. Press the right-arrow button and choose New in the palette's pop-up menu. The Material Editor dialog box appears.
4. Click the Edit box. A black outline indicates that you can paste a material into the box. Press Ctrl+V (Windows) or Cmd+V (Mac) to paste the image into the Edit box.
5. Adjust the settings in the dialog box to set the brightness, gloss, reflectivity, and transparency of the Surface material. See "Material Editor options" on page 168 for information on these settings.
6. Click OK. A directory dialog box appears. Select a location on disk to save the Surface material file.

   ▲ You can keep all your Surface material files together, or create separate folders for each project.

   ▲ When you open a document, if DENEBACAD can't locate the Surface materials assigned to objects in the document, a directory dialog appears in which you select a Surface material file to indicate the location of the document's Surface materials.
7. Click Save. DENEBACAD saves the Surface material file on disk.

Note: You must add a new Surface material to the Fill palette to use it in a document; see "To add a Surface material from disk" on page 167.



To edit a Surface material

1. Open the Fill palette by dragging it from the Surface material button on the Attributes bar. Double-click the Surface material you want to edit. The Material Editor dialog opens.

   ▲ You can click a Surface material to select it and then choose Edit in the palette's pop-up menu to edit the Surface material.

   ▲ You can edit the preset Surface materials that ship with DENEBACAD or any Surface materials that you have added to the tab.

2. To replace the Surface material with a new image, click the Edit box. A black outline indicates you can paste a new image into the dialog box. Press Ctrl+V (Windows) or Cmd+V (Mac) to paste an image from the Clipboard.

3. Adjust the settings in the dialog box to set options for the Surface material. See "Material Editor options" on page 168 for information on these settings.

4. Click OK to replace the selected Surface material with the new settings.

To delete a Surface material

1. Open the Fill palette by dragging it from the Surface material button on the Attributes bar.

2. Select a Surface material you have added to the palette that you want to delete.

3. Click the Trash can icon at the bottom of the Fill palette.

Note: If you try to delete a Surface material that has been applied to objects in a document, a message box tells you that you can't delete the material because it is in use. Click OK in the message box to continue.

To add a Surface material from disk

Surface materials are stored in individual files on disk. You might save hundreds of Surface material files, but you probably don't want to use that many Surface materials in every document, or keep that many in the Fill palette.

At any time, you can add a Surface material to the Fill palette by using the palette's Add command. You need to do this when you create a new Surface material (because creating the Surface material stores it on disk, but doesn't add it to the palette).

1. Open the Surface material tab of the Fill palette. In the pop-up menu, choose Add. A directory dialog box appears.

2. Select the file of the Surface material you want to add.

3. Click Open. DENEBACAD inserts the Surface material into the Surface material tab.

Material Editor options

The settings in the Material Editor dialog box control the appearance of Surface materials in Solid renderings.

You can use the Material Editor to adjust the settings of a new material or edit existing materials.

◆ **To change a setting in the Material Editor:** Drag the slider to the left or right until you reach the desired value. Or, type a value in the text box next to the setting you want to change.

When you adjust settings in the Material Editor, a higher number increases the effect of the particular setting (Gloss, Brightness, Reflectivity, and Transparency). For example, a higher Transparency setting makes the Surface material more transparent.

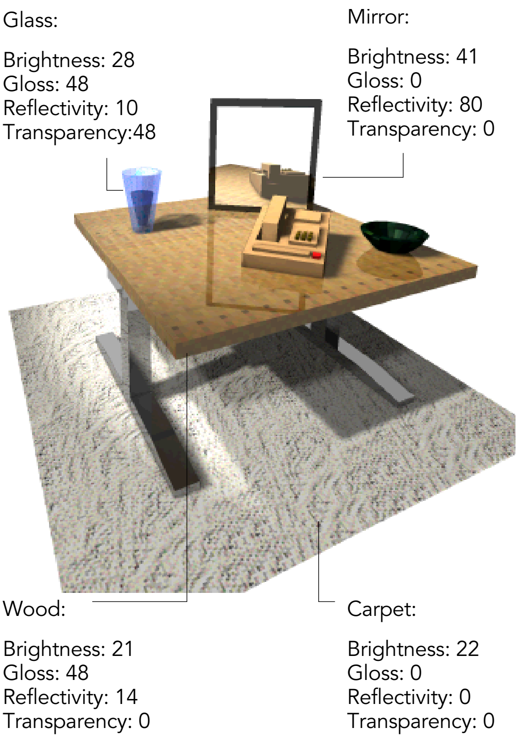


Fig. 183 Surface Material settings

Brightness Gives a Surface material the ability to reflect light. In a Solid rendering, when light shines on a Surface material that has a high degree of Brightness, the colors of the Surface material fade. At a Brightness setting of 100, a Surface material's color fades to white.

Gloss Adds shine to Surface materials. Gloss is similar to Brightness in that it gives the Surface material the ability to reflect light. Gloss, however, does not fade colors, but rather brightens them. When light shines on a Surface material in a Solid rendering, Gloss adds shine to the material by brightening colors but does not add reflectivity.

For example, the shiny surface of a marble wall has gloss, whereas the dull surface of a brick wall has no gloss.

Reflectivity Adds a mirror-like quality to Surface materials. In a Solid rendering, if the Surface material has color and a high value of Reflectivity, the color of the Surface material is not that apparent. The more reflectivity the material has, the less color will be displayed. A Surface material with a Reflectivity setting of 100 displays no color of its own.

For example, to create gray mirror, you can use a gray Surface material with Reflectivity set to 50. The mirror will reflect objects and still retain some of its own color.

Transparency Makes Surface materials clear or partially transparent. In a Solid rendering, you can see through Surface materials that have a high value of Transparency. At a Transparency setting of 100, a Surface material is invisible in a Solid rendering.

For example, you can see the glass in a window, as well as see through the glass, when the Transparency of the glass Surface material is set to 50.

TOOLBARS 12

A variety of toolbars, pop-up menus, buttons, and other controls in the DENEBACAD interface provide quick access to commands, attributes, and settings.

This chapter describes the following toolbars and other controls:

Action buttons A row of buttons on the Attributes bar. Action buttons provide shortcuts for most commands in the Object menu.

Attributes bar A bar near the top of the screen that displays attributes controls and Action buttons.

Display Options pop-up menu A menu in the Status bar that lets you display selected objects and layers.

Extrusion format buttons Three buttons in the Status bar that control the format for Linear and Sweep extrusions.

Help bar A bar at the bottom of the screen that displays command and tool names, and gives feedback on procedures and the progress of operations.

Info bar A bar near the top of the screen containing text boxes and the Coordinate System pop-up menu. The text boxes display coordinate information and let you enter drawing vector data.

Layer bar A group of controls at the bottom of Draft and Sculpt windows that let you work with layers.

Mode pop-up menu Menus at the bottom of document windows that let you select Draft, Sculpt, or Render modes.

Relative View pop-up menu A menu in the Status bar that lets you activate and deactivate Relative Views.

Clipping Plane pop-up menu A menu in the Status bar that lets you activate and deactivate clipping planes.

Snap buttons Two buttons in the Attributes bar that activate snap-to options.

Status bar The current settings for several commands appear in the Status bar. You can choose commands for layers, display modes, extrusion planes, and sectioning from pop-up menus.

3D Plane & 3D Axis pop-up menus Two menus in the Status bar, one of which is displayed according to the extrusion mode, that let you activate extrusion planes and axes.

View Options pop-up menu A menu in the Status bar that lets you control whether 2D and 3D objects are displayed in Draft and Sculpt windows.

Zoom bar A group of controls at the bottom of Draft, Sculpt, and Render windows that let you set zoom levels.

ACTION BUTTONS

You can use the Action buttons in the Attributes bar as shortcuts for common object commands.

The Action buttons can be turned on and off using the Toolbars command in the Layout menu.

The Action buttons are grouped into toolbars. These toolbars act the same way as toolbars in the toolbox.

A right-pointing red arrow in the upper right corner of an Action button indicates that it is part of a toolbar. You can drag from an Action button in a toolbar down to other Action buttons in the toolbar.

The following are the names of the Action button toolbars and the commands available in each:

Reshape Reshape command.

Extrude toolbar Linear, Spin, and Sweep commands (Draft mode). Linear, Spin and Sweep extrusion methods (Sculpt mode).

Trims toolbar Trim, Join, Chamfer, Extend, and Fillet commands.

Path toolbar Convert to Polygon, Round, Offset, and Smooth commands.

Combine toolbar Unite, Intersect, Punch from Front, Punch from Front and Trim, Punch from Back, and Punch from Back and Trim commands.

Move toolbar Move, Move a Copy, and Linear Array commands.

Rotate toolbar Rotate, Rotate a Copy, and Polar Array commands.

Mirror toolbar Mirror and Mirror a Copy commands.

Scale Scale command.

Align toolbar The action buttons in the Align toolbar correspond to the alignment and distribution options in the Align palette.

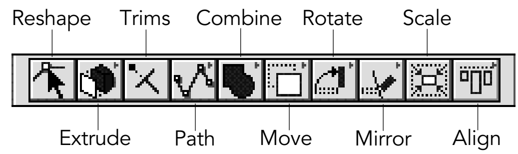


Fig. 184 The Action buttons toolbars

The last command you chose in a toolbar occupies the home position in the toolbar. The Action buttons in the home position are the Action buttons you see without popping a toolbar out.

ALIGN TOOLBAR

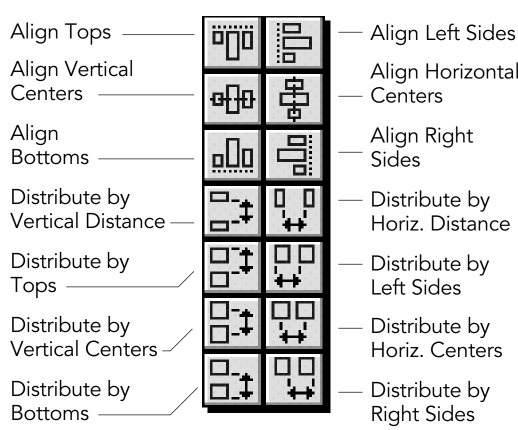


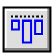
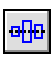
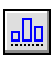
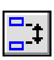
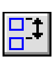
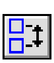
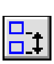
Fig. 185 Align toolbar

To position objects in DENEBACAD, you can use the Align action buttons or the Alignment palette.

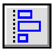
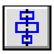
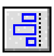
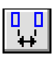
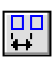
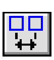
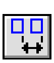
You can align objects horizontally or vertically, or distribute objects within a specified area. As the reference point for alignment and distribution, DENEBACAD uses the objects' bounding rectangles.

When you use the Align action buttons, you can apply one type of alignment or distribution at a time.

Vertical alignment and distribution

Option		**Procedure**
	Top vertical alignment	To line up the tops of selected objects by moving them vertically on-screen, select this button. This lines up the top of each object along an invisible horizontal line that passes through the top of each object.
	Centered vertical alignment	To line up the centers of selected objects by moving them vertically on-screen, select this button. This lines up the center of each object along an invisible horizontal line that passes through the center of each object.
	Bottom vertical alignment	To line up the bottoms of selected objects by moving them vertically on-screen, select this button. This lines up the bottom of each object along an invisible horizontal line that passes through the bottom of each object.
	Inside vertical distribution	To equalize the space between the inside edges of selected objects by moving them vertically on-screen, select this button. This distributes the inside edges of the objects along invisible horizontal lines that pass through the inside edge of each object.
	Top vertical distribution	To equalize the space between the top edges of selected objects by moving them vertically on-screen, select this button. This distributes the inside edges of the objects along invisible horizontal lines that pass through the top edge of each object.
	Centered vertical distribution	To equalize the space between the centers of selected objects by moving them vertically on-screen, select this button. This distributes the centers of the objects along invisible horizontal lines that pass through the center of each object.
	Bottom vertical distribution	To equalize the space between the bottom edges of selected objects by moving them vertically on-screen, select this button. This distributes the bottom edges of the objects along invisible horizontal lines that pass through the bottom edge of each object.

Horizontal alignment and distribution

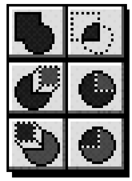
Option		Procedure
	Left horizontal alignment	To line up the left sides of selected objects by moving them horizontally on-screen, select this button. This lines up the left side of each object along an invisible vertical line that passes through the left side of each object.
	Centered horizontal alignment	To line up the centers of selected objects by moving them horizontally on-screen, select this button. This lines up the center of each object along an invisible vertical line that passes through the center of each object.
	Right horizontal alignment	To line up right sides of selected objects by moving them horizontally on-screen, select this button. This lines up the right side of each object along an invisible vertical line that passes through the right side of each object.
	Inside horizontal distribution	To equalize the space between the inside edges of selected objects by moving them horizontally on-screen, select this button. This distributes the inside edges of the objects along invisible vertical lines that pass through the inside edge of each object.
	Left horizontal distribution	To equalize the space between the left edges of selected objects by moving them horizontally on-screen, select this button. This distributes the left edges of the objects along invisible vertical lines that pass through the left edge of each object.
	Centered horizontal distribution	To equalize the space between the centers of selected objects by moving them horizontally on-screen, select this button. This distributes the centers of the objects along invisible vertical lines that pass through the center of each object.
	Right horizontal distribution	To equalize the space between the right edges of selected objects by moving them horizontally on-screen, select this button. This distributes the right edges of the objects along invisible vertical lines that pass through the right edge of each object.

To use the Align action buttons

1. Select the objects you want to align.
2. Using the Align action buttons, choose how you want to align or distribute objects. DENEBACAD aligns or distributes the objects in the manner you specify.

COMBINE TOOLBAR

DENEBACAD provides six buttons in the Combine toolbar: Intersect, Punch from Back, Punch from Back & Trim, Punch from Front, Punch from Front & Trim, and Unite.



Intersect button

The Intersect button executes the Intersect command. The Intersect command is in the Combine submenu in the Object menu.

By using the Intersect command, you can create a new object from the overlapping area of multiple objects. The parts of the objects outside the intersection are then hidden. The Intersect command is available in Draft and Sculpt modes only.

◆ **To use Intersect:** With two or more objects selected, select the Intersect button in the Combine toolbar. Or, click the Intersect button if it's in the home position.

See "Intersect" on page 275 for more information.

Punch from Back button

The Punch from Back button executes the Punch from Back command. The Punch from Back command is in the Combine submenu in the Object menu.

By using the Punch from Back command, you can take an object at the front of the stacking order and use it to "knock out" (or punch) a hole in all of the overlapping objects behind it.

When you use the Punch from Back command on Sculpt mode objects, it's often easier to see the results of the operation in Render mode.

◆ **To use Punch from Back:** With two or more objects selected, select the Punch from Back button in the Combine toolbar. Or, click the Punch from Back button if it's in the home position.

The Punch from Back command does not refer to "Back" view. It refers to taking an object at the front of the object creation order and using it to knock a hole through objects behind it.

See "Punch from Back" on page 277 for more information.

Punch from Back & Trim button

The Punch from Back & Trim button executes the Punch from Back & Trim command. The Punch from Back & Trim button is in the Combine submenu in the Object menu.

By using the Punch from Back & Trim command, you can use an object in the front of the stacking order to "knock out" a hole in all of the overlapping objects behind it.

When you use Punch From Back & Trim, DENEBACAD doesn't recall the original objects when you put the object in Reshape mode, because non-overlapping areas of the front object are "trimmed" to fit the shape of the punched object. If the front object completely overlaps all selected objects behind it, there is no difference between "Punch from Back" and "Punch from Back & Trim."

When you use the Punch from Back & Trim command on Sculpt mode objects, it's often easier to see the results of the operation in Render mode.

◆ **To use Punch from Back & Trim:** With two or more objects selected, select the Punch from Back & Trim button. Or, click the Punch from Back & Trim button if it's in the home position.

The Punch from Back & Trim command does not refer to "Back" view. It refers to taking the object at the front of the stacking order, and using it to knock a hole in objects behind it.

See “Punch from Back & Trim” on page 278 for more information.

Punch from Front button

The Punch from Front button executes the Punch from Front command. The Punch from Front button is in the Combine submenu in the Object menu.

The Punch from Front command knocks holes in the object at the back of the stacking order using the objects in front. Unlike Punch from Back, which uses the front object to punch one hole through all objects behind it, Punch from Front can cut several holes from the back object.

When you use the Punch from Front command on Sculpt mode objects, it’s often easier to see the results of the operation in Render mode.

◆ **To use Punch from Front:** With two or more objects selected, select the Punch from Front button. Or, click the Punch from Front button if it’s in the home position.

The Punch from Front command has no relation to “Front” view. Rather, it refers to taking objects at the front of the object creation order and using them to knock holes through the object in the back of it.

See “Punch from Front” on page 279 for more information.

Punch from Front & Trim button

The Punch from Front & Trim button executes the Punch from Front & Trim command. The Punch from Front & Trim command is in the Combine submenu in the Object menu.

The “Punch from Front & Trim” command knocks holes in the object at the back of the stacking order using the objects in the front, similar to “Punch from Front.” However, unlike “Punch from Front,” areas that don’t overlap the back object are permanently deleted, and cannot be recalled in Reshape mode.

If all objects in front completely overlap the back object, there is no difference in operation between “Punch from Front & Trim” and “Punch from Front.”

When you use this command on Sculpt mode objects, it’s often easiest to see the results of the operation in Render mode.

◆ **To use Punch from Front & Trim:** With two or more objects selected, select the Punch from Front & Trim button. Or, click the Punch from Front & Trim button if it’s in the home position.

The Punch from Front & Trim command does not refer to “Front” view. Rather, it refers to taking objects at the front of the object creation order and using them to knock holes through the object in the back of it.

See “Punch from Back & Trim” on page 278 for information.

Unite button

The Unite button executes the Unite command. The Unite command is in the Combine submenu in the Object menu.

By choosing the Unite command, you can create a new object from the outline of multiple selected objects.

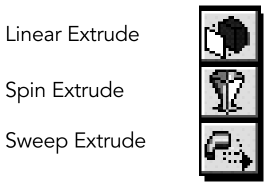
The Unite command provides a simple and powerful way to create complex shapes. After uniting objects, they become one, composite object that you can move, resize, edit and offset as a single object.

◆ **To use Unite:** With two or more objects selected, select the Unite button. Or, click the Unite button if it’s in the home position.

See “Unite” on page 281 for more information.

EXTRUDE TOOLBAR

DENEBACAD provides three buttons in the Extrude toolbar: Linear Extrude, Spin Extrude, and Sweep Extrude. The Extrusion button that occupies the home position of the Extrude toolbar is the active extrusion method.



Linear Extrude button

The Linear Extrude button performs one of three operations: executes the Linear command, changes the active extrusion method, or opens the Extrude Options dialog box. The operation the Linear Extrude button performs depends on how you use it.

◆ **To execute the Linear command in Draft mode:** Select the Linear Extrude button with objects selected. Or, click the Linear Extrude button if it's in the home position. This operation extrudes the selected objects using the Linear extrusion method.

◆ **To activate the Linear extrusion method in Draft mode:** Select the Linear Extrude button with no objects selected.

◆ **To activate the Linear extrusion method in Sculpt mode:** Select the Linear Extrude button.

◆ **To open the Extrude Options dialog box:**

Double-click the Linear Extrude button. The Extrude Options dialog box appears with the Linear tab in front.

See "Extrude submenu" on page 284 for more information about the Linear extrusion method.

Spin Extrude button

The Spin Extrude button performs one of three operations: executes the Spin command, changes the active extrusion method, or opens the Extrude Options dialog box. The operation the Spin Extrude button performs depends on how you use it.

◆ **To execute the Spin command in Draft mode:** Select the Spin Extrude button with objects selected. Or, click the Spin Extrude button if it's in the home position. This operation extrudes the selected objects using the Spin extrusion method.

◆ **To activate the Spin extrusion method in Draft mode:** Select the Spin Extrude button with no objects selected.

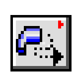
◆ **To activate the Spin extrusion method in Sculpt mode:** Select the Spin Extrude button.

◆ **To open the Extrude Options dialog box:**

Double-click the Spin Extrude button. The Extrude Options dialog box appears with the Spin tab in Front.

See "Extrude submenu" on page 284 for more information about the Spin extrusion method.

Sweep Extrude button



The Sweep Extrude button performs one of three functions: executes the Sweep command, changes the active extrusion method, or opens the Extrude Options dialog box. The operation the Sweep extrude button performs depends on how you use it.

◆ **To execute the Sweep command in Draft mode:** Select the Sweep Extrude button with a sweep path selected.

◆ **To activate the Sweep extrusion method in Draft mode:** Select the Sweep Extrude button with no objects selected.

◆ **To activate the Sweep extrusion method in Sculpt mode:** Select the Sweep Extrude button.

◆ **To open the Extrude Options dialog box:**

Double-click the Sweep Extrude button. The Extrude Options dialog box appears with the Sweep tab in front.

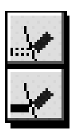
See "Extrude submenu" on page 284 for more information about the Sweep extrusion method.

MIRROR TOOLBAR

The buttons in the Mirror toolbar let you apply Mirror commands to selected objects in Draft and Sculpt modes.

Mirror

Mirror a Copy



Mirror button

The Mirror button executes the Mirror command. The command is also available in the Position submenu in the Object menu.

The Mirror command flips selected objects up to 180 degrees over an axis that you draw. You can set the mirror axis anywhere, and use the Snaps pop-up menu to constrain the axis.

For more information about the Mirror command, see "Mirror" on page 302.

◆ **To apply the Mirror command:** Select the Mirror button when objects you want to mirror are selected. Or, click the Mirror button when it appears in the Mirror toolbar's home position.

Mirror a Copy button

The Mirror a Copy button executes the Mirror a Copy command. The command is also available in the Position submenu in the Object menu.

The Mirror a Copy command flips copies of selected objects up to 180 degrees over an axis that you draw. You can set the mirror axis anywhere, and use the Snaps pop-up menu to constrain the axis.

See "Mirror a Copy" on page 303 for more information about the Mirror a Copy command.

◆ **To apply Mirror a Copy:** Select the Mirror a Copy button when objects you want to mirror are selected. Or, click the Mirror a Copy button when it appears in the Mirror toolbar's home position.

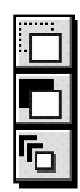
MOVE TOOLBAR

The buttons in the Move toolbar let you apply Position commands to selected objects in Draft and Sculpt modes.

Move

Move a Copy

Linear Array



Linear Array button

The Linear Array button executes the Linear Array command. The command is also available in the Position submenu in the Object menu.

The Linear Array command makes evenly spaced copies of a selected object. You can specify the number of copies. You use the drawing vector to set the distance between copies, or the total distance from the original.

See “Linear Array” on page 301 for more information about the Linear Array command.

◆ **To apply the Linear Array command:**

Select the Linear Array button when objects you want to array are selected. Or, click the Linear Array button when it appears in the Move toolbar’s home position.

Move button

The Move button executes the Move command. The command is also available in the Position submenu in the Object menu.

The Move command repositions selected objects. While you can drag objects with the Selection tool to move them, the Move command lets you specify movement numerically and with the mouse.

See “Move” on page 304 for more information about the Move command.

◆ **To apply the Move command:** Select the Move button when objects you want to move are selected. Or, click the Move button when it appears in the Move toolbar’s home position.

Move a Copy button

The Move a Copy button executes the Move a Copy command. The command is also available in the Position submenu in the Object menu.

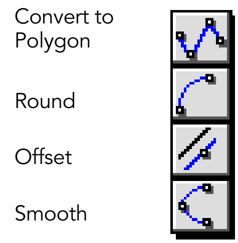
The Move a Copy command copies selected objects to the location you specify, relative to the original objects.

See “Move a Copy” on page 305 for more information.

◆ **To apply Move a Copy:** Select the Move a Copy button when objects you want to copy and move are selected. Or, click the Move a Copy button when it appears in the Move toolbar’s home position.

PATH TOOLBAR

The buttons in the Path toolbar let you apply Path submenu commands.



Convert to Polygon button

The Convert to Polygon button executes the Convert to Polygon command. The command is also available in the Path submenu in the Object menu.

The Convert to Polygon command creates a polygon from selected objects. This operation removes an object’s creation points, and lets you edit the object by moving its segment endpoints.

Curved shapes are converted to segments based on settings in the Preferences dialog box. A higher number of segments makes curves appear smoother, because shorter segments result when you apply the Convert to Polygon command.

See “Convert To Polygon” on page 298 for more information.

◆ **To apply the Convert to Polygon command:** Select the Convert to Polygon button when the object you want to convert is selected. Or, click the Convert to Polygon button when it appears in the Path toolbar’s home position.

Offset button

The Offset button executes the Offset command. The command is also available in the Path submenu in the Object menu.

The Offset command offsets the path of a 2D object to create parallel polylines. The Offset command is not available in Sculpt and Render modes.

See “Offset” on page 298 for more information for more information about the Offset command.

◆ **To execute the Offset command:** Select the Offset button when a 2D object is selected. Or, click the Offset button when it appears in the Path toolbar’s home position.

Round button

The Round button executes the Round command. The command is also available in the Path submenu in the Object menu.

The Round command changes a 2D object’s sharp corners to rounded corners. You can apply the Round command to any object that has at least one vertex. Rounding converts an object to a polyline or polygon (which eliminates specific object properties the object might have had).

The Round command is not available in Sculpt and Render modes. See “Round” on page 299 for more information about the Round command.

◆ **To apply the Round command:** Select the Round button when a 2D object is selected. Or, click the Round action button when it appears in the Path toolbar’s home position.

Smooth button

The Smooth button executes the Smooth command. The command is also available in the Path submenu in the Object menu.

The Smooth command changes polygons and polylines to B-spline curves. The command can be applied to any object that has at least one vertex.

Although you can use the Smooth command on shapes that contain curves, the “smoothing” effect is more pronounced on shapes that have sharp corners. Smoothing a curved object converts the object to a polyline or polygon, which removes any special properties of the original object.

The Smooth command is not available in Sculpt and Render modes. See “Smooth” on page 300 for more information about the Smooth command.

◆ **To apply the Smooth command:** Select the Smooth button when a 2D object you want to smooth is selected. Or, click the Smooth button when it appears in the Path toolbar’s home position.

RESHAPE BUTTON

The Reshape button executes the Reshape command, which puts selected objects into reshape mode.

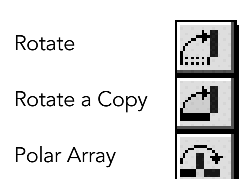
In Reshape mode, an object’s creation points appear as handles that you can drag to reshape the object.

See “Reshape” on page 232 for more information about the Reshape command.

◆ **To apply the Reshape command:** Click the Reshape action button when an object you want to reshape is selected.

ROTATE TOOLBAR

The buttons in the Rotate toolbar let you apply rotation commands to selected objects.



Polar Array button

The Polar Array button executes the Polar Array command. The command is also available in the Position submenu in the Object menu.

The Polar Array command copies objects and places the copies in radial arrangements. See “Polar Array” on page 306 for more information about the Polar Array command.

◆ **To apply the Polar Array command:**

Select the Polar Array button when objects you want to copy are selected. Or, click the Polar Array button when it appears in the Rotate toolbar’s home position.

Rotate button

The Rotate button executes the Rotate command. The command is also available in the Position submenu in the Object menu.

The Rotate command rotates selected objects by the amount you specify. See “Rotate” on page 307 for more information about the Rotate command.

◆ **To apply the Rotate command:** Select the Rotate button when the objects you want to rotate are selected. Or, click the Rotate button when it is in the Rotate toolbar’s home position.

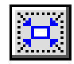
Rotate a Copy button

The Rotate a Copy button executes the Rotate a Copy command. The command is also available in the Position submenu in the Object menu.

The Rotate a Copy command rotates copies of selected objects. See “Rotate a Copy” on page 308 for more information about the Rotate a Copy command.

◆ **To apply Rotate a Copy:** Select the Rotate a Copy button when objects you want to copy and rotate are selected. Or, click the Rotate a Copy button when it appears in the Rotate toolbar’s home position.

SCALE BUTTON



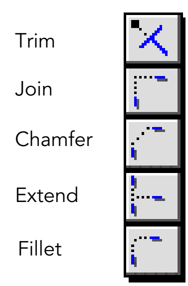
The Scale button executes the Scale command. The command is also available in the Object menu.

The Scale command lets you resize selected objects relative to their original sizes. See “Scale” on page 309 for more information.

◆ **To execute the Scale command:** Click the Scale button when objects you want to scale are selected.

TRIMS TOOLBAR

DENEBACAD provides five buttons in the Trims toolbar: Trim, Join, Chamfer, Extend, and Fillet.



Chamfer button

The Chamfer button executes the Chamfer command. The Chamfer command is in the Trim submenu in the Object menu.

The Chamfer command lets you connect two non parallel lines by joining them together with an angled line.

You can chamfer lines that don’t touch, or lines that intersect.

You can also use the Chamfer command to chamfer the corners of a rectangle. You can chamfer rectangles drawn with the Rectangle Diagonal, Rectangle Center to Corner, or Rectangle 3 Points tool.

When you select the Chamfer command, the Chamfer dialog box opens. You can configure the chamfer settings using this dialog box.

◆ **To execute the Chamfer command:** Select the Chamfer button with or without objects selected. Or, click the Chamfer button if it's in the home position.

The Chamfer command is not available in Sculpt or Render modes. See "Chamfer" on page 311 for more information.

Trim button

The Trim button executes the Trim command. The Trim command is in the Trims submenu in the Object menu.

The Trim command clips, or trims, objects so they end at the cutting edge defined by other objects.

The Trim command is not available in Sculpt and Render modes.

◆ **To execute the Trim command:** Select the cutting edges and then specify the objects to trim or clip, trimming them in sequence.

See "Trim" on page 314 for more information.

Extend button

The Extend button executes the Extend command. The Extend command is in the Trims submenu in the Object menu.

You can use the Extend command to lengthen or shorten a line so that it meets another line. The lines can be two ends of the same open-sided object, or two separate lines. DENEBACAD extends each line so that they meet, but does not combine them into one object.

You cannot use this command on two parallel lines, closed objects, or parallel polylines. You can use this command on lines that intersect.

When two lines do not meet, DENEBACAD lengthens the first line you click until it meets the other line.

When two lines intersect, DENEBACAD trims the first line you click to meet the other line.

Additionally, you can use the Extend command to divide a line into separate line segments. The line you want to divide intersects with the line that divides it.

The Extend command is not available in Sculpt and Render modes.

◆ **To execute the Extend command:** Select the Extend button with or without objects selected. Or, click the Extend button if it's in the home position.

See "Extend" on page 312 for more information.

Fillet button

The Fillet button executes the Fillet command. The Fillet command is in the Trim submenu in the Object menu.

The Fillet command creates a convex round line join between two lines. The lines can be two ends of the same open-sided object, or two separate lines. You can set the radius of the rounded corner; if necessary, DENEBACAD extends the lines so they can form the specified corner. The objects are combined into one object.

You cannot use this command on two parallel lines, closed objects, or parallel polylines.

The Fillet command is not available in Sculpt and Render modes.

◆ **To execute the Fillet command:** Select the Fillet button with or without objects selected. Or, click the Fillet button if it's in the home position.

See "Fillet" on page 313 for more information.

Join button

The Join button executes the Join command. The Join command is in the Trims submenu in the Object menu.

The Join command connects two lines by extending or trimming them until they meet at a vertex. The lines can be two ends of the same open-sided object, or two separate lines. DENEBACAD extends each line so that they meet, but does not combine them into one object.

You cannot use this command on two parallel lines, closed objects, or parallel polylines.

The Join command is not available in Sculpt and Render modes.

◆ **To execute the Join command:** Select the Join button with or without objects selected. Or, click the Join button if it's in the home position.

See "Join" on page 313 for more information.

HELP BAR

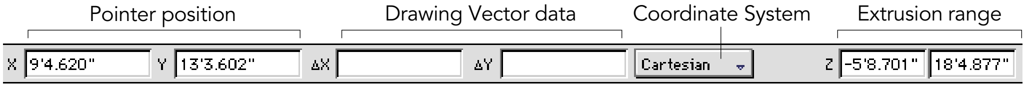
The Help Bar displays several types of useful information. If you point to a command in a menu, or a tool icon in the toolbox, information about the command or tool appears in the Help bar. The Help Bar also displays the amount of free system memory available to DENEBACAD.

When you initiate an operation that requires a process time of over 3 seconds, the Help Bar displays progress bars, which let you see the percentage of the operation completed by the program.

To display or hide the Help bar, use the Toolbars command in the Layout menu. For more information, see "Toolbars" on page 267.

Fig. 186 Help bar

INFO BAR



The Info Bar displays the location of the pointer, the distance and direction an object has been moved, the active coordinate system, and the range of the active set of extrusion planes.

You can also use the Info Bar to create objects by typing their creation points.

The information in the Info Bar is constantly changing as you change modes and views, move the pointer, and move objects. For example, the illustration below displays the Info Bar as it appears in Top view, with the Cartesian coordinate system and a custom set of extrusion planes active.

Coordinate System pop-up menu

The Coordinate System pop-up menu displays the active coordinate system. You can use the pop-up menu to select a new coordinate system.

The coordinate system you choose affects the labels of the text boxes in the Info Bar.

DENEBACAD offers six coordinate systems: Cartesian, Polar, Relational, Bearing, Geographic, and Gradient.

Drawing Vector data

These two text boxes display information for the drawing vector. This information can be displayed in many ways depending on the active coordinate system.

Extrusion range

The extrusion range text boxes display the coordinates of the first and second extrusion planes. The extrusion planes define extrusion depth for linear extrusions.

The values displayed in the text boxes change when you activate a new set of extrusion planes. If one or both of the current extrusion planes is inclined, the values change when you move the pointer because DENEBACAD displays the extrusion range based on the pointer's position.

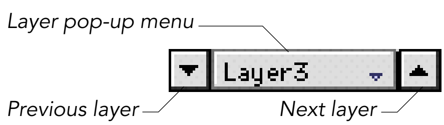
Pointer position

The first two text boxes at the left end of the Info bar display the current pointer coordinates. Depending on the active coordinate system, the pointer position can be displayed as a coordinate position, or as a distance and angle from the Centerpoint. The text boxes' labels also reflect the current coordinate system.

LAYER BAR

You can use the Layer bar at the bottom of the active window to maneuver through the layer structure and open the Layer Manager.

The Layer bar is made up of the Layer pop-up menu in the center, with a Previous Layer button on the left, and a Next Layer button on the right.



Clicking the Previous Layer button activates the next lower layer in the layer structure. Clicking the Next Layer button activates the next higher layer. If the layer you activate is not visible based on the setting in the Layer Manager, the layer becomes visible.

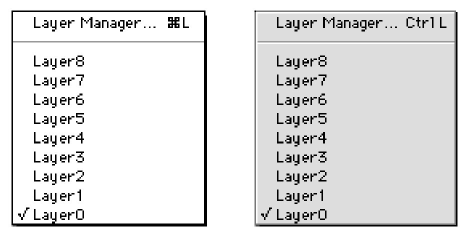
You can use the Layer buttons and pop-up menu the same way in Draft and Sculpt modes.

The Layer pop-up menu

The Layer pop-up menu displays the names of the layers in the current layer structure. If a layer name is to long to fit in the pop-up menu, DENEBACAD displays the beginning of the layer's name.

Open the pop-up menu to activate a layer, or choose the Layer Manager command.

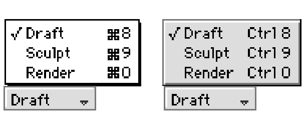
Pressing Ctrl+L (Windows) or Cmd+L (Mac OS) also opens the Layer Manager.



Mac OS *Windows*

MODE MENU

You can use the pop-up menu at the bottom of an active document window to change the window's mode. The Mode pop-up menu is available in Draft, Sculpt, and Render mode windows.



The Mode menu displays the active window's mode

◆ **To change to Draft mode:** Choose Draft in the Mode pop-up menu. Or, press Ctrl+8 (Windows) or Cmd+8 (Mac OS). The window changes to Draft mode.

◆ **To change to Sculpt mode:** Choose Sculpt in the Mode pop-up menu. Or, press Ctrl+9 (Windows) or Cmd+9 (Mac OS). The window changes to Sculpt mode.

◆ **To change to Render mode:** Choose Render in the Mode pop-up menu. Or, press Ctrl+0

(Windows) or Cmd+0 (Mac OS). The window changes to Render mode.

Changing modes changes the tools in the toolbox and the available menu commands.

Setting windows to the same mode

You can use the Mode pop-up menu if you want to have more than one Draft, Sculpt, or Render window open at one time. By choosing a mode from a window's Mode menu, you change the mode of that window, without affecting other open windows.

If you change the mode of any open windows, the Hide commands in the Window menu will correspond to the modes of the open windows. In other words, if there are three Sculpt windows open, the three commands at the top of the Window menu will be "Hide Sculpt" commands.

STATUS BAR

You can use the pop-up menus in the Status Bar to change the way DENEBACAD displays your work, and the way you create objects. You can activate and deactivate extrusion planes, clipping planes, Relative Views, and extrusion formats.

The Status Bar also displays the status of these features. You can tell at a glance what extrusion planes, views, and formats are active.

You can display the Status Bar or remove it from the screen with the Toolbars command in the Layout menu; see "Toolbars" on page 267

The following sections describe the pop-up menus and buttons in the Status Bar in alphabetical order. For more information on commands, refer to the command descriptions in the menu commands chapters.

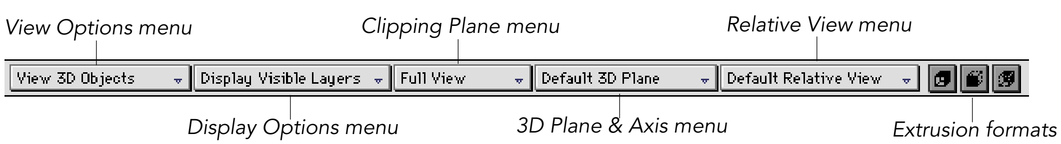


Fig. 187 Pop-up menus and Extrusion format buttons on the Status bar

DISPLAY OPTIONS POP-UP MENU

The commands in the Display Options pop-up menu let you set what layers and objects are visible. These commands are also available in the Layout menu.

Display Visible Layers Displays the objects on visible layers.

Display Active Layer Displays only the objects on the active layer.

Display Selected Objects Displays selected objects only.

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Mac OS Windows

Display Visible Layers, Display Active Layer, and Display Selected Objects are mutually exclusive. If you activate one of these commands, it deactivates the other two.

When you activate a Draft mode or Sculpt mode window, the Status bar shows if Display Visible Layers, Display Active Layer, or Display Selected Objects is in effect.

DISPLAY VISIBLE LAYERS

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+Opt+7
Windows Shortcut: Ctrl+Alt+7

The Display Visible Layers command displays objects on visible layers in the active document window only. To specify visible layers, use the Layer Manager.

See “Display Visible Layers” on page 243 for more information.

DISPLAY ACTIVE LAYER

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+Opt+8
Windows Shortcut: Ctrl+Alt+8

This command displays objects on the active layer only; it affects the active document window only.

See “Display Active Layer” on page 243 for more information.

DISPLAY SELECTED OBJECTS

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+Opt+9
Windows Shortcut: Ctrl+Alt+9

This command displays selected objects on visible layers only; it affects the active document window only.

See “Display Selected Objects” on page 243 for more information.

EXTRUSION FORMAT BUTTONS

The Extrusion Format buttons on the Status bar let you select extrusion formats, which control the shapes created by linear and sweep extrusions. Extrusion formats do not affect Spin extrusions.

When all extrusion formats are active, an extruded 3D object has sides, a front face, and a back face. At least one extrusion format must be active to created 3D objects.

For example, if Front Cap extrusion format is not selected, an extruded object consists of sides and a back face without a front face.

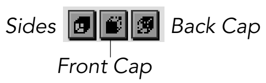


Fig. 188 Extrusion Format buttons

Sides format Objects created when this extrusion format is selected have sides.

For example, a cube extruded from a rectangle will have four sides when Sides format is selected.

Front Cap format Objects created when this extrusion format is selected have a front face. For example, an extruded cube will have a top when Front Cap format is selected.

Back Cap format Objects created when this extrusion format is selected have a back face. For example, an extruded cube will have a bottom when Back Cap format is selected.

If you attempt to extrude objects when all three extrusion formats are deselected, a message tells you that at least one extrusion format must be selected. Click OK to close the message box. Select at least one extrusion format before trying to extrude again.

RELATIVE VIEW POP-UP MENU

A pop-up menu on the Status bar lets you select relative views, including saved relative views and the default relative view. Relative views are available in Sculpt mode only.

CUSTOM RELATIVE VIEW

Mode: Sculpt

When the Status bar displays “Custom Relative View,” the relative view is a custom relative view that you have defined but not saved.

You can’t choose “Custom Relative View” from the Relative View menu. When a custom relative view is active, if you select the default relative view, or a saved relative view, the custom relative view is discarded.

DEFAULT RELATIVE VIEW

Mode: Sculpt
Mac OS Shortcut: Cmd+Opt+Shift+S
Windows Shortcut: Ctrl+Alt+Shift+S

The Default Relative View command selects the Default relative view.

“Default Relative View “ appears in the Status bar until you define a custom relative view.

See “Default Relative View” on page 341 for more information.

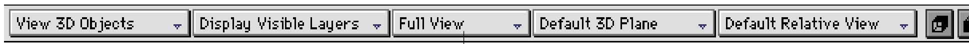
Selecting relative views

In DENEBACAD, there’s always a relative view active; it can be the Default relative view, an unsaved custom relative view, or a saved custom relative view.

Once you save a relative view, you can select it in the Relative View pop-up menu.

CLIPPING PLANE POP-UP MEN4U

Status bar



Clipping Plane pop-up menu

The Clipping Plane pop-up menu commands allow you to activate and deactivate defined clipping planes. The Clipping Plane pop-up menu shows whether a sectioned view is active, or if the full project is visible.

FULL VIEW

Modes: Sculpt, Render
Mac OS Shortcut: Cmd+[
Windows Shortcut: Ctrl+[

You can use the Full View command to deactivate a clipping plane. Once a clipping plane is deactivated, you can reactivate it at any time using the Sectioned View command. This command works the same way in Sculpt mode and Render mode. This command is also available in the Clipping Planes submenu in the View menu. See “Full View” on page 340 for more information.

SECTIONED VIEW

Modes: Sculpt, Render
Mac OS Shortcut: Cmd+]
Windows Shortcut: Ctrl+]

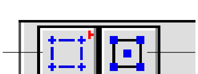
You can use the Sectioned View command to activate a defined clipping plane. Once a clipping plane is activated, you can deactivate it using the Full View command.

This command works the same in Sculpt and Render mode. This command is also available in the Clipping Planes submenu in the View menu. See “Sectioned View” on page 340 for more information.

SNAP BUTTONS

The Snap buttons on the Attributes bar let you select grid snap and object snap, and set the spacing of the snap grid.

Snap grid



Snap to objects

Snap Grid button

The Snap to Grid command restricts the movement of the drawing vector to the dots in the grid, even if the grid is not visible.

By contrast, when Snap to Grid is not active, the drawing vector moves freely even if the grid is visible. When you open a new document, DENEBACAD activates Snap to Grid by default.

You can use the Snap Grid button to execute the Snap to Grid command and the Grid Setup command.

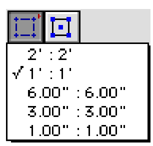
To execute the Snap to Grid command, click the Snap Grid button.

The Snap to Grid command is also available in the Snaps submenu in the Layout menu. See “Snap to Grid” on page 251 for more information.

To choose a grid setup

1. Press the Snap Grid button.
2. Choose a grid measurement in the pop-up menu.

Press to open Grid menu



The Grid Setup command, which lets you set up grid measurements, is available in the Layout menu. See “Grid Setup” on page 245 for more information.

Snap to Objects button

You can use the Snap to Objects button to execute the Snap to Objects command.

◆ **To execute the Snap to Objects command:** Click the Snap to Objects button.

The Snap to Objects command restricts the movement of the drawing vector and can snap the outlines of objects to the grid points.

The Snap to Objects command is also available in the Snaps submenu in the Layout menu. See “Snap to Objects” on page 251 for more information.

3D AXIS POP-UP MENU

Status bar

3D Axis pop-up menu

The the 3D Axis pop-up menu on the Status bar lets you select spin extrusion axes. This menu is available when Spin extrusion method is selected.The menu button displays the name of the active spin extrusions axis for the current view.

CUSTOM 3D AXIS

Modes: Draft, Sculpt

Custom 3D Axis indicates that an unsaved spin extrusion axis is active in the current view.

Until you create a spin extrusion axis, Custom 3D Axis doesn’t appear in the 3D Axis pop-up menu. If you save a spin extrusion axis, the command disappears from the 3D Axis pop-up menu.

When you activate a Draft mode or Sculpt mode window, DENEBACAD shows you if the last spin extrusion axis you defined (but didn't save) is active. When it is, "Custom 3D Axis" appears in the 3D Axis pop-up menu. See "Custom 3D Axis" on page 252 for more information.

DEFAULT 3D AXIS

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+O
Windows Shortcut: Ctrl+Shift+O

The Default 3D Axis command selects the default spin extrusion axis in the current view. In new DENEBACAD documents, the default spin extrusion axis passes through the Absolute Origin, which is the default Centerpoint for new documents. The Default 3D Axis command appears in the 3D Axis pop-up menu the first time you select the spin extrusion method in a view.

The first time you select the Spin extrusion method, DENEBACAD activates Default 3D Axis in the 3D Axis pop-up menu.

In addition, when you activate a Draft mode or Sculpt mode window, DENEBACAD shows you if Default 3D Axis is active. When it is, Default 3D Axis appears in the 3D Axis pop-up menu. See "Default 3D Axis" on page 252 for more information.

Selecting a 3D Axis

Before you select a saved spin extrusion axis, make sure you're in the correct view. The correct view is the view in which you defined the spin extrusion axis. In each view, the names of saved spin extrusion axes appear in the 3D Axis pop-up menu on the Status bar.

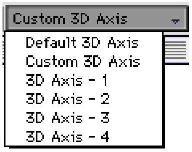


Fig. 189 3D Axis pop-up menu

3D PLANE POP-UP MENU

Status bar

3D Plane pop-up menu

The 3D Plane pop-up menu lets you select extrusion planes. The menu button indicates which extrusion planes are active in the current view.

The 3D Plane pop-up menu is available when Linear or Sweep extrusion method is selected.

CUSTOM 3D PLANE

Modes: Draft, Sculpt

"Custom 3D Plane" appears on the 3D Plane menu button in the Status Bar (and the Layout >

3D Plane submenu) if custom extrusion planes are active in the current view, and have not been saved.

Until you define a set of extrusion planes, “Custom 3D Plane” doesn’t appear in the 3D Plane menu. If you save extrusion planes, “Custom 3D Plane” is removed from the menu.

DEFAULT 3D PLANE

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+O
Windows Shortcut: Ctrl+Shift+O

For each view in DENEBACAD, you can select the default set of extrusion planes by choosing Default 3D Plane in the 3D Plane pop-up menu.

When the default set of extrusion planes is selected, “Default 3D Plane” appears in the 3D Plane pop-up menu.

VIEW OPTIONS POP-UP MENU

Status bar

View Options pop-up menu

The View Options pop-up menu on the Status bar lets you control the display of objects in Draft and Sculpt modes.

The menu button shows which view mode is active, and the pop-up menu is a shortcut for choosing display commands in the View > View options submenu.

You can select the following display options in the View Options menu:

- In Draft mode, you can display only 2D objects, or 2D and 3D objects.
- In Sculpt mode, you can display only 3D objects, or 3D and 2D objects.

When you display both object types, you can lock one type or the other, depending on the window’s mode.

- In Draft mode, you can lock 3D objects when both 2D and 3D objects are displayed.
- In Sculpt mode, you can lock 2D objects when both 3D and 2D objects are displayed.

VIEW 2D & 3D OBJECTS

Modes: Draft
Mac OS Shortcut: Cmd+Opt+5
Windows Shortcut: Ctrl+Alt+5

To display 2D and 3D objects at the same time, choose View 2D & 3D Objects in the View Options pop-up menu. See “View 2D & 3D Objects” on page 347 for more information.

VIEW 2D & LOCK 3D OBJECTS

Modes: Draft
Mac OS Shortcut: Cmd+Opt+6
Windows Shortcut: Ctrl+Alt+6

The View 2D & Lock 3D Objects command locks 3D objects in addition to displaying them in Draft mode. You can’t move, delete, or edit locked objects. This can prevent accidental changes.

See “View 2D & Lock 3D Objects” on page 347 for more information.

VIEW 2D OBJECTS

Mode: Draft
Mac OS Shortcut: Command+Option+4

To view 2D objects only in Draft mode, choose View 2D Objects in the View Options pop-up menu.

See “View 2D Objects” on page 347 for more information.

VIEW 3D & 2D OBJECTS

Modes: Sculpt
Mac OS Shortcut: Cmd+Opt+5
Windows Shortcut: Ctrl+Alt+5

To display 3D and 2D objects in Sculpt mode, choose View 3D & 2D Objects in the View Options pop-up menu.

See “View 2D & 3D Objects” on page 347 for more information.

VIEW 3D & LOCK 2D OBJECTS

Modes: Sculpt
Mac OS Shortcut: Cmd+Opt+6
Windows Shortcut: Ctrl+Alt+6

The View 3D & Lock 2D Objects command locks 2D objects in addition to displaying them in Sculpt mode. You can’t move, delete, or edit locked objects. This can prevent accidental changes.

See “View 3D & Lock 2D Objects” on page 348 for more information.

VIEW 3D OBJECTS

Modes: Sculpt
Mac OS Shortcut: Cmd+Opt+4
Windows Shortcut: Ctrl+Alt+4

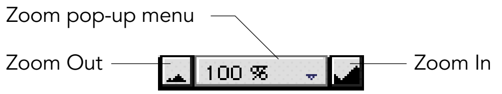
To view 3D objects only in Sculpt mode, choose View 3D Objects in the View Options pop-up menu.

See “View 3D Objects” on page 348 for more information.

ZOOM BAR

You can use the Zoom bar at the bottom of the active window to change the magnification (zoom) level. The Zoom bar is available in Draft and Sculpt modes, and when the Isometric Render tool is selected in Render mode.

The Zoom pop-up menu is located in the center of the Zoom bar. The Zoom bar also contains a Zoom Out button to the left of the Zoom bar and a Zoom In button on the right of the pop-up menu.



◆ **To reduce the zoom level by half:** Click the Zoom Out button.

◆ **To double the zoom level:** Click the Zoom In button.

In Render mode, you must select the Isometric Render tool in the toolbox before you can use the Zoom bar. The Zoom bar is not available in Render mode when the Perspective Render tool is selected.

The Zoom pop-up menu

The Zoom pop-up menu displays the current zoom level. The zoom level appears as a percentage, or as a ratio if the Zoom by Scale option in the Preferences dialog box is selected.

Open the pop-up menu to select a zoom level or a zoom view command. The Zoom pop-up menu contains the following items:

- Preset zoom levels from 1,600% to 6.2% (Mac OS) or 6.3% (Windows). These zoom levels are displayed as percentages, or as ratios if you select the Zoom by Scale option in the Preferences dialog box.

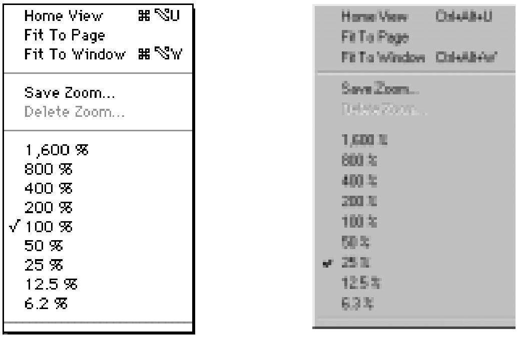
- The Save Zoom command. This command saves zoom levels.

- The Delete Zoom command. This command deletes saved zoom levels.

- The Fit to Window command. This command reduces or increases the zoom level to the maximum zoom level at which the visible objects in the document fill the window.

- The Fit to Page command. This command changes the zoom level to the maximum level at which the defined printing area fills the window. You can define the printing area using the Set Print Area command in the File menu. The default printing area is centered on the Absolute Origin. When you choose this command, DENEBACAD also activates the Show Page Breaks command.

- The Home View command. This command centers the drawing at 100% magnification at the Absolute Origin.



Mac OS Windows

Fig. 190 Zoom pop-up menu

Zoom levels

When you create a new document, DENEBACAD defaults to 100% zoom level. At this zoom level, objects appear on screen at actual scaled size.

The range of available zoom levels depends on the scale that has been set for the drawing. For example, at 1:100 scale, you can set the zoom level from .2% to 1,000,000%.

You can change the zoom level using the zoom buttons, preset zoom levels in the Zoom pop-up menu, and the Zoom tool. Each window maintains its own zoom level independently of other windows.

If the Panned Zoom option is selected in the Preferences dialog box, when you zoom in, DENEBACAD displays a rectangle that corresponds to the visible area after magnification. Move the pointer to position the rectangle, and then click to complete the zoom.

DENEBACAD saves the zoom level with a document and reopens a document at the saved zoom level.

SAVING CUSTOM ZOOMS

You can save custom zoom levels. You can return to a saved custom zoom by choosing it from the Zoom pop-up menu.

This feature makes it simple to use non-preset zoom levels.

To save a custom zoom

1. Choose the Save Zoom command in the Zoom pop-up menu. The Save Zoom dialog box opens.
2. Type the custom zoom value as a percentage or as a ratio if the Zoom by Scale option is selected in the Preferences dialog box. DENEBACAD zooms to the percentage or ratio that you typed.
3. Click OK. The Save Zoom dialog box closes. DENEBACAD adds the custom zoom to the bottom of the Zoom pop-up menu.

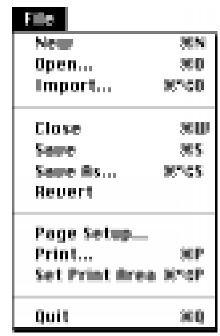
DENEBACAD saves custom zooms with the document. The same custom zooms are available when the document they were created in is reopened. Custom zooms for a document are available in all windows and at all times that the Zoom bar is available.

To delete a custom zoom

1. Select the custom zoom that you want to delete from the Zoom pop-up menu. DENEBACAD zooms in or zooms out to the new zoom level.
2. Choose the Delete Zoom command in the Zoom pop-up menu.
3. A message box appears, asking you to confirm that you want to delete the zoom. Click Yes to remove the custom zoom from the Zoom pop-up menu.

The commands in the File menu let you create, open, close, save, and print documents. The File menu also contains commands used to import and export your work to and from a number of common file formats.

The About DENEBACAD command appears in the Help menu (Windows) or the Apple menu (Mac OS). This command is described first in this chapter, followed by all the commands in the File menu.



Mac OS

Windows

ABOUT DENEBACAD

The startup screen appears when you start DENEBACAD. When the program is running, you can use the About DENEBACAD command in the Help menu (Windows) or the Apple menu (Mac OS) to display the startup screen.

◆ **To display the DENEBACAD startup screen:** Choose About DENEBACAD in the Help menu (Windows) or the Apple menu (Mac OS). The startup screen appears and remains on screen until you click it.

The startup screen displays the name of the registered user and the program version number. It's important to know the program version number if you request technical help from Deneba Software.

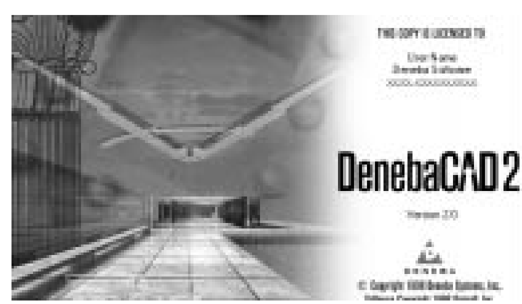


Fig. 191 The startup screen

CLOSE

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+W
Windows Shortcut: Ctrl+F4

To close an active DENEBACAD document without ending the current work session, choose File > Close. Or, press Ctrl+F4 (Windows) or Cmd+W (Mac OS).

When you choose the Close command, DENEBACAD closes all of the windows associated with the document you're closing, but DENEBACAD continues to run.

If you have only one window open, you can also close a document by clicking its Close box or Close button.

To use the Close command

1. Choose File > Close. Or, press Command+W (Mac OS) or Ctrl+F4 (Windows). If the active document has no unsaved changes, DENEBACAD closes all windows of the document.
2. If the active document has unsaved changes, a message asks you if you want to save the changes before closing the document. The dialog box offers three choices:

   ▲ To close the document and save the changes, click Save. If the document hasn't been saved in DENEBACAD format, clicking Save works the same way as the Save As command. Otherwise, it works the same as the Save command. In either case, after saving the document, DENEBACAD closes all windows of the document.

   ▲ To close the document and discard the changes, click Don't Save. DENEBACAD closes all windows of the document without saving changes to the document on disk

   ▲ To keep the document open, click Cancel.

EXIT (WINDOWS)

Modes: Draft, Sculpt, Render
Shortcut: Alt+F4

To end a DENEBACAD work session and close the program, choose File > Exit. Or, press Alt+F4.

If you have an unsaved document open, DENEBACAD asks you if you want to save the document before ending the session.

To quit without saving, click No. To save the document before quitting, click Yes. If the document hasn't been saved in DENEBACAD format, clicking Yes is the same as choosing Save As. Otherwise, clicking Yes is the same as choosing the Save command.

To continue working in DENEBACAD without quitting, click Cancel.

IMPORT

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+O
Windows Shortcut: Ctrl+Alt+O

You can use Import to bring the contents of a file into a DENEBACAD document window. You can import files saved in a variety of file formats that DENEBACAD supports.

When you use the Import command, DENEBACAD integrates the contents of the imported file into the active DENEBACAD document. The result is similar to pasting objects into the active window.

Note: The Import command does not open DENEBACAD files; the Open command does.

To use the Import command

1. Select the window where you want to import a file.

   ▲ Select a Draft window to import all files except QuickDraw 3D.

   ▲ Select a Sculpt window to import DWG, DXF 3D, PTF, and QuickDraw 3D files.

2. Choose File > Import. A directory dialog box appears.
3. Choose a file format in the File Format (Mac OS) or Files of Type (Windows) menu.

   ▲ You can show all files in a folder, even files DENEBACAD can't import. Select All Files in the "Files of type" menu (Windows), or click "Show All Files" (Mac OS).

4. Select the file to import and click Open, or double-click the file name.
5. For some types of files, an options dialog box appears. See the file format descriptions in this chapter for information.

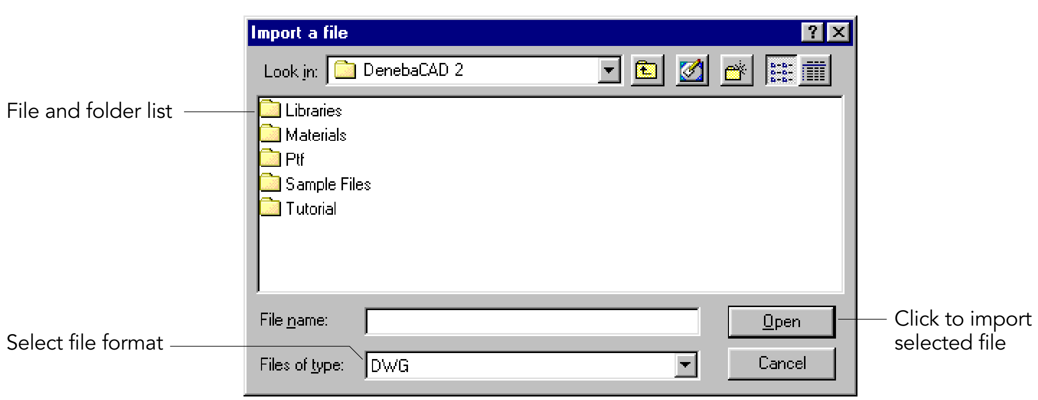


Fig. 192 Import dialog box (Windows)

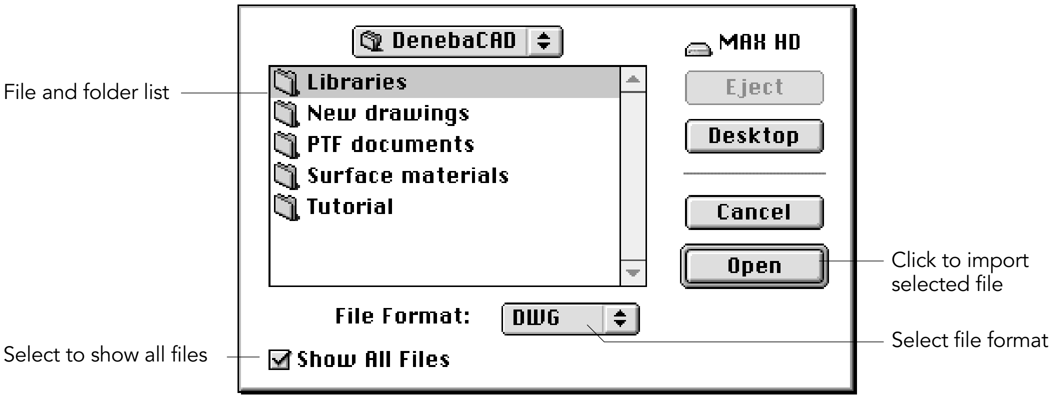


Fig. 193 Import dialog box (Mac OS)

IMPORT FILE FORMATS

DENEBACAD imports these file formats:

BMP A standard Windows file format that supports color and black-and-white raster images.

CGM (Computer Graphics Metafile) An ANSI format for RGB color illustrations. CGM supports 2D vector and raster graphics, layers, and formatted text. See "CGM import options" on page 199.

DWG The proprietary native file format of AutoCAD. DENEBACAD imports documents created in all AutoCAD releases through release 14.01. See "DWG import options" on page 199.

DXF / DXF 3D (Drawing Interchange Format) A metafile format that supports geometric data and text. DXF format is available in Draft mode. DXF 3D is available in Sculpt mode. See "DXF import options" on page 199.

AI EPS / EPS (Encapsulated PostScript) EPS is a format for exchanging PostScript graphics among applications. AI EPS format imports vector objects only. EPS format imports an EPS file as a raster image.

HPGL (Hewlett-Packard Graphics Language)

A vector file format used for plotter files. See "HPGL import options" on page 199.

IGES (Initial Graphics Exchange Specification) A 2D and 3D vector graphics format used in CAD and rendering applications. See "IGES import options" on page 199.

JPEG (Joint Photographic Experts Group) A raster image format with "lossy" compression, typically used to compress photographic images.

PICT A standard Mac OS graphics file format, PICT supports 2D objects, raster images, and text. PICT does not support 3D objects.

PTF (Parametric Text Format) A text file format that describes objects using creation points and attributes from DENEBACAD. When you place a PTF file, a dialog box appears. See "PTF Place Options" on page 401.

QuickDraw 3D A 3D metafile format developed by Apple Computer. QuickDraw 3D is available in Sculpt mode only.

TIFF (Tag Image File Format) A raster image file format that supports high-resolution color, grayscale, and black-and-white images.

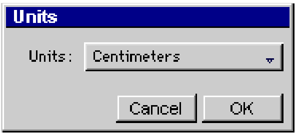
WMF (Windows Metafile Format) A standard Windows file format, WMF supports 2D vector graphics and raster images.

IMPORT OPTIONS

Some file formats have options you can set when importing a file.

CGM import options

When you import a CGM file, the Units dialog box opens.



Choose meters, centimeters, or inches in the Units menu. One CGM unit will be equal to one of the units you choose. Click OK.

DWG import options

When you import a DWG file, the Units dialog box opens. Choose meters, centimeters, or inches in the Units pop-up menu. Click OK.

A message tells you that the document's output scale is being set at 1:1. Click OK to complete the operation and close the message.

The document opens using the measurement system you selected with the output scale set to 1:1. See "Adjusting output scale" on page 199.

DXF import options

When you import DXF file, the Units dialog box opens. Choose meters, centimeters, or inches in the pop-up menu in the dialog box. Click OK.

A message tells you that the document's output scale is being set at 1:1. Click OK to complete the operation and close the message.

The document opens using the measurement system you selected with the output scale set to 1:1. See "Adjusting output scale" on page 199.

HPGL import options

When you import an HPGL file, the Units dialog box opens. Choose meters, centimeters, or inches in the Units pop-up menu. Click OK. One HPGL unit will be equal to one of the units of measure that you choose.

IGES import options

When you import an IGES file, the Units dialog box opens. Choose meters, centimeters, or inches in the Units menu. One IGES unit will be equal to one of the units of measure that you choose. Click OK.

Adjusting output scale

A document in which the output scale is 1:1 could be too large to print on one sheet of paper. An output scale of 1:1 means objects in the document will print at actual size. An object that is 2 feet square will print at 2 feet square if the output scale is 1:1.

To change the output scale of a document, choose Layout > Output Setup. You can select a standard scale or specify a custom scale. See "Output Setup" on page 247.

NEW

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+N
Windows Shortcut: Ctrl+N

You can create a new DENEBACAD document by choosing File > New.

A single window labeled "Untitled-*number*-Draft-Top," appears when you create new document.

- *number* is the document number. The first document you create is labeled Untitled, the second is labeled Untitled-2, the third is labeled Untitled-3, and so on.

- Draft-Top is the current mode and view. New DENEBACAD documents are created with a Draft window in Top view.

- A new document has the default settings for the grid, measurement units, and the output scale.

You can create multiple new documents if your system's resources are sufficient. In general, the number of documents you can have open at once depends on the amount of free memory, and the amount of data contained in the documents.

◆ **To create a new document:** Choose File > New. Or, press Ctrl+N (Windows) or Cmd+N (Mac OS).

The document's name appears with a check mark in the Window menu. The check mark identifies the active document. The names of all open documents appear at the bottom of the menu.

OPEN

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+O
Windows Shortcut: Ctrl+O

You can open an existing DENEBACAD document by choosing File > Open.

When you open a DENEBACAD document, the document opens with the same grid and unit settings, and in the same mode, as when it was saved.

If a document was saved with only a Sculpt mode window only, for example, it opens with a Sculpt mode window. If a document was saved with multiple windows open, it opens with one window in the same mode as the active window at the time the document was saved.

For example, if you open a document that was saved with Draft, Sculpt, and Render mode windows open, and the active window was a Sculpt mode window, the document opens with a Sculpt mode window.

You can have as many documents open at once as your system's resources allow. In general, the number of documents you can have open at once depends on the amount of free memory, and the amount of data contained in the documents.

The easiest way to open a DENEBACAD document stored on disk is to double-click its icon in a folder or on the desktop. If DENEBACAD is not running, this launches the program and opens the document. If DENEBACAD is already running, this makes DENEBACAD active and opens the document.

To use the Open command

1. Choose File > Open. Or, Press Ctrl+O (Windows) or Cmd+O (Mac OS). A directory dialog box appears.
2. In the dialog box, select the DENEBACAD document you want to open, and click Open.

PAGE SETUP (MAC OS)

Mode: Draft

You can specify page size, orientation, and other printing options with the Page Setup command. When you choose Page Setup, a dialog box opens. The options in this dialog box depend on the type and model of printer you are using. If you use more than one printer, you should use the Page Setup command each time you connect to a different printer.

To use the Page Setup command

1. Choose File > Page Setup. The Page Setup dialog box opens.
2. In the dialog box, configure the available options.
3. To implement the new settings, click OK. To keep the original settings, click Cancel.

PRINT

Modes: Draft
Mac OS Shortcut: Cmd+P
Windows Shortcut: Ctrl+P

Printing in DENEBACAD is done in Draft mode only. To print objects from a Sculpt mode window or to print renderings, you can bring the objects or rendering into a Draft mode window.

When you print a DENEBACAD document, the Print dialog box opens. You can specify the settings you want, and then click Print to send a document to the current output device.

Before you print a document, you might need to choose Page Setup (Mac OS) or Printer Setup (Windows) to set the page orientation, paper size, and other options.

To use the Print command

1. Choose File > Print. Or, press Ctrl+P (Windows) or Cmd+P (Mac OS). The Print dialog box opens.
2. In the dialog box, specify the settings you want, and then click Print (Mac OS), or OK (Windows) to output the document.

   ▲ The options in the Print dialog box depend on the type of output device you use and the options available on a particular model of output device. For example, most PostScript devices have a common set of options, which can differ from non PostScript devices. For information on specific output options, consult the documentation for the output device and any printing software that you use.

▲ To close the dialog box without printing, click Cancel.

▲ On Mac OS, to save the settings you specify, click Save Settings.

PRINTER SETUP (WINDOWS)

Mode: Draft

You can specify page size, orientation, and other printing options with the Printer Setup command. When you choose Printer Setup, a dialog box opens. The options in this dialog box depend on the type and model of printer you are using. If you use more than one printer, you should use the Printer Setup command each time you connect to a different printer.

To use the Printer Setup command

1. Choose File > Printer Setup. The Printer Setup dialog box opens.
2. In the dialog box, configure the available options.
3. To implement the new settings, click OK. To keep the original settings, click Cancel.

QUIT (MAC OS)

Modes: Draft, Sculpt, Render
Shortcut: Cmd+Q

To end a DENEBACAD work session and close the program, choose File > Quit, or press Cmd+Q.

If you have an unsaved document open, DENEBACAD asks you if you want to save the document before ending the session.

To quit without saving, click Don’t Save. To save the document before quitting, click Save. If the document hasn’t been saved in DENEBACAD format, clicking Save is the same as choosing Save As. Otherwise, clicking Save is the same as choosing the Save command.

To continue working in DENEBACAD without quitting, click Cancel.

REVERT

Modes: Draft, Sculpt, Render

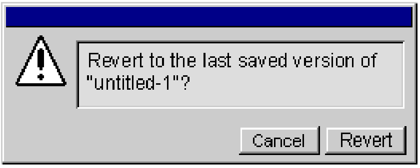
You can discard all changes made to a DENEBACAD document since it was last saved with the Revert command. Essentially, the Revert command is a shortcut for closing the current document and opening the original version stored on disk.

This command cannot be reversed with the Undo command, so be sure you really want to discard all of your changes before you choose Revert.

The Revert command is not available when no documents are open, when the current document has not been saved, and when a document opened from disk has not been changed.

To use the Revert command

1. Choose File > Revert. A warning message appears.



2. Do either:

   ▲ To discard the current document and replace it with the version stored on disk, click Revert.

   ▲ To keep the current document, click Cancel.

SAVE

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+S
Windows Shortcut: Ctrl+S

You can store the current DENEBACAD document on disk by choosing File > Save. Or, press Ctrl+S (Windows), or Cmd+S (Mac OS).

If the Save command is not available (appears dimmed in the menu), the document has not changed since it was last saved.

As you work in DENEBACAD, you should use the Save command often to keep the current version of your work safely stored on disk.

DENEBACAD will not save a document unless you choose the Save command.

If a document hasn't yet been saved in DENEBACAD format, choosing Save is the same as choosing the Save As command. Save As lets you type a name and select a folder in which to store the document. See "Save As" on page 204.

If a document has been saved in DENEBACAD format, the Save command stores a new version of the document on disk; the new version overwrites the previous version. If you want to keep the previous version, use the Save As command to save a copy of the document with a different file name.

SAVE AS

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+Option+S
Windows Shortcut: Ctrl+Alt+S

You can save a copy of the active document on disk with the Save As command. You can also save (export) a file in a variety of standard file formats using Save As.

When you use Save As, you can save an entire document, a particular view of a document, or only the visible objects.

The native format for saving documents is "DENEBACAD." You should always save your work in DENEBACAD format. This ensures that you save all the items contained in a document, including 2D and 3D objects, attributes, layers, library objects and symbols, relative views, and other DENEBACAD-specific features.

Important: Information and objects can't always be converted successfully when you export to other file formats. If you want to be able to edit a document in DENEBACAD, be sure to save the document in DENEBACAD format before you export it using another file format.

Generally, exporting your work from DENEBACAD should be the final step in a project, after you have saved your work in DENEBACAD format.

Exporting renderings

You can use Save As to export images from Render mode. You can select BMP, JPEG, Photoshop, PICT, and QuickTime image file formats in Render mode. Or, you can copy and paste rendered images into a Draft window, then use Save As to export renderings in other file formats.

To use the Save As command

1. Select the mode from which to export.
   - ▲ You can select a Draft window for all formats, except QuickDraw 3D.
   - ▲ You can select a Sculpt window to export to QuickDraw 3D, DWG, DXF, and PTF formats.
   - ▲ You can export images from a Render window in BMP, JPEG, Photoshop, PICT, and QuickTime image formats.
   - ▲ Saving in DENEBACAD format saves the entire DENEBACAD document in any mode.
2. Choose File > Save As. Or, press Ctrl+Alt+S (Windows) or Cmd+Shift+S (Mac OS). The Save As dialog box opens.
3. In the Save As dialog box, open the folder where you want to save the file.
4. Type a file name in the Save As box (Mac OS) or the File Name box (Windows).
5. In the File Format menu (Mac OS) or the Save As Type menu (Windows), choose a format for the saved file. The default format is DENEBACAD.
6. In the Save Scope menu (Mac OS) or the Save Scope area (Windows), choose an option for what to save. These options are described below.
7. Click Save to save the file. For some file formats, a dialog box of options appears. See "Export Options" on page 206.

Save Scope options

Choose an option in the Save Scope menu (Mac OS) or Save Scope area (Windows) to specify what to save.

For DENEBACAD format, the default scope saves the entire document. For other formats, these options might not be available.

Document To save an entire document, choose Document. This option is available for DENEBACAD format only.

Entire View To save all objects in the active window, choose Entire View. This option is not available for DENEBACAD format.

Visible Objects To save only the objects that are visible in the active window, choose Visible Objects.

EXPORT FILE FORMATS

DENEBACAD exports to these file formats:

BMP A standard Windows format that supports color and black-and-white raster images. BMP is available in Draft and Render modes.

CGM (Computer Graphics Metafile) An ANSI format for RGB color illustrations. CGM supports 2D vector and raster graphics, layers, and formatted text. CGM is available for export in Draft mode.

DenebaCAD DENEBACAD's native format. This format is available in Draft, Sculpt, and Render mode. **DenebaCAD Template** format creates a template file for new projects.

DWF (Drawing Web Format) AutoCAD's web format which allows you to publish DENEBACAD documents on the Web. DWF is available for export in Draft mode.

DWG The proprietary native file format of AutoCAD. DWG format is available for export in Draft and Sculpt modes.

When you save a a document in DWG format, DENEBACAD saves the 2D information in AutoCAD release 10 format. You can use AutoCAD release 10 to open and save exported DWG files. AutoCAD release 10 does not import DenebaCAD's 3D information.

AutoCAD releases 11 through 14 can import DENEBACAD's 3D information along with 2D information. You can use release 11 through 14 versions of AutoCAD to open exported files that contain 2D and 3D information.

DXF / DXF 3D (Drawing Interchange Format) A Metafile format that supports plain text, 2D and 3D geometric data. "DXF" appears in Draft mode; "DXF 3D" appears in Sculpt mode.

DXF files do not use a measurement system (English or metric) to define units. Instead, DXF units are generic measurement units.

EPS (Encapsulated PostScript) A file format for exchanging PostScript graphics among page layout and graphics programs, including Deneba Canvas. EPS is available for export in Draft mode.

GIF (Graphics Interchange Format) An 8-bit raster image format frequently used for graphics on World Wide Web pages. GIF is available for export in Draft mode.

HPGL (Hewlett-Packard Graphics Language)

A vector graphics format generally used for plotter printing files. HPGL is available for export in Draft mode.

IGES (Initial Graphics Exchange Specification) A vector graphics format used in CAD and rendering applications. IGES is available for export in Draft mode.

JPEG (Joint Photographic Experts Group) A raster image format with "lossy" compression. JPEG is available in Draft and Render modes.

Photoshop A raster image format developed by Adobe. Photoshop format is available for export in Draft and Render modes.

PICT A standard Mac OS graphics file format, PICT supports 2D objects, raster images, and text. It does not support 3D objects. PICT is available for export in Draft and Render modes.

PNG (Portable Network Graphics) A new cross-platform format for storing raster and vector graphics with compression. PNG is available for export in Draft mode.

PTF (Parametric Text Format) A text file format for describing DenebaCAD objects. PTF is available for export in Draft and Sculpt modes.

When you save a file in PTF format, a dialog box presents several options. See “Options for saving PTF files” on page 400.

QuickDraw 3D A 3D file format, also called “3DMF,” developed by Apple Computer. QuickDraw 3D is available for export in Sculpt mode.

QuickTime Image A raster image format developed by Apple Computer. QuickTime Image is available in Draft and Render modes.

TIFF (Tag Image File Format) A raster image file format that supports high-resolution color and grayscale images. TIFF is available for export in Draft mode.

WMF A standard Windows metafile format that supports vector graphics and raster images. WMF is available for export in Draft mode.

EXPORT OPTIONS

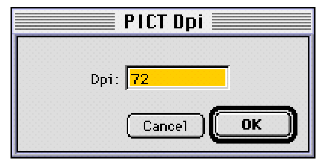
Some file formats have options you can choose when you save a file.

Similar file formats have the same options presented in a dialog box when you import a file. The options shared by related file formats are described together here.

Image resolution option

When you save images in a raster image file format, you can specify the resolution of the exported image.

A dialog box labeled “Dpi” (Dots per inch) appears when you use the following file formats: BMP, EPS, JPEG, Photoshop, PICT, and QuickTime Image.



Type a resolution value expressed as pixels per inch in the box.

Click OK to save an image at the specified resolution in the selected file format.

DXF/DWG options

When you save a file in DXF or DWG format, you can specify a measurement unit. The unit you choose will be converted to DWG/DXF units in the exported file.

Select meters, centimeters, or inches in the Units menu. Click OK to save the file in the specified file format.

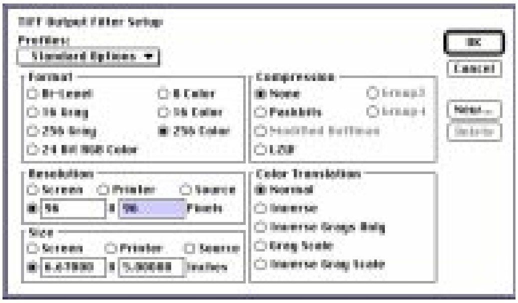
To ensure compatibility with the file format of a particular version of AutoCAD, select the version from the AutoCAD Version menu in the DXF/DWG dialog box.

Use Top Coordinates exports a drawing with coordinates that correspond to Top view in DenebaCAD. Otherwise, the coordinates are based on the current view.

Output Filter Setup options

Several file formats have export options that appear in dialog boxes labeled “Output Filter

Setup.” These file formats are CGM, DWF, GIF, HPGL, IGES, PNG, TIFF, and WMF.



The Output Filter Setup options described next generally work the same for the various file formats. Not every option appears in the dialog box for every file format.

Background Color

For PNG format, select an option for the background color of the image.

Default saves an image with a white background.

To specify a color, select the **User Define** option. Type RGB (red, green, blue) color values in the boxes. Values can range from 0 (no color) to 255 (100% color). Setting all values to 255 produces white. Setting all values to 0 produces black.

Background Rectangle

Select this option for WMF and IGES formats to include a background rectangle object in the exported file. The rectangle corresponds to the bounding box of the saved objects.

Color Translation

Select Normal in the Color Translation area, unless you want to alter or inverse the colors in the image. Gray Scale converts colors to shades of gray. Inverse reverses white and black, and changes colors to their complementary colors.

Compression options

In the Compression area, select a compression method to store a TIFF image in a compressed format.

Some compression methods are available with particular color formats only. For example, Group 3 and Group 4 (CCIT G3 and CCIT G4) are available only when Bi-Level is selected in the Format area.

Fill Mode

Click an option button to select the mode for the fill colors of objects in the exported file.

Color Format options

In the Color Format area, click a button to select a color format for the exported file.

Bi-Level or **Monochrome** saves a black-and-white image; color information is not saved.

16 Gray uses a palette of 16 gray shades.

256 Gray uses a full range of gray tones.

24 Bit RGB Color uses a full range of colors.

8 Color uses an eight-color palette.

16 Color uses a 16-color palette.

256 Color uses a 256-color (8-bit) palette.

YCC color uses a device-independent standard color format.

Interlace

None stores an image in standard, non-interlaced PNG file.

Interlaced stores an image in an split (interlaced) PNG format. This allows a low-resolution image to appear during a download operation.

Line Mode

Click an option button to select Device or Stroked mode for lines in the exported file.

Line Cap Mode

Click an option button to select Device or Stroked mode for line caps in the exported file.

Resolution

In the Resolution area, select an option to specify the resolution of an exported image.

Screen, Printer, Source saves an image at 72 ppi (pixels per inch).

To enter a specific resolution, click the button at the left of the text boxes and enter resolution values in pixels per inch. Use the same values in the boxes, unless you want to stretch one dimension.

Retain Gradients

Select this option for WMF export if you want to preserve color gradients in the file. If you do not select this option, gradients are not exported.

Size

To specify the size for an exported image, select an option in the Size area.

Screen saves an image at the screen size.

Printer saves an image at the printer's paper size.

Source saves an image at its original size.

To enter a custom size, select the button at the left of the text boxes and enter size values. Use the same value in each box unless you want to distort the image.

Transparency

This option affects the appearance of an image when the image is viewed in an application, such as a web browser, that supports transparency for certain image format files.

None saves an image with all colors defined to appear opaque.

To specify a transparent color, select the **Transparent Color** option. Type the RGB (red, green, blue) color values for the transparent color in the boxes. RGB values range from 0 to 255.

Saving output settings

You can save the settings in an Output Filter Setup dialog box as a profile. You can select a profile from the Profiles pop-up menu the next time you save a file.

1. Select the options you want in the Output Filter Setup dialog box.
2. Click New. A dialog box opens.
3. Type a profile name in the text box. In the pop-up menu, you can select an existing profile on which to base the new profile.
4. Click OK. The Profile dialog box closes and you return to the Output Filter Setup dialog box.
5. Click OK. DENEBACAD saves the file with the options you selected.

To delete the current profile, click Delete. DENEBACAD deletes the profile from the list.

SET PRINT AREA

Modes: Draft
Mac OS Shortcut: Cmd+Opt+P
Windows Shortcut: Ctrl+Alt+P

To specify which part of a document to print, choose File > Set Print Area. The printable area is the part of the page available to DENEBACAD for printing documents. The size of the printable area depends on the current printer or plotter driver, and the options selected in the Page Setup or Printer Setup dialog box.

When you choose the Set Print Area command, the printable area appears on screen as a gray rectangle, which you can move to enclose different areas of a drawing. After you choose Set Print Area, the printable area remains visible on screen if Show Page Breaks is in effect. The displayed area can change if you adjust the settings in the Page Setup or Printer Setup dialog box. To hide the printable area, choose Layout > Show Page Breaks.

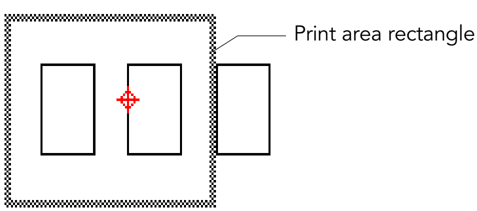


Fig. 194 Setting the print area

To use the Set Print Area command

1. Choose File > Set Print Area. Or, press Ctrl+Alt+P (Windows) or press Cmd+Opt+P (Mac OS). The pointer changes to a cross centered in a gray rectangle. The gray rectangle represents the printable area.

   ▲ *Note:* Depending on the view magnification, you might not see the entire rectangle. In this case, reduce the view magnification.

2. Move the pointer to position the rectangle over the part of the document you want to print. The gray rectangle follows the pointer. Center the area you want to print within the rectangle.
3. Click to set the printable area. The printable area remains visible until you close the document, or make sure Layout > Show Page Breaks isn't in effect.

If you adjust the options in the Page Setup dialog box, the size of the printable area can change. For more information, see "Page Setup (Mac OS)" on page 201.

EDIT MENU COMMANDS

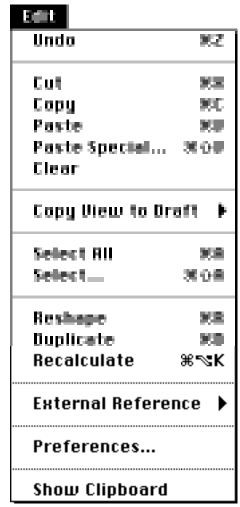
The Edit menu contains commands for working with selections, objects, and the program environment.

Select, Copy, and Paste commands let you manipulate objects through selection and duplication. Reshape lets you edit object shapes.

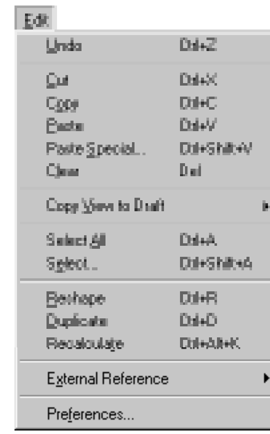
The External Reference commands let you share objects among documents.

The Undo and Redo commands let you cancel and uncancel actions.

The Preferences command lets you adjust the DENEBACAD environment.



Mac OS



Windows

CLEAR

Modes: Draft, Sculpt
Shortcut: Delete or Backspace

The Clear command removes selected objects and highlighted text selections from the active window. The command is available when an object or text is selected in the active window.

Choosing the Clear command is the same as pressing the Delete key to remove selected objects or text from the active window. Choosing Clear doesn’t place the selected objects or text on the Clipboard, nor does it replace the Clipboard contents. If you want to be able to paste objects or text after removing them from a document, use the Cut command instead of Clear.

If you delete objects by mistake, you can restore the objects by choosing Edit > Undo. However, Undo will not restore a text selection that has been deleted with the Clear command.

To use the Clear command

1. In a Draft or Sculpt window, select the objects you want to remove, or highlight a text selection that you want to delete.
2. Choose Edit > Clear. The selection disappears.

COPY

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+C
Windows Shortcut: Ctrl+C

The Copy command puts a copy of selected objects, text selections, or renderings on the Clipboard. Doing this makes it possible to paste the contents of the Clipboard into DENEBACAD documents and into other programs.

The Copy command is available when an object or text is selected in the active Draft or Sculpt window. The Copy command is also available when the Render window is active.

Using Copy to place copies of selected objects, text, or renderings on the Clipboard does not remove the originals from the active Draft or Sculpt window or affect the rendering in the Render window. However, the operation does replace the contents of the Clipboard with the copied selection or rendering.

The Clipboard, which is part of your computer's operating system, stores only one item at a time. Each time you choose Cut or Copy in any application, the current contents of the Clipboard are replaced by the new selection. If you copy items to the Clipboard that you want to use repeatedly, make sure you paste the items into a document before using the Cut or Copy command again.

To make copies of objects without affecting the Clipboard, use the Duplicate command.

To see the contents of the Clipboard, choose Edit > Show Clipboard in DENEBACAD (Mac OS only).

To copy objects or text to the Clipboard

1. Select the objects or text you want to copy in the active Draft or Sculpt window.
2. Choose Edit > Copy. DENEBACAD copies the selection to the Clipboard.

Copying the Render mode window

When the Render mode window is active, the Copy command copies the window contents to the Clipboard as an image. You can then paste the rendered image into a Draft mode window, or into other programs, such as page-layout and image-editing applications.

To copy renderings to the Clipboard

1. Set the rendering options, focal points, and window size for the rendered image.
2. Choose Wireframe, Hidden Line, or Solid in the Rendering menu to generate a rendering of the document.
3. Choose Edit > Copy. DENEBACAD copies the rendered image to the Clipboard.

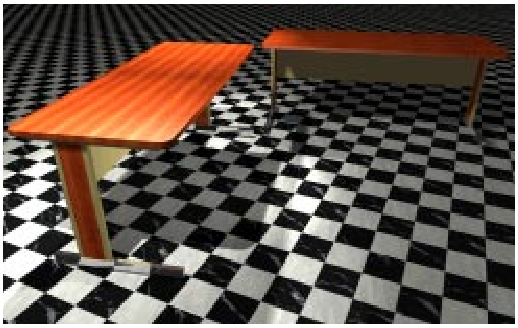


Fig. 195 Rendering copied from Render window

COPY VIEW TO DRAFT

The Copy View to Draft submenu appears in the Edit menu in Sculpt mode only. The Copy View to Draft commands create renderings of 3D objects and insert the renderings into Draft mode, where they can be printed.

HIDDEN LINE

Mode: Sculpt
Mac OS Shortcut: Cmd+J
Windows Shortcut: Ctrl+J

The Hidden Line command in the Copy View to Draft submenu creates a 2D copy of the 3D objects in a Sculpt mode window, and inserts the objects into the Draft window.

This command is useful for extracting elevations from a 3D model.

To use the Hidden Line command

1. Make sure a Sculpt window is active.
2. Choose Edit > Copy View to Draft > Hidden Line. A hidden line rendering appears in Draft mode.

An inserted rendering appears in Draft mode in the same view as in Sculpt mode. For example, if you choose Hidden Line in Sculpt Top view, the rendering appears in Top view in Draft mode.

In Draft mode, a hidden line rendering creates objects that you can move, reshape, and extrude like any other objects in Draft mode.

If the rendering contains multiple objects, the objects are grouped. Choose Object > Ungroup to separate the objects.

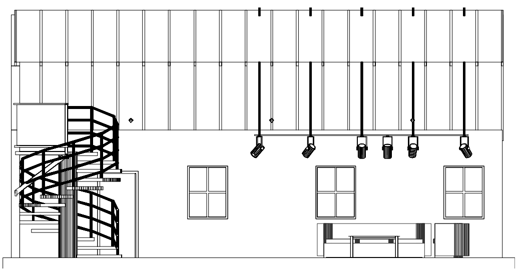


Fig. 196 Hidden Line rendering

SOLID

Mode: Sculpt
Mac OS Shortcut: Cmd+Opt+J
Windows Shortcut: Ctrl+Alt+J

The Solid command in the Copy View to Draft submenu creates a 2D raster image in Draft mode of the 3D objects in a Sculpt mode window. The image created in Draft mode is similar to a Solid rendering created in a Render mode window.

DENEBACAD uses Surface materials, light sources, and the settings in the Rendering Options dialog box when creating a Solid rendering.

To use the Solid command

1. Make sure the active window is a Sculpt mode window.
2. Choose Edit > Copy View to Draft > Solid. Or, press Ctrl+Alt+J (Windows) or Cmd+Opt+J (Mac OS). DENEBACAD creates an image in Draft mode based on the 3D objects in the active Sculpt mode window.

The image appears in the same view as the view of the Sculpt mode window when you choose the command.

For example, if you choose Solid while the active window is a Sculpt mode window in Front view, the objects appear in Draft mode, Front view.

An image created in Draft mode with the Solid command is equal in size to the displayed area of the active Sculpt mode window.

For example, if the active Sculpt mode window displays an area of 30' by 50', the image created in the draft window will be 30' by 50'.

You can resize the image. However, the individual objects in the image can not be resized, reshaped, or extruded.

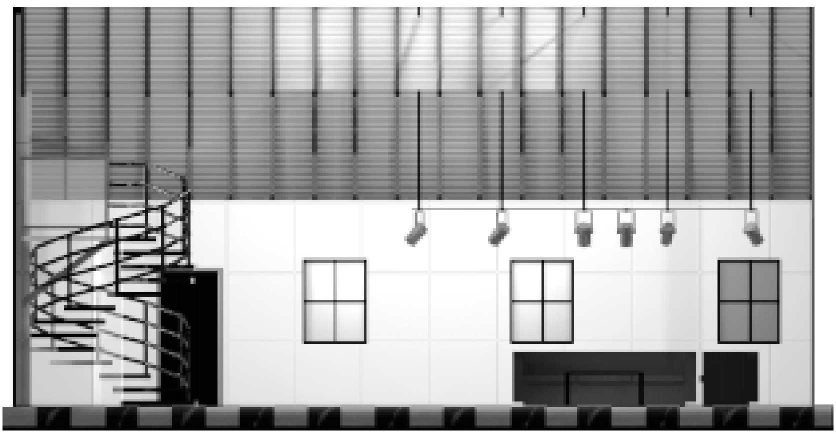


Fig. 197 Solid rendering

CUT

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+X
Windows Shortcut: Ctrl+X

The Cut command removes selected objects or text from a document and transfers the selection to the Clipboard.

The Cut command is available when one or more objects is selected or text is highlighted in Draft or Sculpt mode.

When you use the Cut command, selected objects or text characters disappear from the active Draft or Sculpt window. DENEBACAD places the selection on the Clipboard, and you can then use the Paste and Paste Special commands to place it in a DENEBACAD drawing.

You can also use the Paste command in another application to insert a copy of the Clipboard contents into another program. However, if you paste an object cut from DENEBACAD into a document created in another application, the object might lose some of its properties.

The Clipboard, which is part of the Mac OS operating system, stores only one item at a time. Each time you choose Cut or Copy in any application, the current contents of the Clipboard are replaced by the new selection. If you copy items to the Clipboard that you want to use repeatedly, make sure you paste the items into a document before using the Cut or Copy command again.

To see the contents of the Clipboard, choose Edit > Show Clipboard in DENEBACAD.

To remove an item without affecting the Clipboard, choose Edit > Clear.

To use the Cut command

1. Select the objects or highlight the text that you want to remove from the document.
2. Choose Edit > Cut. Or, press Ctrl+X (Windows) or Cmd+X (Mac OS). DENEBACAD removes the selection from the document and places it on the Clipboard.

DUPLICATE

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+D
Windows Shortcut: Ctrl+D

The Duplicate command makes duplicate copies of selected objects in the Draft or Sculpt window.

The Duplicate command is available when one or more objects is selected in Draft or Sculpt mode. Duplicate is not available when the Render window is active or when a text object is in edit mode.

When you choose Duplicate, DENEBACAD makes a copy of the selected objects in the active Draft or Sculpt window without replacing the Clipboard contents.

DENEBACAD places the duplicated objects in the same position as the originals. The duplicated objects are selected.

Because a duplicate appears in the same position as the original, the duplicate isn't visible as a separate object until you move it away from the original. You can press the keyboard arrow keys or drag a duplicated object to move it away from the original object.

Use the Duplicate command to copy objects in a single step, rather than the two steps required when you copy objects using the Copy and Paste commands.

To use the Duplicate command

1. Select one or more objects that you want to duplicate. You can select 2D, 3D, and Text objects, but not highlighted characters within a Text object.
2. Choose Edit > Duplicate. Or, press Ctrl+D (Windows) or Cmd+D (Mac OS). DENEBACAD places the duplicate objects in the same position as the original selected objects.

EXTERNAL REFERENCE

Commands in the External Reference submenu let you compose multi-scale objects from many DENEBACAD, DWG, or DXF documents in a single DENEBACAD document for presentation and output.

External References, or “X-Refs,” create dynamic associations between an area (“frame”) in a DENEBACAD document, and a DXF or DWG file. X-Refs can also create an association between an entire DENEBACAD document and a DXF or DWG file. X-Refs make it possible to share information between DENEBACAD documents on either Windows or Mac OS platforms.

X-Refs attach a pointer to the external reference file. X-Ref objects appear in the current document, although the objects themselves are not added to the document.

When you attach an X-Ref to a document, changes you make to the original drawing file containing the external reference are automatically reflected in the document that references it. Changes appear each time you open the document containing the external reference. If you know that the original drawing was modified, you can reload the external reference or update the frame layers while you work.

When you attach an X-Ref, its layers, line types, text styles, and other attributes are not added to the current document. Rather, these elements are loaded from the linked file each time you update the X-Ref. Attached X-Refs can themselves contain other, nested X-Refs. When you attach an X-Ref, any nested references contained in the file also appear in the current document.

You can attach as many copies of an X-Ref as you want, each with a different position and scale.

DENEBACAD X-Refs are available in both Draft and Sculpt modes. X-Ref frames are available in Draft mode only.

X-Ref frames

X-Ref frames are “snapshots” of the visible information displayed inside the frame border in the original DENEBACAD document. Frames create dynamic associations between pre-defined areas of DENEBACAD documents.

Using frames, you can automatically update an X-Ref frame appearing in several different documents whenever you change the original object.

One practical use for frames is to create a Draft mode document containing objects drawn at different scales. To do this, you create a frame object, then attach it in either the same or another document for printing.

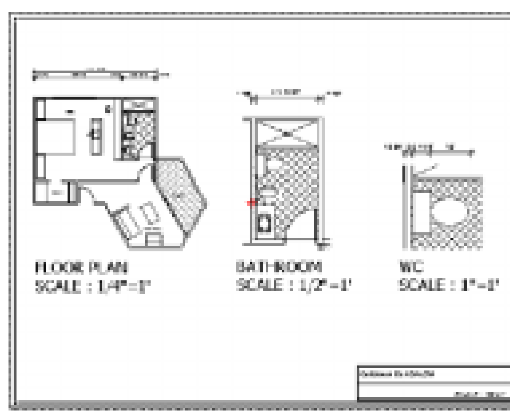


Fig. 198 A multi-scale X-Ref frame in a document

If you change and save a frame, you can refresh its association in a document with the Update Frame Layers command.

In DENEBACAD, frames operate in Draft mode. You can create 2D frame objects from a Draft mode window and attach them in Draft mode. The Frame submenu commands are not available in Sculpt and Render modes.

Attach links an external reference in the active window of the current document.

Reload updates the selected X-Ref with its currently saved information.

Bind copies the selected X-Ref into the current document.

Detach completely removes the selected X-Ref from the current document.

Path updates the directory information of the selected X-Ref.

Scale modifies the selected X-Ref with the selected scale settings.

Create Frame makes the selected area inside the frame border in the active window of the current document an external reference.

Update Frame Layers updates the selected frame so that an object within the frame border on any visible layer is included in the X-Ref.

Show/Hide Frame Borders toggles the display of gray borders around frames.

X-Ref Manager... displays a list and graphical thumbnail of all external references attached to the current document. The X-Ref Manager palette also contains a pop-up menu for manipulating the X-Refs attached to the current document.

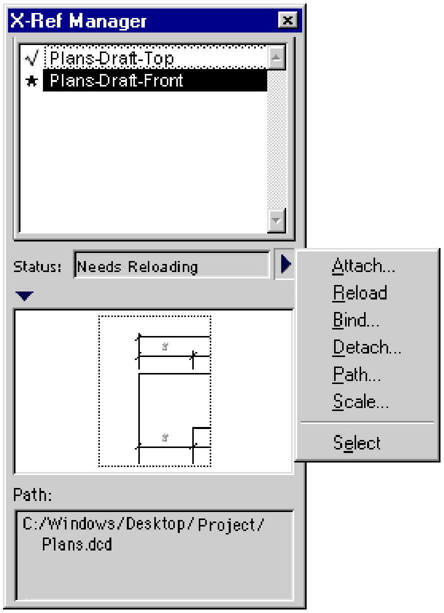


Fig. 199 X-Ref Manager

ATTACH

Mode: Draft and Sculpt

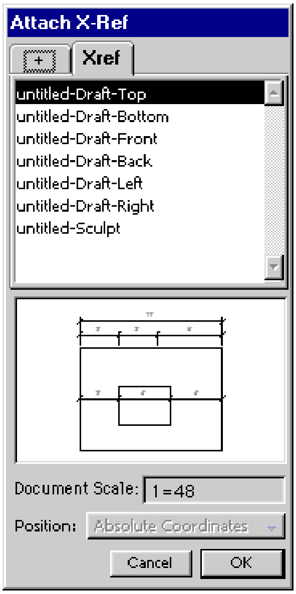
Attaching a separate drawing to the current document creates an external reference. The external reference appears in the current document as an X-ref, whose objects are linked rather than added to the current document. If you modify the linked drawing, you can update the X-Ref so that the current document reflects the latest version of the external reference.

When you attach an X-Ref to the current document, its layers, linetypes, text styles and other attributes are not added to the current document. Rather, these attributes are also loaded from the external reference file each time you update the external reference. Attached reference files can themselves contain other, nested reference files. When you attach an external reference, any nested references contained in the file also appear in the current document. You can attach as many copies of an external reference file as you want. Each copy can have a different position, scale and rotation angle.

To attach an external reference file

1. Choose Edit > External Reference > Attach. A directory dialog box appears.
2. Double-click the document to attach, or select it and click OK.

If the file you select is a DWG, DXF or DENEBACAD file without a frame, it will be attached. If it is a DENEBACAD file that contains one or more frames, the Attach Frames dialog box lets you select the frames to attach.



RELOAD

Mode: Draft and Sculpt

If you modify a drawing that is attached to the current document as an external reference, you can reload the current drawing to display the latest version of the X-Ref drawing.

To reload an X-Ref

1. Choose Reload in the X-Refs submenu. A directory dialog box appears.
2. Using the file and folder list, locate the external reference file you want to update.
3. In the list, double-click the document name or select the DENEBACAD document and click OK.

You can click Cancel to close the dialog box and not reload the external reference.

BIND

Mode: Draft and Sculpt

X-Refs are not part of the document. Rather, they are links to an externally referenced file. To provide a copy of a drawing containing external references to someone else, you must also provide all the external reference files. In addition, the person receiving the drawings must either re-create the same paths you used when linking the external reference file or change the paths for the X-Ref.

The Bind command in the X-Refs submenu lets you insert an X-Ref file into the current document. To provide a copy of a drawing that contains an X-Ref, it is best to Bind the X-Ref file.

If you Bind a scaled X-Ref, you will get a warning explaining that dimension objects will be converted in separated Text, Lines and Arrows, and that clipped objects contained inside frames will be bound as PIC objects.

To bind an X-Ref

1. Choose Bind in the X-Refs submenu. A directory dialog box appears.
2. Using the file and folder list, locate the external reference file you want to insert.
3. In the list, double-click the document name or select the DENEBACAD document and click OK. You can continue to select additional files to bind by selecting them and clicking the OK button.
4. When you are finished, click OK to close the dialog box, or click Cancel to close the dialog box and not bind the external reference file(s).

DETACH

Mode: Draft and Sculpt

To completely remove an X-Ref file from your document, you must detach the reference file. Deleting the external reference does not remove the attributes such as layers and linetypes added to the current document. This command is only available once an external reference file (X-Ref) has been attached to the current document.

To detach an X-Ref

1. Choose Detach in the X-Refs submenu. A directory dialog box appears.
2. Using the file and folder list, locate the external reference file you want to detach.
3. In the list, double-click the document name or select the DENEBACAD document and click OK. You can continue to select additional files to detach by selecting them and clicking the OK button.
4. When you are finished, click OK to close the dialog box, or click Cancel to close the dialog box and not detach the external reference file(s)

PATH

If the file associated with an X-Ref is moved to a different directory or renamed, DENEBACAD displays a message indicating that it can not update the external reference file. You can re-establish the link to the file by changing the path for the external reference file.

To change the X-Ref path

1. Choose Path in the X-Refs submenu. A directory dialog appears. Using the file and folder list, locate the external reference file you want to re-establish the path to.
2. In the list, double click the document name or select the DENEBACAD document and click OK. You can click Cancel to close the dialog box and not re-establish the path to the external reference file.

SCALE

The Scale command in the X-Refs submenu sets the scale for and resizes X-Refs. When you Scale an X-Ref, all objects, except for text, dimension text and arrows will be scaled. Text, dimension text and arrows will be scales according to the settings in Output Setup, unless the “Resize text and arrows” checkbox is checked.

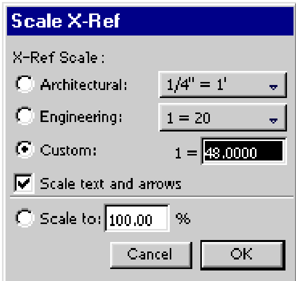


Fig. 200 X-Ref Scale dialog box

If you resize an X-Ref or X-Ref frame, all objects, including text, dimension text and arrows are resized accordingly.

To set the scale of an X-Ref or X-Ref frame

1. Select the X-Ref or X-Ref frame in the active window or select its name from the X-Ref Manager's list.
2. Choose Scale in the X-Ref submenu. You get the Scale X-Ref dialog box.
3. Select the scale factor or percentage you want to apply to the X-Ref. Note: the Resize text and arrows option is disabled when the Resize X-Ref option is selected.
4. Click OK to set the X-Ref Scale, or Cancel to close the dialog box.

CREATE FRAME

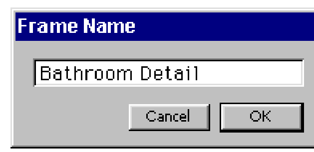
Mode: Draft

The Create Frame command designates one or more selected 2D objects, or a specified area in a Draft window, as an X-Ref which can be attached to any number of additional documents.

The Create Frame command is only available when a Draft mode window is active. When you create an X-Ref, DenebaCAD saves an internal reference file containing the linked selection. When you create frames, the objects must be on visible layers.

To create a frame

1. Select the objects in DenebaCAD you want to include in a frame. If you want to designate an area as a frame, skip to Step 4.
2. Choose Create Frame in the X-Refs submenu. A dialog box appears in which you can name the X-Ref frame.



3. Enter the frame name and click OK. You can click Cancel to close the dialog box and not create the frame.
4. If you did not select objects to frame, a cross pointer appears in the drawing window. Drag the pointer to define a rectangular area as the frame in the drawing.
5. Enter the frame name in the dialog box that appears when you release the mouse and click OK. You can click Cancel to close the dialog box and not create the frame.

Using frame borders

When you create an X-Ref frame, and the Show Frame Borders command is active, DenebaCAD displays a gray rectangle around the frame.

The rectangular border DenebaCAD displays around an X-Ref frame is like a picture window into the external reference file. DenebaCAD references objects that are within this border. You can resize and move the border and move objects into and out of it to change the contents of the frame.

To move an Edition border

1. If necessary, show the frame border by choosing Show Frame Borders in the X-Refs submenu.
2. Drag the frame border to a new location.

To resize the frame border

1. If necessary, show the frame border by choosing Show Frame Borders in the X-Refs submenu.
2. Select the frame.
3. Drag the selection handles of the frame border to resize the area.

SHOW/HIDE FRAME BORDERS

Mode: Draft

Frames are surrounded by a gray rectangle by default. This border helps you distinguish these areas from other objects in a document. However, borders can be distracting and add clutter to the project. You can hide and display the borders as needed.

The Show Frame Borders command displays borders around frames in a drawing. The Hide Frame Borders command turns off the display of these borders. Only one of these commands is available in the X-Refs submenu at a time when a Draft mode window is active.

When the Hide Borders command is active, a border still appears when you select a frame. To do this, you must click the edge of the frame.

◆ **To use Hide frame borders:**
Choose Hide Frame Borders in the X-Refs submenu to make the shaded boundaries invisible. This command is available when borders are displayed.

◆ **To use Show Frame Borders:**
Choose Show Frame Borders in the X-Refs submenu to make the shaded borders visible. This command is available when borders are hidden.

UPDATE FRAME LAYERS

Mode: Draft

When you create an X-Ref frame, all visible layers are included inside the frame border. If you change the visibility of layers containing frames, you can use the Update Frame Layers command to refresh, or update your X-Ref with the latest information. This ensures that all objects on visible layers are included in X-Ref.

To update an X-Ref's layer structure

1. Select the X-Ref frame whose layer structure has been modified or needs updating.
2. Choose Update Frame Layers in the X-Refs submenu.
3. In the list, double-click the document name or select the DENEBACAD document and click OK. You can click Cancel to close the dialog box and not update the layer structure of the X-Ref.

PASTE

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+V
Windows Shortcut: Ctrl+V

The Paste command inserts the contents of the Clipboard into the active Draft or Sculpt window.

The Paste command is available when a Draft or Sculpt window is active and the Clipboard contains objects that can be pasted into a DENEBACAD drawing. The Paste command is not available in Render mode and when the Clipboard contains nothing that can be pasted.

You can place selected objects and highlighted text selections on the Clipboard by using the Cut or Copy commands.

Pasting objects

You can use the Paste command to make copies of objects in the same document or a different DENEBACAD document. You can also use the Paste command in other applications to place objects copied from DENEBACAD into another application's documents.

When you choose the Paste command, DENEBACAD inserts the Clipboard contents in the center of the active window. The pasted objects appear on the active layer.

- To paste the Clipboard contents in a specific location, use the Paste Special command in the Edit menu.
- Because the Paste command does not remove the Clipboard contents, you can use the Paste command to insert multiple copies of an object.
- You can view the contents of the Clipboard with the Show Clipboard command (Mac only).

Viewing pasted objects

The View Options submenu affects the visibility of pasted objects. For example, you can paste 3D objects into a window set to Draft mode, but you won't be able to see the objects unless View 2D & 3D Objects is in effect in the View Options submenu. Conversely, you can paste 2D objects in a window set to Sculpt mode, but you won't see the objects unless View 3D & 2D Objects is in effect.

Pasting renderings in Draft mode

If the Render window contains a Solid rendering, and you use the Copy command to place the contents of the Render window on the Clipboard, you can paste the image into a Draft window. After you paste the rendering, you can use the Recalculate command in the Edit menu to make quick changes to Surface materials, textures, or colors in the rendering. This technique lets you make changes without having to repeatedly re-render every object to see the changes.

To use the Paste command

1. Use the Cut or Copy command in the Edit menu of DENEBACAD or another application to place a selection on the Clipboard.
2. Make sure the window where you want to paste the Clipboard contents is active. You can open a new window using a Window menu command, or click a Draft or Sculpt window to make it active.

3. Choose Edit > Paste. Press Ctrl+V (Windows) or Cmd+V (Mac OS). DENEBACAD pastes the Clipboard contents in the center of the active window on the current layer.

Note: If the view magnification is high, you might need to reduce the view magnification to see the pasted object.

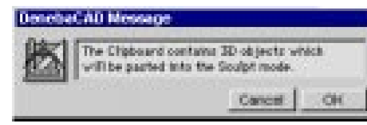
Pasting in other modes

If the type of object on the Clipboard does not match the mode of the active window, the result of choosing the Paste command depends on the view setting of the active window.

If the view setting would make the pasted selection invisible in the active window, DENEBACAD displays a message to tell you that the Clipboard contents will appear in another window.

Pasting 2D objects If the Clipboard contains 2D objects, and the Sculpt window is active when you choose Paste, a message informs you that the contents will appear in the Draft environment. Click OK in the message box to paste into the Draft window. If the Draft window where the selection originated isn't open, DENEBACAD doesn't paste the Clipboard contents.

Pasting 3D objects If the Clipboard contains 3D objects and the Draft window is active when you choose Paste, a message tells you that the contents will appear in the Sculpt environment. Click OK in the message box to paste the Clipboard contents into the Sculpt window. If the Sculpt window is not open, DENEBACAD doesn't paste the Clipboard contents.



If the view setting of the active window is View 2D & 3D Objects, or View 2D & Lock 3D Objects, this allows a pasted selection to be seen even if it doesn't match the mode of the active window. In this case, DENEBACAD pastes the selection in the appropriate window, even if the window isn't active, without displaying a message, because the pasted selection can be seen in the active window.

For example, you copy a 3D object from a Sculpt window, and the Draft window is active when you choose Paste. If the view mode of the Draft window is "View 2D & 3D Objects," DENEBACAD pastes the 3D object in the Sculpt window, but the object also appears in the Draft window.

PASTE SPECIAL

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+V
Windows Shortcut: Ctrl+Shift+V

The Paste Special command inserts the contents of the Clipboard into the active Draft or Sculpt window at a location you specify.

The Paste Special command is available when a Draft or Sculpt window is active and the Clipboard contains objects that can be pasted into a DENEBACAD drawing.

To use the Paste Special command

1. Use the Cut or Copy command to place objects on the Clipboard.
2. Choose Edit > Paste Special. The Paste Special dialog box appears.
3. Choose an option in the Paste pop-up menu and click OK.

   - If you choose Absolute or Relative, DENEBACAD pastes the Clipboard contents in the specified manner.

   - If you choose Cursor Position, the pointer changes to a cross. Position the center of the cross and click in the drawing to place the top-left corner of the pasted selection's bounding rectangle.

Paste Special choices

In the Paste Special dialog box, the Paste pop-up menu contains three positioning methods: Absolute, Relative, and Cursor Position.

Absolute pastes the contents of the Clipboard in the same position in the document as the original objects.

Relative pastes the contents of the Clipboard in the center of the active window. The Paste command pastes objects using this method also.

Cursor Position lets you specify a point at which to paste the top-left corner of the bounding rectangle of the Clipboard contents. For a rendering copied from the Render window, the size of the bounding rectangle depends on the size of the Render window.

Pasting in other modes

When you use the Paste Special command, DENEBACAD displays a message if objects you are pasting do not match the active window mode. For more information, see "Pasting in other modes" on page 223.

PREFERENCES

Mode: Draft, Sculpt, Render

The Preferences command lets you customize the DENEBACAD work environment. The Preferences command is available when any DENEBACAD document is open.

When you choose the Preferences command, the Preferences dialog box appears.

The Preferences dialog box organizes options on two tabs. To bring a tab to the front, click the tab.

- The tab marked with a window icon contains General preference settings.

- The tab marked with an arrow icon contains Tools preference settings.

Changing preferences

Preferences affect the operation of DENEBACAD at the program level. The current preferences affect open DENEBACAD documents and new documents you create. When you click OK in the Preferences dialog box, all preferences take effect immediately.

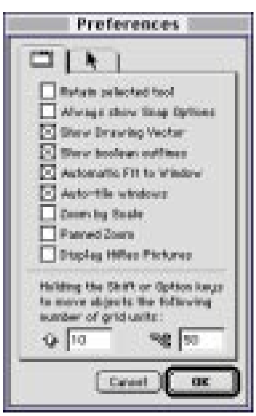
However, existing objects do not change when you implement new preferences that affect objects, such as the “# of Snap Points” and “Round Radius” preferences.

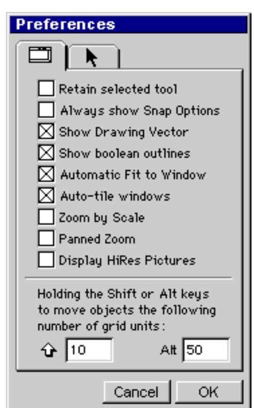
When you save a document, DENEBACAD updates the current preferences information.

To change preference settings

1. Choose Edit > Preferences. The Preferences dialog box opens.
2. Configure the preferences in the dialog box. To change tabs, click the icon at the top of the tab to bring the back tab to the front.
3. To make the new preferences take effect, click OK. To leave the last saved preferences unchanged, click Cancel.

GENERAL TAB





A window icon appears at the top of the General preferences tab.

Retain selected tool

If you select this option, DENEBACAD activates the Selection tool after you use a drawing tool, so you can move the object you just created. After you move the object, you need to click an empty area of the active window to switch back to the previous drawing tool.

Even if you do not move the object you just created, you need to click an empty area of the active window to switch back to the previous drawing tool.

If this option is not checked, DENEBACAD activates the Selection tool after you use any other tool. The Selection tool remains active even after you use it to select or move objects.

Always show snap options

If you check this option, DENEBACAD displays the Snaps pop-up menu whenever you press the pointer in the document. This means the Snaps menu appears if you use a drawing tool and you drag to draw an object. However, the Snaps menu doesn’t appear if you click (rather than drag) in the document to set the first point of an object, but the menu does appear if you press (not click) to set subsequent creation points of an object.

If “Always show snap options” is not checked, the Snaps menu appears when you hold down the Control key and press the pointer (Mac OS), or right-click (Windows) in a drawing.

Show drawing vector

If you check this option, DENEBACAD displays the drawing vector when you draw, move, extrude, rotate, place, or otherwise manipulate the position of an object.

The drawing vector is a hollow arrow that indicates direction and distance when you draw or perform operations on objects.

As you move or drag the mouse, the drawing vector follows the pointer. Its exact position depends on the operation you perform. If you draw with the Circle Radius tool, for example, the drawing vector extends from the center of the circle to its perimeter as you draw. If the Snap to Grid command is active, the pointer and the tip of the drawing vector snap to points on the grid. If the Snap to Grid command isn't active, the drawing vector and pointer do not snap to the grid.

Show boolean outlines

If you check this option, DENEBACAD displays a red outline around a combined object when the object is in reshape mode. Combined objects are groups of individual objects that have been merged with a command in the Combine submenu in the Object menu. The red outline helps you remember the combined shape as you reshape individual objects using the Reshape command.

When "Show Boolean outlines" is not checked, the red outline does not appear when combined objects are in reshape mode.

Automatic Fit to Window

If you check this option, DENEBACAD calculates the magnification of a new Draft or Sculpt window so that all objects appear and are centered within the newly opened window.

If "Automatic Fit to Window" is not checked, objects in the new Draft or Sculpt window appear at the same magnification as the active window. This means the objects might not be visible when you open a new window.

Auto-tile windows

If you check this option, DENEBACAD arranges windows on screen so they do not overlap when you open a new Draft, Sculpt, and Render window using the "Show" commands in the Window menu. When you open a Draft, Sculpt, and Render window, this option places the Draft window on the left half of the screen, with the Sculpt window above the Render window on the right half. When you open two windows, each occupies half of the screen.

If you close a window, DENEBACAD does not resize or arrange the remaining windows.

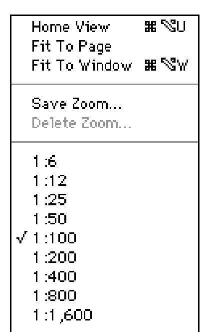
If "Auto-tile windows" is not checked, each new window you open fills the screen.

Zoom by Scale

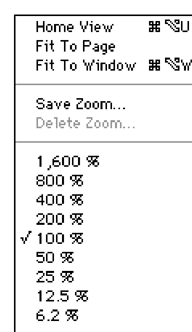
If you check the Zoom by Scale option, the magnification levels in the zoom pop-up menu appear as ratios based on the current drawing scale. For example, if the drawing scale is 1/4" = 1' (one-quarter inch equals one foot), the 100% magnification level appears as the ratio 1:48, and the 50% magnification level appears as the ratio 1:96.

You can set the drawing scale with the Output Setup command in the Layout menu.

If the Zoom by Scale option is not selected, the magnification levels in the Zoom pop-up menu appear as percentages, with 100% indicating normal view magnification.



Zoom by Scale



Percentage zoom levels

Fig. 201 The Zoom pop-up menu

Panned Zoom

If you check the Panned Zoom option, you can select the view area in a drawing when you increase the zoom level. Selecting this preference makes the Zoom pop-up menu work more like the Zoom tool. With the Zoom tool, you always specify the view area when you increase magnification.

When the Panned Zoom option is checked and you choose an increased zoom level in the Zoom pop-up menu, DENEBACAD displays a gray rectangle in the drawing. The rectangle indicates the area that will be visible in the window at the new zoom level. Move the pointer to position the rectangle, and click to zoom in.

The size and proportions of the Panned Zoom rectangle matches the size and proportions of the drawing window.

When the Panned Zoom option is not checked, the zoom level changes as soon as you choose a zoom level in the Zoom pop-up menu. When you increase the zoom level, DENEBACAD zooms in to the current view in the drawing window.

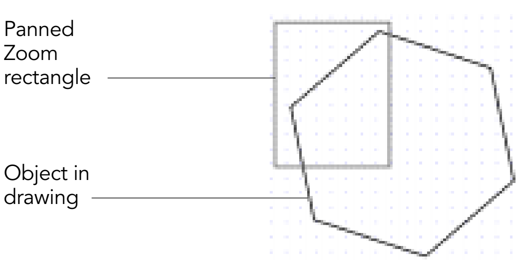


Fig. 202 Zooming with the Panned Zoom rectangle

Display HiRes Pictures

If you check this option, DENEBACAD displays Solid renderings pasted into Draft mode windows at the resolution specified in the Rendering Options dialog box.

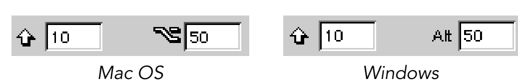
If you do not check “Display HiRes Pictures”, DENEBACAD displays Solid renderings at 72 ppi, regardless of the resolution specified in the Rendering Options dialog box. This option is for display purposes only. The “Display HiRes Pictures” option doesn’t affect the actual resolution of a solid rendering. If you don’t check “Display HiRes Pictures,” it has no effect on printing solid renderings, or saving drawings.

Arrow key settings

The text boxes at the bottom of the General Preferences tab let you specify the number of grid units a selected object moves when you use a combination of modifier and keyboard arrow keys.

In the box on the left, type the number of grid increments to move objects when you hold down the Shift key and press a keyboard arrow key.

In the box on the right, type the number of grid increments to move selected objects when you hold down the Option key (Mac OS), or the Alt key (Windows), and then press a keyboard arrow key.

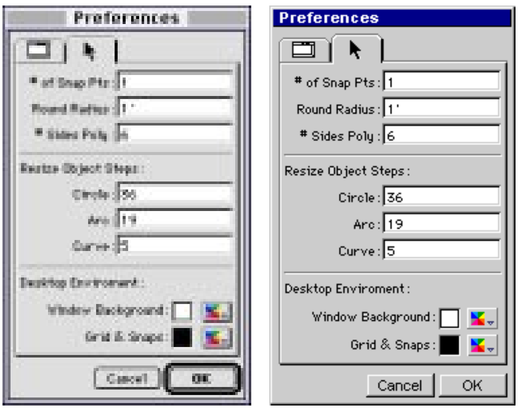


For example, with the settings shown above, pressing Shift+Left Arrow moves a selected object 10 grid units to the left, and pressing Option+Right Arrow moves a selected object 50 grid units to the right.

To specify units, use the Units Setup command in the Layout menu.

TOOLS TAB

A solid arrow appears at the top of the Tools tab in the Preferences dialog box. The options on the tab let you specify settings for objects you create with drawing tools. You can also use Desktop Environment options to change the colors of the window background, drawing grid, and snap objects.



of Snap Pts.

The "# of Snap Pts." preference sets the number of snap points that appear on a line segment, arc, or curve. Type the number of points in the text box.

DENEBACAD displays snap points on selected objects when you point to an object and choose Snap Points in the Snaps pop-up menu or press the Spacebar.

- For squares and rectangles, snap points appear evenly spaced between the corner points.
- For polygons, snap points appear evenly spaced between each vertex.
- For curved polylines, snap points appear evenly spaced between the points through which the curve is defined.
- For arcs, snap points appear evenly spaced between the end points. DENEBACAD treats circles as four arcs.

Round Radius

The Round Radius preference controls the radius of the corners of rounded rectangles. To change the value, type the radius measurement in the text box.

You can change the Round Radius amount at any time. Changing this value does not affect existing objects, but does affect the next rounded rectangle that you draw.

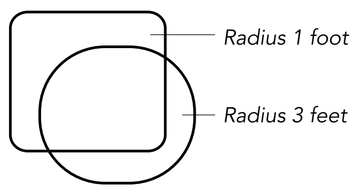


Fig. 203 Effect of Round Radius settings

Sides Poly

The "#Sides Poly" preference specifies how many sides the Polygon Vertex and Polygon Midpoint tools create when you use these tools to draw polygons.

You can change this preference by double-clicking the Polygon Vertex or Polygon Midpoint tool icons in the toolbox. In the dialog box that appears, type a number in the "# of sides" text box and click OK to change the preference.

Resize Object Steps

Curved objects become polylines or polygons when you perform certain operations on them.

An arc, circle, or curve becomes polylines or polygons when you do the following:

- Apply the Convert to Polygon command to a 2D object. The resulting object is a polygon made of multiple segments.
- Apply the Unchain command to a 2D object. The result is a series of separate polyline objects.
- Extrude a 2D object in Draft mode or draw a 3D curved object in Sculpt mode. The resulting object is made of a series of linked polygons.

The "Resize Object Steps" preference controls the number of steps in these converted or extruded objects. "Steps" refers to separate polygons in 3D objects, and separate polylines or polygon segments in 2D objects.

For example, if you unchain a circle, the Circle Step preference determines the number of segments in the resulting polygon. When you draw a 3D cylinder with a circle tool, the Circle Step preference determines the number of facets or faces of the 3D object. Higher Circle Step values result in smoother and more complex 3D objects.

Circle Step sets the number of steps in an extruded or polygonized circle.

Arc Step sets the number of steps in an extruded or polygonized arc.

Curve Step sets the number of steps in an extruded or polygonized curve.

DESKTOP ENVIRONMENT

The Desktop Environment preferences on the Tools tab let you specify background and grid colors for drawing windows.

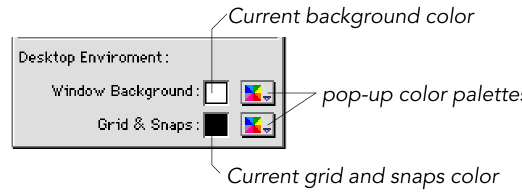


Fig. 204 Desktop Environment options

Window Background

The Window Background preference lets you pick a background color for Draft and Sculpt mode windows. Render mode windows do not use the Window Background color. You choose the Render window background in the Rendering Options dialog box.

The current background color appears in the box. You can choose a preset or custom color from the pop-up color palette.

To change the background color, press the pop-up color palette and drag to the color in the grid that you want to use. To select a custom color, drag to the "Other..." area at the bottom of the grid.

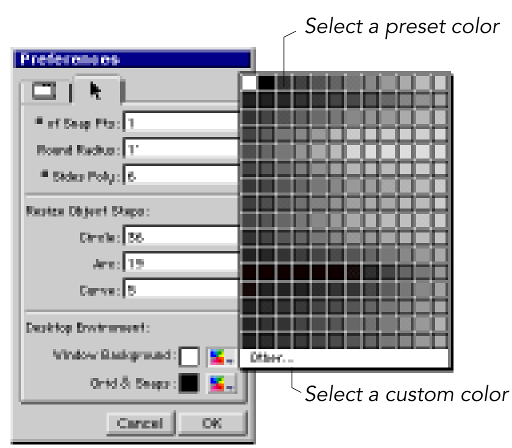


Fig. 205 Options in the pop-up color palette

Choosing a Custom color

When you select “Other...” in the pop-up color palette, the Mac OS Color Picker appears. This dialog box lets you specify custom colors from a color wheel or by entering color values numerically.

To specify a custom color, click in the color wheel, or enter color values in the Hue, Saturation, Brightness or Red, Green, Blue text boxes. Click OK to accept the current color.

In some cases, a different dialog box appears when you select “Other...” This happens if you use a system extension that replaces the standard Mac OS Color Picker. Some color management utilities do this. For information on other color pickers, refer to your system documentation or to the documentation for your color management software.

Grid & Snaps

The Grid & Snaps preference lets you pick a color for the drawing grid in Draft and Sculpt mode windows. Render mode windows do not use the Grid & Snaps color because the grid doesn’t appear in Render mode.

The current grid color appears in the color box. You can choose a preset or custom color from the pop-up color palette.

To change the Grid & Snaps color, press the pop-up color palette and drag to the color in the grid that you want to use. To select a custom color, drag to the “Other...” area at the bottom of the grid to open the Color Picker dialog box.

RECALCULATE

Mode: Draft
Mac OS Shortcut: Cmd+Option+K
Windows Shortcut: Ctrl+Alt+K

The Recalculate command updates a rendering, created with the Solid command, that has been pasted into a Draft mode window. You can use the Recalculate command to make the rendered image match the current attributes of the objects, after you change the attributes of any object that appears in the rendering.

Recalculate is available when a pasted rendering is selected in the Draft mode window.

Using renderings in Draft mode

You can copy Solid renderings from the Render mode window, and then use the Paste command to insert the renderings into a Draft mode window. You can then change the color or Surface materials of objects, and use the Recalculate command to re-render only the objects you’ve changed. In this way, you can avoid re-rendering all 3D objects in the document when you just want to experiment with different colors or Surface materials.

Note: To recalculate a rendering, it must be in the same DENEBACAD document as the 3D model on which it is based.

Using the Recalculate command makes it easy to change some aspects of a rendering without having to set camera angles and re-render all 3D objects.

For example, you can paste into a Draft mode window a rendering of a room that contains many rendered elements. Then, change the color of the carpet, and use the Recalculate command to assess the effect of the change without having to re-render every object in the entire room.

When you choose Recalculate, new fill colors and Surface materials appear in the Draft mode copy of a rendering only. Recalculate does not change the current rendering in the Render mode window.

If you do want to see new fill attributes in Render mode, you must use the Solid command to create a new rendering with colors and Surface materials in Render mode.

Before you can use the Recalculate command, you must copy a rendering and paste it into a Draft mode window. You must also make a modification to an object in Sculpt mode.

To paste a rendering into Draft mode

1. Create a rendering in the Render mode window using the Solid command. Choose Edit > Copy.
2. Open or activate a Draft window. Choose Edit > Paste. Or, press Ctrl+V (Windows), or Cmd+V (Mac OS). A copy appears in the center of the window.

Note: You can choose Edit > Copy View to Draft > Solid to copy a rendering to a Draft window.

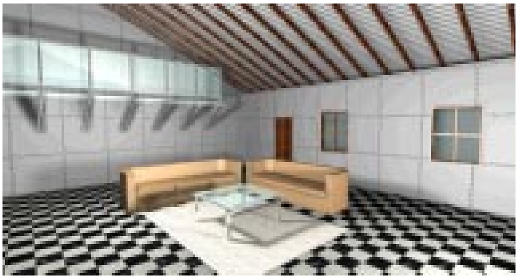
To change colors or Surface materials

1. In a Sculpt window, select the objects to apply colors or surface material.
2. Select a Pen color in the Pen palette, or, select a Surface material in the Fill palette.

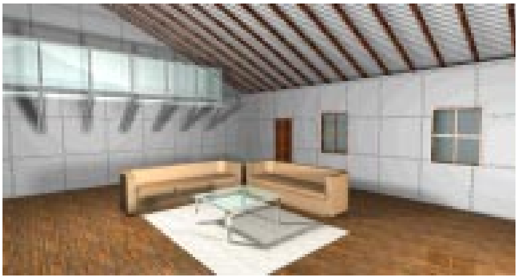
Note: An object's Pen color appears as its "fill" color in Solid renderings, if the object does not have a surface material.

To use the Recalculate command

1. If necessary, follow the above procedures to paste a rendering into Draft mode and change the color or Surface material of 3D objects in Sculpt mode.
2. Select the rendering in Draft mode.
3. Choose Edit > Recalculate. Or, press Ctrl+Alt+K (Windows) or Cmd+Option+K (Mac OS). DENEBACAD recalculates the rendering in the active Draft mode window to match the current Pen color or Surface material of the objects in the rendering.



A Solid rendering pasted into Draft mode



A wood surface material was applied to the floor in Sculpt mode, then Recalculate updated the rendering

REDO

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Z
Windows Shortcut: Ctrl+Z

The Redo command cancels the effect of the Undo command. Redo is available in the Edit menu after you apply the Undo command.

Once you choose Undo, Redo replaces the Undo command in the Edit menu until you perform another editing action.

◆ **To use the Redo command:** To reinstate a canceled action, choose Edit > Redo. Or, press Ctrl+Z (Windows) or Cmd+Z (Mac OS) immediately after choosing Undo.

Note: If you can redo a command, *Redo command name* appears in the Edit menu, where *command name* is the name of the command you can reinstate.

RESHAPE

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+R
Windows Shortcut: Ctrl+R

The Reshape command puts selected objects in reshape mode. In this mode, an object's creation points appear as handles that you can drag to reshape the object.

The Reshape command is available when an object that can be reshaped is selected in the active Draft or Sculpt window. The Reshape action button duplicates the function of the Reshape command.

Fig. 206 Reshape action button

You can reshape selected objects at any stage in the design process.

In reshape mode, you can reshape an object by editing its creation points. On screen, an object's creation points appear as tiny black squares. In general, an object in Reshape mode behaves like a selected object; if you press Delete, for example (or choose Clear), the object disappears.

The locations of creation points depend on how the object was created. For example, the creation points of a rectangle drawn with the Rectangle Diagonal tool in Draft mode, are the points you set to create the rectangle. By contrast, the creation points of a cube drawn with the Rectangle Diagonal tool in Sculpt mode, appear at each corner of the object.

For Boolean objects, creation points are at the creation points of each of the constituent objects. (A Boolean object is two or more objects combined using a command in the Combine submenu.)

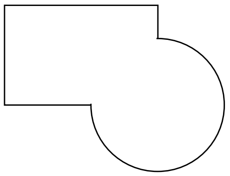
As you reshape a Boolean object, you can see the Boolean object's original outline if the preference "Show Boolean outlines" is active. This is one of DENEBACAD's most useful features.

After you finish reshaping a Boolean object, DENEBACAD reapplies the original Boolean operation. If you're working with "Show Boolean outlines" off, you can still use the Reshape command on Boolean objects, but you won't be able to see the Boolean object's original outline as you work. In general, to make reshaping Boolean objects easier, you should leave "Show Boolean outlines" selected.

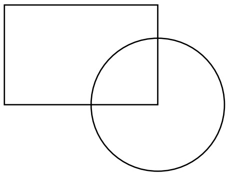
To use the Reshape command

1. Select one or more objects to reshape.
2. Choose Edit > Reshape. Or, click the Reshape action button. The creation points of the selected objects appear.

   ▲ If you selected a Boolean object, creation points appear on each object in the shape.
3. To reshape the object, drag its creation points.
4. To exit Reshape mode, click an empty area of the document. The creation points of the selected objects disappear and the objects are no longer selected.



Original Boolean object



The original objects are outlined in reshape mode when "Show Boolean outlines" is selected

Fig. 207 Effect of Show Boolean Outlines preference

SELECT

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+A
Windows Shortcut: Ctrl+Shift+A

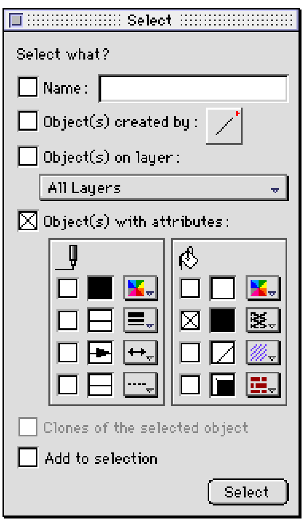
The Select command lets you select objects in a document based on their attributes, the tools used to create the objects, and other properties.

The Select command is available whenever a Draft or Sculpt mode window is active.

By using the Select command, you can select only the objects with the properties you specify. When you choose this command, DENEBACAD displays the Select palette. You can then specify the properties of the objects you want to select.

To use the Select command

1. Choose Edit > Select. The Select palette appears.
2. Configure the selection options in the palette. You must check the options for the properties you want DENEBACAD to use for selecting objects with the corresponding properties. These options are described later in this section.
3. To select the objects, click Select. DENEBACAD selects the objects with the specified properties.



SELECTION METHODS

In the Select palette, you can select objects by the following properties:

- name
- the tool used to create the objects
- layers
- Pen and Fill attributes

You can also select clones of selected objects, or add to a current selection.

You can use a single selection property or multiple properties. When you use more than one property, DENEBACAD selects objects that match all of the properties, and not objects that meet *any* of the properties. This type of selection can be described as a logical "And" selection.

For example, if you choose the "Objects created by" option and the Line tool icon, and also check the Pen color option and choose a pen color, DENEBACAD selects objects drawn with the Line tool and that also match the color you specified.

On the other hand, if you want to perform a logical "Or" selection, you can do this in the Select palette by specifying one property at a time, clicking Select, and then using the "Add to selection" option to select a series of objects.

Select palette options

Name If this option is checked, DENEBACAD selects all objects with the name you type in the Name text box.

Object(s) created by If this option is checked, DENEBACAD selects all objects created by the tool you choose in the adjacent pop-up menu.

Object(s) on layer If this option is checked, DENEBACAD selects all objects on the specified layer. Choose the layer from the pop-up menu.

Object(s) with attributes If this option is checked, DENEBACAD selects all objects with the Pen and Fill attributes that you specify. Use the individual options to select the attributes you want to include in the selection criteria. The first column contains Pen attributes, and the second column contains Fill attributes.

For example, to include arrowheads in the selection criteria, check the arrowhead checkbox and choose the arrowhead example from the pop-up palette.

Clones of the selected object If this option is checked, DENEBACAD selects all copies of the selected object, regardless of how the copies were added to the document. This option is available only when an object is selected.

Add to selection When this option is checked and you click the Select button, DENEBACAD leaves selected any objects that are currently selected, and then adds to the selection any objects that meet the current selection criteria in the Select palette.

SELECT ALL

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+A
Windows Shortcut: Ctrl+A

The Select All command selects all visible, unlocked objects in the current Draft or Sculpt mode window.

Note: You can't select objects in Render mode.

- You can select 3D objects in Draft mode and 2D objects in Sculpt mode by choosing the correct option in the View Options submenu, or the View Options pop-up menu.

◆ **To use the Select All command:** Choose Edit > Select All. Or, press Ctrl+A (Windows) or Cmd+A (Mac OS).

SHOW CLIPBOARD / HIDE CLIPBOARD (MAC OS)

Modes: Draft, Sculpt, Render

The Show Clipboard command displays the contents on the system Clipboard in a floating window. The Hide Clipboard command removes the Clipboard window from the screen.

When you choose the Show Clipboard command, a window displays the current Clipboard contents so you can see what's stored on the Clipboard without interrupting your work.

You can ensure that you're pasting the correct selection by displaying the Clipboard before choosing the Paste command. Also, before choosing the Cut or Copy commands, you can display the Clipboard to avoid accidentally replacing a selection that you want to keep.

◆ **To use the Show Clipboard command:** To display the system Clipboard, choose Edit > Show Clipboard.

If the Clipboard window is open but not active, you can choose Show Clipboard to bring the Clipboard window to the front and make it active.

When the Clipboard window is the active window, the Show Clipboard command is not available, and the Hide Clipboard command appears in the Edit menu. You can choose this command to hide the system Clipboard.

◆ **To use the Hide Clipboard command:** To hide the Clipboard window, choose Edit > Hide Clipboard.

UNDO

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Z
Windows Shortcut: Ctrl+Z

The Undo command cancels the most recent editing action.

To undo an action, you must choose Undo immediately after you perform the action. You can cancel only the most recent action with the Undo command.

Almost any editing action or command can be canceled with the Undo command. However, you usually cannot cancel any action that doesn't change objects in the document. The following are some actions that can't be canceled with the Undo command:

- saving a file
- changing the view magnification
- opening a palette
- adjusting the settings in a dialog box
- opening a file
- importing a file

If an action cannot be undone, *Can't Undo* appears in the Edit menu in place of the Undo command.

◆ **To use the Undo command:** To cancel your last editing action, choose Edit > Undo.

Note: When a command can be canceled, the Undo command appears in the menu as *Undo command name*, where *command name* is the name of the command just performed.

Restoring a canceled action with Redo

After you choose Undo, Redo replaces Undo in the Edit menu until you perform another editing action. The Redo command lets you restore an action canceled with Undo.

◆ **To use the Redo command:** To reinstate a canceled action, choose Edit > Redo immediately after choosing Undo.

Note: When you can redo a command, "Redo *command name*" appears in the Edit menu, where *command name* is the name of the command you can reinstate.

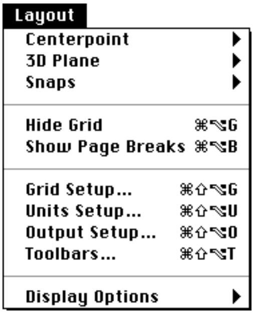
LAYOUT MENU COMMANDS 15

Layout menu commands let you set up a document's drawing scale, measurement units, toolbars, and output settings. You can also define, save, and select extrusion planes and extrusion axes in Draft and Sculpt modes.

The commands used to set up a document include Grid Setup, Units Setup, Output Setup, and Toolbars.

The submenus in the Layout menu are Centerpoint, 3D Plane, 3D Axis, Snaps, and Display Options.

Note: The 3D Plane and 3D Axis submenus replace each other in the Layout menu depending on the current extrusion method.



Mac OS

Windows

CENTERPOINT SUBMENU

Modes: Draft, Sculpt

A Centerpoint is a reference point in a DENEBACAD document. The Centerpoint submenu contains commands to set, activate, save, and delete Centerpoints. The submenu also contains the names of saved custom Centerpoints.

This section describes the following Centerpoint submenu commands:

Custom Centerpoint Appears in the Layout > Centerpoint submenu if a custom Centerpoint is active.

Default Centerpoint Activates the default Centerpoint, which is the Absolute Origin of the DENEBACAD workspace.

Delete Centerpoint Lets you delete custom Centerpoints that you have saved.

Edit Centerpoint Lets you edit the coordinates of the active custom Centerpoint.

Save Centerpoint Saves the active custom Centerpoint so you can select it again.

Set Position Lets you set a custom Centerpoint at any location in the document.

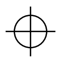


Fig. 208 Centerpoint

Working with Centerpoints

The Centerpoint is a point of reference in a DENEBACAD document. Usually, the Centerpoint is the zero point from which coordinates and distances are calculated. The Centerpoint appears on screen as a red circled cross (*figure 208*).

In a new DENEBACAD document, the Centerpoint is the Absolute Origin of the document, at the Cartesian coordinates X=0, Y=0, Z=0 (in 3D space). This Centerpoint is the default Centerpoint, which you can return to by choosing the Default Centerpoint command.

You can set new Centerpoints in a document so you can refer to other points as “virtual” origins. You can also assign other coordinates to a custom Centerpoint, so that you can refer to these coordinates without working at the actual coordinate location in the document.

When you activate a Centerpoint, the coordinates of objects in the document can change, because the positions of objects are defined relative to the active Centerpoint.

One Centerpoint is always active in a document. You can activate a different Centerpoint by choosing Layout > Centerpoint > Default Centerpoint, or the name of a saved Centerpoint. You can set a new Centerpoint by choosing Layout > Centerpoint > Set Position.

When you set a Centerpoint, and then save it, the Centerpoint’s name appears in the Layout > Centerpoint submenu in every view in Draft and Sculpt modes.

Note: After you choose a Centerpoint, you can’t choose Undo to return to the previous Centerpoint.

CUSTOM CENTERPOINT

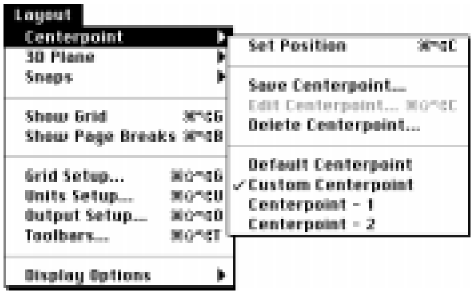
Modes: Draft, Sculpt

When “Custom Centerpoint” appears in the Layout > Centerpoint submenu, it means that a custom Centerpoint, which you set using the Set Position command, is active.

“Custom Centerpoint” appears until you activate another Centerpoint by choosing the Default Centerpoint command, or the name of a saved Centerpoint.

If the custom Centerpoint is active, and you save it, the name you specify appears at the bottom of the Centerpoint submenu, a checkmark indicates that this is the active Centerpoint, and “Custom Centerpoint” no longer appears.

Be sure to save a custom Centerpoint if you want to activate it again. For more information, see “Save Centerpoint” on page 240.



Custom Centerpoint appears if you set a Centerpoint

DEFAULT CENTERPOINT

The Default Centerpoint command activates the default Centerpoint.

When you create a DENEBACAD document, a check mark next to “Default Centerpoint” in the Layout > Centerpoint submenu indicates that the default Centerpoint is active.

The default Centerpoint is the Absolute Origin for the DENEBACAD workspace, at X=0, Y=0, Z=0. The coordinates of the default Centerpoint can’t be changed.

◆ **To activate the default Centerpoint:**

Choose Layout > Centerpoint > Default Centerpoint. DENEBACAD activates the default Centerpoint and centers it in the active window.

Note: After you choose a Centerpoint, you can’t choose Undo to return to the previous Centerpoint.

DELETE CENTERPOINT

Modes: Draft, Sculpt

The Delete Centerpoint command lets you delete Centerpoints that you have saved.

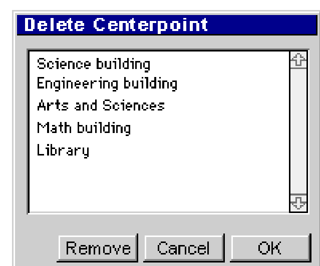
To delete a Centerpoint

1. Choose Layout > Centerpoint > Delete Centerpoint. The Delete Centerpoint dialog box appears.

2. Select the names of Centerpoints that you want to delete from the document.

   ▲ To select (or deselect) a contiguous range of Centerpoints, press Shift and click the first and last Centerpoints’ names.

   ▲ To select (or deselect) several non-contiguous Centerpoints, press Ctrl (Windows) or Cmd (Mac OS) and click each Centerpoint’s name.



3. To remove the selected Centerpoints, click Remove.

4. To permanently delete the removed Centerpoints, click OK.

EDIT CENTERPOINT

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+Shift+C
Windows Shortcut: Ctrl+Alt+Shift+C

The Edit Centerpoint command lets you edit the Cartesian coordinates of a saved Centerpoint. This command is not available until you save a Centerpoint.

In addition, the Edit Centerpoint command can be used to determine a Centerpoint's true coordinate location.

Using the Edit Centerpoint command

Editing a Centerpoint lets you use a small drawing area, but also refer to a "virtual" Centerpoint that simulates a larger drawing area. With this technique, you don't need to separate structures or elements of a project by their true distances apart just to ensure accurate coordinates across the document.

For example, if you want to draw two structures separated by 1 mile in the same document, you can put the first structure at the default Centerpoint (the Absolute Origin). Then, set a custom Centerpoint nearby, and use the Edit Centerpoint command to assign "virtual" coordinates to the custom Centerpoint. You can draw the second structure, using the edited Centerpoint as the reference point, without having to work at the true coordinate location of the second structure.

To edit a Centerpoint

1. Choose Layout > Centerpoint > the saved Centerpoint to edit. This activates the saved Centerpoint, which DENEBACAD centers in the window.

2. Choose Layout > Centerpoint > Edit Centerpoint. The Edit Centerpoint dialog box appears. You can change the Centerpoint's name or coordinates by typing in the text boxes.

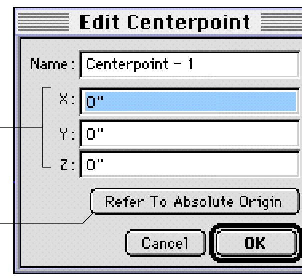
   ▲ The first time you choose the Edit Centerpoint command for a custom Centerpoint, the coordinates are X=0, Y=0, Z=0. Changing these values doesn't move the Centerpoint relative to objects in the document or to the Absolute Origin. However, the Centerpoint's new coordinates are the reference point for object coordinates and measurements when the Centerpoint is active.

   ▲ To display the Centerpoint's true coordinates relative to the Absolute Origin, click "Refer to Absolute Origin." The coordinates appear in the X, Y, and Z text boxes.

3. Click OK to implement the settings

Type new values to change the coordinates

Click to display coordinates from the Absolute Origin



SAVE CENTERPOINT

Modes: Draft, Sculpt

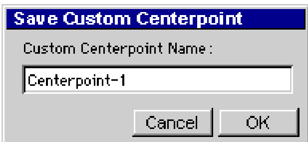
After you set a custom Centerpoint using the Set Position command, you can save the Centerpoint. The names of saved Centerpoints appear at the bottom of the Centerpoint submenu. Saved Centerpoints are available in Draft and Sculpt mode and in all views.

If a saved Centerpoint is active, a check mark appears next to its name in the Layout > Centerpoint submenu.

Using saved Centerpoints let you quickly display various parts of a plan by activating the saved Centerpoints using the Centerpoint submenu.

To save a Centerpoint

1. Use the Set Position command to set a Centerpoint.
2. Choose Layout > Centerpoint > Save Centerpoint. The Save Custom Centerpoint dialog box appears.



3. To change the default name, type a different name in the text box.
4. Click OK to save the Centerpoint. The Centerpoint's name appears at the bottom of the Centerpoint submenu.

SET POSITION

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+C
Windows Shortcut: Ctrl+Alt+C

The Set Position command lets you set Centerpoints in DENEBACAD documents.

Every document has a default Centerpoint, which is the document's Absolute Origin, at X=0, Y=0, and Z=0. A Centerpoint that you set with the Set Position command is a virtual origin, which initially has the coordinates X=0, Y=0, Z=0.

A custom Centerpoint's virtual coordinates can also be changed, by using the Edit Centerpoint command; see "Edit Centerpoint" on page 239 for more information.

While you can set more than one Centerpoint in a document, only one is active at a time. The active Centerpoint has the coordinates X=0, Y=0, Z=0 (unless you change them). This makes the active Centerpoint the origin of a user-defined coordinate system.

When you create a custom origin, you don't lose the Absolute Origin. To see a Centerpoint's true coordinates, choose the Centerpoint, and then choose the Edit Centerpoint command. Click the button "Refer to Absolute Origin" to see the actual coordinates of the Centerpoint.

The Set Position command has many practical uses. For example, rather than moving an entire drawing to the default Centerpoint, you can create a Centerpoint to accommodate the drawing. Also, you can create several Centerpoints for different areas of a large plan. This makes it easy to switch to these areas by activating saved Centerpoints.

To set a Centerpoint

1. With a Draft mode or Sculpt mode window active, choose Layout > Centerpoint > Set Position.
2. The pointer becomes a cross followed by a shadow Centerpoint. Click to place the Centerpoint. (If Snap to Grid, Snap to Objects, or Snap Objects are active, the pointer snaps to the nearest grid, object, or snap point.)

When you set a Centerpoint, it becomes the document's origin, and is centered in the document window.

DISPLAY OPTIONS SUBMENU

The Display Options submenu contains commands that let you visually organize a document based on display criteria. These actions are for display purposes only and are independent of any attributes you've applied to objects in a document.

You can organize objects based on Rapidograph color-coding information. You can also distinguish objects based on layers or dimensions. You can display selected objects only, visible layers only, or layers in their creation order.

To avoid confusion, keep these issues in mind when working with the Display Options submenu:

- These commands are active or inactive. If a command is active, a check mark appears next to the command in the Display Options submenu.
- Activating one of these commands overrides other attributes assigned to objects, such as Pen or Fill colors. When you deactivate the command, assigned attributes reappear.
- These commands affect the active document window only.
- Not every command in the Display Options submenu can be active at the same time. This section explains which commands can be active simultaneously.
- Because the Display Options submenu commands affect the visibility of objects, they can affect the appearance of renderings.

The Display Options submenu contains the following commands:

Display Visible Layers Displays objects on visible layers.

Display Active Layer Displays objects on the active layer only.

Display Selected Objects Displays selected objects only.

- If you activate one of these commands, DENEBACAD deactivates the other two.
- The Display Options pop-up menu on the Status bar shows the active display command.

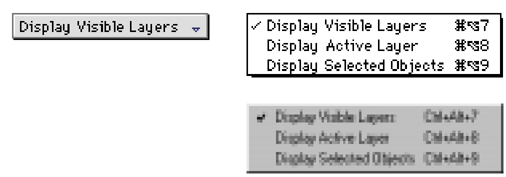


Fig. 209 The Display Options pop-up menu

Display Layer Colors Displays objects based on layer colors in the Layer Manager.

Display Pen Weight Colors Displays 2D objects in colors based on the Rapidograph pen color standard.

Display 2D & 3D Mode Colors Lets you override the red-blue display when you activate a 2D and 3D design environment.

- If you activate one of the above three commands, DENEBACAD deactivates the other two.

Display Locked Objects in Gray Displays all locked objects with gray Pen color.

Display by Layer Creation Order Displays objects based on the absolute order in which the layers were created in the Layer Manager.

Note: The Display Options submenu commands are described in this section in the order they appear in the submenu, not alphabetically.

DISPLAY VISIBLE LAYERS

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+Opt+7
Windows Shortcut: Ctrl+Alt+7

The Display Visible Layers command displays or hides objects on visible layers in the active document window. To change which layers are visible, use the Layer Manager command.

To show objects on all visible layers, choose Display Visible Layers in the Display Options pop-up menu on the Status bar or the Display Options submenu. Objects on visible layers become visible. This deactivates Display Active Layer and Display Selected Objects, if either command was active.

DISPLAY ACTIVE LAYER

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+Opt+8
Windows Shortcut: Ctrl+Alt+8

Display Active Layer displays objects on the active layer only, in the active document window.

To show objects on the active layer only, choose Display Active Layer in the Display Options pop-up menu on the Status bar, or the Display Options submenu.

Choosing this command deactivates Display Visible Layers and Display Selected Objects, if either command was active.

DISPLAY SELECTED OBJECTS

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+Opt+9
Windows Shortcut: Ctrl+Alt+9

This command displays only the objects that are selected and are on visible layers, in the active document window.

To show selected objects on visible, unlocked layers only, choose Display Selected Objects in the Display Options pop-up menu on the Status bar, or the Display Options submenu. This deactivates Display Visible Layers and Display Active Layer, if either command was active.

DISPLAY LAYER COLORS

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+Opt+A
Windows Shortcut: Ctrl+Alt+A

The Display Layer Colors command displays all objects that are on visible layers using the layers' colors as the colors of objects.

For more information on assigning layer colors, refer to “Layer Manager” on page 363.

To display all visible objects in the colors assigned to their layers, choose Display Layer Colors in the Display Options submenu.

To hide layer colors when this command is active, you can choose Display Layer Colors again to deactivate the command, or choose Display Pen Weight Colors, or Display 2D & 3D Mode colors.

DISPLAY PEN WEIGHT COLORS

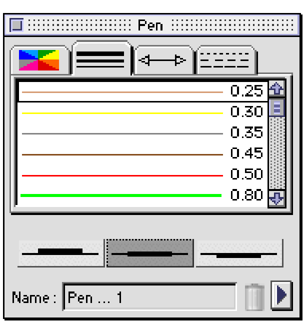
Mode: Draft
Mac OS Shortcut: Cmd+Opt+E
Windows Shortcut: Ctrl+Alt+E

The Display Pen Weight Colors command displays objects with the colors associated with the objects' Pen weights. This command does not affect 3D objects.

To use Display Pen Weight Colors, in Draft mode, choose Display Pen Weight Colors in the Display Options submenu. DENEBACAD shows the Rapidograph pen colors of 2D objects.

To hide the pen colors, deactivate Display Pen Weight Colors, or activate Display Layer Colors, or Display 2D & 3D Mode Colors.

DENEBACAD assigns Pen weight colors based on the Rapidograph pen color standards. Rapidograph pen weights are measured in millimeters; to see how colors and pen weights are related, you can open the Pen palette and choose Display Pen Weight Colors.



The Pen tab in the Pen palette displays Rapidograph pen colors if Display Pen Weight Colors is active

Fig. 210 Pen tab displaying Pen weight colors

The following table summarizes the Rapidograph pen colors.

Pen	Color
0.25 mm	Beige
0.30 mm	Yellow
0.35 mm	Gray
0.45 mm	Brown
0.50 mm	Red
0.80 mm	Green
1.20 mm	Orange
1.80 mm	Blue

DISPLAY 2D & 3D MODE COLORS

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+D
Windows Shortcut: Ctrl+Alt+D

Whenever you display 2D and 3D objects at the same time (by using the commands in the View Options submenu), DENEBACAD activates Display 2D & 3D Mode Colors. This causes 2D objects to appear in red, and 3D objects to appear in blue, to distinguish the objects types.

A message appears when Display 2D & 3D Mode Colors is active. This happens when 2D and 3D objects are displayed in the same window.

When the Display 2D & 3D Mode Colors command is active, the command has a checkmark in the Display Options submenu. However, while the command might be active, if you have not chosen to display both 2D and 3D objects, the command appears dimmed in the submenu to indicate that it is not available.

◆ **To show 2D & 3D Mode colors:** To show 2D objects in red and 3D objects in blue, choose Display 2D & 3D Mode Colors in the Display Options submenu (if the command isn't already active). This deactivates Display Pen Weight Colors or Display Layer Colors, if either was active.

◆ **To hide 2D & 3D mode colors:** Choose Display 2D & 3D Mode Colors in the Display Options submenu to deactivate the command, so that 2D and 3D objects appear with their assigned colors.

DISPLAY LOCKED OBJECTS IN GRAY

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+Opt+H
Windows Shortcut: Ctrl+Alt+H

The Display Locked Objects in Gray command displays objects that are on locked layers with gray outlines. This command affects the active document window only.

You can lock layers using the Layer Manager. You can also lock objects using the View Options submenu, by choosing "View 2D & Lock 3D Objects" or "View 3D & Lock 2D Objects." When you lock 2D or 3D objects this way, DENEBACAD activates the Display Locked Objects in Gray command.

◆ **To display locked objects in Gray:** Choose Display Locked Objects in Gray in the Display Options submenu. All objects on locked layers then appear with gray outlines.

DISPLAY BY LAYER CREATION ORDER

Modes: Draft, Sculpt, Render

The Display by Layer Creation Order command shows objects on visible, unlocked layers based on the original order in which the layers were created in the Layer Manager.

This command is in effect by default. Choosing the command turns the option on or off.

◆ **To display objects by layer creation order:** Choose Display by Layer Creation Order in the Display Options submenu. When you activate this command, DENEBACAD displays objects based on the original layer creation order, regardless of how you've reordered layers in the Layer Manager.

GRID SETUP

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+Shift+G
Windows Shortcut: F8

The Grid Setup command lets you configure the DENEBACAD grid. You can also do this using the Snap Grid action button.

The grid spacing reflects the current units and measurement system. You can specify the current units and measurement system in the Units Setup dialog box.

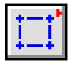


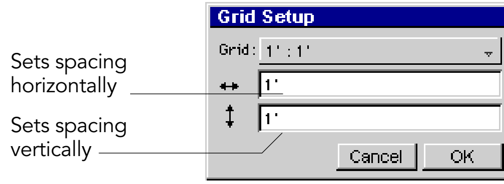
Fig. 211   The Snap Grid button

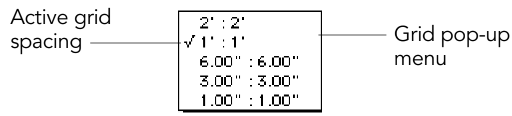
Use the text boxes in the Grid Setup dialog box to set up the spacing of the dots in the grid. For example, you can make the horizontal spacing of the grid 10 units horizontally, and 15 units vertically.

The drawing vector and the pointer snap to the grid if Snap to Grid is active.

To use the Grid Setup command

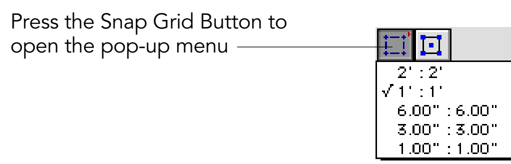
1. Choose Layout > Grid Setup. The Grid Setup dialog box appears.
2. In the Grid pop-up menu, choose the grid setting that you want to change. The choices in the Grid pop-up menu vary, depending on the current units and measurement system.
3. To specify custom spacing for the grid, type spacing values in the text boxes.
4. Click OK to implement the grid setup.





To use the Snap Grid button

Press the Snap Grid button on the Attributes bar and choose the grid spacing you want in the pop-up menu. DENEBACAD resets the grid spacing



Note: If you click the Snap Grid button, you activate or deactivate the Snap to Grid command.

HIDE GRID / SHOW GRID

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+G
Windows Shortcut: F4

The Hide Grid command hides the grid when it is displayed in Draft Mode and Sculpt mode windows. The Show Grid command displays the grid in Draft and Sculpt windows.

Only one of these commands is available in the Layout menu at a time. When the Hide Grid command appears in the menu, the Show Grid command is active. When the Show Grid command appears, the Hide Grid command is active.

Note: The drawing vector can snap to the dots in the grid, even if the grid is hidden, if the Snap to Grid command is active.

If the grid scale or magnification is too low, DENEBACAD won't show the dots in the grid, even if Show Grid is in effect.

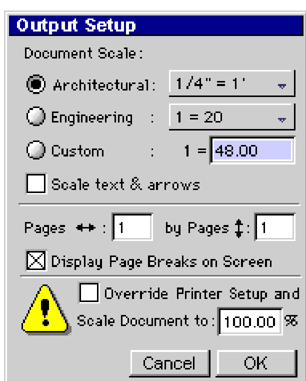
◆ **To use the Hide Grid command:** Choose Layout > Hide Grid. DENEBACAD hides the grid.

◆ **To use the Show Grid command:** Choose Layout > Show Grid. DENEBACAD displays the grid.

OUTPUT SETUP

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+Opt+Shift+O
Windows Shortcut: F11

The Output Setup command lets you set up a DENEBACAD project for final output. Although you can configure Output Setup options in Draft, Sculpt, and Render modes, you can print or plot in Draft mode only.



Many factors affect document output, such as your Printer Setup (Windows) or Page Setup (Mac OS) configuration, and the selected printer or plotter driver. When you output a document, DENEBACAD lets you permanently override the scaling factor in the Printer/Page Setup dialog.

To use the Output Setup dialog box

1. Choose Layout > Output Setup. The Output Setup dialog box appears.
2. Configure the options, which are explained in this section, and then click OK to make them take effect.

Document Scale area

In this area, you can select a preset Architectural output scale or an Engineering output scale from the pop-up menus, or specify a custom scale value.

Architectural Select this option to choose an architectural output scale in the pop-up menu.

Engineering Select this option to choose an engineering output scale in the pop-up menu.

Custom Select this option to use a custom output scale. Type the scale value in the “1=” text box.

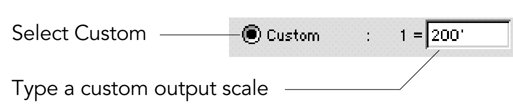


Fig. 212 Custom output option

Scale text & arrows Select this option to scale text and arrows (along with all other objects in the document) when you change the output scale of a document.

This option is primarily used when you import DWG or DXF files. DENEBACAD imports these files at 1:1 scale. If you want to output the document at any other scale, you must change the scale in the Document Scale area. If you do not change the output scale, the document's scale remains set at 1:1. A document with an output scale set at 1:1 might be very large because the objects will print at actual size.

Once you select a new output scale from the Document Scale area, you can select the "Scale text & arrows" option so that the text and arrows are output at the same scale as the rest of the document.

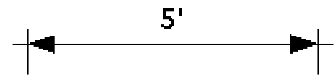
If you do not select "Scale text and arrows," but you change the output scale, all objects in the document will be output at the scale you choose, but text and arrows will be output at 1:1 scale.

Every time you open the Output Setup dialog box, the "Scale text & arrows" checkbox is not selected. You must select the option each time you change the output scale of a document to scale the text and arrows.

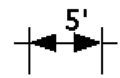
To see what happens to text and arrows in a DWG or DXF document when you change the output scale from 1:1, select the "Display Page Breaks on Screen" option in the Output Setup dialog box. With this option selected, you can see the change in document scale in relation to the printable area.

For example, select 3/4"=1' in the Architectural pop-up menu in the Document Scale area. Do not select the "Scale text & arrows" option. When you click OK, the dialog box closes. Objects in the document change scale, but text and arrows do not. The text and arrows become too large for the document, and they are no longer in the correct locations.

If you select 3/4"=1' in the Architectural pop-up menu, and select the "Scale text & arrows" option, you see that objects, text, and arrows change scale, and that text and arrows appear in their correct locations.



Output scale has been changed from 1:1 to 3/4"=1' with "Scale text & arrows" selected.



Output scale has been changed from 1:1 to 3/4"=1'. "Scale text & arrows" was not selected.

Fig. 213   Scaling text and arrows

Pages area

In the Pages Area, you can specify the number of horizontal and vertical pages to output. You can also choose to display or hide page breaks on screen

Pages   Type the number of horizontal pages you want to print from the document.

by Pages   Type the number of vertical pages you want to print from the document.

Display Page Breaks on Screen   Select this option to show page breaks on screen. Page breaks appear on screen as gray rectangles. If you don't see the current page break on screen after choosing this command and clicking OK, try reducing the view magnification.

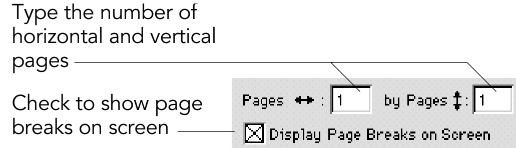


Fig. 214   Page options

Overriding Printer/Page Setup document scaling

In DENEBACAD, you can override the current scaling factor specified in the Printer/Page Setup dialog box. Overriding the Page Setup scaling can produce unexpected results when used improperly.

Fig. 215 Output scaling options

To override output scaling

1. To override the current scaling factor in the Printer Setup or Page Setup dialog box, click the “Override Printer/Page Setup” check box.

   ▲ *Important*: You must select this option to override the scale setting in the Printer/Page Setup dialog, although DENEBACAD lets you type override percentages in the “Scale Document to” box even without this option selected.

2. To scale a document to the output percentage you want, type the percentage in the “Scale Document to” text box.
3. To implement the setting, click OK.

SHOW PAGE BREAKS

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+B
Windows Shortcut: F12

The Show Page Breaks command displays or hides non-printing gray rectangles that represent page boundaries on screen.

Note: You can choose Show Page Breaks in Sculpt mode. However, DENEBACAD prints only in Draft mode, and prints the contents of the current Draft window only.

DENEBACAD determines the current page boundaries based on the paper size and the printable area. Page breaks appear in the active document window and the current view only.

The Show Page Breaks command is active or inactive. If the command is active, page breaks appear on screen, and a check mark appears next to the command in the Layout menu. Otherwise, page breaks do not appear and no check mark appears.

DENEBACAD displays as many page break borders as the number of pages you specify in the Output Setup dialog box. By using Show Page Breaks, you can see how DENEBACAD will divide the document into pages when you print it.

◆ **To make page breaks visible:** Choose Layout > Show Page Breaks when page breaks are not displayed in the Draft window.

◆ **To hide page breaks:** Choose Layout > Show Page Breaks when page breaks are displayed.

SNAPS SUBMENU

By using the commands in the Snaps submenu, you can restrict the movement of objects and the drawing vector in various ways.

You can snap the drawing vector to the grid, to the snap points of 2D and 3D objects, and to snap objects made with the Create Snap Objects command. In many cases, restricting the drawing vector's movement makes it easier to draw precisely.

The following Snaps submenu commands are described in this section:

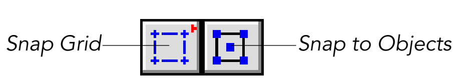
Create Snap Objects Lets you make custom grids by converting 2D objects to snap objects.

Snap to Grid Restricts the drawing vector and objects to the grid, even if the grid is not visible.

Snap to Objects Restricts the drawing vector and objects to the snap points of objects in a drawing, even if snap points are not displayed. All 2D and 3D objects contain snap points.

The Snap to Grid and Snap to Objects commands are active or inactive. When active, a check mark appears next to the command in the Snaps submenu. Otherwise, no check mark appears.

You can activate the Snap to Grid and Snap to Objects commands by clicking the Snap Grid button and the Snap to Objects button.



These commands are not mutually exclusive; You can activate them simultaneously. This lets you snap to the grid, to objects, to snap objects, to all, or to none.

CREATE SNAP OBJECTS

Modes: Draft

The Create Snap Objects command creates snap grids from selected objects. For example, to create a radial grid, draw several concentric circles and convert them to snap objects. You can move or delete the original objects, and the snap points remain visible on screen and act as a custom grid. Custom snap points don't print.

To use the Create Snap Objects command

1. Select the 2D object that you want to make into a snap object.
2. Choose Layout > Snaps > Create Snap Objects.
3. A message asks you to confirm your choice. Click OK to make the selected object a snap object.

Notice that the object's snap points are visible, even though the original object is deselected. You can move or delete the original object, but the snap points remain visible on screen.

Additionally, you can move, resize, and reshape the snap points grid as you would any 2D object.

To delete the snap points, select them and press Delete.

To set the number of snap points in objects

1. Choose Edit > Preferences. The Preferences dialog box opens. Click the Drawing tab to bring it to the front, if necessary.
2. In the "# of Snap Pts" text box, type the number of snap points for snap objects.
3. To implement the settings, click OK.

SNAP TO GRID

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+\
Windows Shortcut: Ctrl+\

The Snap to Grid command causes the drawing vector and objects to snap to the dots in the grid, even if the grid is hidden.

When Snap to Grid is not active, the drawing vector moves freely, even if the grid is visible.

When you create a new document, DENEBACAD activates Snap to Grid by default.

◆ **To activate Snap to Grid:** Choose Layout > Snaps > Snap to Grid. DENEBACAD restricts the movement of the drawing vector and objects to the dots in the grid.

▲ If you select Snap to Objects in addition to Snap to Grid, the drawing vector snaps to the snap points of objects.

◆ **To deactivate Snap to Grid:** Choose Layout > Snaps Snap to Grid when grid snap is active.

SNAP TO OBJECTS

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+\
Windows Shortcut: Ctrl+Alt+\

The Snap to Objects command causes the drawing vector and objects to snap to the snap points in 2D and 3D objects.

◆ **To activate Snap to Objects:** Choose Layout > Snaps > Snap to Objects.

▲ If you select Snap to Objects in addition to Snap to Grid, the drawing vector snaps to the snap points of objects.

◆ **To deactivate Snap to Objects:** Choose Layout > Snaps > Snap to Objects when object snap is active.

3D AXIS SUBMENU

The 3D Axis submenu lets you set, save, and delete spin extrusion axes. The following commands appear in the 3D Axis submenu:

Custom 3D Axis In the 3D Axis pop-up menu and the 3D Axis submenu, this indicates that a custom axis has been set up for spin extrusions, and has not been saved. See “Custom 3D Axis” on page 252 for more information.

Default 3D Axis Selects the default axis for spin extrusions. Refer to “Default 3D Axis” on page 252 for more information.

Define Lateral 3D Axis Lets you set an axis in Left view for spin extrusions. Refer to “Define Lateral 3D Axis” on page 254 for more information.

Define Frontal 3D Axis Lets you set an axis in Front view for spin extrusions. Refer to “Define Frontal 3D Axis” on page 253 for more information.

Define Vertical 3D Axis Lets you set an axis in Top view for spin extrusions. Refer to “Define Vertical 3D Axis” on page 254 for more information.

Delete 3D Axis Deletes a saved spin extrusion axis. Refer to “Delete 3D Axis” on page 255 for more information.

Save 3D Axis Saves an axis for spin extrusions. Refer to "Save 3D Axis" on page 256 for more information.

WORKING WITH SPIN EXTRUSION AXES

The 3D Axis submenu appears in the Layout menu when you select the Spin extrusion method.

A spin extrusion axis is an axis around which 2D objects extrude when you use the Spin extrusion method. A spin extrusion axis appears on screen as a horizontal or vertical gray line. DENEBACAD lets you set up to 255 spin extrusion axes for each view.

You can set a spin extrusion axis in all views in DENEBACAD, including Relative Views. You use a spin extrusion axis in the same view (Top, Bottom, Front, Back, Left, or Right) from which you choose Define Frontal 3D Axis, Define Vertical 3D Axis, or Define Lateral 3D Axis. For example, if you choose Define Frontal 3D Axis in Top view to set a spin extrusion axis in Front view, you use the spin extrusion axis in Top view.

Keep these issues in mind when working with the commands in the 3D Axis:

- It's best to display the Info bar and the Status bar before you begin working with spin extrusion axes. To do this, choose Layout > Toolbars.
- There's always a spin extrusion axis active in each view. The active spin extrusion axis can be the default spin extrusion axis, a spin extrusion axis that you set, or a spin extrusion axis you have saved.
- The first time you select the Spin extrusion method, the default spin extrusion axis appears. The default spin extrusion axis passes through the Absolute Origin (even if a Centerpoint that doesn't correspond with the Absolute Origin is active). For more information on Centerpoints, refer to "Centerpoint submenu" on page 237.
- The extrusion format buttons on the far right of the status bar (Front Cap, Back Cap, and Sides) are not available when you select the Spin extrusion method. These button do not affect spin extrusions.
- You can't spin extrude an object that overlaps, crosses, or intersects a spin extrusion axis.

CUSTOM 3D AXIS

Modes: Draft, Sculpt

"Custom 3D Axis" appears on the Status bar, and in the 3D Axis submenu with a check mark, when a custom 3D axis is active and has not been saved.

Until you set a custom 3D axis, "Custom 3D Axis" doesn't appear in the 3D Axis submenu.

When you save a custom 3D axis, the name of the saved 3D axis replaces "Custom 3D Axis" in the 3D Axis submenu and the Status bar.

If you select a saved axis or the default 3D axis when a custom axis is active, the custom axis is discarded, and "Custom 3D Axis" disappears from the 3D Axis submenu and the Status bar.

DEFAULT 3D AXIS

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+O
Windows Shortcut: Ctrl+Shift+O

The Default 3D Axis command activates the default spin extrusion axis in the current view.

When you select Spin extrusion method, the Default 3D Axis command is available in the 3D Axis pop-up menu on the Status bar and the 3D Axis submenu.

If you set a new spin extrusion axis, the default spin extrusion axis still passes through the Absolute Origin, and you can activate it by choosing the Default 3D Axis command.

The default spin extrusion axis passes through the Absolute Origin in Top, Bottom, Front, Back, Left, and Right views.

The default extrusion axis appears as a vertical gray line in Draft and Sculpt modes. The default extrusion axis appears the first time you select the Spin extrusion method. Thereafter, the last active 3D axis appears whenever you select Spin extrusion method in a particular view.

If the default spin extrusion axis is active, "Default 3D Axis" appears in the Status bar and, in the 3D Axis submenu, a check mark appears next to the Default 3D Axis command.

To activate the default 3D Axis

1. Choose the view in which you want to use the default spin extrusion axis.
2. Choose Layout > 3D Axis > Default 3D Axis, or use the pop-up menu on the Status bar.

SETTING A SPIN EXTRUSION AXIS

A spin extrusion axis is the axis around which 2D objects extrude when you use the Spin extrusion method.

DENEBACAD provides three commands for setting a spin extrusion axis: Define Frontal 3D Axis, Define Vertical 3D Axis, and Define Horizontal 3D Axis. Each command is explained in this section.

When you choose a Define 3D Axis command, you move the pointer to set the coordinates of the spin extrusion axis in Front, Left, or Top view. The coordinates you choose set the position of objects extruded into 3D space. Refer to "Using a Define 3D Axis command" on page 257 for an illustration of this procedure.

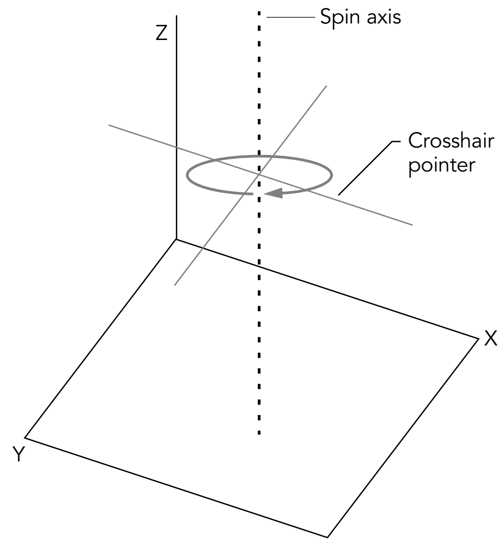


Fig. 216 A Spin extrusion axis

Define Frontal 3D Axis

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+F
Windows Shortcut: Ctrl+Alt+F

The Define Frontal 3D Axis command lets you set a spin extrusion axis in Front view.

The Define Frontal 3D Axis command is available in Top, Bottom, Left, or Right view.

If the document window is set to Top or Bottom view when you choose Define Frontal 3D Axis, you'll create a vertical spin extrusion axis.

If the document window is set to Left or Right view when you choose Define Frontal 3D Axis, you'll create a horizontal spin extrusion axis.

Using Define Frontal 3D Axis temporarily switches the document window to Front view so you can set a spin extrusion axis in Front view. After you set the spin extrusion axis, DENEBACAD switches the document window to the original view (Top, Bottom, Left, or Right), and activates the spin extrusion axis you set.

To use the Define Frontal 3D Axis command, refer to "To use a Define 3D Axis command" on page 254.

Define Lateral 3D Axis

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+L
Windows Shortcut: Ctrl+Alt+L

The Define Lateral 3D Axis command lets you set a spin extrusion axis in Left view.

The Define Lateral 3D Axis command is available in Top, Bottom, Front, or Back view.

Whether the document window is set to Top, Bottom, Front, or Back view when you choose Define Lateral 3D Axis, you'll create a horizontal spin extrusion axis.

Using Define Lateral 3D Axis temporarily switches the document window to Left view so you can set the spin extrusion axis in Left view. After you set the spin extrusion axis, DENEBACAD switches the document window to the original view (Top, Bottom, Front, or Back), and activates the spin extrusion axis you set.

To use the Define Lateral 3D Axis command, refer to "To use a Define 3D Axis command" on page 254.

Define Vertical 3D Axis

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+V
Windows Shortcut: Ctrl+Alt+V

The Define Vertical 3D Axis command lets you set a spin extrusion axis in Top view.

The Define Vertical 3D Axis command is available in Front, Back, Left, or Right view.

Whether the document window is set to Front, Back, Left, or Right view when you choose Define Vertical 3D Axis, you'll create a vertical spin extrusion axis.

Using Define Vertical 3D Axis temporarily switches the document window to Top view so you can place the spin axis in Top view. After placing the spin axis, DENEBACAD switches to the original view (Front, Back, Left, or Right), and activates the Custom 3D Axis.

To use the Define Vertical 3D Axis command, refer to "To use a Define 3D Axis command," next.

To use a Define 3D Axis command

Use this procedure to set a 3D Axis in Front, Left, or Top view.

1. Choose a Define 3D Axis command in the Layout > 3D Axis submenu. You can choose Define Frontal 3D Axis, Define Vertical 3D Axis, or Define Lateral 3D Axis, depending on the current view.

   ▲ See "Define Frontal 3D Axis" on page 253 for the views in which this command is available.

   ▲ See "Define Lateral 3D Axis" on page 254 for the views in which this command is available.

▲ See “Define Vertical 3D Axis” on page 254 for the views in which this command is available.

2. DENEBACAD switches to the view in which you set the 3D Axis. A spin pointer appears. To help you place the 3D axis, a full-screen crosshair follows the spin pointer.
3. Move the pointer and click at the coordinates where you want to set the spin axis.
4. The document window returns to the original view. The axis appears as a gray line in the drawing window.

Defining an angled spin extrusion axis

Using a relative view lets you set a spin extrusion axis for extruding objects that are non-parallel or non-perpendicular in orthogonal views.

For example, to draw an angled satellite dish, you can set a spin extrusion axis in a relative view and use it to extrude the dish, which will appear angled in orthogonal views.

To set a spin extrusion axis in a relative view

1. In Sculpt mode, set up a relative view. See “Relative View submenu” on page 341.
2. Set the 3D Axis as described under “To use a Define 3D Axis command” on page 254.
3. Draw the object.
4. Choose View > Relative View > Default Relative View, or use the Status bar, to return to an orthogonal view.

In the default relative view, the object is angled based on the angled spin extrusion axis that you set using the relative view.

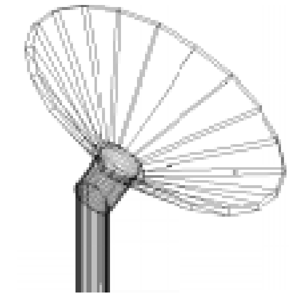


Fig. 217 Satellite dish in Front view

DELETE 3D AXIS

Modes: Draft, Sculpt

The Delete 3D Axis command deletes a spin extrusion axis. This command is available after you save a spin extrusion axis in a particular view.

When you delete a spin extrusion axis, it is no longer available in the 3D Axis pop-up menu on the Status bar, or in the Layout > 3D Axis submenu.

If you try to delete a saved axis that is active, a warning message appears. Click OK in the message box to continue. The saved axis becomes an unsaved, active “Custom 3D Axis.”

To delete a 3D axis

1. Choose Layout > 3D Axis > Delete 3D Axis.
2. A dialog box appears. Select the 3D axis (or axes) to delete.

   ▲ To select several continuous items, press Shift and click the first and last items. To select non-contiguous items, press Ctrl (Windows) or Cmd (Mac OS) and click each item.

3. To remove the selected items, click Remove, and then click OK to delete them.

SAVE 3D AXIS

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+Shift+P
Windows Shortcut: Ctrl+Alt+Shift+P

The Save 3D Axis command saves a custom spin extrusion axis that is active, but has not been saved. The command is not available if the custom axis has been saved before.

When you save a custom axis, the name you save it with appears in the 3D Axis pop-up menu on the Status bar, and in the Layout > 3D Axis submenu, so you can select the axis again. The saved axis name replaces "Custom 3D Axis" in the 3D Axis menus.

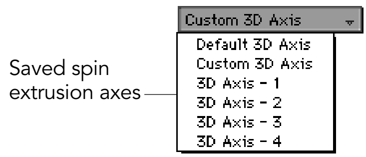
To save a custom 3D axis

1. Choose Layout > 3D Axis > Save 3D Axis.

2. Type a name in the dialog box and click OK.

Selecting a saved extrusion axis

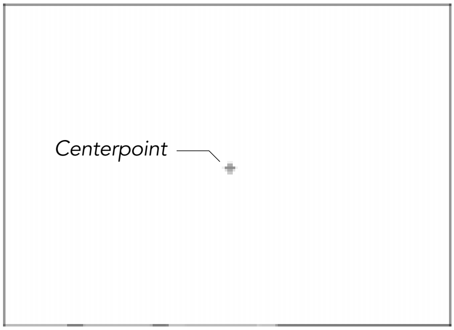
In each view, the names of saved spin extrusion axes are available in the 3D Axis pop-up menu on the Status bar, and the Layout > 3D Axis submenu.



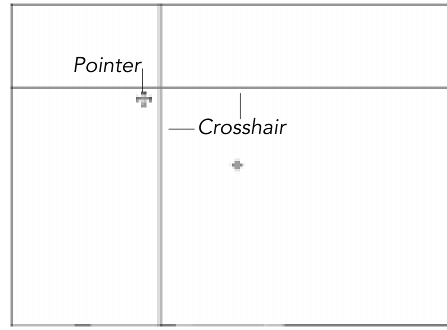
The name of the active axis appears in the Status bar, and has a check mark in the 3D Axis submenu.

◆ **To select a saved extrusion axis:** Choose the name of the 3D Axis from the pop-up menu on the Status bar, or in the Layout > 3D Axis submenu.

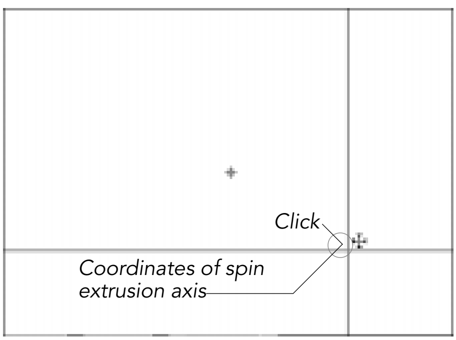
Using a Define 3D Axis command



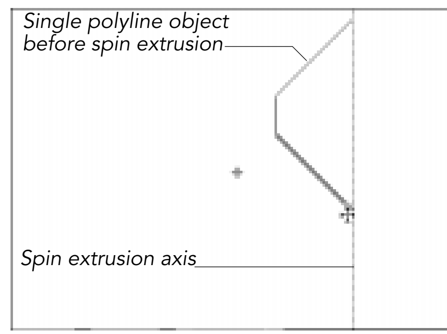
1. Choose a Define 3D Axis command in the view in which you want to extrude.



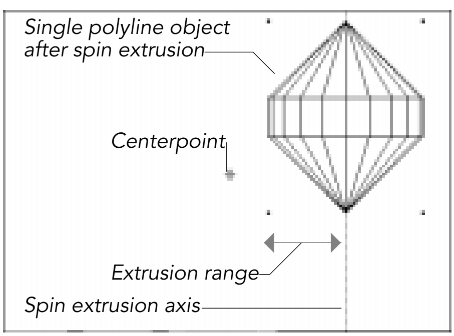
2. Move the pointer to position the spin axis. The full-screen crosshair follows the pointer.



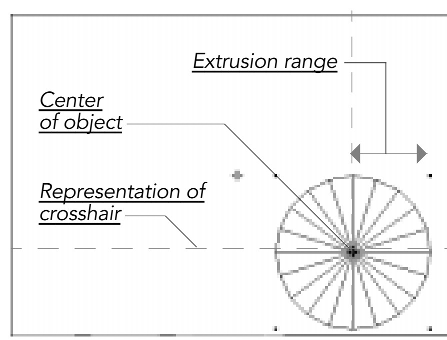
3. Click to set the coordinates of the spin axis. DENEBACAD switches back to the original view.



4. The spin extrusion axis appears on-screen as a vertical gray line. Draw an object to extrude.

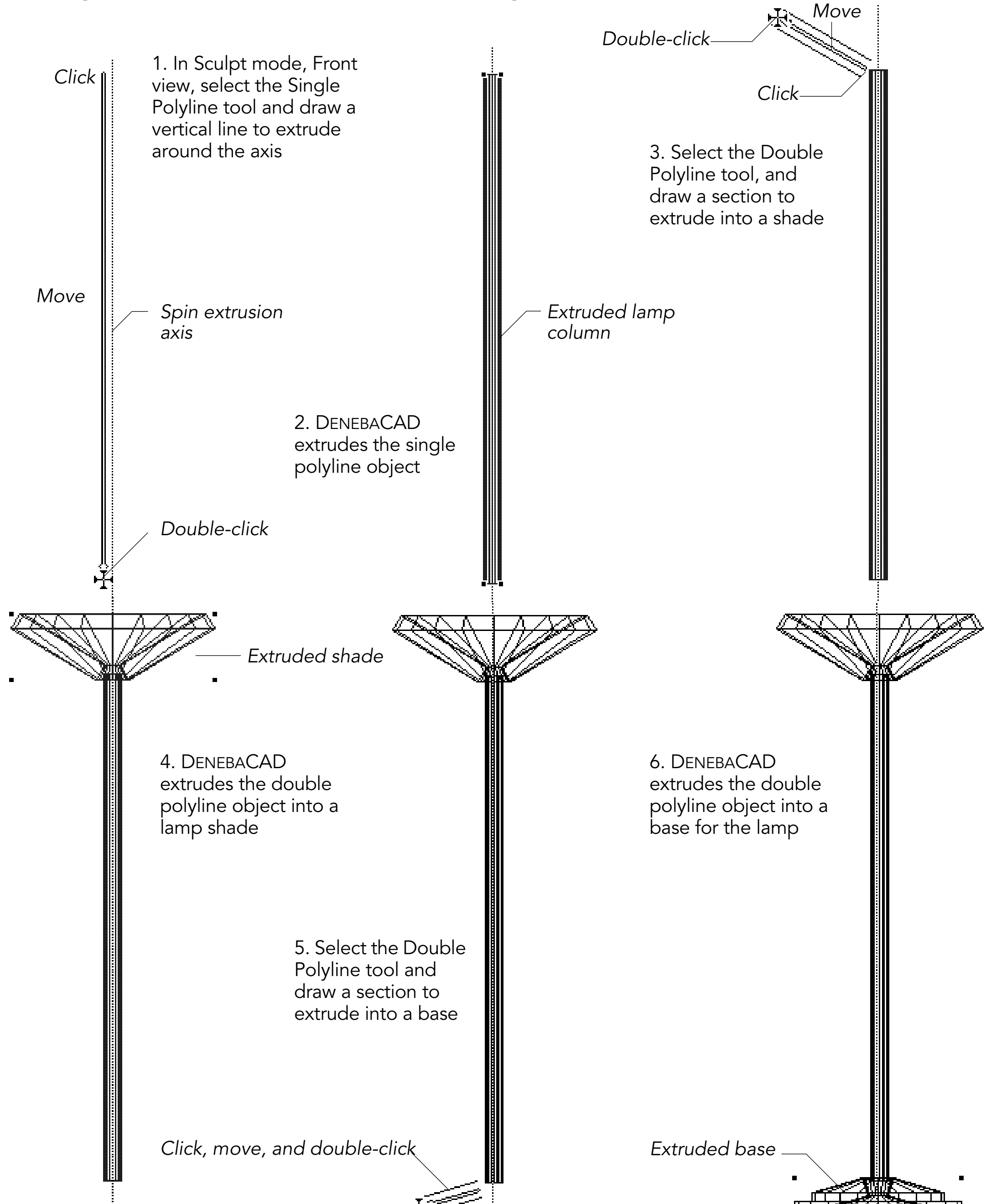


5. The object's extrusion range is based on the distance of the original object from the axis. Here, the object is next to the extrusion axis.



6. Switch to Front view to see that the position of the object in 3D space is based on the coordinates you set in Step 3.

Using a spin extrusion axis to create a halogen lamp



3D PLANE SUBMENU

Modes: Draft, Sculpt

The 3D Plane submenu appears in the Layout menu when you select Linear or Sweep extrusion method. 3D Plane commands let you set, save, and delete extrusion planes.

The general concept of extrusion planes is described next, followed by descriptions of each 3D Plane command and examples of their use.

ABOUT 3D PLANES

In DENEBACAD, the term *3D Plane* refers to a pair of planes that control linear extrusions. In other words, a "3D Plane" is two *extrusion planes*. These planes limit the range of extrusion when a 2D object is projected into 3D space.

The process of linear extrusion projects an object between the active extrusion planes. In effect, a 2D object is projected from one extrusion plane to the other extrusion plane to form a 3D object.

For example, if you draw a circle in Top view, you can use linear extrusion to extend the circle into a 3D cylinder. The active extrusion planes control the cylinder's height and vertical position in 3D space. The ends, or *caps*, of the extruded cylinder rest on the extrusion planes.

Extrusion planes can be parallel to each other or inclined. Their orientation affects the shape of extruded objects.

Parallel extrusion planes produce 3D objects with flat ends, which are parallel to each other and to the view plane. Inclined extrusion planes slant one or both ends relative to the view plane.

Consider a column that supports a sloped ceiling. To create this, you can extrude a circle, with one extrusion plane corresponding to the floor and the other at the ceiling height and angle. The bottom of the resulting column will be flat on the floor, and its top will be angled at the ceiling.

It's helpful to display the Info bar and the Status bar before you begin working with extrusion planes. To do this, choose Layout > Toolbars, and select the Info bar and Status bar options.

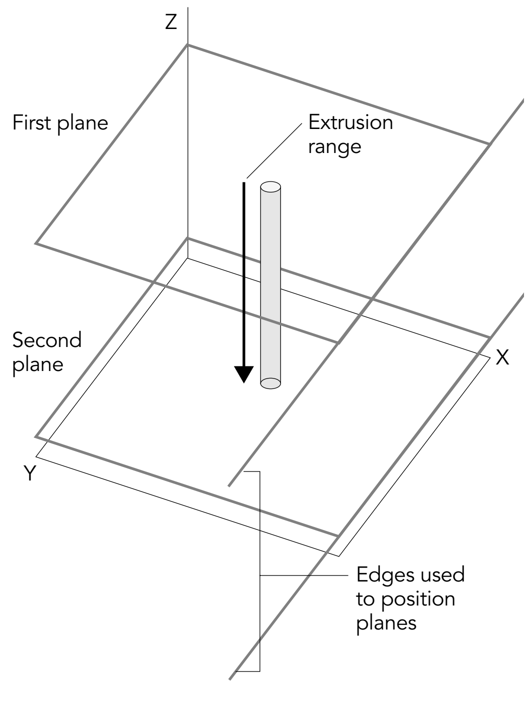


Fig. 218 Linear extrusion planes

Views and extrusion planes

You use "Define 3D Plane" commands to set up extrusion planes for each view in DENEBACAD, including relative views.

A different set of extrusion planes is active in each orthogonal (and relative) view. You can save 255 sets of extrusion planes for each view.

Extrusion planes must be used in the view they were set up for. Extrusion planes set up for Front view, for example, can't be used for extrusion in Back view or any other view except Front.

Extrusion planes are view-dependent, but are mode-independent. You can set up extrusion planes in Draft mode, Top view, and use them in Sculpt mode, Top view — but not in other views in Draft or Sculpt mode.

Draft and Sculpt modes differ in that extrusion is optional in Draft mode and automatic in Sculpt mode. In Draft mode, you draw 2D objects, and you can extrude them with the Linear command, which creates 3D objects that appear in Sculpt mode. When you draw in Sculpt mode, shapes are immediately extruded into 3D objects.

SETTING UP EXTRUSION PLANES

To set up extrusion for a particular view, you must use a different view. You will be perpendicular to extrusion planes you are setting up, and parallel to them when you extrude objects.

You can set up extrusion planes for any view. When you choose a command to define the extrusion planes, DenebaCAD will switch to a perpendicular view, either Top, Front, or Left.

In terms of coordinate systems, the view where you extrude objects determines the dimension that extrusion adds. It's probably easiest to begin working in the Cartesian coordinate system, which displays X, Y, and Z information in the Info bar. This can help you visualize how you set up extrusion planes in Top, Front, or Left view.

For example, if you extrude objects from Top or Bottom view, extrusion adds the Z dimension to the objects. If you extrude objects from Front or Back view, extrusion adds the Y dimension. If you extrude from Left or Right view, extrusion adds the X dimension to create 3D objects.

Knowing this can help you visualize the views used for setting up extrusion planes. For example, if you draw in Top view, 2D objects have X and Y dimensions, as shown in the Info bar.

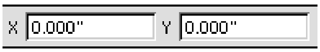


Fig. 219 2D coordinates in Top view

Extruding an object in Top view will add height along the Z axis. However, you can't draw in the Z direction in Top view, you can draw along the X and Y axes only.

In visual terms, you can't see the height of an object in a 3D model if you're above the object in Top view. Instead, you have to look at the 3D model from the side.

For you to set up Top view (Z-axis) extrusions, DenebaCAD switches the view so you can see your 3D model from the front or side.

The commands for setting up 3D planes in Top view are Define Frontal 3D Plane and Define Lateral 3D Plane. Define Frontal 3D Plane shows you the Front elevation of your model and Define Lateral 3D Plane shows you the Left elevation as you set up extrusion planes. You can use either perspective to set up extrusion planes along the Z axis. These extrusion planes will determine the vertical start, end, and height of Top view extrusions.

If the extrusion planes will be flat (parallel to the XY axes) there is no difference between using Define Frontal or Define Lateral to set them up. You can choose whichever view lets you align the planes with significant objects in the 3D model.

However, if you want to set either extrusion plane at an angle, your orientation to the model does matter.

Consider a house model, where Front view shows the front elevation of the house and Left view shows the left side. You want to set the upper extrusion plane to match the slope of a ceiling. If the ceiling slopes left to right, choose Define Frontal 3D Plane. If the ceiling slopes front to back, choose Define Lateral 3D Plane.

See page 265 for illustrations of angled extrusion planes.

After you set up extrusion planes, DENEBACAD switches back to the original view. The range of extrusion defined by the planes is shown on the right of the Info bar.

Fig. 220 Default extrusion range

The text boxes show the position of the two extrusion planes. The distance from one plane to the other is the total extrusion depth. The default depth is 10 feet.

For more information on setting up extrusion planes, refer to the “Define 3D Plane” commands, next, and the illustrations “Defining parallel extrusion planes” on page 263, and “Defining inclined extrusion planes” on page 264.

CUSTOM 3D PLANE

Modes: Draft, Sculpt

“Custom 3D Plane” appears in the 3D Plane submenu and in the Status bar when the last set of extrusion planes you created in the current view are active, but have not been saved with the Save 3D Plane command.

In each view, “Custom 3D Plane” appears only after you define a set of extrusion planes. Then, “Custom 3D Plane appears” with a check mark in the Layout > 3D Plane submenu, and on the 3D Plane pop-up menu in the Status bar.

If you choose the Default 3D Plane command, or a saved set of extrusion planes, the custom extrusion planes are discarded. “Custom 3D Plane” is replaced on the 3D Plane pop-up menu and the Layout 3D plane submenu. Therefore, you should save any set of extrusion planes that you want to use more than once.

DEFAULT 3D PLANE

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+O
Windows Shortcut: Ctrl+Shift+O

You can activate the default set of extrusion planes in each view by choosing Default 3D Plane in the 3D Plane submenu or the 3D Plane pop-up menu on the Status bar.

The extrusion range of the default set of extrusion planes for new DENEBACAD documents is 10 feet. To change the extrusion range of the default set of extrusion planes for the current view, choose Object > Extrude > Extrude Options.

In each view, the default set of extrusion planes is active until you define extrusion planes using a “Define 3D Plane” command.

When the default set of extrusion planes is active, “Default 3D Plane” appears in the Status bar, and the Default 3D Plane command has a check mark in the 3D Plane submenu.

DEFINE FRONTAL 3D PLANE

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+F
Windows Shortcut: Ctrl+Alt+F

The Define Frontal 3D Plane command lets you set up extrusion planes from Front view.

You can use Define Frontal 3D Plane to set up planes for Top, Bottom, Left, or Right view extrusion. One of these must be the current view for Define Frontal 3D plane to be available.

When you choose Define Frontal 3D Plane, DENEBACAD switches the active document window to Front view.

If the view is Top or Bottom when you choose the command, the planes you set in Front view appear as horizontal lines that you can angle up or down.

If the view is Left or Right when you choose the command, the planes you set in Front view appear as vertical lines that you can angle left or right.

To set parallel extrusion planes, see fig. 221 (page 263). To set inclined extrusion planes, see fig. 222 (page 264).

After you define extrusion planes in Front view, DENEBACAD switches to the original view. “Custom 3D Plane” appears in the Status bar and in the 3D Plane submenu. To save the extrusion planes, use the Save 3D Plane command.

DEFINE LATERAL 3D PLANE

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+L
Windows Shortcut: Ctrl+Alt+L

The Define Lateral 3D Plane command lets you set up extrusion planes from Left view.

You can use Define Lateral 3D Plane to set up planes for Top, Bottom, Front, or Back view extrusion. One of these must be the current view for the command to be available.

When you choose Define Lateral 3D Plane, DENEBACAD switches the active document window to Left view.

If the view is Top or Bottom when you choose the command, the planes you set in Left view appear as horizontal lines that you can angle up or down.

If the view is Front or Back when you choose the command, the planes you set in Left view appear as vertical lines that you can angle left or right.

To set parallel extrusion planes, see fig. 221 (page 263). To set inclined extrusion planes, see fig. 222 (page 264).

After you define extrusion planes in Left view, DENEBACAD switches to the original view. “Custom 3D Plane” appears in the 3D Plane submenu and the Status bar. To save the extrusion planes, use the Save 3D Plane command.

DEFINE VERTICAL 3D PLANE

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+V
Windows Shortcut: Ctrl+Alt+V

The Define Vertical 3D Plane command lets you set up extrusion planes from Top view.

You can use Define Vertical 3D Plane to set up planes for Front, Back, Left, or Right view extrusion. One of these must be the current view for the command to be available.

When you choose Define Vertical 3D Plane, DENEBACAD switches the active document window to Top view.

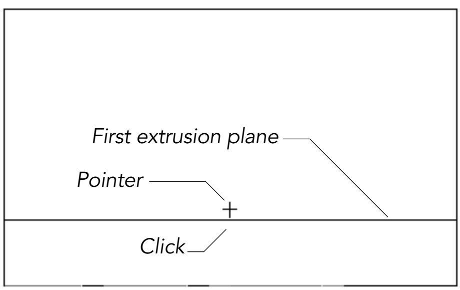
If you are in Front or Back view when you choose the command, the planes you set in Top view are horizontal lines you can angle up or down.

If you are in Left or Right view when you choose the command, the planes you set in Front view are vertical lines you can angle left or right.

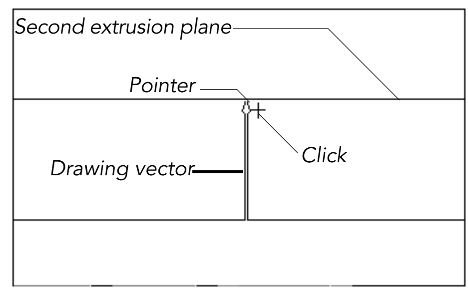
To set parallel extrusion planes, see fig. 221 (page 263). To set inclined extrusion planes, see fig. 222 (page 264).

After you define extrusion planes in Top view, DENEBACAD switches to the original view. “Custom 3D Plane” appears in the 3D Plane submenu and the Status bar. To save the extrusion planes, use the Save 3D Plane command.

Parallel horizontal extrusion planes

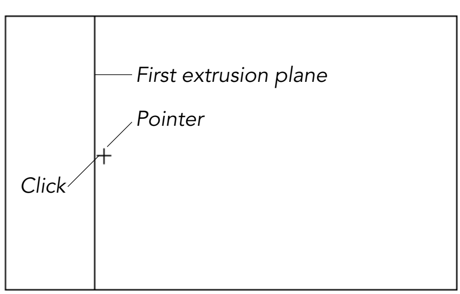


1. Move the pointer to position the first extrusion plane, and then click to set the plane.

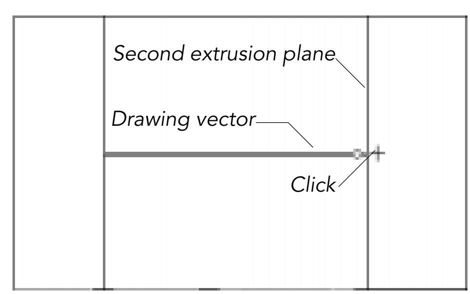


2. Move the pointer to position the second extrusion plane. Click to set the second extrusion plane.

Parallel vertical extrusion planes



1. Move the pointer to position the first extrusion plane, and then click to set the plane.



2. Move the pointer to position the second extrusion plane. Click to set the second extrusion plane.

Fig. 221 Defining parallel extrusion planes

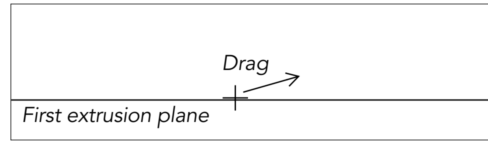
Defining inclined extrusion planes

Setting extrusion planes at angles lets you create complex 3D shapes easily.

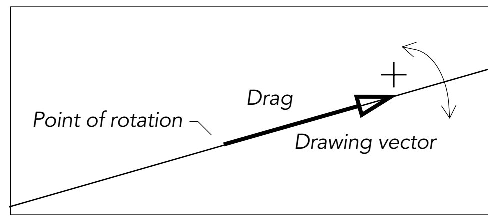
1. Choose Layout > 3D Plane > Define Frontal 3D Plane, Define Lateral 3D Plane, or Define Vertical 3D Plane.
2. The view changes to the one where you will set up the extrusion planes. The pointer is a cross, with a line that represents the first extrusion plane.
3. Do either:

   ▲ To set the plane without rotating it, move to position the plane, and click to set the plane.

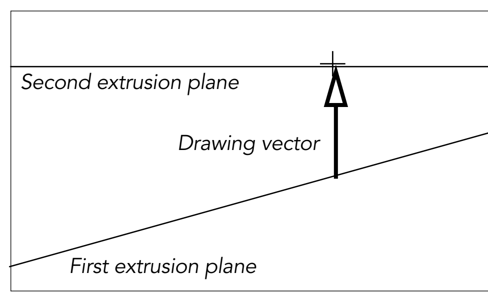
   ▲ To angle the first plane, move to position it, and drag to rotate the plane around the point where you begin dragging. The drawing vector indicates the rotation angle, and acts as a lever that rotates the plane. If you lengthen the vector, the plane rotates less when you move the mouse. Release the mouse to set the plane.
4. Repeat the previous step to set the second plane, with or without rotating it.
5. The view changes to the original view. “Custom 3D Plane” appears on the Status bar, indicating that the custom planes you set are active.



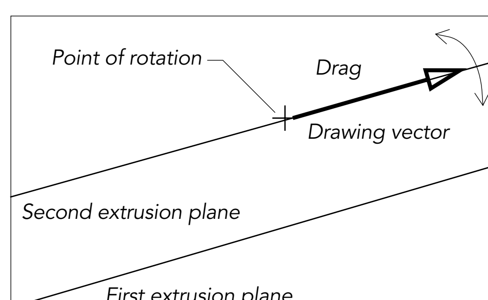
Move the pointer to position the first plane. Click to set it, or begin dragging to rotate it.



Continue dragging to incline the first plane, and then release the mouse button.



Move the pointer to position the second plane. Click to set it, or drag to set the angle.



Release the mouse button to set the angle of the second plane.

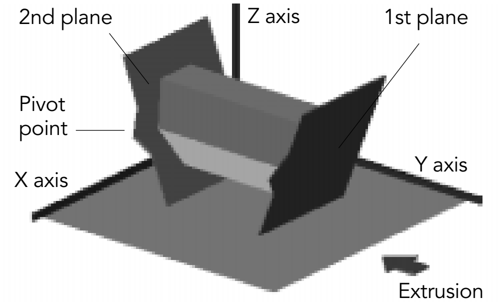
Fig. 222 Defining inclined extrusion planes

Extrusion planes in various views

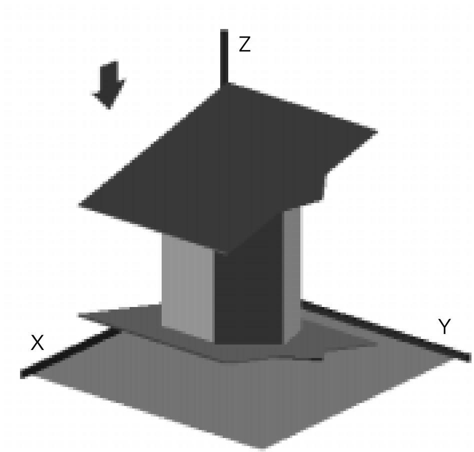
You can define extrusion planes from Left (Define Lateral 3D Plane), Front (Define Frontal 3D Plane) or Top (Define Vertical 3D Plane). You choose a "Define" command in the view where you will draw, and DENEBACAD switches to a perpendicular view where you set up the extrusion planes.

For the extrusion shown at right, Define Lateral 3D Plane was chosen in Front view. The planes were set in Left view, and then the hexagon was drawn in Front view.

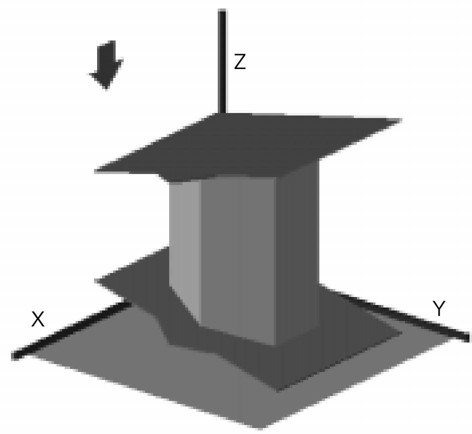
Extrusion planes can be parallel or inclined. You can and the planes away from horizontal or vertical, depending on which view you use to set them up. In these illustrations, a point on the edge of each plane represents the pivot point. The gray arrows indicate the direction of extrusion.



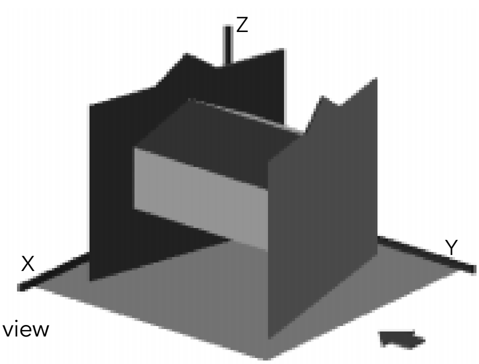
Extrusion planes set in Left view (inclined left and right), hexagon drawn in Front view



Extrusion planes set in Front view
Object drawn in Top view



Extrusion planes set in Left view
Object drawn in Top view



Extrusion planes set in Top view
Object drawn in Front view

Fig. 223 Orientation of extrusion planes

DELETE 3D PLANE

Modes: Draft, Sculpt

The Delete 3D Plane command deletes saved extrusion planes. Deleting a set of extrusion planes removed the set’s name from the 3D Plane pop-up menu and 3D Plane submenu.

To delete 3D planes

1. Choose Layout > 3D Plane > Delete 3D Plane. The Delete 3D Plane dialog box appears.
2. In the dialog box, select the saved extrusion planes to delete.

   ▲ To select (or deselect) a range of items, press Shift and click the first and last items.

   ▲ To select (or deselect) non-contiguous items, press Ctrl (Windows) or Cmd (Mac OS) and click each item.
3. To remove the selected extrusion planes, click Remove. To delete the extrusion planes, click OK.

If you try to delete the active set of extrusion planes, a message appears. Click OK to continue. DENEBACAD will keep the deleted extrusion planes active as (unsaved) custom extrusion planes. See “Custom 3D Plane” on page 261 for more information.

SAVE 3D PLANE

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+Shift+P
Windows Shortcut: Ctrl+Alt+Shift+P

After defining extrusion planes, you can save them as a named set.

In each view, named extrusion planes appear in the 3D Plane submenu, and in the 3D Plane pop-up menu on the Status bar.

You can use saved extrusion planes in the view they were created for. For example, if you save the active extrusion planes in Top view, you can use them in Top view only.

To save extrusion planes

4. With custom extrusion planes active, choose Layout > 3D Plane > Save 3D Plane. The Save 3D Plane dialog box appears.

5. Type a name for the extrusion planes in the text box.
6. Click OK. In the current view, DENEBACAD adds the name of the extrusion planes to the 3D plane pop-up menu on the Status bar, and to the bottom of the 3D Plane submenu.

Working with saved extrusion planes

In each view, the names of saved sets of extrusion planes appear at the bottom of the 3D Plane submenu, and in the 3D Plane pop-up menu on the Status bar.

If saved extrusion planes are active, their name appears on the Status bar.

To use saved extrusion planes

1. Choose the view with the saved extrusion planes you want to use.
2. Choose the name of the extrusion planes in the 3D Plane submenu, or the 3D Plane pop-up menu on the Status bar.

TOOLBARS

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+Shift+T
Windows Shortcut: Ctrl+Alt+Shift+T

You can use the Toolbars command to display or hide DENEBACAD’s toolbars.

Four toolbars are available. The Action buttons, Info bar, and Status bar appear below the menu bar. The Help bar appears at the bottom of the screen.

Action buttons Provides quick access to commands and common editing features, such as reshape mode, extrusion methods, path editing, Rotate, Mirror, Scale, and Align.

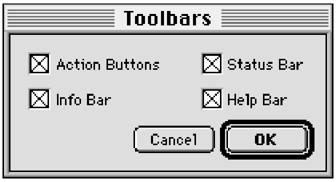
Info bar Shows coordinate information for pointer movements. You can use the Info Bar to enter coordinates when you create objects and perform other operations. You can also change coordinate systems using the Coordinate System pop-up menu.

Status bar Lets you select Extrusion formats and options in pop-up menus. The Status bar contains five pop-up menus: View Options, Display Options, 3D Planes (or 3D Axes), Sections, and Relative Views.

Help bar Displays tips and instructions for using tools and commands. It also displays prompts for actions, file information, rendering progress, boolean vector progress, and memory status.

To show or hide toolbars

1. Choose Layout > Toolbars. The Toolbars dialog box opens. Active toolbars have a checkmark.

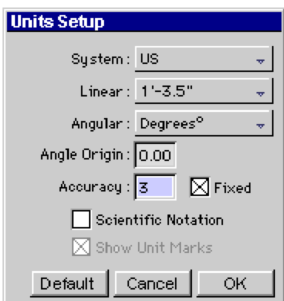


2. To hide an active toolbar, deselect its checkbox. To display a toolbar, select the checkbox.
3. Click OK to implement the toolbar settings. Click Cancel to leave the current toolbar setup the same.

UNITS SETUP

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+Opt+Shift+U
Windows Shortcut: F10

Use the Units Setup command to configure units and measurement system for a project. For example, when you use a Dimensioning tool, the dimensions use the units and measurement system specified in the Units Setup dialog box.



◆ **To use the Units Setup command:** Choose Layout > Units Setup. In the Units Setup dialog box, select the options you want and click OK.

System Lets you choose a measurement system for the active document. You can choose the US (English) measurement system, or the Metric measurement system.

Linear Lets you choose the units for the measurement system. For example, if you chose Metric in the System pop-up menu, you can choose kilometers, meters, centimeters, or millimeters in the Linear pop-up menu.

Angle Origin Lets you specify the angle origin for your the document.

Angles are measured from 0 to 360 degrees, except in the Gradient coordinate system, in which a circle is divided into 400 degrees. By using the Angle Origin text box you, you can specify a new origin for angles and angular measurements in DENEBACAD.

Accuracy Lets you specify the decimal precision for the project. DENEBACAD allows precision up to 16 decimal places.

Click the “Fixed” check box to suppress values smaller than the assigned number of decimal places.

Scientific Notation If you select this option, numbers are displayed in scientific notation.

Show Unit Marks When this option is selected, DENEBACAD includes the current measurement units in dimensions. If you select the US measurement system, and a combination of feet and inches in the Linear pop-up menu, Show Unit Marks is not available. In addition, if you click the Show Unit Marks check box, and then choose a combination of feet and inches in the Linear pop-up menu, DENEBACAD deselects this option.

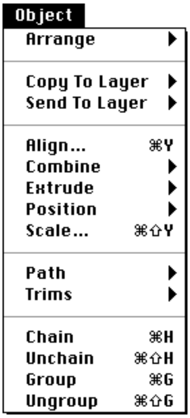
Default Click the Default button to set the current settings as the default settings for new documents.

The Object menu and the corresponding Action buttons provide a many functions for manipulating objects in DENEBACAD.

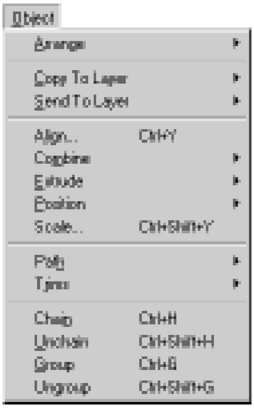
In general, you must select one or more objects before you can choose a command in the Object menu (or use an Action button to apply an Object menu command).

The Object menu contains the following commands: Align, Scale, Chain, Unchain, Group, and Ungroup.

The Object menu also contains several submenus. The commands in the submenus let you arrange objects, copy or send them to layers, configure extrusion options, change the position of objects, manipulate object paths, trim object shapes, and combine objects using Boolean operations.



Mac OS



Windows

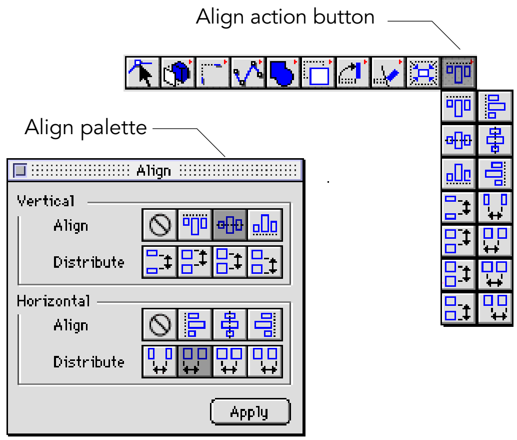
ALIGN

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Y
Windows Shortcut: Ctrl+Y

To quickly and precisely position objects in DENEBACAD, you can use the Align palette or the Align action button. The Align palette can stay on screen while you work.

You can align selected objects horizontally or vertically. Or, you can equally distribute selected objects within a specified area.

As a reference point for alignment and distribution, DENEBACAD uses the objects' bounding rectangles. You can align or distribute objects in separate or combined operations.



To use the Align palette

1. Select the objects you want to align.
2. Choose Object > Align. The Align palette appears.
3. Click to select an option in the Vertical section and one in the Horizontal section.
4. Click Apply. DENEBACAD aligns or distributes the objects in the specified manner.

Align action buttons

You can apply one alignment or distribution option at a time using an Align action button. The last Align button you used appears in the Attributes bar. Click the button to apply the same alignment option, or press the button to choose another option from the Align toolbar.

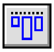
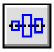
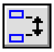
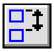
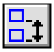
ALIGN OPTIONS

DENEBACAD aligns or distributes objects relative to their bounding rectangles.

When you use the Align palette, you can combine horizontal and vertical alignment and distribution. However, you cannot combine vertical alignment and vertical distributions, or horizontal alignment and horizontal distribution.

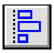
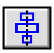
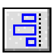
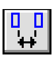
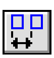
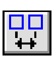
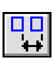
Vertical alignment and distribution

When you align or distribute objects vertically, DENEBACAD moves the selected objects vertically to arrange them relative to each other.

Icon	Vertical options	Procedure
	No alignment	To specify no vertical alignment, select this button
	Top vertical alignment	Lines up the tops of selected objects by moving them vertically
	Centered vertical alignment	Lines up the centers of selected objects by moving them vertically on-screen
	Bottom vertical alignment	Lines up the bottoms of selected objects by moving them vertically
	Inside vertical distribution	Equalizes space between the inside edges of selected objects by moving them vertically
	Top vertical distribution	Equalizes space between the top edges of selected objects by moving them vertically
	Centered vertical distribution	Equalizes space between the centers of selected objects by moving them vertically
	Bottom vertical distribution	Equalizes space between the bottom edges of selected objects by moving them vertically

Horizontal alignment and distribution

When you align or distribute objects horizontally, DENEBACAD moves the selected objects horizontally to arrange them relative to each other.

Icon	Horizontal options	Procedure
	No horizontal alignment	To specify no horizontal alignment, select this button
	Left horizontal alignment	Lines up the left sides of selected objects by moving them horizontally
	Centered horizontal alignment	Lines up the centers of selected objects by moving them horizontally
	Right horizontal alignment	Lines up the right sides of selected objects by moving them horizontally
	Inside horizontal distribution	Equalizes space between the inside edges of selected objects by moving them horizontally
	Left horizontal distribution	Equalizes space between the left edges of selected objects by moving them horizontally
	Centered horizontal distribution	Equalizes space between the centers of selected objects by moving them horizontally
	Right horizontal distribution	Equalize space between the right edges of selected objects by moving them horizontally

ARRANGE SUBMENU

The Arrange submenu contains commands that change the stacking order of objects on a layer.

Bring To Front Moves the selected objects to the front of the current layer's stacking order. The objects you bring to the front maintain their stacking order relative to each other, but are in front of all other objects.

Send to Back Moves the selected objects to the back of the current layer's stacking order. The objects you send to the back maintain their stacking order relative to each other, but are behind all other objects.

Shuffle Forward Moves each of the selected objects one step towards the front of the current layer's stacking order.

Shuffle Backward Moves each of the selected objects one step towards the back of the current layer's stacking order.

To use the Arrange submenu

1. Select the object or objects you want to move in the stacking order.
2. To arrange the object or objects, choose *command name* in the Arrange submenu, where *command name* is the name of the Arrange submenu command you want to choose.

STACKING ORDER AND LAYERS

To avoid confusion, remember the internal stacking order for objects on a layer differs from the order of layers themselves.

You can create up to 255 layers for each view (Top, Bottom, Front, Back, Left, Right, or relative) in Draft mode. Using the Layer Manager, you can change the order of layers. If you do this, it changes the relation of objects in your project. In the Layer Manager, objects on higher layers block objects on lower layers, regardless of the internal stacking order of the objects on any particular layer.

To remove an object from a layer and send it to a different layer, use the Send to Layer command in the Object menu. To send a copy of an object to another layer, use the Copy to Layer command.

BRING TO FRONT

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+F
Windows Shortcut: Ctrl+F

By using the Bring to Front command, you can move selected objects to the front of the current layer's stacking order. In DENEBACAD, every layer has a stacking order. The stacking order influences the way objects relate to each other on the layer. Objects in the front of the stack obscure objects in the back of the stack.

When you draw a new object on a layer, it gets placed in the "front" of the stack. Existing objects then move toward the "back." If you paste an object from the Clipboard, it appears in the front of the stack.

Unless you rearrange the objects on a layer using the commands in the Arrange submenu, the most recent object is always in the "front," and the oldest object is in the "back." This means stacking order is an important consideration when using the commands in the Combine submenu (Punch from Front, Punch from Back, Punch from Front & Trim, Punch from Back & Trim) to create Boolean objects.

To use the Bring to Front command

1. Select the object you want to move to the front.
2. Choose Bring to Front in the Arrange submenu. DENEBACAD moves the object to the front of the current layer's stacking order.

SEND TO BACK

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+B
Windows Shortcut: Ctrl+B

By using the Send to Back command, you can move selected objects to the back of the current layer's stacking order. In DENEBACAD, every layer has a stacking order. The stacking order influences the way objects relate to each other on the layer. Objects in the front of the stack obscure objects in the back of the stack.

When you draw a new object on a layer, it gets placed in the "front" of the stack. Existing objects then move toward the "back." If you paste an object from the Clipboard, it appears in the front of the stack.

Unless you rearrange the objects on a layer using a command in the Arrange submenu, the most recent object is always in the “front,” and the oldest object is in the “back.” This means stacking order can be an important consideration when using the commands in the Combine submenu (Punch from Front, Punch from Back, Punch from Front & Trim, Punch from Back & Trim) to create Boolean objects.

To use the Send to Back command

1. Select the object you want to move to the back.
2. Choose Send to Back in the Arrange submenu. DENEBACAD moves the object to the back of the current layer’s stacking order.

SHUFFLE BACKWARD

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+B
Windows Shortcut: Ctrl+Shift+B

By using the Shuffle Backward command, you can move selected objects one step backward in the current layer’s stacking order. In DENEBACAD, every layer has a stacking order. The stacking order influences the way objects relate to each other on the layer. Objects in the front of the stack obscure objects in the back of the stack.

When you draw a new object on a layer, it gets placed in the “front” of the stack. Existing objects then move toward the “back.” If you paste an object from the Clipboard, it appears in the front of the stack.

Unless you rearrange the objects on a layer using a command in the Arrange submenu, the most recent object is always in the “front,” and the oldest object is always in the “back.” This means the stacking order on a layer can be an important consideration when using the commands in the Combine submenu (Punch from Front, Punch from Back, Punch from Front & Trim, Punch from Back & Trim) to create Boolean objects.

To use the Shuffle Backward command

1. Select the object you want to move one step backward in the stacking order.
2. Choose Shuffle Backward in the Arrange submenu.

SHUFFLE FORWARD

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+F
Windows Shortcut: Ctrl+Shift+F

By using the Shuffle Forward command, you can move selected objects one step forward in the current layer’s stacking order. In DENEBACAD, every layer has a stacking order. The stacking order influences the way objects relate to each other on the layer. Objects in the front of the stack obscure objects in the back of the stack.

When you draw a new object on a layer, it gets placed in the “front” of the stack. Existing objects then move toward the “back.” If you paste an object from the Clipboard, it appears in the front of the stack.

Unless you rearrange the objects on a layer using the commands in Arrange submenu, the most recent object is always in the “front” of the stack,” and the oldest object is in the “back.” This means stacking order can be an important consideration when you use the commands in the Combine submenu (Punch from Front, Punch from Back, Punch from Front & Trim, Punch from Back & Trim) to create Boolean objects.

To use the Shuffle Forward command

1. Select the object you want to move one step forward in the stacking order.
2. Choose Shuffle Forward in the Arrange submenu. DENEBACAD moves the object one step backward in the stacking order for the current layer.

CHAIN

Mode: Draft
Mac OS Shortcut: Cmd+H
Windows Shortcut: Ctrl+H

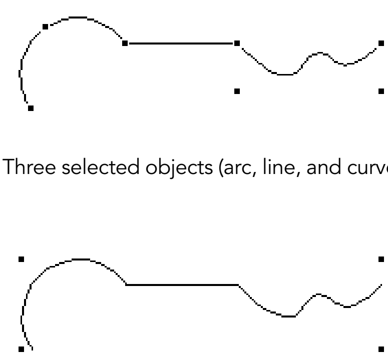
The Chain command connects selected lines, arcs, and open polylines to form a single polygon or polyline. This command is useful for creating shapes by drawing individual segments first.

For example, you can draw a shape that contains an arc, a line, and a B-spline curve, arrange their endpoints so they touch, and chain them to create one object. In addition, you can use the Chain command to re-connect objects that you unchained with the Unchain command.

The Chain command can't be used to connect objects that are not touching each other, such as parallel lines, or closed objects.

To use the Chain command

1. Select the objects you want to chain. An endpoint of each object you want to chain must touch at least one endpoint of another selected object.
2. Choose Chain in the Object menu. DENEBACAD joins the selected segments at the endpoints.



Three selected objects (arc, line, and curve)

One object created with Chain command

Fig. 224 Chaining objects

COMBINE SUBMENU

The Combine submenu in the Object menu contains commands that let you combine 2D or 3D objects to create complex shapes without losing the ability to edit the original objects. In DENEBACAD, these are referred to as *Boolean operations*.

If you have the Preference “Show boolean outlines” selected, the shape of the combined object remains visible as you reshape its constituent objects. For example, you can combine a rectangle and a six-sided polygon to create a bay window in a wall. Later, if you decide the window should be in a different place and a different size, the Reshape action button (or the Reshape command) displays the original rectangle and polygon for editing. In addition, when the preference “Show Boolean outlines” is on, the outline of Boolean objects is displayed in red.

DENEBACAD has six Boolean operations:

Unite Lets you create a new object from the outline of multiple objects in Draft and Sculpt.

Intersect Lets you create a new object from the overlapping area of multiple objects. This hides the parts of the objects outside the intersection.

Punch from Front The Punch from Front command knocks holes in the object at the back of the stacking order using the objects in front.

Punch from Front & Trim The “Punch from Front & Trim” command knocks holes in the object at the back of the stacking order using the objects in the front, similar to “Punch from Front.” However, unlike “Punch from Front,” areas that don't overlap the back object are permanently deleted, and cannot be recalled in Reshape mode.

Punch from Back The Punch from Back command takes an object at the front of the stacking order and uses it to “knock out” (or punch) a hole in all of the overlapping objects behind it.

Punch from Back & Trim The Punch from Back & Trim command uses an object in the front of the stacking order to “knock out” a hole in all of the overlapping objects behind it. However, unlike “Punch from Back,” areas that don't overlap the front object are permanently deleted, and cannot be recalled in Reshape mode.

INTERSECT

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+E
Windows Shortcut: Ctrl+E

By using the Intersect command, you can create a new object from the overlapping area of multiple objects. The parts of the objects outside the intersection are then hidden. The Intersect command is available in Draft and Sculpt modes.

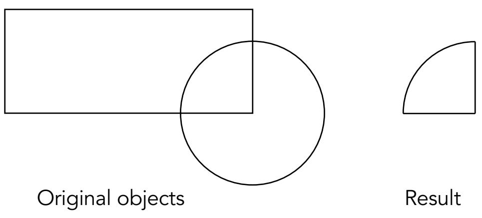


Fig. 225 Result of Intersect command

If necessary, you can use the Reshape command in the Edit menu to reshape an object you create with the Intersect command.

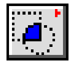


Fig. 226 Intersect action button

To use the Intersect command

1. Select the objects you want to combine with the Intersect command. The objects must overlap.
2. Choose Intersect in the Combine submenu (or click the Intersect action button). DENEBACAD finds the overlapping area of the shapes and removes the rest.

To reshape combined objects

You can use the Reshape command in the Edit menu to edit objects combined with the Intersect command.

1. Select the combined objects you want to reshape.
2. Choose Reshape in the Edit menu (or click the Reshape action button). The original shapes of the constituent objects that comprise the intersected Boolean object reappear.
3. Use the Selection tool to move, resize, and reshape the objects. If you have the Preference "Show boolean outlines" on, the shape of the intersected object will be visible as you reshape its constituent objects.
4. When you are done editing the shapes, click outside the selection to reapply the Intersect operation and end Reshape mode.

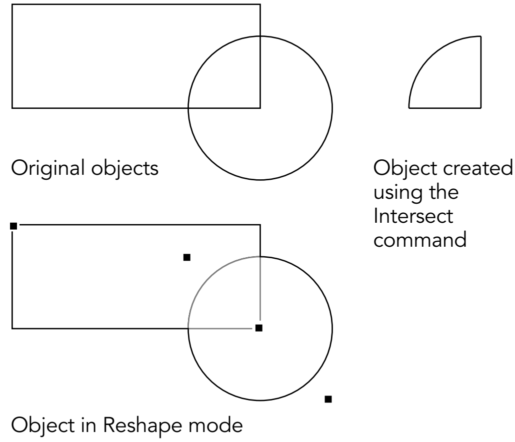


Fig. 227 Reshape applied to combined objects

Intersecting when objects have different attributes

If you intersect objects with different attributes, such as Fill colors, hatches, or Surface materials, the new object takes on the attributes of the rearmost object in the stacking order of the objects being combined. When two objects overlap, you can tell which object is rearmost in the stacking order because objects in the "front" obscure objects in the "back."

Intersecting objects on different layers

If you intersect objects on different layers, DENEBACAD places the intersected object on the currently active layer. For example, if you intersect an object on Layer 1 and an object on Layer 3, but Layer 2 is the active layer, the intersected object is placed on Layer 2. In addition, if the objects you're intersecting have different attributes, the intersected object takes on the attributes of the object on the currently active layer.

PUNCH FROM BACK

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+=
Windows Shortcut: Ctrl+=

By using the Punch from Back command, you can take an object at the front of the stacking order and use it to “knock out” (or punch) a hole in all of the overlapping objects behind it.

When you use the Punch from Back command on Sculpt mode objects, it’s often easier to see the results of the operation in Render mode.

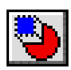
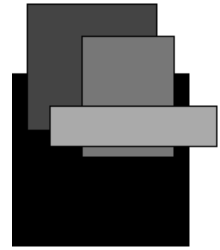


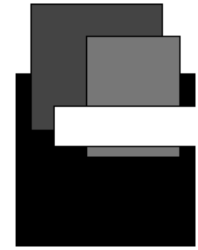
Fig. 228 Punch from Back action button

To avoid confusion, remember the Punch from Back command does not refer to Back view. It refers to taking an object at the front of the object creation order and using it to knock a hole through objects behind it.

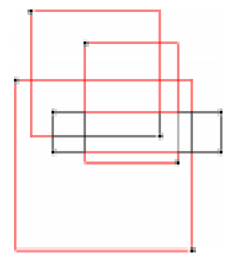
If necessary, you can use the Reshape command to reshape objects formed with the Punch from Back command.



Original objects



Result of Punch from Bac



Punched objects shown in Reshape mode

To use the Punch from Back command

1. Make sure the object you want to use to punch through objects is at the front of the object hierarchy. If necessary, select the object and choose Bring to Front in the Arrange submenu to bring it to the front of the hierarchy.
2. Select the objects you want to combine with the Punch from Back command.

   ▲ All overlapping selected objects will be affected.

   ▲ Non-overlapping objects won’t be affected by the punch operation.
3. Choose Punch from Back in the Combine submenu. DENEBACAD removes the front object from each of the objects behind it.

To reshape combined objects

You can use the Reshape command in the Edit menu to edit shapes created with the Punch from Back command.

1. Select the punched shape you want to edit.
2. Choose Reshape in the Edit menu or click the Reshape action button. The original front object removed from the back objects reappears.
3. Use the Selection tool to move, resize, and reshape the objects.

   ▲ If “Show boolean outlines” is selected in the Preferences dialog box, the shape of the combined object remains visible as you reshape its constituent objects.
4. When you are done editing the shapes, click outside the selection to reapply the combine operation and end Reshape mode.

Applying Punch from Back to objects on different layers

If you combine objects on different layers with the Punch from Back command, DENEBACAD places the combined object on the currently active layer. For example, if you punch an object on Layer 3 through an object on Layer 2, and Layer 2 is the active layer, the new object is placed on Layer 2.

PUNCH FROM BACK & TRIM

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+=
Windows Shortcut: Ctrl+Shift+=

By using the Punch from Back & Trim command, you can use an object in the front of the stacking order to “knock out” a hole in all of the overlapping objects behind it.

When you use Punch From Back & Trim, DENEBACAD doesn't recall the original objects when you put the resulting object in Reshape mode, because non-overlapping areas of the front object are “trimmed” to fit the shape of the punched object. There is no difference between “Punch from Back” and “Punch from Back & Trim” when no part of the front object extends past the outlines of the back objects.

Punch from Back & Trim is available in Draft and Sculpt only. However, when you use this command on Sculpt mode objects, it’s often easier to see the results of the operation in Render.

To avoid confusion, remember the Punch from Back & Trim command does not refer to Back view. It refers to taking the object at the front of the stacking order, and using it to knock holes in objects behind it.

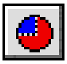


Fig. 229 Punch from Back & Trim action button

To use the Punch from Back & Trim command

1. Make sure the object you want to use to punch through objects is in front. If necessary, select the object and choose Bring to Front in the Arrange submenu.
2. Select the objects you want to combine with the Punch from Back & Trim command. All selected objects directly behind the front object will be affected. Non-overlapping objects stay the same.
3. Choose Punch from Back & Trim in the Combine submenu. DENEBACAD removes the front object from each of the objects behind it.

To edit combined objects

In Reshape mode, you can see how this command "trims" areas of objects that don't overlap the back object

1. Select an object to which you applied the "Punch from Back & Trim" command.
2. Choose Reshape in the Edit menu, or click the Reshape action button.
3. Use the Selection tool to move, resize, and reshape the objects.
4. When you are done editing the shapes, click outside the selection to reapply the "Punch from Back & Trim" operation and end Reshape mode.

Applying Punch From Back & Trim to objects on different layers

If you combine objects on different layers with the Punch from Back & Trim command, DENEBACAD places the combined object on the currently active layer. For example, if you punch an object on Layer 3 through an object on Layer 2, and Layer 2 is the active layer in the Layer Manager, the new object is placed on Layer 2.

PUNCH FROM FRONT

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+-
Windows Shortcut: Ctrl+-

The Punch from Front command knocks holes in the object at the back of the stacking order using the objects in front. Unlike Punch from Back, which uses the front object to punch one hole through all objects behind it, Punch from Front can cut several holes from the back object.

When you use the Punch from Front command on Sculpt mode objects, it's often easier to see the results of the operation in Render mode.

To avoid confusion, remember the Punch from Front command has no relation to Front view. Rather, it refers to taking objects at the front of the object creation order and using them to knock holes through the object at the back of it.

If necessary, you can use the Reshape command to reshape objects formed with the Punch from Front command.

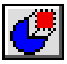


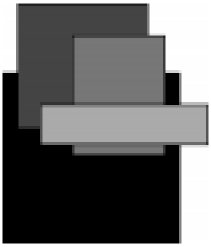
Fig. 230 Punch from Front action button

To use the Punch from Front command

1. Make sure the object you want to punch holes is at the back of the stacking order. If necessary, select the object and choose Send to Back in the Arrange submenu.
2. Select the objects you want to combine. All selected objects in front will punch holes in the back object. If a front object doesn't overlap the back object, the object will disappear.
3. Choose Punch from Front in the Combine submenu. DENEBACAD removes the front objects from the back object.

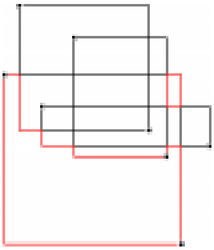
To reshape combined objects

You can use the Reshape command in the Edit menu to edit shapes created with the Punch from Front command.



Original objects

Result of Punch from Front



Punched object shown in Reshape mode

1. Select the object you punched holes in.
2. Choose Reshape in the Edit menu, or click the Reshape action button. The original shapes that punched the holes in the back object reappear.
3. Use the Selection tool to move, resize, and reshape the objects.
4. When you are done editing the shapes, click outside the selection to reapply the "Punch from Front" operation and end Reshape mode.

PUNCH FROM FRONT & TRIM

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+-
Windows Shortcut: Ctrl+Shift+-

The "Punch from Front & Trim" command knocks holes in the object at the back of the stacking order using the objects in the front, similar to "Punch from Front." However, unlike "Punch from Front," areas that don't overlap the back object are permanently deleted, and cannot be recalled in Reshape mode.

If all objects in front are completely overlapped by the back object, there is no difference in between "Punch from Front & Trim" and "Punch from Front."

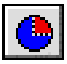


Fig. 231 Punch from Front & Trim action button

Punch from Front & Trim is available in Draft and Sculpt modes only. However, when you use this command on Sculpt mode objects, it's often easier to see the results of the operation in Render mode.

To avoid confusion, remember the Punch from Front & Trim command does not refer to Front view. Rather, it refers to taking objects at the front of the object creation order and using them to knock holes through the object in the back of it.

To use the Punch from Front & Trim command

1. Make sure the object you want to punch holes in is behind the objects that will make the holes. If necessary, select the object and choose Send to Back in the Arrange submenu.
2. Select the objects you want to combine. If a front object doesn't overlap the back object, the object will disappear.
3. Choose Punch from Front & Trim in the Combine submenu. DENEBACAD removes the front objects from the back object.

To edit combined objects

In Reshape mode, you can see how this command "trims" areas of objects that don't overlap the back object.

1. Select an object to which you applied the "Punch from Front & Trim" command.
2. Choose Reshape in the Edit menu, or click the Reshape action button.
3. Use the Selection tool to move, resize, and reshape the objects.
4. When you are done editing the shapes, click outside the selection to reapply the "Punch from Front & Trim" operation and end Reshape mode.

Applying Punch From Front & Trim to objects on different layers

If you combine objects on different layers with the Punch from Front & Trim command, DENEBACAD places the combined object on the currently active layer. For example, if you punch an object on Layer 3 through an object on Layer 2, and Layer 2 is the active layer in the Layer Manager, the new object is placed on Layer 2.

UNITE

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+U
Windows Shortcut: Ctrl+U

By choosing the Unite command, you can create a new object from the outline of multiple selected objects. The Unite command is available in Draft and Sculpt modes only.

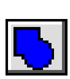


Fig. 232 Unite action button

The Unite command provides a simple and powerful way to create complex shapes. After uniting objects, you can move, resize, edit, and offset the resulting single object.

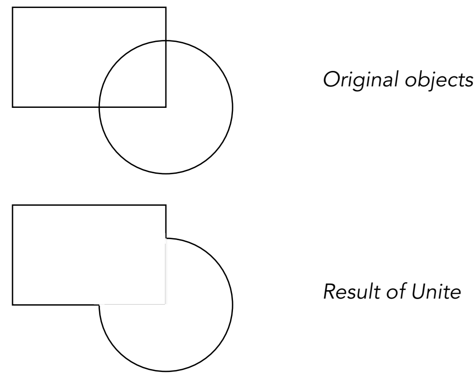


Fig. 233 The Unite command

To use the Unite command

1. Select the objects you want to combine. The objects must overlap.
2. Choose Object > Combine > Unite (or click the Unite action button). DENEBACAD finds the union of the shapes and removes interior lines.

To reshape combined objects

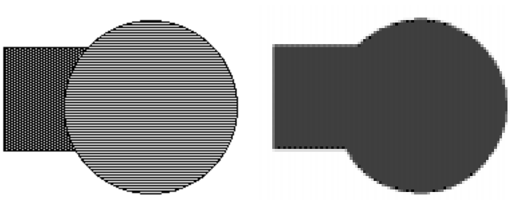
You can use the Reshape command to edit objects combined with the Unite command.

1. Select the combined object you want to reshape.
2. Choose Edit > Reshape (or click the Reshape action button). The original objects that comprise the united object appear.
3. Use the Selection tool to move, resize, and reshape the objects. If the Preference "Show boolean outlines" is selected, the combined object is visible as you reshape its constituent objects.

4. When you finish editing, click outside the selection to end Reshape mode.

Uniting objects with different attributes

If you unite objects on the same layer with different attributes, such as fill colors, hatches, or materials, the new object takes on the attributes of the rearmost object in the stacking order. When two objects overlap, you can tell which object is rearmost in the stacking order because objects in the "front" obscure objects in the "back."



Original objects *Combined objects*

A combined object (right) takes the attributes of the back object

Uniting objects on different layers

If you unite objects on different layers, DENEBACAD places the united object on the currently active layer. For example, if you unite an object on Layer 1 and an object on Layer 3, but Layer 2 is the active layer in the Layer Manager, the united object is placed on Layer 2. In addition, if the objects you're uniting have different attributes, the united object takes on the attributes of the object on the currently active layer.

Offsetting complex shapes

When you want to offset a complex shape, keep in mind that shapes created with the Double Polyline tool combine differently than single-line shapes. Although a double polyline acts like a line, it behaves like a polygon when combined with other objects.

Rather than creating objects with the Double Polyline tool, and then combining them with the Unite command, it's easier to combine polyline objects, and then offset the combined object.

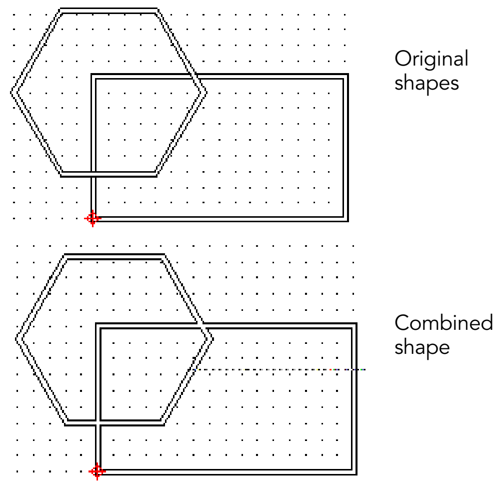


Fig. 234 Combining double polyline objects

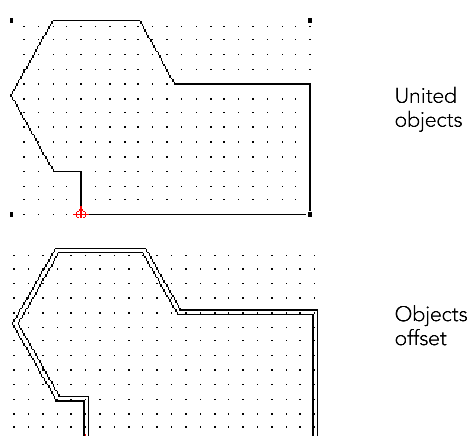


Fig. 235 Offsetting combined polyline objects

COPY TO LAYER

Mode: Draft, Sculpt

By using the Copy To Layer command in the Object menu, you can copy objects from one layer to another. This command duplicates selected objects and places them in the same position on the specified layer.

The Copy to Layer command does not use the system Clipboard and will not change the Clipboard contents.

To copy objects to another layer

1. Select the object or objects you want to copy to another layer.
2. In the Object menu, open the Copy To Layer submenu, and choose the layer to which you want to copy the selection. In the submenu, all the layers for the current view are listed. The active layer has a check mark next to it. You can copy objects to visible layers only. After you select a layer, DENEBACAD copies the selection and places the duplicate in the same position as the original.

Although you can copy an object to the same layer, the Duplicate command in the Edit menu is better for this purpose.

GROUP

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+G
Windows Shortcut: Ctrl+G

By using the Group command, you can unite separate objects that you want to keep together as a single unit. This command is available in Draft and Sculpt modes only.

You can group objects that are themselves part of a grouped object. You can group 2D objects with other 2D objects, and group 3D objects with other 3D objects. As long as the objects you want to group are visible in a Draft or Sculpt mode window, it doesn't matter whether you're working in Draft or Sculpt mode for purposes of grouping objects. However, you cannot group a 2D object with a 3D object.

If you group objects on the same layer, it might affect the stacking order of the objects on that layer. For example, suppose you have three overlapping 2D objects in Draft mode with different fill colors. If you group the front and back objects, the grouped object moves to the front of the stacking order, and the middle object moves to the back.

To use the Group command

1. Select the objects you want to group.
2. Choose Group in the Object menu, press Ctrl+G (Windows), or Cmd+G (Mac). DENEBACAD unites the selected objects

into one unit and replaces the bounding rectangles of the individual objects with a single bounding rectangle.

If you group objects on different layers, a message box appears to tell you that some objects will be moved to the active layer. Click OK to complete the operation and close the dialog box.

Grouping objects on different layers places the grouped object on the active layer in the Layer Manager.

EXTRUDE SUBMENU

Commands in the Extrude submenu in the Object menu affect the creation of 3D objects.

You can use the Linear, Spin, and Sweep commands to create 3D objects that are based on 2D objects you have drawn in Draft mode. These commands also have associated Action buttons. You can use the Action buttons to set the current extrusion method for drawing 3D objects directly in Sculpt mode, or for extruding 2D objects into 3D objects in Draft mode.

Also, in Draft mode, you can use the Action buttons to extrude selected 2D objects into 3D objects.

EXTRUDE OPTIONS

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+Shift+E
Windows Shortcut: Ctrl+Alt+Shift+E

You can use the Extrude Options command to access the Extrude options dialog box. Or, you can double-click one of the Extrude action buttons to access the Extrude Options dialog box.

Before you extrude a 2D object in Draft mode, or create a 3D object in Sculpt mode, you can choose the Extrude Options command to configure the extrusion settings.

The Extrude Options dialog box organizes settings on three tabs. To bring a tab to the front, click the tab.

- The tab marked with a Linear extrusion icon contains Linear extrusion settings.
- The tab marked with a Spin extrusion icon contains Spin extrusion settings.
- The tab marked with a Sweep extrusion icon contains Sweep extrusion settings.

To use the Extrude Options command

1. Choose the Extrude Options command in the Object menu. The Extrude Options dialog box appears.
2. Configure the settings in the dialog box. To change tabs, click the icon at the top of the tab to bring the tab to the front.
3. To make the new settings take effect, click OK.

Solid Faces

In the Extrude Options dialog box, the Extrusion Format buttons appear on each tab under the label Solid Faces; these buttons duplicate the operation of the Extrusion Format buttons on the Status bar.

When you activate the Extrusion Formats using the Extrude Options dialog box, you also activate the corresponding Extrusion Format buttons in the Status bar.

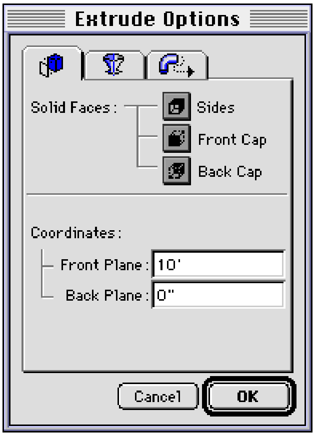
If you have the Status bar turned off, you can change the Extrusion Formats using the Extrude Options dialog box.

You can see these buttons on all three tabs in the Extrude Options dialog box. You can turn these buttons on and off when you're using the Linear or Sweep tabs. The extrusion format buttons also appear on the Spin tab, but they do not affect spin extrusions.

When you open a new DENEBACAD document, all three Extrusion Format buttons are turned on. This configuration produces solid objects. For more information on the extrusion format buttons, refer to "Extrusion formats" on page 80. To see an illustration of how the extrusion format buttons affect objects, see "Extrusion formats" on page 5.

LINEAR EXTRUSION TAB

A Linear extrusion icon appears at the top of the Linear extrusion tab.



Coordinates

You can change the coordinates of the default extrusion planes by entering new values in the Front Plane and Back Plane text boxes.

Front Plane refers to the first plane you set. Back Plane refers to the second plane you set.

When you open a new DENEBACAD document, the default extrusion planes are set to extrude objects from 0 to 10 feet.

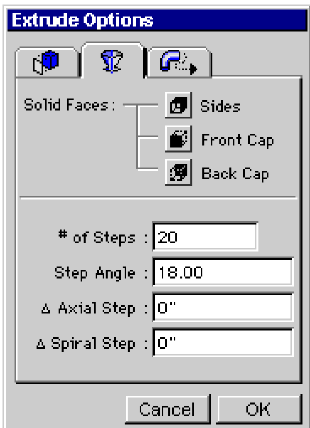
In metric units the default Front Plane is 3.048 meters and the default Back Plane is 0.000 meters.

When you enter new values in the Coordinates text boxes, you change the extrusion range of the default 3D extrusion planes for the current document only.

SPIN EXTRUSION TAB

A Spin extrusion icon appears at the top of the Spin extrusion tab.

Note: The extrusion format icons (Sides, Front Cap, and Back Cap) appear on this tab, but do not affect spin extrusions.



of Steps

You can enter a value in the # of Steps text box to specify the number of sides an object will have once you spin extrude it.

# of Steps works in conjunction with Step Angle to create complete or partial spins around the spin axis.

When you open the Extrude Options text box for the first time in a new document, the default value in the # of Steps text box is 20. This means the object will spin through 20 steps.

When you open the Extrude Options dialog box for the first time in a new document, the default value in the Step Angle text box is 18. This means each step will spin through 18-degrees.

With the # of Steps set to 20 and the Step angle set to 18, the object spins 360 degrees around the spin axis, and the completed object has 20 equal sides.

If you type 10 in the # of Steps text box, and enter 18 in the Step Angle text box, the completed object is extruded 180 degrees around the spin axis and has 10 equal sides.

Step Angle

You can enter a value in the Step Angle text box to specify the number of degrees an object will spin around the 3D axis.

Step Angle works in conjunction with # of Steps to create complete or partial spins around the spin extrusion axis.

You can determine if you are creating an object that completes a spin around the 3D axis by multiplying the step angle by the number of sides.

For example, if the Step Angle is 18 degrees and the # of Steps is 20, you know that the object spins around the axis 360 degrees.

If you change the Step Angle to nine degrees, and leave the number of sides at 20, the object spins 180 degrees.

Δ Axial Step

You can enter a value in the Δ Axial Step text box to specify the direction and offset of an object as it spins around and parallel to the spin axis.

When you enter a positive number in the Δ Axial Step text box, the object spins around the 3D axis and in the positive direction parallel to the spin axis.

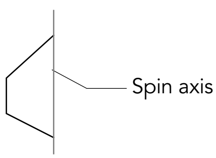
When you enter a negative number in the Δ Axial Step text box, the object spins around the spin axis and in the negative direction parallel to the spin axis.

Δ Spiral Step

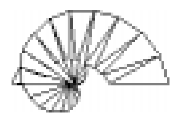
You can enter a value in the Δ Spiral Step text box to specify the spiral offset of an object from the spin axis.

When you enter a positive number in the Δ Spiral Step text box, the object spins out, or away, from the spin axis.

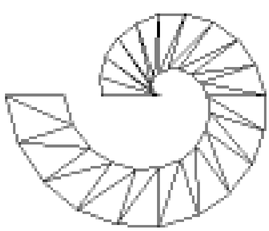
When you enter a negative number in the Δ Spiral Step text box, the object spins in, or towards the spin axis.



2D object shown in Draft mode: Front view



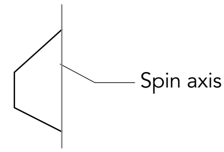
3D object created using a negative number for the Spiral Step. Shown in Sculpt mode: Top view



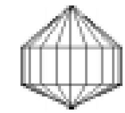
3D object created using a positive number for the Spiral Step. Shown in Sculpt mode: Top view

Fig. 236 Spiral steps in Spin extrusions

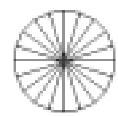
The following figures illustrate several spin extrusion configurations. They were created using the same Draft mode object and the same spin axis, but different spin extrusion configurations.



2D object shown in Draft mode: Front view



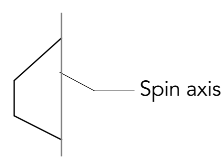
3D object shown in Sculpt mode: Front view



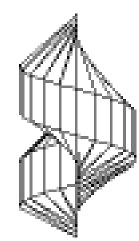
3D object shown in Sculpt mode: Top view

Spin extrusion configuration
# of Steps: 20
Step Angle: 18
Δ Axial Step: 0”
Δ Spiral Step: 0”

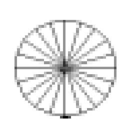
Fig. 237 Spin extrusion



2D object shown in Draft mode: Front view



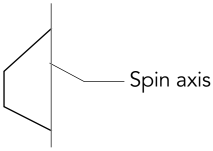
3D object shown in Sculpt mode: Front view



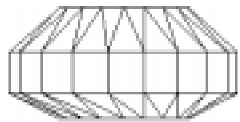
3D object shown in Sculpt mode: Top view

Spin extrusion configuration
# of Steps: 20
Step Angle: 18
Δ Axial Step:10”
Δ Spiral Step: 0”

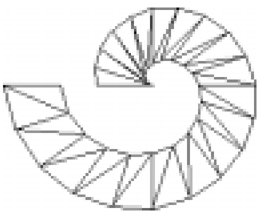
Fig. 238 Spin extrusion with Axial Step



2D object shown in Draft mode: Front view



3D object shown in Sculpt mode: Front view



3D object shown in Sculpt mode: Top view

Spin extrusion configuration

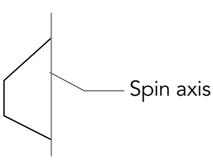
# of Steps: 20

Step Angle: 18

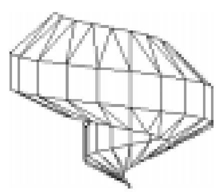
Δ Axial Step: 0”

Δ Spiral Step: 10”

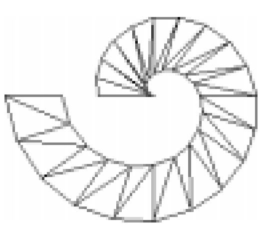
Fig. 239 Spin extrusion with Spiral Step



2D object shown in Draft mode: Front view



3D object shown in Sculpt mode: Front view



3D object shown in Sculpt mode: Top view

Spin extrusion configuration

# of Steps: 20

Step Angle: 18

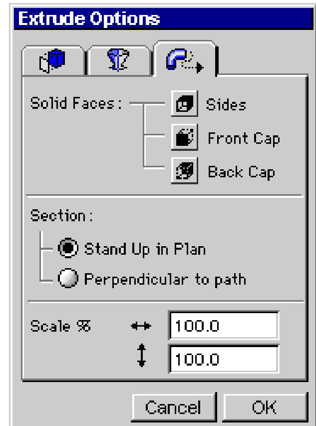
Δ Axial Step: 10”

Δ Spiral Step: 10”

Fig. 240 Spin extrusion with Axial and Spiral step

SWEEP EXTRUSION TAB

A Sweep extrusion icon appears at the top of the Sweep extrusion tab.



Section

You can choose the way DENEBACAD extrudes a sweep section by selecting Stand Up in Plan or Perpendicular to path.

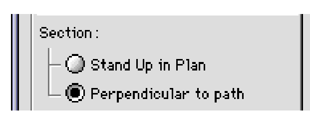


Fig. 241 Section buttons on the Sweep tab

Stand Up in Plan

Click the Stand Up in Plan button if you want the sweep section to be perpendicular to the plan.

When you use this option, the end caps of the extruded object are also perpendicular to the plan.

The plan in this case is defined as Top view. No matter which view you work in, a sweep extrusion extrudes the sweep section to be perpendicular to Top view.

Perpendicular to path

Click the Perpendicular to path button if you want the sweep section to be perpendicular to the path you use to create the extrusion.

When you use this option, the end caps of the extruded object are also perpendicular to the path.

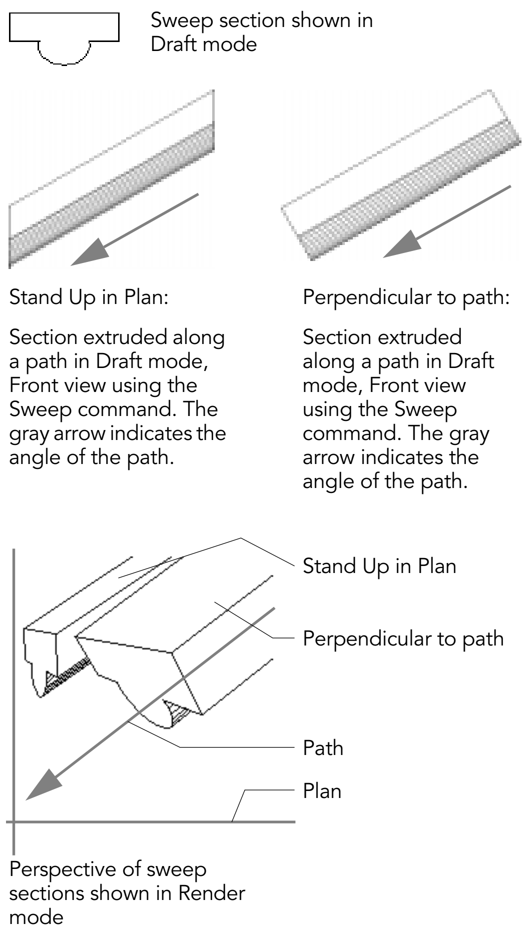


Fig. 242 Sweep extrusions options

Scale%

You can enlarge or reduce a sweep section, as it sweeps along a path, by using the Scale% option. If you change the percent values in the Scale% text boxes, you are changing the proportion of the second end cap of the extrusion as it relates to the first end cap.

For example, if you type 50 percent in the horizontal text box, and type 50 percent in the vertical text box, the extrusion proportionally reduces itself along the path by half. The edges of the finished extrusion's second end cap are 50 percent smaller than the first end cap's sides.

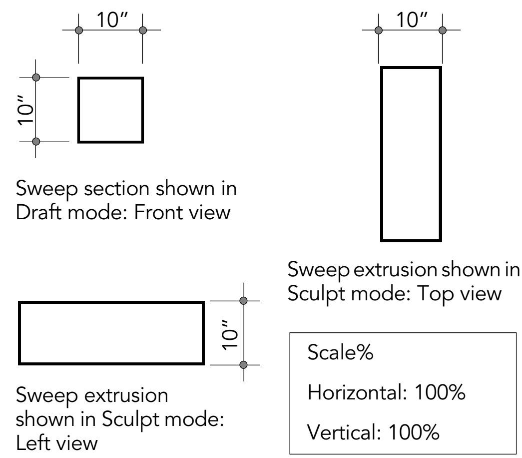


Fig. 243 Scale% option

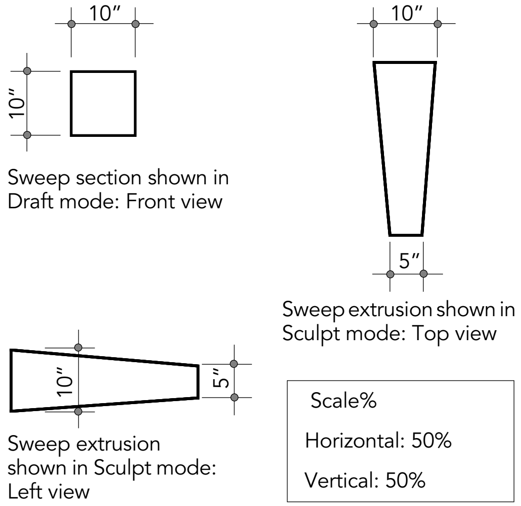


Fig. 244 Scale% option

LINEAR

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+E
Windows Shortcut: Ctrl+Shift+E

In Draft mode, you can create a 3D object by extruding a 2D object by choosing the Linear command.

In Sculpt mode, you can change the current extrusion method to the Linear extrusion method by choosing the Linear command.

In Draft mode The Linear command extrudes an object between the first and second planes that define the 3D extrusion.

To create a 3D object using the Linear extrusion method, you first draw an object, and then choose the Linear command to extrude it.

For example, to create a cube, you draw a rectangle, and then choose the Linear command to extrude the object into a cube.

In Sculpt mode You can choose the Linear command to change the current extrusion method to the Linear extrusion method. You do not have to use the command if the current extrusion method is Linear. The Extrude action button displays the Linear Extrude icon when Linear is the current extrusion method.

Once you set the extrusion method to Linear, you do not have to use the command again. Any object you create in Sculpt mode will be a Linear extrusion, until you choose another extrusion method.

To create a 3D object in Sculpt mode using the Linear extrusion method, you first choose the Linear command, if necessary, and then draw the object.

For example, to draw a cube, you choose the Linear command to change the current extrusion method to Linear before you draw the rectangle. Then, when you draw the rectangle, DENEBACAD extrudes it into a cube.

Action buttons In Draft and Sculpt mode, you can use the Linear Extrude action button to change the current extrusion method to the Linear extrusion method.

Additionally, in Draft mode, you can use the Action button to extrude 2D objects into 3D objects.

The Linear Extrude action button is located with the Extrude action buttons. To use the Extrude action buttons you must have the Action buttons turned on. You can turn on Action buttons by choosing the Toolbars command in the Layout menu.

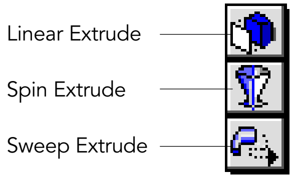


Fig. 245 The Extrude action buttons

To extrude objects in Draft mode

1. Open or activate a Draft mode window.

   ▲ Before you extrude an object, you might want to open and tile a Sculpt mode window also, so you can see the 3D object appear.

   ▲ You can use the Extrude Options command to change the extrusion configuration before you extrude the object.

2. Select the 2D object you want to extrude.

3. You can select a saved set of extrusion planes from the 3D Plane pop-up menu in the Status bar. Or, you can define a new set of planes for the extrusion using one of the 3D Plane commands in 3D Plane submenu in the Layout menu.

   ▲ If you do not specify the extrusion planes, DENEBACAD uses the active set of planes.

4. Choose the Linear command in the Extrude submenu. Or, click the Linear Extrude action button if it is visible. The 3D object is extruded.

   ▲ If the current extrusion method is not Linear, you can select the Linear Extrude action button from the Extrude action buttons. Note that when you select the Action button this way, you also change the current extrusion method to the Linear extrusion method. The Extrude action button displays the Linear Extrude icon.

   ▲ Choosing the Linear command does not change the current extrusion method.

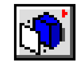


Fig. 246 Linear Extrude action button

To use the Linear extrusion method in Sculpt mode

1. Open or activate a Sculpt mode window.

2. If the current extrusion method is not Linear, choose the Linear command in the Extrude submenu. The Linear extrusion method is the current extrusion method. The Extrude action button displays the Linear Extrude icon.

   ▲ You can also select Linear Extrude from the Extrude action buttons.

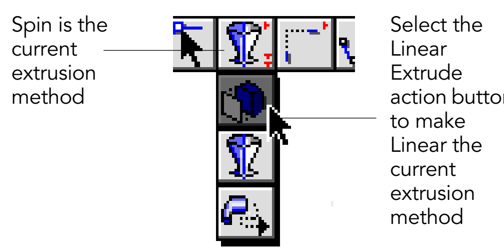


Fig. 247 Selecting the Linear Extrude action button

Using Action buttons to choose extrusion methods

1. Open or activate a Draft or Sculpt window.

2. Select the Linear Extrude action button from the Extrude action buttons. The Linear extrusion method is the current extrusion method. The Extrude action button displays the Linear Extrude icon.

- ▲ In Draft mode, selecting the Linear Extrude action button, with an object selected, changes the current extrusion method to Linear and extrudes the object. Selecting the Linear Extrude action button with no object selected changes the extrusion method only.
- ▲ In Sculpt, selecting the Linear Extrude action button, with an object selected, changes the current extrusion method to Linear but does not affect the object.

SPIN

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+C
Windows Shortcut: Ctrl+Shift+C

In Draft mode, you can create a 3D object by extruding a 2D object by choosing the Spin command.

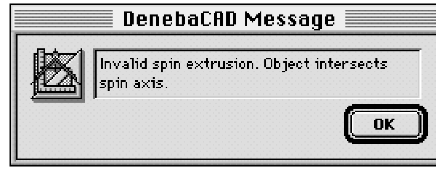
In Sculpt mode, you can change the current extrusion method to the Spin extrusion method by choosing the Spin command.

In Draft mode The Spin command creates a 3D object by rotating a selected object in steps around a spin extrusion axis.

To create a 3D object using the Spin extrusion method, you draw the cross section of an object in 2D and then choose the Spin command to extrude it.

For example, to create a sphere, you draw an arc so that its endpoints touch the spin extrusion axis, and then choose the Spin command to extrude the object into a sphere.

You can't use the Spin command to extrude objects that overlap, cross, or intersect a spin extrusion axis. If you attempt to extrude an object that overlaps a spin extrusion axis, DENEBACAD displays a message:



Click OK to close the dialog, and then draw an object that doesn't cross the spin extrusion axis. For more information on spin extrusion axes, refer to the "3D Axis submenu" on page 251.

In Sculpt mode You can choose the Spin command to change the current extrusion method to the Spin extrusion method. You do not have to use the command if the current extrusion method is Spin. The Extrude action button displays the Spin Extrude icon when Spin is the current extrusion method.

Once you set the extrusion method to Spin, you do not have to use the command again. Any object you create in Sculpt mode will be a Spin extrusion, until you choose another extrusion method.

To create a 3D object in Sculpt mode using the Spin extrusion method, you first choose the Spin command, if necessary, and then draw the object.

For example, to draw a sphere, you choose the Spin command to change the current extrusion method to Spin before you draw the arc. Then, when you draw the arc, DENEBACAD extrudes it into a sphere.

Action buttons In Draft and Sculpt mode, you can use the Spin Extrude action button to change the current extrusion method to the Spin extrusion method.

Additionally, in Draft mode, you can use the Action button to extrude 2D objects into 3D objects.

The Spin Extrude action button is located with the Extrude action buttons. To use the Extrude action buttons you must have the Action buttons turned on. You can turn on Action buttons by choosing the Toolbars command in the Layout menu.

To extrude objects in Draft mode

1. Open or activate a Draft mode window.

   ▲ Before you extrude an object, you might want to open and tile a Sculpt mode window also, so you can see the 3D object appear.

   ▲ You can use the Extrude Options command to change the extrusion configuration before you extrude the object.

2. Select the 2D object you want to extrude.

3. You can select a saved spin extrusion axis from the 3D Axis pop-up menu in the Status bar. Or, you can define a new axis for the extrusion using one of the 3D Axis commands in the 3D Axis submenu in the Layout menu.

   ▲ If you want to define a spin extrusion axis using one of the 3D Axis commands, the Spin extrusion method must be the current extrusion method. If it is not the current method, select the Spin Extrude action button to change the current extrusion method to Spin.

   ▲ If you do not select a spin extrusion axis, the object is extruded around the active axis.

4. Choose the Spin command in the Extrude submenu. Or, click the Spin Extrude action button if it is visible. The 3D object is extruded.

   ▲ If the current extrusion method is not Spin, you can select the Spin Extrude action button from the Extrude action buttons. Note that when you select the Action button this way, you also change the current extrusion method to the Spin extrusion method. The Extrude action button displays the Spin Extrude icon.

   ▲ Choosing the Spin command does not change the current extrusion method.

Fig. 248 Spin Extrude action button

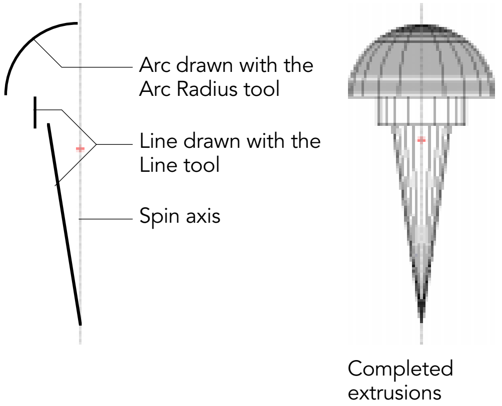


Fig. 249 Examples of Spin extrusions in Sculpt mode

To use the Spin extrusion method in Sculpt mode

1. Open or activate a Sculpt mode window.

2. If the current extrusion method is not Spin, choose the Spin command in the Extrude submenu. The Spin extrusion method is the current extrusion method. The Extrude action button displays the Spin Extrude icon.

   ▲ You can also select the Spin Extrude action button from the Extrude action buttons.

Using Action buttons to choose extrusion methods

1. Open or activate a Draft or Sculpt mode window.
2. Select the Spin Extrude action button from the Extrude action buttons. The Spin extrusion method is the current extrusion method. The Extrude action button displays the Spin Extrude icon.

   ▲ In Draft mode, selecting the Spin Extrude action button, with an object selected, changes the current extrusion method to Spin and extrudes the object. Selecting the Spin Extrude action button with no object selected changes the extrusion method only.

   ▲ In Sculpt mode, selecting the Spin Extrude action button, with an object selected, changes the current extrusion method to Spin but does not affect the object.

SWEEP

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+B
Windows Shortcut: Ctrl+Alt+B

In Draft mode, you can create a 3D object by extruding a 2D object by choosing the Sweep command.

In Sculpt mode, you can change the current extrusion method to the Sweep extrusion method by choosing the Sweep command.

In Draft mode The Sweep command extrudes a predefined cross section of an object along a path. You can refer to "Creating a sweep section" under the Sweep Sections command on page 297 for more information on how to create a section.

To create a 3D object using the Sweep extrusion method, you first draw an object, the path, and then choose the Sweep Extrude command to extrude the section along the path.

Running base moulding around a room is a good example of how to use a Sweep extrusion. A cross section of a base moulding can be extruded along the perimeter of a room with the Sweep command.

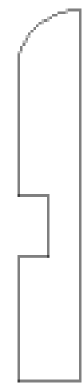


Fig. 250 Cross section of a base moulding

In Sculpt mode You can choose the Sweep command to change the current extrusion method to the Sweep extrusion method. You do not have to use the command if the current extrusion method is Sweep. The Extrude action button displays the Sweep Extrude icon when Sweep is the current extrusion method.

Once you set the extrusion method to Sweep, you do not have to use the command again. Any object you create in Sculpt mode will be a Sweep extrusion, until you choose another extrusion method.

To create a 3D object in Sculpt mode using the Sweep extrusion method, you first choose the Sweep command, if necessary, and then draw the object.

For example, to draw the base moulding, you choose the Sweep command to change the current extrusion method to Sweep before you draw the object that will be the Sweep path. Then, when you draw the object, DENEBACAD extrudes it into the base moulding.

Action buttons In Draft and Sculpt mode, you can use the Sweep Extrude action button to change the current extrusion method to the Sweep extrusion method.

Additionally, in Draft mode, you can use the Action button to extrude 2D objects into 3D objects.

The Sweep Extrude action button is located with the Extrude action buttons. To use the Extrude action buttons you must have the Action buttons turned on. You can turn on Action buttons by choosing the Toolbars command in the Layout menu.

To extrude objects in Draft mode

1. Make sure you are working in the document that contains the section you want to extrude.

2. Open or activate a Draft mode window.

▲ Before you extrude an object, you might want to open and tile a Sculpt mode window also, so you can see the 3D object appear.

▲ You can use the Extrude Options command to change the extrusion configuration before you extrude the object.

3. You can select a saved set of extrusion planes from the 3D Plane pop-up menu in the Status bar. Or, you can define a new set of planes for the extrusion using one of the 3D Plane commands in 3D Plane submenu in the Layout menu.

▲ If you do not select extrusion planes, DENEBACAD uses the active set of planes, and extrudes the section on the lower active plane.

4. You can select the section you want to extrude using the Sweep Sections command in the Extrude submenu. If you do not specify a section, DENEBACAD extrudes the current section.

▲ Refer to "Sweep Sections" on page 296 for information on how to use the Sweep Sections command.

5. Select or create an object you want the section to sweep along. In other words, create a path for the section to follow.

▲ An example of this would be to select a wall you to which you want to apply a base moulding. The extrusion will follow the outline of the wall.

▲ When you extrude a section, DENEBACAD places the new 3D object on the lower extrusion plane. If the Info bar is visible, you can determine which is the lower extrusion plane by looking at the values in the text boxes at the far right of the Info bar. The smaller number indicates which plane is the lower extrusion plane.

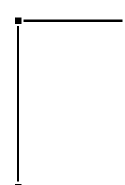


Fig. 251 Selected object for a sweep path

6. Choose the Sweep command in the Extrude submenu. Or, click the Sweep Extrude action button if it is visible. The currently selected section is extruded along the path.

▲ If you clicked the Sweep Extrude action button, the Sweep Section palette opens at the same time the object is extruded.

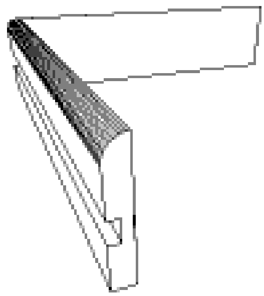


Fig. 252 Rendering of extruded section

▲ If the current extrusion method is not Sweep, you can select the Sweep Extrude action button from the Extrude action buttons. Note that when you select the Action button this way, you also change the current extrusion method to the Sweep extrusion method. The Extrude action button displays the Sweep Extrude icon. Also, when you select the Action button, the Sweep Sections palette opens.

▲ Choosing the Sweep command does not change the current extrusion method.

Fig. 253 Sweep Extrude button

To use the Sweep extrusion method in Sculpt mode

1. Open or activate a Sculpt mode window.
2. If the current extrusion method is not Sweep, choose the Sweep command in the Extrude submenu. The Sweep extrusion method is the current extrusion method. The Extrude action button displays the Sweep Extrude icon and the Sweep Sections palette opens.

▲ You can also select the Sweep Extrude action button from the Extrude action buttons.

Using Action buttons to choose extrusion methods

1. Open or activate a Draft or Sculpt mode window.
2. Select the Sweep Extrude action button from the Extrude action buttons. The Sweep extrusion method is the current method. The Extrude action button displays the Sweep Extrude icon and the Sweep Sections palette opens.

▲ In Draft mode, selecting the Sweep Extrude action button, with an object selected, changes the current extrusion method to Sweep and extrudes the object. Selecting the Sweep Extrude action button with no object selected changes the extrusion method only.

▲ In Sculpt mode, selecting the Sweep Extrude action button, with an object selected, changes the current extrusion method to Sweep but does not affect the object.

SWEEP SECTIONS

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+Shift+B
Windows Shortcut: Ctrl+Alt+Shift+B

The Sweep Sections command opens the Sweep Sections palette. In the palette, you can select from saved sections you want to extrude along a path.

You must work in the document that contains the original section object you want to extrude.

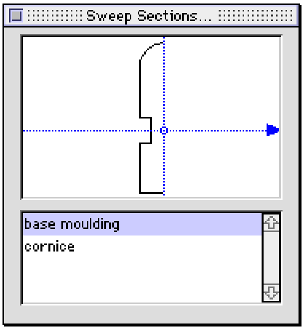
If you edit the original section object, the Sweep Sections palette reflects the revision.

If you delete the original section object, it disappears from the Sweep Sections palette.

To use the Sweep Sections command

1. Make sure you are working in the document that contains the section you want to view.
2. Open or activate a Draft or Sculpt mode window.
3. Choose the Sweep Sections command in the Extrude submenu. The Sweep Sections palette opens. The Sweep Sections palette displays the available sweep sections.

   ▲ If you have not created any sweep sections, the palette is empty.

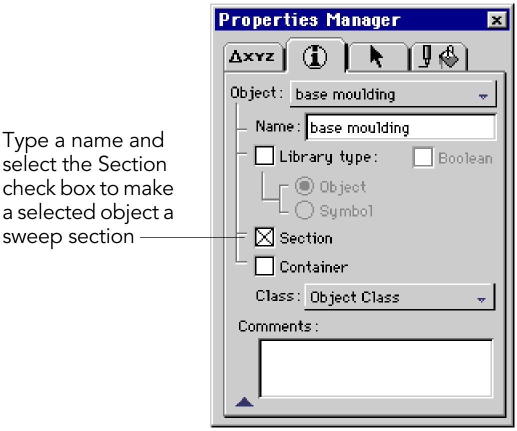


Creating a sweep section

1. Open or activate a Draft mode window. Set the window to the view you want to work in.

   ▲ Because you can create a sweep section in one view and use it another, you can work in any view when you create the sweep section.

2. Create an object using any of the object drawing tools in the toolbox. You can also create more than one object and group them.
3. Double-click the object, or group of objects. The Properties Manager appears. Click the Information tab to bring it to the front.



4. Type a name for the section in the Name text box. Click the Section checkbox. The object is now a section. You can close the Properties Manager.
5. To change the location of the insertion point of the new section, select the original object and use the Reshape command.

   ▲ It is important to know the location of the section's insertion point. The section sweeps along the path and is placed, on the lower active plane, at its insertion point.

PATH SUBMENU

The Path submenu in the Object menu contains commands that let you change the shape or structure of an object's outline, or *path*.

You can use the Path submenu to create an offset path, smooth or unsmooth a path, and convert an object to a polygon.

CONVERT TO POLYGON

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+P
Windows Shortcut: Ctrl+Shift+P

In DENEBACAD, objects such as ovals, rectangles, and regular polygons have special attributes and editing features. For example, in Reshape mode, a regular polygon has two handles that let you specify the size and rotation of the polygon. Regardless how you move these handles, the polygon maintains its basic shape and proportions.

By using the Convert to Polygon command, you can distort, skew, or completely reshape a specialized object. This command removes an object's specialized editing features, and lets you freely edit the shape using its control points. For shapes with curves, the number of control points depends on the setting in the Preferences dialog box. More control points makes the curved shapes appear smoother.

The Convert to Polygon command is available in Draft and Sculpt modes only. However, you can use the Convert to Polygon command on Wireframe and Hidden Line renderings pasted into Draft mode, or if you use the Copy View to Draft command to copy a Hidden Line rendering into a Draft mode window.

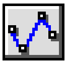


Fig. 254 Convert to Polygon action button

To use the Convert to Polygon command

1. Select the object or objects you want to convert to polygons.
2. Choose Convert To Polygon in the Path submenu.

To see the difference between a regular object in reshape mode and a polygon, draw an oval, select it, then choose Reshape in the Edit menu. Notice it has two handles that let you control the size of the oval. While still in Reshape mode with the oval selected, choose Convert To Polygon in the Path submenu. The oval now has several control points; when you drag a point, it distorts the shape in a way you can control.

OFFSET

Mode: Draft
Mac OS Shortcut: Cmd+Shift+O
Windows Shortcut: Ctrl+Shift+O

In Draft mode, you can change any line or polygon to a parallel polyline using the Offset command in the Path submenu, or the Offset Path action button. This feature is especially useful for creating walls of a specific thickness from polygons and other shapes. The Offset command is not available in Sculpt and Render modes.

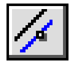


Fig. 255 Offset Path action button

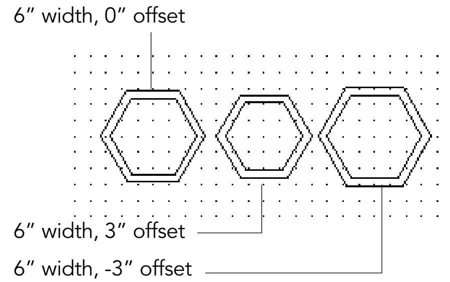


Fig. 256 Offset shapes

To use the Offset command

Use this procedure to apply or change the offset for any object.

1. Select the objects you want to offset. You can select objects that already have offset paths.
2. Click the Offset Path action button, or choose Offset in the Path submenu. The Offset dialog box appears.
3. In the Width text box, type the amount of space you want between the parallel polylines. For drawings of walls, this value equals the thickness of the walls.
4. In the Offset text box, type an offset amount. To make the offset path appear to the right of the original path, enter a positive number. To make the offset path appear to the left of the original path, enter a negative number

   ▲ For example, to create an 4-inch thick wall offset to the right of the original path, type 4" in the Width text box and 8" in the Offset text box.

5. To offset the path, click OK

ROUND

Mode: Draft
Mac OS Shortcut: Cmd+Shift+D
Windows Shortcut: Ctrl+Shift+D

By using the Round command, you can change an object's sharp corners to rounded corners with a specific radius. This command is available in Draft mode only.

You can apply the Round command to any object with at least one vertex, including Wireframe and Hidden Line renderings pasted into a Draft mode window, or brought into Draft mode using the Copy View to Draft submenu. When using the Round command, DENEBACAD rounds only an object's corners. The radius is in the current units in the Units Setup dialog box.

Although you can round objects with smooth corners, the rounding effect is more pronounced on object's with sharp corners. Keep in mind that rounding an object automatically converts the shape to a polyline or polygon; this eliminates any special properties the object might have had in Reshape mode.

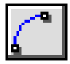
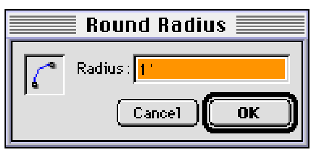


Fig. 257   Round action button

To use the Round command

1. Select the objects for which you want to smooth corners.
2. Click the Round action button, or choose Round in the Path submenu. The Round Radius dialog box opens.



3. In the dialog box, type a number to set the radius of the rounded corner. A larger radius creates more rounded corners. A smaller number creates less rounded corners.
4. Click OK to smooth corners of objects.

When rounding parallel polylines, keep in mind that DENEBACAD considers the end caps to be corners, and applies the radius to them.

Fig. 258   Rounded end caps and corners

To round corners but leave end caps flat, draw a polyline shape first, apply the Round command, then use the Offset command to add thickness.

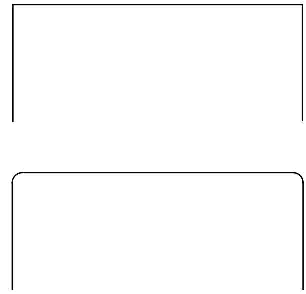


Fig. 259   Polyline shape with rounded corners

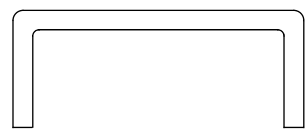


Fig. 260   Path offset

SMOOTH

Mode: Draft
Mac OS Shortcut: Cmd+Opt+Z
Windows Shortcut: Ctrl+Alt+Z

By using the Smooth command, you can change polygons and polylines to B-Spline curves. Unlike the Round command, the Smooth command changes all straight lines to curves. The Smooth command is available in Draft mode only.

In addition, the Smooth command will work on any object with at least one vertex. Although you can use the Smooth command on shapes that contain curves, the “smoothing” effect is more pronounced on shapes with sharp corners. Also, smoothing an object converts the shape to a polyline or polygon, removing any special properties the object has in Reshape mode.

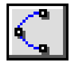


Fig. 261 Smooth action button

To create smooth objects

1. Select the object (or objects) you want to convert to B-Spline curves.
2. Click the Smooth action button, or choose Smooth in the Path submenu. DENEBACAD smooths the object.

   ▲ To create rounded parallel polylines, you should first work with basic, single-line shapes, smooth them, then offset them to create parallel polylines. Smoothing polylines might produce unexpected results.

UNSMOOTH

Mode: Draft,
Mac OS Shortcut: Cmd+Shift+Z
Windows Shortcut: Ctrl+Shift+Z

The Unsmooth command is used to convert B-Spline curves to polylines. Unsmooth can reverse the effects of applying the Smooth command on an object. This command is available in Draft mode only.

To unsmooth objects

1. Select the objects that you want to convert from B-Spline curves to polylines.
2. Choose Unsmooth in the Path submenu. DENEBACAD straightens all curves.

POSITION SUBMENU

The position submenu contains commands that let you reposition objects by rotating, mirroring, and arraying them. You can create duplicates of an object with Position submenu commands, or simply reposition the original selected objects.

LINEAR ARRAY

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+Shift+M
Windows Shortcut: Ctrl+Alt+Shift+M

By using the Linear Array command in the Position submenu, you can make multiple, evenly-spaced copies of a selection. You can decide how many copies to make and the distance between each copy. In addition, you can also specify the total area that you want the duplicates to fill, and DENEBACAD evenly distributes the objects inside this area.

The Linear Array can make many tasks easy. For example, you can place several windows along a wall, add trusses to a roof, and so on.

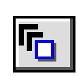


Fig. 262 Linear Array action button

To use the Linear Array command

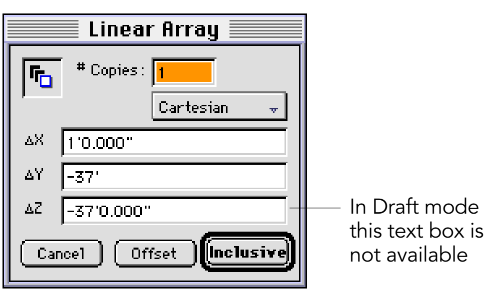
To make using this command easier, read and follow the hint lines in the Help bar.

1. Select the object or objects you want to copy and distribute.
2. Choose Linear Array in the Position submenu (or click the Linear Array action button).
3. Depending on how you want to distribute objects, do one of the following:

   ▲ To specify the distance between copies, drag from the selected objects in the direction you want to place the first copy; an outline of the selection follows the pointer. Release the mouse button when the outline is in the position you want.

   ▲ To specify the area that you want the copies to fill, drag from the selection in the direction you want to place the last copy; an outline of the selection follows the pointer. Release the mouse button when the outline is in the position you want.

The Linear Array dialog box appears.



4. In the Linear Array dialog box, type the number of copies you want to place in the "# of Copies" text box.
5. To change the coordinate system, choose an option in the pop-up menu.

   ▲ Depending on the coordinate system you're working in, the three text boxes in the Linear Array dialog box show the distance you dragged the mouse. These distances are shown in the values of the current coordinate system. For example, if you're working in a cartesian coordinate system, the distance that you dragged the mouse is shown in the delta x, y, and z text boxes.
6. To distribute the objects within the area you defined, click Inclusive. To separate each copy by the distance you defined, click Offset.

MIRROR

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+/
Windows Shortcut: Ctrl+/

The Mirror command in the Position submenu lets you flip selected objects 180 degrees around an axis you specify. You can set the axis at any angle and distance from the selected objects. In addition, you can set constraints for the axis using the Snaps pop-up menu.

The Mirror command is available in Draft and Sculpt modes only when you have objects selected.

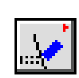
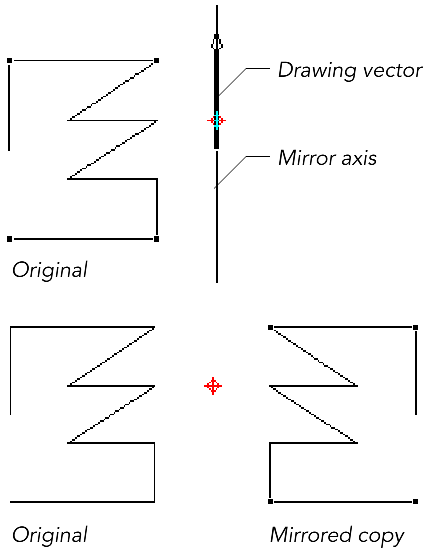


Fig. 263 Mirror action button



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

Fig. 264 Using the Mirror command

To use the Mirror command

1. Select the object (or objects) you want to flip.
2. Choose Mirror in the Position submenu (or click the Mirror action button). A new pointer followed by a vertical line appears. The line is the axis around which DENEBACAD will flip the object.
3. Click to anchor the axis at a point; you should try to anchor the axis at the point you want to mirror the object around. Depending on how you want to mirror the object, using the Snaps pop-up menu might make it easier to anchor the axis exactly where you want it. After you click to anchor the axis, the axis starts rotating.
4. Move the pointer where you want to place the second anchor for the axis. As you move the pointer, you can see the angle of the axis change. When the axis is in the position you want, click to set the second anchor point. DENEBACAD moves and flips the selected object around the axis.

Note: Working in a Polar coordinate system can make it easier to define precise angles with the pointer.

MIRROR A COPY

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+/
Windows Shortcut: Ctrl+Shift+/

The Mirror a Copy command in the Position sub-menu lets you flip a duplicate set of selected objects 180 degrees around an axis that you set. The axis can be at any angle and distance from the selected objects, and you can set constraints for the axis using the Snaps pop-up menu.

The Mirror a Copy command is available in Draft and Sculpt modes only and you must have an object selected to use this command.

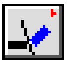


Fig. 265 Mirror a Copy action button

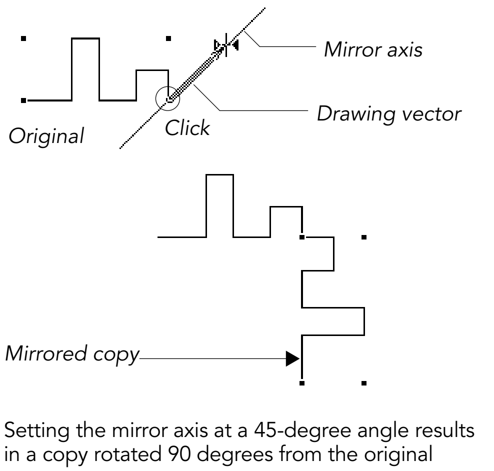


Fig. 266 Using the Mirror a Copy command

To use the Mirror a Copy command

To make using this command easier, refer to the hint line instructions on the Help bar.

1. Select the object or objects you want to flip and duplicate.
2. Choose Mirror a Copy in the Position submenu (or click the Mirror a Copy action button). A new pointer followed by a vertical line appears. The line is the axis around which DENEBACAD will flip the duplicate objects.
3. Click to anchor the axis at a point; you should try to anchor the axis at the exact point you want to flip the objects around. Depending on how you want to duplicate and mirror the objects, using the Snaps pop-up menu might make it easier to anchor the axis exactly where you want it. After you click to anchor the axis, the axis starts rotating at the point you click.
4. Move the pointer where you want to place the second anchor for the axis. As you move the pointer, you can see the angle of the axis change. When the axis is in the position you want, click to set the second anchor point. DENEBACAD duplicates the selected objects and flips them around the axis.

MOVE

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+M
Windows Shortcut: Ctrl+Shift+M

You can change the position of selected objects by using the Move command in the Position submenu (or the Move action button). The Move command is available in Draft and Sculpt modes only. You must have an object selected to use this command.

Although you can perform the same action using the Selection tool, the Move command is easier to use for large selections because it requires less precision with the mouse.

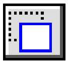


Fig. 267 Move action button

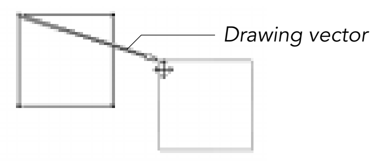
To use the Move command

When you use the Move command, you can get help by reading the hint line at the bottom of your screen.

1. Select the object (or objects) you want to move.
2. Choose Move in the Position submenu (or click the Move action button). A new pointer appears.

3. Click to set the first anchor point of the drawing vector you'll use to move the object. You don't have to click the object itself; you can set the first anchor point in any location in your drawing.

4. Drag to where you want to move the objects. The drawing vector extends from the anchor point and an outline of the selection follows the pointer. You don't have touch the selection with the pointer, although this might make it easier to see what you are doing. Release the mouse button where you want to place the objects



The drawing Vector shows the relative movement of the reference point

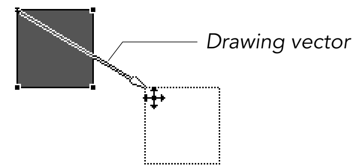
Fig. 268 Using the Move command

MOVE A COPY

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+M
Windows Shortcut: Ctrl+Alt+M

The Move a Copy command is used to place a copy of selected objects at a location you specify. The original objects stay in their positions.

The Move a Copy command is available in Draft and Sculpt mode only. To use this command, you must have objects selected.



The drawing Vector shows the relative movement of the reference point

Fig. 269 Using the Move a Copy command

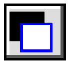


Fig. 270 Move a Copy action button

To use the Move A Copy command

1. Select the objects you want to move and copy.

2. Choose Move a Copy in the Position submenu (or click the Move a Copy action button). A new pointer appears.

3. Click to set the first anchor point of the drawing vector you'll use to move the copies. You don't have to click the objects you're copying themselves; you can set the point in any location in the drawing. Once you set the first anchor point, the drawing vector extends from this point, and outlines of the objects you're moving follow the pointer.

4. Drag in the direction you want to place the duplicates of the objects; outlines of the selection follow the pointer. Release the mouse button where you want to copy the objects.

POLAR ARRAY

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+Shift+R
Windows Shortcut: Ctrl+Alt+Shift+R

The Polar Array command lets you make evenly spaced copies of objects distributed radially from one another by a specific distance and angle. You can use this command in Draft and Sculpt modes.

When using the Polar Array command in Draft mode, the last text box in the Polar Array dialog box appears dimmed. In Sculpt mode, by contrast, this text box is active and it lets you enter a radial displacement for the radial copies of 3D objects along a third axis. This has several useful applications, such as making it easy to make a spiral staircase out of a simple Sculpt mode object.

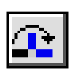


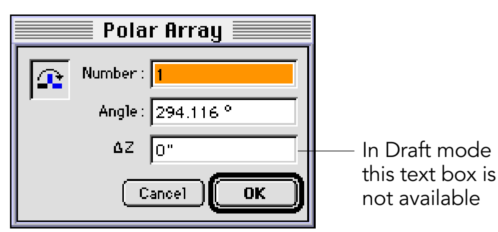
Fig. 271 Polar Array action button

To use the Polar Array command

1. Select the object or objects you want to copy and distribute radially.
2. Click the Polar Array action button, choose Polar Array in the Position submenu, press Ctrl+Alt+Shift+R (Windows), or Cmd+Opt+Shift+R. The pointer changes to a crosshair.
3. Click to set the center of rotation. This is often the corner of the object you're copying, or near the corner of an entire selection, but it can be anywhere in your document. After you set the center of rotation, the drawing vector extends from the center of rotation (the point you click).
4. Drag the drawing vector and then click. This sets the baseline of rotation. Don't confuse the baseline of rotation with the "baseline" of your object (or objects); they're not the same thing. After you define the baseline of rotation, a second drawing vector appears. You define the angle of rotation for your object by creating an angle between this drawing vector and the baseline of rotation.

Note: You can bypass defining a baseline of rotation by clicking the Polar Array action button, and then double-clicking on the rotation center with the rotation pointer. This displays the Polar Array dialog box. You can then enter information for the Polar Array immediately. If you do this, proceed to step 6 below.

5. To specify the angle of rotation, drag the drawing vector from the center of rotation to a new position in your document. This is the angle of rotation for your object: the rotation angle is defined as the angle between the rotation baseline and the point you click with the drawing vector.
6. After defining the angle of rotation, the Polar Array dialog box opens.



7. In the Number text box, enter the number of copies you want to spin radially around the center of rotation.

8. In the angle text box, enter the number of degrees you want to rotate each copy. The number of degrees you rotate each copy also sets the distance between the objects in the polar array.

   ▲ If you're working in Draft mode, click OK to create the polar array. Click Cancel to leave your object unchanged.

   ▲ If you're working in Sculpt mode you can displace copies of the objects in your polar array along the vertical axis (specified in the Units Setup dialog box). In the ΔX, ΔY, or ΔZ text box, enter the displacement along the vertical axis in the current units. Click OK to create the polar array. Click Cancel to leave your object unchanged.

Note: Depending on the current view, you'll see a ΔX, ΔY, or ΔZ text box on the Polar Array dialog box.

ROTATE

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+R
Windows Shortcut: Ctrl+Shift+R

By using the Rotate command, you can precisely rotate selected objects a specific number of degrees. You can use the Rotate command in Draft and Sculpt modes.

To use the Rotate command, you first set the center of rotation for your objects. You can then visually specify a baseline of rotation, and from this baseline, rotate your objects in 2D or 3D space. DENEBACAD then shows you the exact degrees of rotation for verification before completing the operation.

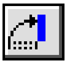


Fig. 272 Rotate action button

To use the Rotate command

To make using the Rotate command easier, follow the hint line instructions in the Help bar.

1. Select the object (or objects) you want to rotate.

2. Choose Rotate in the Position submenu (or click the Rotate action button). A new pointer appears.

3. Click to set the center of rotation. This is often the corner of the object you're rotating, or near the corner of an entire selection, but it can be anywhere in your document. After you set the center of rotation, the drawing vector extends from the center of rotation (the point you click).

4. Drag the drawing vector and then click. This sets the baseline of rotation. Don't confuse the baseline of rotation with the "baseline" of your object (or objects); they're not the same thing. After you define the baseline of rotation, a second drawing vector appears. You define the angle of rotation for your object by creating an angle between this drawing vector and the baseline of rotation.

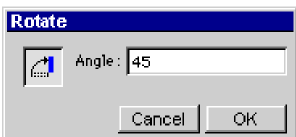
   ▲ You can bypass defining a baseline of rotation by clicking the Rotate action button, and then double-clicking on the rotation center with the rotation pointer. This displays the Rotate dialog box. You can then enter an angle of rotation immediately. If you do this, proceed to step 6 below.

5. To specify the angle of rotation, drag the drawing vector from the center of rotation to a new position in your document. This is the

angle of rotation for your object. Again, the rotation angle is defined as the angle between the rotation baseline and the point you click with the drawing vector.

▲ Working in a Polar coordinate system can make it easier to define precise angles with the drawing vector.

6. After defining the angle of rotation, the Rotate dialog box opens.



▲ In the dialog box, you can verify the exact number of the degrees of rotation. The number in the Angle text box shows you the degrees of rotation you defined relative to the baseline of rotation.

7. To change the degrees of rotation, type a new value in the Angle text box.

8. To rotate the selection, click OK. Click Cancel to leave the objects as they were.

ROTATE A COPY

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+R
Windows Shortcut: Ctrl+Alt+R

By using the Rotate A Copy command, you can precisely rotate copies of selected objects a specific number of degrees. You can use the Rotate A Copy command in Draft and Sculpt modes.

To use the Rotate A Copy command, you first set the center of rotation for your object. You can then visually specify a baseline of rotation, and from this baseline, rotate a copy of your object in 2D or 3D space. DENEBACAD then shows you the exact degrees of rotation for verification before completing the operation.

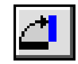


Fig. 273 Rotate A Copy action button

To use the Rotate a Copy command

To make using the Rotate A Copy command easier, follow the hint line instructions in the Help bar.

1. Select the object (or objects) you want to rotate.

2. Choose Rotate in the Position submenu (or click the Rotate action button). A new pointer appears.

3. Click to set the center of rotation. This is often the corner of the object you're rotating a copy of, or near the corner of an entire selection, but it can be anywhere in your document. After you set the center of rotation, the drawing vector extends from the center of rotation (the point you click).

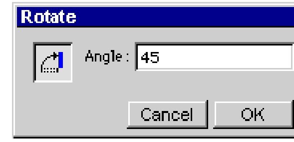
4. Drag the drawing vector and then click. This sets the baseline of rotation. Don't confuse the baseline of rotation with the "baseline" of your object (or objects); they're not the same thing. After you define the baseline of rotation, a second drawing vector appears. You define the angle of rotation for your object you're copying by creating an angle between this drawing vector and the baseline of rotation.

▲ You can bypass defining a baseline of rotation by clicking the Rotate A Copy action button, and then double-clicking on the rotation center with the rotation pointer. This displays the Rotate dialog box. You can then enter an angle of rotation immediately. If you do this, proceed to step 6 below.

5. To specify the angle of rotation, drag the drawing vector from the center of rotation to a new position in your document. This is the angle of rotation for your object. Again, the rotation angle is defined as the angle between the rotation baseline and the point you click with the drawing vector.

   ▲ Working in a Polar coordinate system can make it easier to define precise angles with the drawing vector.

6. After defining the angle of rotation, the Rotate dialog box opens.



7. In the dialog box, you can verify the exact number of the degrees of rotation. The number in the Angle text box shows you the degrees of rotation you defined relative to the baseline of rotation.

8. To change the degrees of rotation, type a new value in the Angle text box.

9. To rotate the selection, click OK. Click Cancel to leave the objects as they were.

SCALE

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+Y
Windows Shortcut: Ctrl+Shift+Y

You can change the size of selected objects relative to their current size by choosing the Scale command in the Object menu.

You can scale objects horizontally, vertically, or proportionally, based on a reference point you specify. This command is available in Draft and Sculpt modes only.

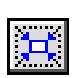


Fig. 274 Scale section button

To use the Scale command

1. Select the object or objects you want to scale.

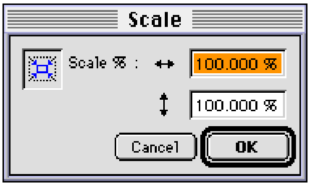
2. Choose Scale in the Object menu (or click the Scale action button). A new pointer appears.

3. Click in the document where you want to set the reference point for scaling. The Scale dialog box opens.

   ▲ In general, the reference point is a corner of the selection that you want to remain stationary as the rest of the selection resizes. However, the reference point can be anywhere in the document.

4. In the Scale dialog box, type percentages by which you want to scale the selection. You can type different values for horizontal and vertical scaling, or the same values to scale proportionally.

5. Click OK to scale the selection, or Cancel to close the dialog box without changing the selection.



SEND TO LAYER

Mode: Draft, Sculpt

By using the Send To Layer command in the Object menu, you can move objects from one layer to another. This command removes selected objects from the current layer and places them in the same position on a layer you choose.

Also, you can move one or more selected objects from one layer to another using the Layer Manager; see “To send objects to a layer” on page 367.

This operation does not use the system Clipboard and will not change the contents of the Clipboard.

To use the Send to Layer command

1. Select the object or objects you want to send to another layer.
2. In the Object menu, open the Send To Layer submenu and choose the layer to which you want to send the selection.

   ▲ In the submenu, all the layers for the current mode and view are listed. The active layer has a check mark next to it. You can send objects to visible layers only. After you select a layer, DENEBACAD removes the selection from the current layer and places it in the same position on the layer you choose.

TRIMS SUBMENU

The Trims submenu contains commands that let you alter the paths of objects by trimming segments, chamfering the intersection of two lines, extending separate objects to create an intersection, joining objects, and filleting objects.

CHAMFER

Mode: Draft
Mac OS Shortcut: Cmd+Shift+K
Windows Shortcut: Ctrl+Shift+K

The Chamfer command lets you connect two non-parallel lines by joining them together with an angled line. The objects are combined into one object.

The lines you want to chamfer do not have to touch, and you can chamfer lines that intersect.

You can chamfer the corners of rectangles drawn with the Rectangle Diagonal, Rectangle Center to Corner, or Rectangle 3 Points tool.

When you select the Chamfer command, the Chamfer dialog box opens. You configure the chamfer settings using this dialog box.

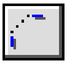
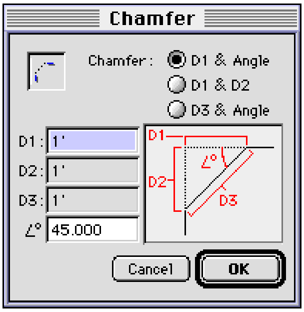


Fig. 275 Chamfer action button

To use the Chamfer command

1. Choose the Chamfer command in the Trims submenu. You can also use the Chamfer action button in the Attributes bar. The Chamfer pointer appears.
2. Click the first line you want to chamfer.
3. Move the pointer to the second line you want to chamfer. The drawing vector follows from the first line to the second line.
4. Click the second line. The Chamfer dialog box appears.



5. Select a Chamfer option. The option determines the calculation used to chamfer the corner.
   - ▲ Select the D1 & Angle Chamfer option to enter values in the D1 and Angle boxes.
   - ▲ Select the D1 & D2 Chamfer option to enter values in the D1 and D2 boxes.
   - ▲ Select the D3 & Angle Chamfer Option to enter values in the D3 and Angle boxes.
6. Click OK to chamfer the lines.
   - ▲ You can click Cancel to leave the drawing unchanged

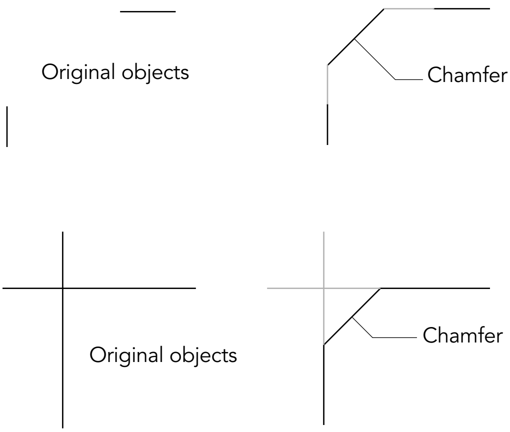


Fig. 276   Chamfer examples

EXTEND

Mode: Draft
Mac OS Shortcut: Cmd+Shift+X
Windows Shortcut: Ctrl+Shift+X

You can use the Extend command to lengthen or shorten a line so that it meets another line. The lines can be two ends of the same open-sided object, or two separate lines. DENEBACAD extends each line so that they meet, but does not combine them into one object.

You cannot use Extend parallel lines, closed objects, or parallel polylines.

When two lines do not meet, DENEBACAD lengthens the first line until it meets the second.

When two lines intersect, DENEBACAD trims the first line you click to meet the other line.

Additionally, the Extend command divides a line into separate line segments. The line you want to divide intersects with the line that divides it.

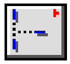


Fig. 277   Extend action button

To extend a line

1. Choose Object > Trims > Extend. The Extend pointer appears.
2. Click the line you want to extend.
3. Move the pointer and click the second line, which the first line will meet.

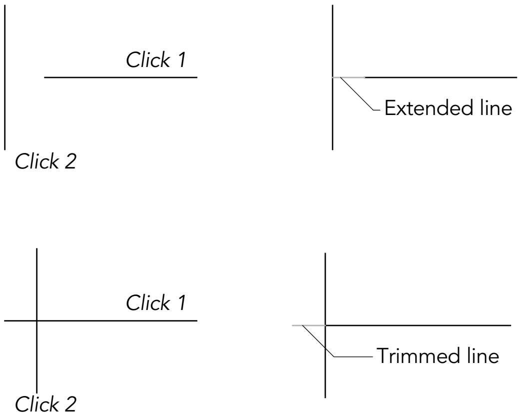
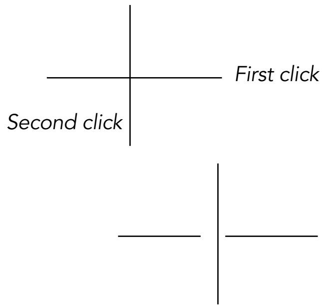


Fig. 278   Extending and trimming lines

To divide a line

1. Choose Object > Trims > Extend. The Extend pointer appears.
2. Press and hold the Option key and click the line you want to divide.
3. While holding down the Option key, move the pointer to the line that intersects the first line. The drawing vector follows from the first line to the second line.

4. Click the second line. DENEBACAD divides the first line where it intersects the second.



Divided lines *(moved for clarity)*

FILLET

Mode: Draft
Mac OS Shortcut: Cmd+Shift+L
Windows Shortcut: Ctrl+Shift+L

The Fillet command creates a convex line join between two lines. The lines can be two ends of the same open-sided object, or two separate lines. You can set the radius of the rounded corner; if necessary, DENEBACAD extends the lines so they can form the specified corner. The objects are combined into one object.

You cannot use this command on parallel lines, closed objects, or parallel polylines.

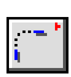


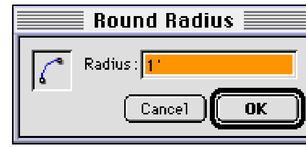
Fig. 279 Fillet action button

To use the Fillet command

1. Choose Object > Trims > Fillet. The Fillet pointer appears.

2. Click the first line.

3. Move the pointer and click the second line. The drawing vector follows from the first line to the second line. A dialog box appears.



4. In the Radius text box, specify the length of the radius for the rounded line join in the current units. You must enter a positive radius value. DENEBACAD extends or trims the lines so their endpoints meet the endpoints of the radius corner.

Original lines New radius corner

JOIN

Mode: Draft
Mac OS Shortcut: Cmd+Shift+J
Windows Shortcut: Ctrl+Shift+J

The Join command connects two lines by extending or trimming them until they meet at a vertex. The lines can be two ends of the same open-sided object, or two separate lines. DENEBACAD extends each line so that they meet, but does not combine them into one object.

You cannot use this command on two parallel lines, closed objects, or parallel polylines.

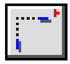


Fig. 280 Join action button

To use the Join command

1. Choose the Object > Trims > Join, or click the Join action button. The Join pointer appears.
2. Click the first line you want to join.
3. Move the pointer and click the second line. The drawing vector follows from the first line to the second line. DENEBACAD extends or trims the lines so they meet at the vertex

TRIM

Mode: Draft
Mac OS Shortcut: Cmd+Shift+T
Windows Shortcut: Ctrl+Shift+T

The Trim command clips, or trims, selected objects so they end at the cutting edge defined by other objects.

You can trim arcs, circles, lines, and open two-dimensional polylines. An object can be both a cutting edge and one of the objects being trimmed.

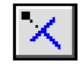
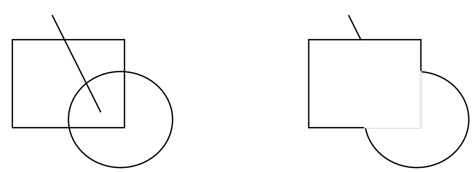


Fig. 281 Trim action button

To use the Trim command

1. Choose Object > Trims > Trim, or click the Trim action button. The Trim pointer appears.
2. Select one or more objects which will become cutting edges.
3. Move the pointer over the object to trim. The cutting edge will change color as the pointer moves over it.
4. Click the mouse button to trim the object.
5. If more than one object is selected, you can continue to trim their cutting edges.
6. Click outside the selected objects to complete the trim operation.



Original objects Trimmed objects

UNCHAIN

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+H
Windows Shortcut: Ctrl+Shift+H

In Draft mode, the Unchain command separates the segments of selected objects into lines, arcs, and open polylines.

You can also use Unchain to unchain Chain Dimension objects (but not other dimension objects) into separate parts you can edit .

In Sculpt mode, the Unchain command separates the facets of 3D objects into individual objects

To use the Unchain command

1. Select the objects you want to unchain.
2. Choose Object > Unchain. DENEBACAD separates the segments of the object at the endpoints

UNGROUP

Mode: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+G
Windows Shortcut: Ctrl+Shift+G

By using the Ungroup command, you can separate objects united with the Group command. You can also use the Ungroup command to separate objects combined using any of the commands in the Combine submenu.This command is available in Draft and Sculpt modes only. If any of the objects in the group are themselves group objects, DENEBACAD keeps them together as a group. If you want to ungroup them, use the Ungroup command again.

In addition, DENEBACAD does not return ungrouped objects to their original position in the stacking order; the program retains the stacking order for the objects resulting from the group operation.

To use the Ungroup command

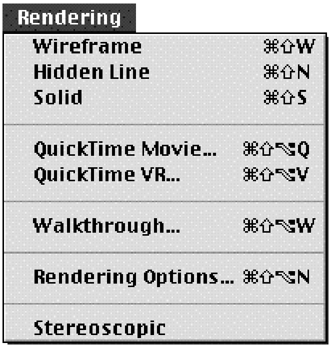
1. Select the group object you want to ungroup.
2. Choose Ungroup in the Object menu, press Ctrl+Shift+G (Windows), or Cmd+Shift+G (Mac). DENEBACAD disunites the group, leaving the individual objects selected.

RENDERING MENU COMMANDS 17

Commands in the Rendering menu let you create renderings in a DENEBACAD document.

You can create Wireframe, Hidden Line, and Solid renderings of 3D objects.

Other Rendering menu commands let you set up options for renderings, and create QuickTime movies, QuickTimeVR scenes, and stereoscopic renderings in movie and VR formats.



Mac OS

Windows

ABOUT RENDER MODE

Render mode lets you view perspective and isometric views of a 3D model. You can also use Render mode to generate a variety of animated walkthroughs.

To begin working in Render mode, you open or activate a Render mode window.

The Rendering menu

When a Render mode window is active, the commands in the Rendering menu are available.

The Wireframe, Hidden Line, and Solid commands generate renderings in the Render mode window. The other Rendering menu commands generate renderings that are stored in QuickTime file formats.

Wireframe A rendering generated with the Wireframe command is a line drawing that displays the Pen colors assigned to 3D objects. The rendered objects appear with outlines only; no indication of surfaces is displayed. Because there is no surface color or shading, objects appear to be transparent. By default, Render mode windows display Wireframe renderings. When a Hidden Line or Solid rendering is displayed, you can choose the Wireframe command to generate a Wireframe rendering.

Hidden Line A rendering generated with the Hidden Line command is a line drawing that displays objects as solids. You can generate Hidden Line renderings using the Hidden Line command.

Solid A rendering generated with the Solid command is a photo-realistic image of 3D objects. This type of rendering can include lighting effects, shadow casting, reflectivity, gloss, and transparency.

QuickTime Movie This command saves photo-realistic animations of a 3D model in Apple Computer's QuickTime movie format. You can view QuickTime movie renderings outside of the DENEBACAD environment using other applications.

QuickTime VR The QuickTime VR command saves photo-realistic interactive pictures of a 3D model. You can view QuickTime VR scenes using player applications outside of the DENEBACAD environment.

Walkthrough The Walkthrough command creates an animated walkthrough of a wireframe rendering of a 3D model, using a path that you specify.

Rendering Options The Rendering Options command opens the Rendering Options dialog box. You use this dialog box to set most rendering options, including frame size, rendering quality, lens angle, and background color, as well as settings for artificial lighting and sunlight.

Stereoscopic You can use the Stereoscopic command with the Solid, QuickTime Movie, and QuickTime VR commands to create stereoscopic 3D renderings and movies. When viewed through red-blue 3D glasses on standard computer monitors, these renderings appear to have true 3D depth.

Background color in renderings

When you open a new Render mode window, the default background color is white. The grid that you use in Draft and Sculpt mode windows isn't visible in the Render mode window.

Solid renderings display a background color if you have not enabled artificial lighting in the Rendering Options dialog box. When you enable only artificial lighting, the background color is black. When you enable only sunlight, or both artificial lighting and sunlight, the background color is a color you select in the Rendering Options dialog box.

You can change the background color using the Background pop-up color palette in the Rendering Options dialog box.

THE RENDER MODE TOOLBOX

The toolbox displayed when you activate a Render mode window contains five tools: Horizontal Focal Point, Vertical Focal Point, Dual Focal Point, Perspective Render, and Isometric Render tools.

Focal Point tools

The Focal Point tools establish the height, distance, and angle of a focal point, or camera, as it relates to the objects in a perspective. You can use these tools to adjust your view before rendering a scene.

Rendering tools

The Perspective Render and the Isometric Render tools let you select between the display of perspective and isometric views of the 3D model.

When you switch from one rendering view to another, the window displays a Wireframe rendering.

HIDDEN LINE

Mode: Render
Mac OS Shortcut: Cmd+Shift+N
Windows Shortcut: Ctrl+Shift+N

The Hidden Line command generates line drawings that display objects as solids in the Render mode window. The objects do not have surface colors or shading, but surfaces are implied by the removal of hidden lines.

Hidden Line renderings display objects with their assigned Pen colors.

The Render mode window displays a Hidden Line rendering until you click in another window. When you click in another window, the Render window displays a Wireframe rendering.

Lighting in Hidden Line renderings

Because Hidden Line renderings are line drawings, they do not show lighting or shadows. However, you do see symbols that represent artificial light sources if you have placed artificial light sources in Sculpt mode.

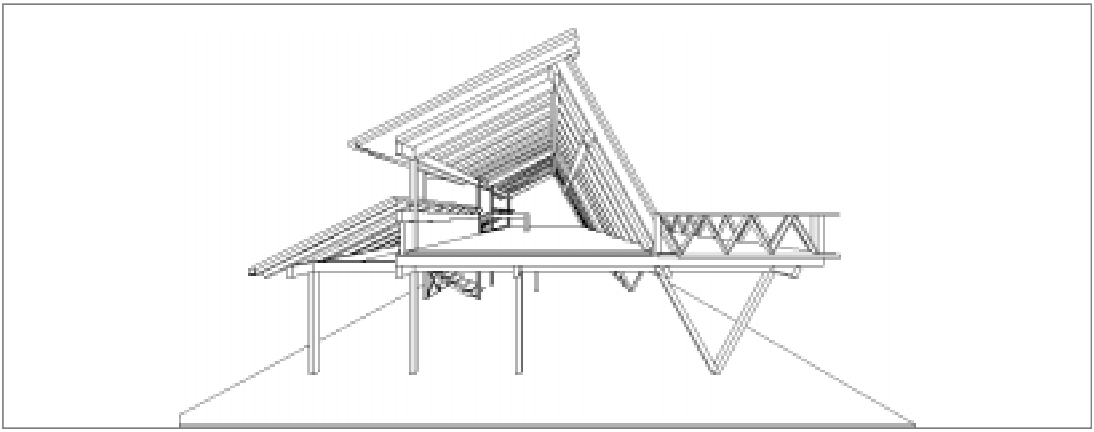


Fig. 282 Rendering generated by the Hidden Line command

To create a Hidden Line rendering

1. Create 3D objects and assign pen colors to the objects.
2. Open or activate a Render mode window. The window initially displays a Wireframe rendering.
3. Use the Focal Point tools to establish horizontal and vertical points of view and sight lines. If you want to configure screen settings and lens angle for the rendering, choose Rendering Options in the Rendering menu.
4. Choose Hidden Line in the Rendering menu. DENEBACAD generates a Hidden Line rendering. You can monitor the generation of the rendering by watching the progress indicators in the Help bar.

QUICKTIME MOVIE

Mode: Render
Mac OS Shortcut: Cmd+Opt+Shift+Q
Windows Shortcut: Ctrl+Alt+Shift+Q

Apple Computer's QuickTime technology lets you record and play digital movies. QuickTime software is an extension included with Mac OS system software and is also available for Windows systems.

DENEBACAD creates QuickTime movies by "photographing" a 3D scene from the point of view of a "camera" moving along a path that you have defined.

QuickTime movies created in DENEBACAD can display colors, Surface materials, artificial lighting, sunlight, and cast shadows to create a photo-realistic animation of an environment.

CAMERA PATHS

Before creating a QuickTime movie, you must create and save a camera path. The camera path is the path from which the "camera" records the scene when you generate a QuickTime movie.

You create and save camera paths in Sculpt mode using the Camera Path tool. For more information, see "Camera Path tool" on page 88.

Testing camera paths

Before you generate a QuickTime Movie, you can use the Walkthrough command to check the camera path you intend to use for the movie.

The Walkthrough command generates animated wireframe renderings, called *walkthroughs*, using a defined camera path, which can also be used to generate a QuickTime movie. This is a quick and easy way for you to detect possible problems with a camera path.

After viewing a walkthrough of a 3D model in Render mode, you can switch back to Sculpt mode to refine the camera path. You can revise the placement and angle of the cameras, the angle and direction of sight lines, and the shape of the path in Sculpt mode.

Once you fine-tune the camera path and view the finished walkthrough, you are ready to generate a QuickTime movie with the QuickTime movie command.

To generate a QuickTime movie rendering

1. Extrude 3D objects in Draft mode, or create 3D objects in Sculpt mode and apply Surface materials, to the objects, if desired.

▲ Without Surface materials, objects appear to be filled with their Pen colors in QuickTime movie renderings.

2. In Sculpt mode, place artificial lights in the model using the Light Source tools if you want to use artificial lighting. See "Light Source tools" on page 114 for more information.

3. In Sculpt mode, use the Camera Path tool to create and save a camera path from which to "photograph" the QuickTime movie.

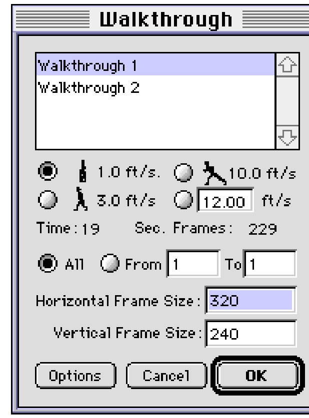
4. Open or activate a Render mode window.

5. Choose the Rendering Options command to access the Rendering Options dialog box. You can use the Lighting tab to enable artificial lighting and sunlight.

▲ You can also configure screen settings and lens angle for a QuickTime movie rendering in the Rendering Options dialog box.

6. Choose QuickTime Movie in the Rendering menu. The Walkthrough dialog box opens. Select the camera path, speed, and frame size in this dialog box; these options are described later in this section. When you finish, click OK to close the Walkthrough dialog box.

▲ To configure settings for QuickTime compression, click the Options button in the Walkthrough dialog box.



7. Click OK. A Save As dialog box opens. Open the folder where you want to store the file, and type a file name. Click Save to create the QuickTime movie file.

DENEBACAD creates the QuickTime Movie frame by frame. You can monitor the generation of the movie by watching the progress bars in the Help bar (if the Help bar is displayed).

The time it takes to generate a movie varies. The number of artificial lights, the size of the texture mapping, and the type of antialiasing all affect the time it takes to generate a movie.

For example, generating a QuickTime movie with Antialiasing set to Best takes eight times longer than generating the same movie with Antialiasing set to Normal.

WALKTHROUGH OPTIONS

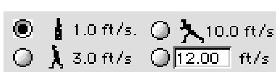
The following options appear in the Walkthrough dialog box. Use these options to specify settings before you generate a QuickTime movie. When you choose the QuickTime command, the Walkthrough dialog box appears.

Camera path

Select one of the saved camera paths from the list in the Walkthrough dialog box. If no camera paths appear in the scrolling list, you need to use the Camera Path tool in Sculpt mode to create and save a camera path.

Walkthrough speed

Select one of the preset walkthrough speed radio buttons, or enter a walkthrough speed in the text box. The lower the foot per second ratio, the slower the walkthrough speed, and the longer it takes to complete the walkthrough path.



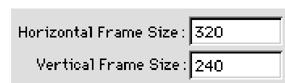
Frames to generate

Select an option to specify which frames of the walkthrough to generate in the QuickTime movie.

- Click All to generate all frames based on the walkthrough path and walkthrough speed setting.
- Click From to specify a range of frames. Type the starting frame number in the first text box and the ending frame number in the second text box.

Frame size

To specify the frame size of the QuickTime movie, type the horizontal and vertical frame measurement in pixels in the Horizontal Frame Size and Vertical Frame Size text boxes



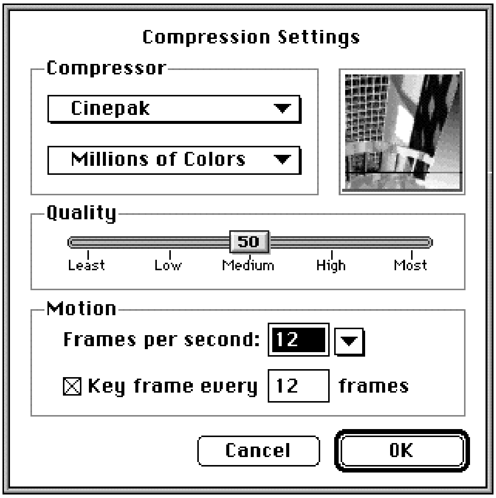
The frame size settings have a direct effect on the amount of information in the QuickTime movie, and therefore on the storage space requirements of the movie.

For example, at a frame size of 200 x 200 pixels, each frame of the resulting QuickTime movie contains 40,000 pixels. If each pixel requires 1 byte of storage space, each frame of the movie contains about 39 KB of information.

At a frame size of 600 x 600 pixels, each movie frame contains 360,000 pixels. At 1 byte per pixel, each frame of the movie contains 351.5 KB of information.

QuickTime compression options

When you click the Options button in the Walkthrough dialog box, the Compression Settings dialog box appears.



In the Compressor area, choose the compression method and the number of colors. Use the Quality slider to fine-tune the compression method. Some compression methods let you adjust the quality and some do not.

In the Motion area, select the number of frames per second from the pop-up menu or type the number of frames in the text box. Select the "Key frame every" checkbox and type the key frame interval in the text box to set up key frames.

Animation is typically created at 24 frames per second to produce a smooth and seamless movie. Compression settings, frame rates, and walk-through options determine the file size of the movie. The higher the quality of the movie, the larger the file size.

QUICKTIME VR

Mode: Render
Mac OS Shortcut: Cmd+Opt+Shift+V
Windows Shortcut: Ctrl+Alt+Shift+V

The QuickTime VR command generates a navigable picture, one that you can move through and turn around in by panning left, right, up, or down.

QuickTime VR is a standard for creating and viewing photo-realistic environments and real world objects. Users interact with QuickTime VR content with a complete 360 degree perspective and control their viewpoint through the mouse in a realistic visual environment.

Before generating a QuickTime VR picture, you establish the location and position of a stationary camera. You can use the Focal Point tools in the Render mode toolbox to set the location and height of the camera.

For example, to give the impression of standing in the middle of a 3D model and turning 360 degrees, place the camera in the center of the 3D model using the Horizontal Focal point tool.

To set the height of the camera to eye level, use the Vertical Focal Point tool.

To use the QuickTime VR command

1. Extrude 3D objects in Draft mode, or create 3D objects in Sculpt mode and assign Pen colors and Surface materials.
2. In Sculpt mode, place artificial lights in the model using the Light Source tools. See "Light Source tools" on page 114 for more information.
3. Open or activate a Render mode window.
4. Set the camera position using the Focal Point tools.

5. Choose the Rendering Options command to access the Rendering Options dialog box. Select the Artificial Lighting tab, the middle tab, to enable artificial lighting and sunlight.

   ▲ You can configure the other screen settings and lens angle of the rendering in the Rendering Options dialog box.

6. Choose the QuickTime VR command in the Rendering menu. The Save As dialog box appears.
7. In the Save As dialog box, locate the folder or directory where you want to store the file.
8. Type a file name in the Save as text box.
9. Select the file location, enter a file name and click Save. DENEBACAD generates a QuickTime VR picture. You can monitor the generation of the picture by watching the Progress bars in the Help bar.

The time it takes to create the picture can vary. The amount of artificial lights, the size of the texture mapping and the type of antialiasing all affect the time it takes to generate a QuickTime VR picture.

For example, generating a QuickTime VR picture with Antialiasing set to Best takes eight times longer than generating the same picture with the Antialiasing set to Normal.

RENDERING OPTIONS

Mode: Render
Mac OS Shortcut: Cmd+Opt+Shift+N
Windows Shortcut: Ctrl+Alt+Shift+N

The Rendering Options command lets you configure lens angle, units, and background color of Wireframe, Hidden Line, and Solid renderings, as well as QuickTime movies and QuickTime VR scenes.

Rendering Options also lets you configure the antialiasing, resolution and lighting parameters for Solid renderings, QuickTime movies, and QuickTime VR scenes. Also, the frame size of a Solid rendering can be specified in the Rendering Options dialog box.

The Rendering Options command is available whenever any DENEBACAD document window is open.

Before you generate a rendering, movie, or VR picture, you can choose the Rendering Options command to configure the settings. The Rendering Options dialog box organizes settings on three tabs. To bring a tab to the front, click the tab.

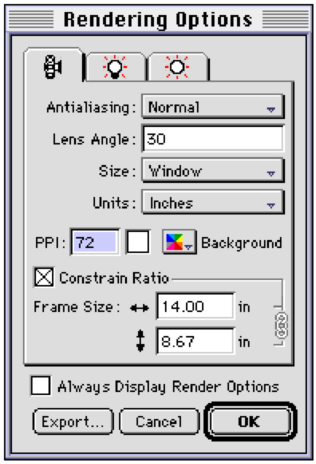
- The tab marked with a camera icon contains the Screen settings.
- The tab marked with a light bulb icon contains the Lighting settings.
- The tab marked with the sun icon contains the Sunlight settings.

To use the Rendering Options command

1. Choose the Rendering > Rendering Options. The Rendering Options dialog box appears.
2. Configure the settings in the dialog box. To change tabs, click the icon at the top of the tab to bring the tab to the front.
3. To make the new settings take effect, click OK. To leave the last saved settings unchanged, click Cancel.

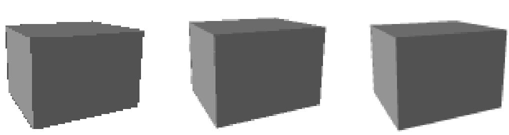
SCREEN TAB

A camera icon appears at the top of the Screen tab in the Rendering Options dialog box.



Antialiasing

Antialiasing reduces the jagged appearance of diagonal lines or edges in an image. DENEBACAD offers three choices of antialiasing that affect the quality of Solid renderings, QuickTime movies, or QuickTime VR scenes: Normal, Better, and Best.



Normal Better Best

Fig. 283 Antialiasing options

Normal Choose Normal in the Antialiasing pop-up menu to render the scene with the least amount of antialiasing. Normal renders the quickest of the three options.

When quality is not as important as how quickly DENEBACAD generates an image, choose Normal.

Better The Better option generates a rendering, movie, or VR picture that has smoother edges than one generated with Normal antialiasing. Better takes four times longer to generate an image than one generated with Normal antialiasing.

Choose the Better command in the Antialiasing pop-up menu when you need increased quality in a rendering, movie, or VR picture.

Best The Best option requires eight times longer to generate a rendering, movie, or VR picture than one generated with Normal antialiasing. Best antialiasing is the smoothest and most realistic type of antialiasing. Choose Best in the Antialiasing pop-up menu when you need the highest quality finished product.

Lens Angle

Lens Angle defines the depth of field of a rendering, QuickTime movie, or QuickTime VR picture. Type a value in the Lens Angle text box to change the current setting.

The higher the number, the shallower the depth of field. A shallower depth of field brings objects closer to the camera.

This option is not available when you are in Draft or Sculpt mode.

Size

The Size option lets you choose the frame size of a Solid rendering. You can choose from a Custom, Window, or preset frame size.

This option is not available when you are in Draft or Sculpt mode.

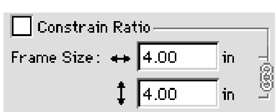
Window Choose the Window command in the Size pop-up menu to generate a Solid rendering to fit the Render mode window.

Custom Choose Custom in the Size pop-up menu to enter your own values in the Frame Size text boxes. If the Constrain Ratio check box is selected, the frame size you enter will have the same proportions as the current frame size.

There are five preset frame sizes. They are measured in pixels. When you choose one of these, the values are shown in the Frame Size text boxes.

Units

The Units pop-up menu lets you choose the unit of measurement of the frame size. You can choose Pixels, Inches, or Centimeters. An abbreviation for the current units follows the Frame Size text boxes.



PPI

For digital images, *resolution* measures the size of the pixels that make up the image, expressed as pixels per inch (ppi). This applies to digital images from any source, whether scanned, created on computer, or captured by a digital camera.

The resolution of an image tells you how much digital information the image contains. Each pixel making up the image is a dot of solid color, and the smaller the pixels, the finer the image appears.

Higher resolutions results in larger images that require more memory. Larger images make your document files larger. Also, the larger the image, the longer DENEBACAD takes to generate and save the rendering.

When you set the resolution in the Rendering Options dialog box, you are setting the image resolution for a Solid rendering as it relates to the output resolution.

To determine the appropriate resolution, consider the halftone screen frequency that will be used for printing. The resolution required of digital images for printing is typically 1.5 to 2 times the halftone screen frequency. For example, if the halftone screen frequency is 150 lines per inch, the required image resolution is 225 to 300 ppi.

To change the resolution, type a new value in the PPI text box.

If you are generating renderings for viewing on screen only, you can use the default resolution of 72 ppi. However, if the images will be enlarged and you want to maintain the screen resolution, you might need to create QuickTime movies or QuickTime VR pictures at a higher resolution. In that case, use the enlargement factor to calculate the required image resolution.

Background

The Background pop-up color palette lets you pick a background color for the Render mode window.

The current background color appears in the box. You can choose a preset or custom color from the pop-up color palette.

To change the background color, press the pop-up color palette and select a color. To use a custom color, select Other at the bottom of the grid. When you select Other, a system Color Picker appears. Use this dialog box to specify custom colors.

Constrain Ratio

If you select this option, the values you enter in the Frame Size text boxes are proportional to those of the current values.

For example, with a frame size of 4" wide by 8" high, the frame height is twice its width. If Constrain Ratio is selected and you enter 5" in the width text box, the height changes to 10". Or, you can enter 10" in the height text box and the width changes to 5".

This option is not available in Draft or Sculpt mode.

Frame Size

Frame size refers to the dimensions of a Solid rendering in the Render window.

The Frame Size text boxes show the current frame size in the units you specify. The unit abbreviations are “pix” for pixels; “in” for inches; and “cm” for centimeters.

To enter custom frame sizes, choose Custom in the Size pop-up menu, and then type the dimensions in the Frame Size text boxes.

You can not enter custom frame sizes in Draft or Sculpt mode.

Always Display Render Options

If you select this option, the Rendering Options dialog box appears each time you choose the Solid or QuickTime VR command in the Rendering menu.

Export

Choose Export to save a Solid rendering to a file.

To export a rendering

1. Open or activate a Render mode window.
2. Choose Rendering > Rendering Options. The Rendering Options dialog box appears.
3. Configure the screen and lighting settings in the Rendering Options dialog box.
4. Click the Export button. A Save dialog box appears.
5. In the Save dialog box, select the folder where you want to store the file. Type a file name in the text box.
6. Click Save to store the rendering file on disk. DENEBACAD creates the Solid rendering. You can monitor the generation of the rendering by watching the Progress bars in the Help bar.

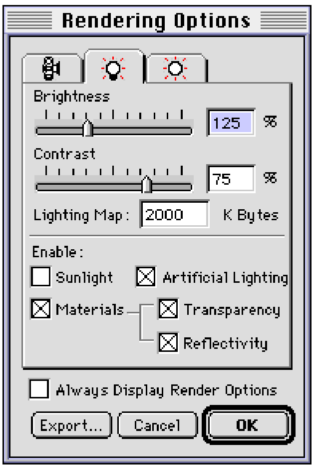
Export memory requirements

The time and system memory required to export a rendering depends on several factors. The number of artificial lights, reflectivity and transparency effects, texture mapping, and the type of antialiasing all affect the resources needed to generate and save a Solid rendering.

In general, you should allocate as much memory as possible to DENEBACAD before you export a high resolution or complex Solid rendering to disk.

LIGHTING TAB

A light bulb icon appears at the top of the Lighting tab in the Rendering Options dialog box.



Brightness and Contrast

You can adjust the brightness and contrast of a Solid rendering, QuickTime movie, or QuickTime VR picture. Brightness refers to the overall lightness of an image. Contrast is the difference among the various lightness levels in an image.

Because the Brightness control adjusts all pixels equally, if you use these controls to lighten an image that appears too dark, the image can lose shadow detail.

Brightness Enter a value from 0-400 in the Brightness text box. Higher values can wash out midtones and shadows. Lower values can dull highlights.

Contrast Enter a value from 0-100 in the Contrast text box. Increasing contrast moves the color values of pixels to the extremes of the brightness spectrum. Decreasing contrast moves color values toward medium gray.

Lighting Map

The Lighting Map value is the amount of memory (in kilobytes) used for calculating the lighting in a Solid rendering, QuickTime movie, or QuickTime VR picture. The more memory you allocate to calculate lighting, the smoother the shading appears in a finished rendering.

DenebaCAD uses the amount of RAM memory you specify to calculate lighting from one light source at a time. For example, if Lighting Map is 2000 KB, and the model has 10 artificial lights, DenebaCAD uses 2000 KB of RAM for the first light, and then releases the memory. It then uses 2000 KB of RAM for the second light, and so on.

Artificial Lighting

If you select this option, DenebaCAD renders the artificial lighting for a Solid rendering, QuickTime movie, or QuickTime VR picture, based on the light sources in the model.

You place light sources in a model using the Light Source tools in Sculpt mode. See “Light Source tools” on page 114 for more information.

When Artificial Lighting is selected, rendering an image or movie requires more time than when this option is not selected.

You can select Artificial Lighting alone, or with the Sunlight option.

When you create a new document, Artificial Lighting is selected.

Sunlight

If you select this option, DenebaCAD renders the sunlight for a Solid rendering, QuickTime movie, or QuickTime VR picture. The effects of sunlight are based on the settings on the Sunlight tab.

An image or movie takes longer to generate when the Sunlight option is selected. You can select Sunlight alone, or with the Artificial Lighting option.

Materials

If you select this option, DENEBACAD renders the Surface materials of 3D objects in a Solid rendering, QuickTime movie or QuickTime VR picture.

An image or movie takes longer to generate when Surface materials are rendered. When the Materials option is not selected, 3D objects are rendered using their Pen colors as surface colors.

When you create a new document, Materials is selected.

Transparency

If you select this option, DENEBACAD renders transparent Surface materials realistically.

When you create a new Surface material in the Material Editor, you can specify its transparency.

You can select the Transparency option only if the Materials option is selected.

Reflectivity

If you select this option, DENEBACAD displays the reflectivity of Surface materials realistically.

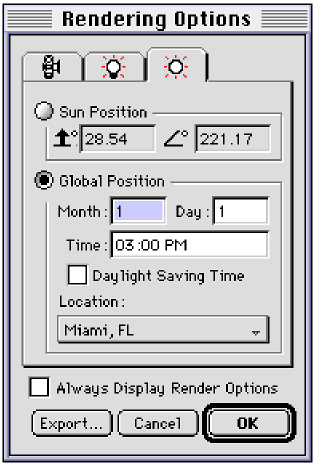
When you create a new Surface material in the Material Editor, you can specify its reflectivity.

You can select the Reflectivity option only if the Materials option is selected.

SUNLIGHT TAB

A sun icon appears at the top of the Sunlight tab in the Rendering Options dialog box. This tab lets you define the position of the sun for renderings when the Sunlight option is selected.

You can define the sun's position based on a geographic location and time of day, or you can set the sun's vertical angle and compass position.



Sun Position

Select the Sun Position button to customize the position of the sun.

When you select the Sun Position option, the Global Position option is deselected.

Enter the sun's vertical angle in degress in the first text box. Enter the compass angle in the second text box. For the compass angle, North is 0, east is 90, south is 180, and west is 270 degrees.

Global Position

Select the Global Position button to simulate the sun's position in a particular city at the time you specify. DENEBACAD calculates the vertical angle and direction of the sun for renderings when the Sunlight option is selected.

When you use this option, the Sun Position text boxes display the sun's vertical angle and direction in degrees. These are preset values that can't be changed. However, you can select the Global

Position option, select a city, and then select the Sun Position option and adjust the vertical angle and direction values.

Month To change the month, type a number from 1-12 in the text box.

Time To specify the time of day, type the hour in the first text box, and the minutes in the second box. Select the PM box for times after noon.

DENEBACAD calculates the sun position at the time of day you specify.

Daylight Saving Time Select this box to use Daylight Saving time.

Location Select a city from the Location menu. Use this option whenever you use Global Position.

You can use the following commands in the Location pop-up menu to customize locations.

Define Location Choose Define Location to add a new city to the list of locations. Once you add a location, you can use it in any drawing.

- ▲ Type the name of the location you are defining in the name text box.
- ▲ Type the latitude in degrees in the Latitude text box. Select the N box for latitudes in the Northern Hemisphere.
- ▲ Type the longitude in degrees in the Longitude text box. Select the E box for longitudes that are east of the prime meridian in Greenwich, England.
- ▲ Type the time difference from GMT in the text boxes. Type the difference in whole hours in the first box and the difference in minutes in the second box. Select the check box labeled "+" for times ahead (east) of GMT.

Click OK to accept the values you entered. The new location appears in the Location pop-up menu.

Edit Location Choose Edit Location to edit the global position of the previously selected location. The name of the selected location appears in the Name text box in the Edit Location dialog box. Make the changes you want and click OK.

Delete Location Choose Delete Location to delete locations from the Location menu. A dialog box appears. Select locations to delete and click Remove. Click OK to delete the selected locations and close the dialog box.

When you add a location, these settings determine the sun position for Solid renderings with the Sunlight option

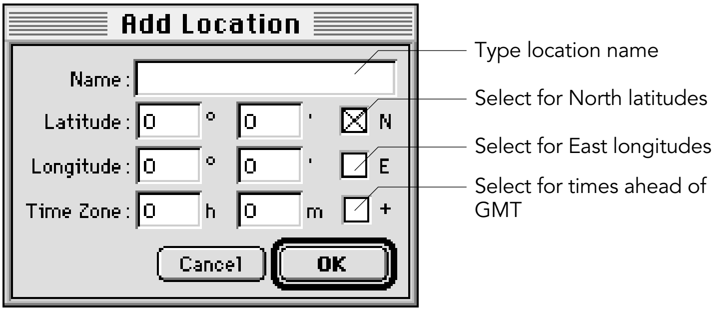


Fig. 284 Add Location dialog box

SOLID

Mode: Render
Mac OS Shortcut: Cmd+Shift+S
Windows Shortcut: Ctrl+Shift+S

The Solid command generates realistic renderings in the Render mode window. Solid renderings can display lighting effects, cast shadows, and textured Surface materials.

To render a 3D model as a Solid rendering, choose the Solid command in the Rendering menu.

A Solid rendering displays an object's assigned pen color or Surface material. If an object has both a pen color and Surface material, you can check the Materials option in the Rendering Options dialog box to render its Surface material in a Solid rendering.

A Solid rendering displays objects that have only pen colors and objects that have both pen colors and Surface materials.

Lighting

Solid renderings can display artificial lighting and cast shadows if you have placed light sources in the model. You place artificial lighting in a model in Sculpt using the Light Source tools.

You can use sunlight in addition to artificial lighting, or as the only lighting in a Solid rendering.

Artificial lighting will generate a realistic image because of the light and cast shadows that fall across objects. However, a rendering with artificial light will take longer to generate because DENEBACAD calculates the light and shadows as well as the texture mapping of Surface materials.

A Solid rendering with both sunlight and artificial lighting also takes longer to generate than a rendering with no lighting because DENEBACAD calculates the light and shadows as well as the texture mapping of Surface materials.

You can generate a Solid rendering much quicker if you disable both artificial lighting and sunlight. When lighting is not enabled, there are no shadows cast in the rendering. However, this method lets you see the position, Surface materials and colors of objects quickly.

To use the Solid command

1. Extrude 3D objects in Draft mode, or create 3D objects in Sculpt mode and assign pen colors and Surface materials.
2. In Sculpt mode, place artificial lights in the model using the Light Source tools.
3. Open or activate a Render mode window.
4. To view the Solid rendering from another perspective, change the location of the focal point using the Focal Point tools in the toolbox.
5. Choose the Rendering Options command to access the Rendering Options dialog box. Select the middle tab to enable artificial lighting and sunlight.

   ▲ You can configure the other screen settings and lens angle of the rendering in the Rendering Options dialog box.
6. To create a Solid rendering, choose the Solid command in the Rendering menu. DENEBACAD generates a Solid rendering. You can monitor the generation of the rendering by watching the Progress bars in the Help bar.

Rendering time and memory requirements

The amount of time and system memory it takes to produce a Solid rendering depends on a number of factors: the number of artificial lights, reflectivity and transparency effects, texture mapping, and the type of antialiasing.

For example, generating a Solid rendering with antialiasing set to Best takes eight times longer and requires more system memory than generating the same rendering with antialiasing set to Normal. Similarly, rendering a scene at 300 ppi resolution takes significantly more time and memory than rendering the same scene at 72 ppi resolution.

Before you generate a complex or high resolution Solid rendering, you should allocate as much system memory as possible to DENEBACAD. Otherwise, the program might not be able to complete the rendering.

If you try to start a rendering with insufficient system memory allocated, DENEBACAD displays a message.

Click OK to close the message box, and then allocate more system memory to DENEBACAD.

If DENEBACAD can't complete a Solid rendering because an insufficient amount of memory is allocated to the program, a message appears. You can click OK in the message box to complete the rendering on disk.

Specify a location for the Solid rendering in the directory dialog box. Type a name for the rendering file, and then click Save to export the rendering to the specified location.

Note: This operation is equivalent to using the Export option in the Rendering Options dialog box. For more information on exporting, see "Export" on page 327.

If you click Cancel, DENEBACAD creates a low-resolution version of the Solid rendering, which requires less memory.

If there is insufficient memory to store a low resolution version of the rendering, DENEBACAD displays a message that there isn't enough memory to create a low-resolution copy of the rendering.

Allocating memory to DENEBACAD

Note: Adjustment of memory allocation is available in Mac OS only. Windows automatically compensates for application memory requirements.

To allocate memory to DENEBACAD, close the program, select the DENEBACAD program icon, and choose File > Get Info in the Mac OS Finder. Change the value in the Preferred size text box.

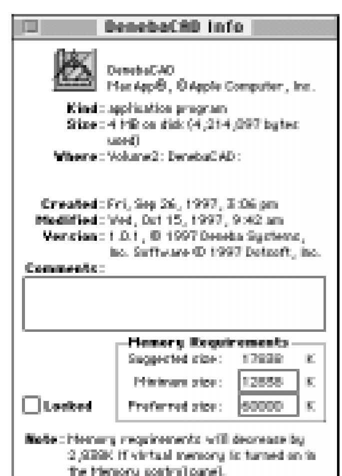


Fig. 285 Rendering created by the Solid command

STEREOSCOPIC

You can use the Stereoscopic command in conjunction with the Solid, QuickTime Movie and QuickTime VR commands to create a different kind of rendering or movie.

This command produces renderings and movies with objects that have the appearance of three dimensions.

Stereoscopic rendering is a process that generates a rendering that appears to be truly 3D by having a depth in addition to a height and width when you view it with the Stereoscopic 3D glasses.

In this process, the objects are first rendered with their pen color or Surface materials. A blue and red transparent overlay of the object is offset from the original object.

You need to view the on screen result with the Stereoscopic 3D glasses to see the illusion of depth that this process creates

To use the Stereoscopic command:

1. Open or activate a Render mode window.
2. Choose Rendering > Stereoscopic. A check mark appears next to the command. This means the command is active and will affect any Solid rendering, movie or VR picture you generate.
3. Choose Solid, QuickTime Movie, or QuickTime VR in the Rendering menu. DENEBACAD generates the Stereoscopic rendering or movie. You can monitor the generation of the rendering by watching the Progress bars in the Help bar.

The Render mode window displays the Stereoscopic Solid rendering until you click in either a Draft or Sculpt mode window. Once you click in another window, the Solid rendering reverts to a Wireframe rendering.

WALKTHROUGH

Mode: Render
Mac OS Shortcut: Cmd+Opt+Shift+W
Windows Shortcut: Ctrl+Alt+Shift+W

The Walkthrough command lets you view a walkthrough path through a wireframe representation of a 3D model.

The Render mode window displays the wireframe objects in a walkthrough in their assigned Pen colors.

The walkthrough path is also referred to as the camera path. You create and save a camera path in Sculpt mode using the Camera Path tool.

Using the Walkthrough command is the quickest way for you to view a camera path.

To use the Walkthrough command

1. Extrude 3D objects in Draft mode, or create 3D objects in Sculpt mode and apply Pen colors and Surface materials.
2. In Sculpt mode use the Camera Path tool to create and save the camera path.
3. Open or activate a Render mode window.
4. Choose the Rendering Options command to access the Rendering Options dialog box. You can configure the screen settings and lens angle of the walkthrough in the Rendering Options dialog box.
5. Choose the Walkthrough command in the Rendering menu. The Walkthrough dialog box appears.
6. Select one of the saved walkthrough paths from the list in the Walkthrough dialog box. You can change the settings in this dialog box to speed up or slow down the walkthrough. You can also change the frame size of the walkthrough window.
7. Select one of the preset walkthrough speed radio buttons or enter a walkthrough speed in the text box. The lower the feet per second ratio, the slower the walkthrough, and therefore, the slower the QuickTime movie.

   ▲ The time and number of frames it takes to create the walkthrough are shown below the walkthrough speed buttons.

   ▲ Select All to view the entire walkthrough. Or, specify only the frames you want to see by entering the frame numbers in the text boxes.

   ▲ You can also specify the frame size of the walkthrough by entering values in the horizontal and vertical frame size text boxes.

   ▲ A larger frame size and a slower walkthrough speed results in a longer, slower walkthrough.
8. Click OK to generate the walkthrough. DENEBACAD generates the wireframe walkthrough.

WIREFRAME

Mode: Render
Mac OS Shortcut: Cmd+Shift+W
Windows Shortcut: Ctrl+Shift+W

To render a 3D model as a Wireframe rendering, choose the Wireframe command in the Rendering menu.

Once you render a 3D model using either the Hidden Line or Solid commands, you can use the Wireframe command to view the rendering as a Wireframe rendering again.

By default a Render mode window displays a Wireframe rendering of the 3D objects in the document. A Wireframe rendering is a line drawing that uses the pen color assigned to objects. Only the outline of an object appears with no indication of its surfaces. Because there is no surface color or shading, objects appear transparent.

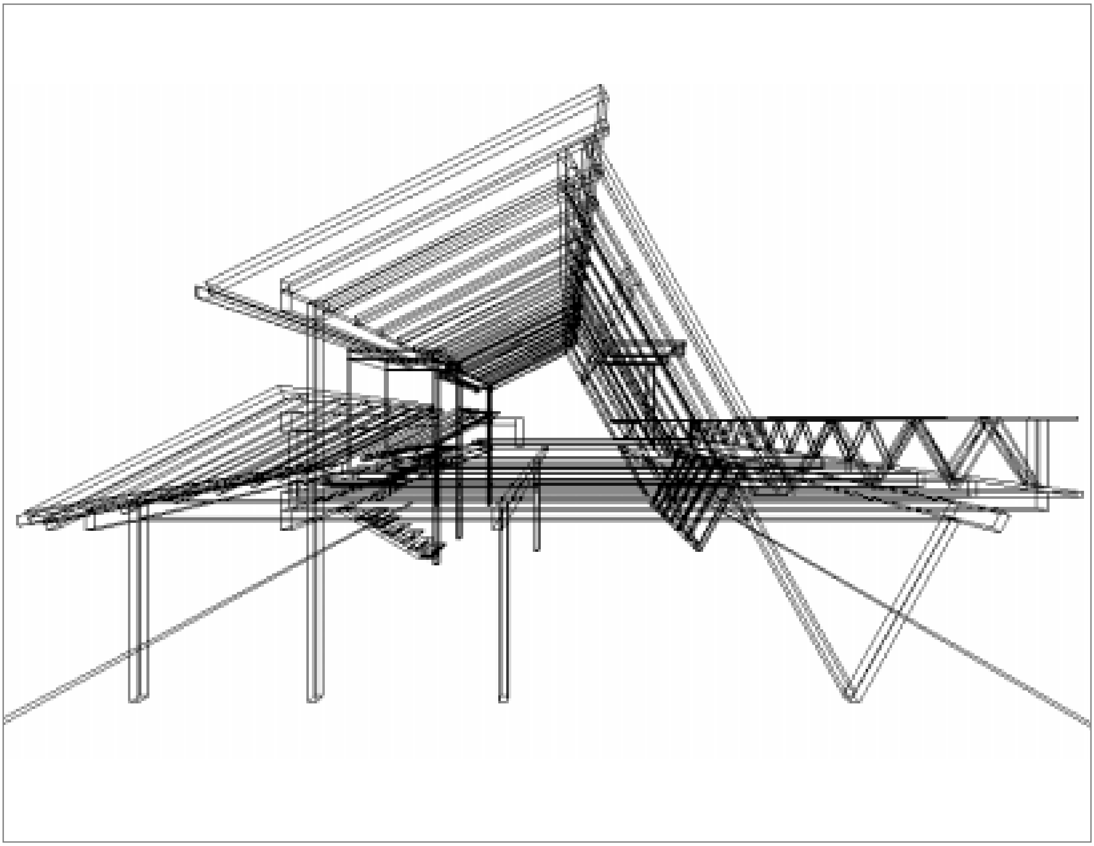


Fig. 286 Rendering generated by the Wireframe command

Lighting

You don't see lighting or cast shadows in a Wireframe rendering. If you have placed artificial lights in your rendering, you see the symbols, drawn in their assigned pen color, that represent the artificial light sources.

To use the Wireframe command

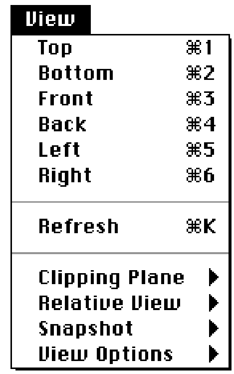
1. Extrude 3D objects in Draft mode, or create 3D objects in Sculpt mode and apply pen colors and Surface materials.
2. Open or activate a Render mode window.
3. To view the Wireframe rendering from another perspective, change the location of the focal point using the Focal Point tools in the toolbox. Refer to "Render tools" on page 141 for more information on the Focal Point tools.

   ▲ If you need to configure the screen settings and lens angle of the rendering choose the Rendering Options command in the Rendering menu.
4. To create a Wireframe rendering, choose the Wireframe command in the Rendering menu. DENEBACAD generates a Wireframe rendering.

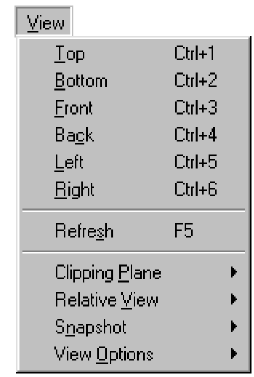
VIEW MENU COMMANDS 18

The View menu contains commands that let you change the display of objects, redraw the screen, clip objects, and create relative views.

Some of the View menu commands are also available from pop-up menus in the Status bar. The pop-up menus also show the current state of some view options. To display the Status bar, use the Toolbars command in the Layout menu.



Mac OS



Windows

ORTHOGONAL VIEW COMMANDS

The first six commands in the View menu — Top, Bottom, Front, Back, Left, and Right — set the view of the active Draft or Sculpt window. When you choose a View command, you change your orientation to the drawing. The six orthogonal view commands are described in menu order.

TOP

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+1
Windows Shortcut: Ctrl+1

DENEBACAD provides six standard views: Top, Bottom, Front, Back, Left, and Right. These are orthogonal views of the DENEBACAD workspace. Top view is commonly referred to as “plan” view.

To choose Top view

1. Before choosing Top view, activate a Draft or Sculpt mode window.
2. Choose View > Top. Or, press Ctrl+1 (Windows), or Cmd+1 (Mac OS). The active window shows the workspace from above.

BOTTOM

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+2
Windows Shortcut: Ctrl+2

To choose Bottom view

1. Activate a Draft or Sculpt mode window.
2. Choose View > Bottom. Or, press Ctrl+2 (Windows), or Cmd+2 (Mac OS).

FRONT

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+3
Windows Shortcut: Ctrl+3

Front view is an elevation view.

To choose Front view

1. Activate a Draft or Sculpt mode window.
2. Choose View > Front. Or, press Ctrl+3 (Windows), or Cmd+3 (Mac OS).

BACK

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+4
Windows Shortcut: Ctrl+4

Back view is an elevation view.

To choose Back view

1. Activate a Draft or Sculpt mode window.
2. Choose View > Back. Or, press Ctrl+4 (Windows), or Cmd+4 (Mac OS).

LEFT

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+5
Windows Shortcut: Ctrl+5

Left view is an elevation view.

To choose Left view

1. Activate a Draft or Sculpt mode window.
2. Choose View > Left. Or, press Ctrl+5 (Windows), or Cmd+5 (Mac OS).

RIGHT

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+6
Windows Shortcut: Ctrl+6

Right view is an elevation view.

To choose Right view

1. Activate a Draft or Sculpt mode window.
2. Choose View > Right. Or, press Ctrl+6 (Windows), or Cmd+6 (Mac OS).

CLIPPING PLANE SUBMENU

The Clipping Plane submenu commands let you view particular sections of your project, activate defined clipping planes, and deactivate clipping planes. Clipping planes create a section of an object by “slicing through” the object at the location of each clipping plane.

DEFINE CLIPPING PLANE

Modes: Sculpt, Render

You can use the Define Clipping Plane command to view a section of your project.

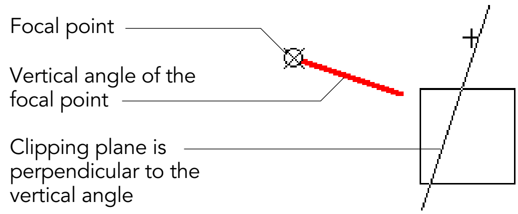
To set clipping planes in Sculpt mode

1. Choose View > Define Clipping Plane. DENEBACAD changes to a Front view if you were in Top or Bottom view or to Top view if you were in an elevation view.
2. Move the pointer to set the location of the first clipping plane. A line follows the pointer. This line is one of the clipping planes.
3. Click to set the location of the clipping plane.
4. Move the pointer to set the location of the second clipping plane. A second line, parallel to the first, follows the pointer. This line represents the second clipping plane.
5. Click to set the location of the second clipping plane. DENEBACAD returns to the original view. DENEBACAD displays the objects and parts of objects that occupy the space between the two clipping planes.

To set clipping planes in Render mode

1. Choose View > Define Clipping Plane. DENEBACAD changes to a Sculpt mode window set to Front view.
2. Move the pointer to set the location of the first clipping plane. A line, perpendicular to the vertical angle of the camera focal point, follows the pointer. This line is one of the clipping planes.

   ▲ The angle of the clipping plane is defined by the angle of the vertical focal point. (You define the vertical focal point using the Vertical Focal point tool in Render mode.) The focal point is represented by a circle with an X. A red line indicates the vertical angle of the focal point. The clipping plane is perpendicular to the vertical angle.



3. Click to set the location of the clipping plane.
4. Move the pointer to set the location of the second clipping plane. A second line, parallel to the first, follows the pointer. This line represents the second clipping plane.
5. Click to set the location of the second clipping plane. The active window returns to Render mode. DENEBACAD displays the objects and parts of objects that occupy the space between the two clipping planes.

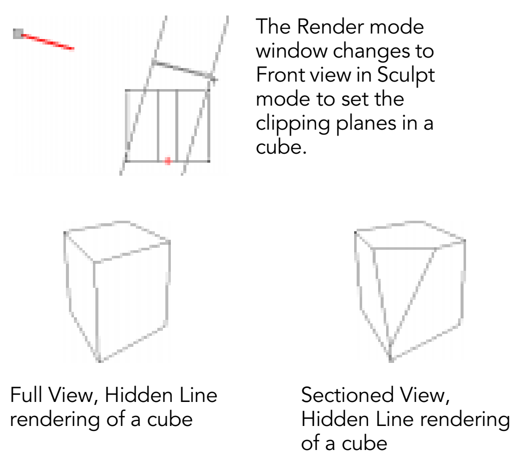


Fig. 287 Clipping planes in Render mode

If you change the focal point after setting clipping planes, DENEBACAD activates Full View in the Sections submenu. This displays the entire rendering. By choosing Sectioned View in the Sections pop-up menu, you can view a clipped section of your rendering from the sightline orientation defined by the new focal point you set. The orientation of the clipping planes is now perpendicular to the new sight line.

FULL VIEW

Modes: Sculpt, Render
Mac OS Shortcut: Cmd+[
Windows Shortcut: Ctrl+[

You can use the Full View command to deactivate clipping planes. Once clipping planes are deactivated, you can reactivate them at any time using the Sectioned View command in the Clipping Plane submenu. This command works the same way in Sculpt mode and Render mode. This command is also available in the Sections pop-up menu on the Status bar.

◆ **To deactivate clipping planes:** Choose View > Clipping Plane > Full View. Or, press Ctrl+[(Windows), or Cmd+[(Mac OS).

SECTIONED VIEW

Modes: Sculpt, Render
Mac OS Shortcut: Cmd+]
Windows Shortcut: Ctrl+]

You can use the Sectioned View command to activate clipping planes. Once clipping planes are activated, you can deactivate them using the Full View command.

Sectioned View works the same in Sculpt and Render modes. The command is also available in the Sections pop-up menu on the Status bar.

◆ **To activate clipping planes:** Choose View Clipping Plane > Sectioned View. Or, press Ctrl+] (Windows), or Cmd+] (Mac OS).

REFRESH

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+K
Windows Shortcut: F5

By using the Refresh command, you can refresh the display in the active Draft or Sculpt document window and redraw all visible objects onscreen. This might be necessary when you scroll or change the view magnification, or change the stacking order for objects.

◆ **To use the Refresh command:** Choose View > Refresh. Or, press F5 (Windows), or Cmd+K (Mac OS). DENEBACAD redraws all visible objects in the active document window.

RELATIVE VIEW SUBMENU

In addition to the six standard orthogonal views, DENEBACAD can show a project from non-standard views based on angles you define graphically or numerically. These non-orthogonal views are called *relative views*. Because relative views are non-orthogonal, they can be used to integrate various parts of your project from non-orthogonal perspectives. In other words, relative views let you view 3D objects from angles that are not parallel to any axis in a standard view.

A relative view can be thought of as a separate project workspace, which is relative to the DENEBACAD workspace. When you activate a relative view, you can draw objects, create layer sets, define extrusion planes, and generally work in a workspace related to, but independent of the DENEBACAD workspace. You can relate the work you do in a relative view to the DENEBACAD workspace by choosing Default Relative View. (The DENEBACAD workspace is the default relative view.) In the default relative view, you can view objects drawn in a relative view based on the relative view's specified angle.

While relative views can be created, activated, or deactivated only in Sculpt mode, they affect the way you see your project in both Draft and Sculpt modes. To see a relative view affect objects in Draft mode, you must set your View Options to show 2D and 3D objects. This functions in a manner similar to the six standard views which act independently in Draft mode like separate drawings.

CUSTOM RELATIVE VIEW

Mode: Sculpt

Custom Relative View is an indicator that an unsaved relative view is in effect in Sculpt mode. "Custom Relative View" appears with a check mark in the View > Relative View submenu, and appears in the Status bar on the Relative View pop-up menu button.

Custom Relative View is not a command; you can not choose it from the menu to restore an unsaved custom relative view. Instead, if you switch from an unsaved relative view to the default relative view or a saved relative view, the unsaved custom relative view is discarded.

DEFAULT RELATIVE VIEW

Mode: Sculpt
Mac OS Shortcut: Cmd+Opt+Shift+S
Windows Shortcut: Ctrl+Alt+Shift+S

The Default Relative View command selects the Default relative view. The Default Relative View command appears in the Relative View pop-up menu when you select it, or until you define a relative view

In addition, when you activate a Sculpt mode window, DENEBACAD shows you if the Default Relative View command is active. If so, Default Relative View appears in the Relative View pop-up menu on the Status bar, and a check mark appears next to the command's name in the Relative View submenu.

Fig. 288 Default Relative View active

To use the Default Relative View command

1. If necessary, open and activate a Sculpt mode window.
2. To activate the Default relative view, choose Default Relative View in the Relative View pop-up menu on the Status bar or in the Relative View submenu.

DEFINE HORIZONTAL ANGLE

Mode: Sculpt

You can use the Define Horizontal Angle command to create a relative view. You define the relative view by choosing a point and angle to view the document.

To define a relative view

1. Choose View > Relative View > Define Horizontal Angle. DENEBACAD changes to Top view and the pointer changes to a relative view pointer.

   ▲ After using the Define Horizontal Angle command, the window remains in Top view.
2. Place the pointer where you want to set the Relative point.
3. Click to set the point. The relative view sight line is represented by the drawing vector as it extends and revolves around the set point. A cone of vision also extends from the set point.
4. Click to set the horizontal direction of the sight line.

You now view the objects in the window from the horizontal angle of the point you set.

DEFINE NUMERICALLY

Mode: Sculpt

You can use the Define Numerically command to create a relative view. You define the relative view by entering numbers for the horizontal and vertical angle from which you want to view the objects in your document.

To define a relative view numerically

1. Choose View > Relative View > Define Numerically. The Relative View dialog box opens.

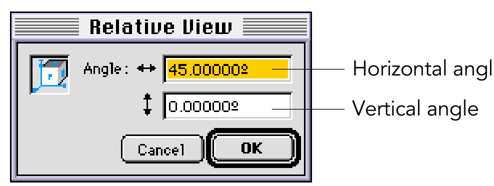


Fig. 289 Defining a relative view numerically

2. Type the relative view's horizontal and vertical angles in the text boxes (*fig. 289*), and then Click OK.

You now view the objects in the window from the vertical and horizontal angles you set.

DEFINE VERTICAL ANGLE

Mode: Sculpt

You can use the Define Vertical Angle command to create a relative view. You define the relative view by choosing a point and angle to view the document.

To define a relative view

1. Choose View > Relative View > Define Vertical Angle. DENEBACAD changes to Right view and the pointer changes to a relative view pointer.

   ▲ After using the Define Vertical Angle command, the document window is set to "Top" view.

2. Place the pointer where you want to set the Relative point.
3. Click to set the point. The relative view sight line is represented by the drawing vector as it extends and revolves around the set point. A cone of vision also extends from the set point.
4. Click to set the vertical direction of the sight line. DENEBACAD switches to "Top" view.

You now view the objects in the window from the vertical angle of the point you set.

DELETE RELATIVE VIEW

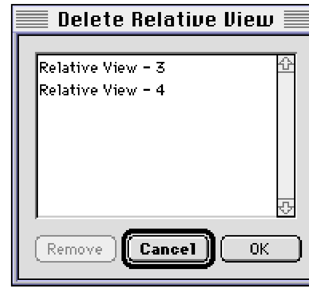
Modes: Sculpt

The Delete Relative View command deletes saved relative views. When you delete a relative view, its name disappears from the Relative View pop-up menu on the Status bar, and the Relative View submenu.

If you try to delete the active relative view, a warning message appears. In this case, DENEBACAD maintains the current relative view as the Custom Relative View.

To use the Delete Relative View command

1. Choose View > Relative View > Delete Relative View. The Delete Relative View dialog box appears.



2. In the dialog box, select the names of the relative views you want to delete.

   ▲ To select (or deselect) a continuous range of relative views, press Shift and click the first and last relative views' names.

   ▲ To select (or deselect) several non-contiguous relative views, press Ctrl (Windows) or Cmd (Mac OS) and click each relative view's name.

3. To remove the relative views, click Remove.
4. To permanently delete the removed relative views, click OK.

If you click Cancel, removed relative views are restored and DENEBACAD closes the dialog box.

SAVE RELATIVE VIEW

Mode: Sculpt

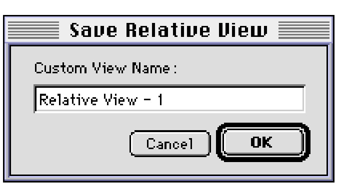
The Save Relative View command saves a relative view. When you save a relative view, the names of relative views appear in the Relative View pop-up menu on the Status bar, and at the bottom of the Relative View submenu.

For example, when you define a relative view, you activate this custom relative view; “Custom Relative View” appears in the Status bar and has a check mark in the View > Relative View submenu. You can use the Save Relative View command to save the relative view with an assigned name.

To avoid confusion, remember “Custom Relative View” in the Relative View submenu is not a saved custom relative view. Anytime you define a relative view, you create “custom” relative view. When you use the Save Relative View command, DENEBACAD saves the custom relative view with the name you enter. This name appears in the View > Relative View submenu and the pop-up menu on the Status bar, and replaces “Custom Relative View.”

To use the Save Relative View command

1. If necessary, open or activate a Sculpt mode window.
2. If necessary, create a relative view. “Custom Relative View” appears in the Status bar.
3. Choose View > Relative View > Save Relative View. In the Save Relative View dialog box, type a name and click OK.



Selecting a saved relative view

The names of saved relative views appear in the Relative View pop-up menu on the Status bar, and at the bottom of the View > Relative View submenu.

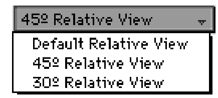


Fig. 290 Saved relative views

In DENEBACAD, there’s always a relative view active; it can be the Default relative view, an unsaved custom relative view, or a saved relative view. Once you save a relative view, it appears in the Relative View pop-up menu.

In addition, when you activate a Sculpt mode window, DENEBACAD shows you if a saved relative view is active. If so, the name of the relative view appears in the Status bar, and a check mark appears next to the name in the View > Relative View submenu.

To select a saved relative view

1. If necessary, open or activate a Sculpt mode window.
2. To select a saved relative view, choose the name of the saved relative view in the Relative View pop-up menu, or in the View > Relative View submenu.

SNAPSHOT SUBMENU

To make viewing and navigating easier, you can take “snapshots” of a drawing. A snapshot records the zoom level and visible area of the active window. After you save snapshots, you can choose one in the Snapshot submenu to return to the same view of a drawing.

The commands in the Snapshot submenu let you save, use, and delete snapshots. These commands are available in Draft, Sculpt, and Render modes.

The Snapshot submenu appears in the View menu in all modes. This section describes the Snapshot submenu commands.

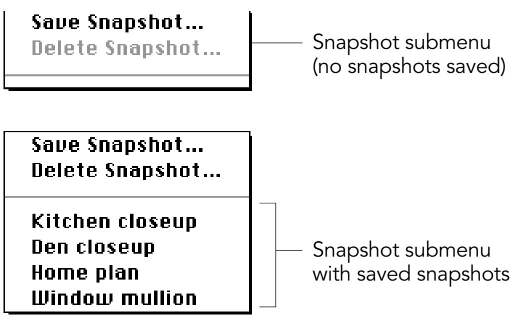


Fig. 291 Snapshot submenu

SAVE SNAPSHOT

Modes: Draft, Sculpt, Render

The Save Snapshot command remembers the current zoom level and the part of the document visible in the active window.

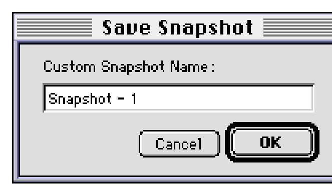
The Save Snapshot command lets you save snapshots that show different areas of your drawing at different zoom levels.

When you save a snapshot, you give it a name; the name appears at the bottom of the Snapshot submenu. By choosing the name in the submenu, it’s easy to switch to different areas of a drawing at different zoom levels. You can even use snapshots to change modes and views.

Saving snapshots in Render mode lets you easily preserve Focal Point positions (camera angles) for viewing renderings. After using the Save Snapshot command, you can make multiple changes that affect a rendering, without having to reconfigure camera angles.

To use the Save Snapshot command

1. Choose View > Snapshot > Save Snapshot. The Save Snapshot dialog box appears.



2. Type a name for the snapshot. The name can contain up to 24 characters.

   ▲ If you intend to save many snapshots taken in different modes and views, it’s important to assign names that indicates the mode and view of each snapshot. Otherwise, it will be hard to know whether choosing a particular snapshot will change the mode or view of the active window.

3. Click OK. DENEBACAD saves the snapshot, and adds its name to the bottom to the Snapshot submenu in the View menu.

CHOOSING SNAPSHOTS

You must save a snapshot with the Save Snapshot command before you can choose the snapshot from the Snapshot submenu.

You can choose a saved snapshot in any mode or view. Keep in mind that DENEBACAD will switch the active window to the mode and view in which the snapshot was taken.

◆ **To choose a snapshot:** Choose *saved snapshot* in the View > Snapshot submenu, where *saved snapshot* is the name of the snapshot you want to use. DENEBACAD switches the active window to the zoom level and visible area “remembered” by the snapshot. If necessary, DENEBACAD also switches to the mode and view in which you took the Snapshot.

DELETE SNAPSHOT

Modes: Draft, Sculpt, Render

The Delete Snapshot command lets you delete snapshots you have saved with the Save Snapshot command.

The Delete Snapshot command is not available if no saved snapshots exist in the document.

Note: You cannot use the Undo command to restore snapshots that you have deleted with the Delete Snapshot command.

To use the Delete Snapshot command

1. Choose View > Snapshot > Delete Snapshot. The Delete Snapshot dialog box appears.
2. In the dialog box, select the name of the Snapshot (or snapshots) you want to delete. When you select a snapshot, the name is highlighted and the Remove button becomes available.

   ▲ To select (or deselect) a continuous range of snapshots, press Shift and click the first and last snapshots’ names.

   ▲ To select (or deselect) several non-contiguous snapshots, press Ctrl (Windows) or Cmd (Mac OS) and click each snapshot’s name.
3. To remove the selected snapshots from the list, click Remove.
4. To permanently delete the snapshots removed from the list, click OK. The Delete Snapshot dialog box closes and DENEBACAD removes the snapshots’ names from the Snapshot submenu.

   ▲ If you decided to close the dialog box without deleting the snapshots marked for deletion, click Cancel.

VIEW OPTIONS SUBMENU

The commands in the View Options submenu let you view 2D and 3D objects in either Draft or Sculpt mode.

When the Status bar is displayed, you can choose View Options commands and see the active command.

VIEW OPTIONS IN DRAFT MODE

To choose View 2D Objects, View 2D & 3D Objects, or View 2D & Lock 3D Objects, you must have a Draft mode window active.

Choosing a View Options command affects the active window only. If you have multiple Draft windows open, you need to activate each one to change its View Options.

The active View Option appears in the View Options pop-up menu at the left end of the Status bar. Additionally, a check appears next to the active View Option in the View Options submenu.

When you choose a View Option, a dialog box appears to remind you that you've activated a 2D and 3D design environment. 2D objects are displayed in red and 3D objects are displayed in blue. This overrides all other display options for the objects in that window.

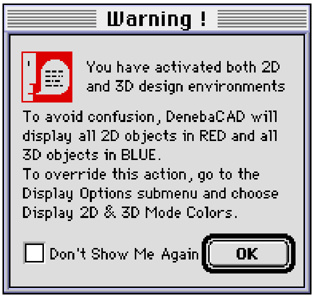


Fig. 292 Warning dialog box

VIEW 2D OBJECTS

Mode: Draft
Mac OS Shortcut: Cmd+Opt+4
Windows Shortcut: Ctrl+Alt+4

View 2D Objects displays 2D objects only.

◆ **To view 2D objects only:** Choose View > View Options > View 2D Objects. Or, press Ctrl+Alt+4 (Windows), or Cmd+Opt+4 (Mac OS). Or use the pop-up menu in the Status bar.

VIEW 2D & 3D OBJECTS

Modes: Draft
Mac OS Shortcut: Cmd+Opt+5
Windows Shortcut: Ctrl+Alt+5

View 2D & 3D Objects displays 2D objects in red and 3D objects in blue.

◆ **To display 2D and 3D objects:** Choose View > View Options > View 2D & 3D Objects. Or, press Ctrl+Alt+5 (Windows), or press Cmd+5 (Mac OS). Or, use the pop-up menu in the status bar.

VIEW 2D & LOCK 3D OBJECTS

Mode: Draft
Mac OS Shortcut: Cmd+Opt+6
Windows Shortcut: Ctrl+Alt+6

View 2D & Lock 3D Objects displays 2D objects in red and 3D objects in blue, with 3D objects locked. You can't move, delete, or edit locked 3D objects.

◆ **To view 2D and locked 3D objects:** Choose View > View Options > View 2D & Lock 3D Objects. Or, press Ctrl+Alt+6 (Windows), or Cmd+Opt+6 (Mac OS). Or, use the pop-up menu in the Status bar.

After choosing this command, you can choose Display Locked Objects in Gray in the Display Options submenu to display locked 3D objects in gray instead of blue.

VIEW OPTIONS IN SCULPT MODE

To choose View 3D Objects, View 3D & 2D Objects, or View 3D & Lock 2D Objects, you must have a Sculpt mode window active.

Choosing a View Options command affects the active window only. If multiple Sculpt windows are open, activate each one to use View Options.

The active View Option appears in the View Options pop-up menu at the left end of the Status bar. Additionally, the active View Option in the View Options submenu has a check mark.

When you choose a View Option, a message reminds you that you've activated a 2D and 3D design environment. 2D objects are displayed in red and 3D objects are displayed in blue. This overrides all other display options for the objects in that window.

VIEW 3D OBJECTS

Mode: Sculpt
Mac OS Shortcut: Cmd+Opt+4
Windows Shortcut: Ctrl+Alt+4

View 3D Objects displays 3D objects only.

◆ **To view 3D objects only:** Choose View > View Options > View 3D Objects. Or, press Ctrl+Alt+4 (Windows), or Cmd+Opt+4 (Mac OS). Or, use the pop-up menu in the Status bar.

VIEW 3D & 2D OBJECTS

Mode: Sculpt
Mac OS Shortcut: Cmd+Opt+5
Windows Shortcut: Ctrl+Alt+5

View 3D & 2D Objects displays 2D objects in red and 3D objects in blue.

◆ **To view 3D and 2D objects:** Choose View > View Options > View 3D & 2D Objects. Or, press Ctrl+Alt+5 (Windows), or press Cmd+Opt+5 (Mac OS). Or, use the pop-up menu at the left end of the Status bar.

VIEW 3D & LOCK 2D OBJECTS

Mode: Sculpt
Mac OS Shortcut: Cmd+Opt+6
Windows Shortcut: Ctrl+Alt+6

View 3D & Lock 2D Objects displays 2D objects in red and 3D objects in blue, with 2D objects locked. You can't move, delete, or edit locked 2D objects.

◆ **To view 3D and locked 2D objects:**

Choose View 3D & Lock 2D Objects in the View Options submenu, press Ctrl+Alt+6 (Windows), or press Cmd+Opt+6 (Mac OS). Or, choose View 3D & Lock 2D Objects in the pop-up menu at the left end of the Status bar.

After choosing this command, you can choose Display Locked Objects in Gray in the Display Options submenu to display locked 2D objects in gray instead of red.

WINDOW MENU COMMANDS 19

Commands in the Window menu let you manage multiple windows to draft, model, and render your drawings. Commands in this menu also display palettes that let you manage objects and information in a DENEBACAD document.

Show and Hide commands

The first part of this chapter describes how to use the “Show” and “Hide” commands in the Window menu to manage Draft, Sculpt, and Render windows.

Palette commands

The Window menu contains several commands that display palettes for managing objects, layers, and data. The following commands, and the associated palettes, are described later in this chapter:

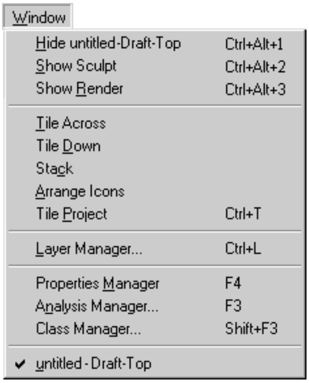
Analysis Manager Displays a palette for data analysis (see “Analysis Manager” on page 354).

Class Manager Displays a palette for managing Classes and class data (see “Class Manager” on page 360).

Layer Manager Displays a palette for managing layers in Draft and Sculpt mode (see “Layer Manager” on page 363).

Properties Manager Displays a palette containing editable object data for modifying and managing objects (see “Properties Manager” on page 375).

Mac OS



Windows

USING SHOW / HIDE WINDOW COMMANDS

The three commands that appear at the top of the Window menu let you manage Draft, Render, and Sculpt windows. You use these windows for 2D drafting, 3D modeling, and rendering in DENEBACAD.

Show Draft / Hide Draft Opens a Draft mode window or closes the same window.

Show Sculpt / Hide Sculpt Opens a Sculpt mode window or closes the same window.

Show Render / Hide Render Opens a Render mode window or closes the same window.

Each of the window management commands begins with the word *Show* or *Hide.* When you use one of the Show commands, the command is replaced by a Hide command. Each open window has a corresponding Hide command.

For example, if you open Draft and Render windows for a document, the following commands appear in the Window menu:

Hide *name*-**Draft-Top**
Show Sculpt
Hide *name*-**Render**

The *name* that appears in the command is the name of the document. Before you save a document with a different name, the word *untitled* appears as the document's name.

Each Show or Hide command also includes the mode and the view that the window is set to. For example, a window in Draft mode in Top view would have a corresponding menu command of **Hide** *untitled*-**Draft-Top**.

When you create a new document, DENEBACAD opens a single Draft mode window. In this case, the first command in the Window menu is Hide *name*-Draft-Top, which lets you hide the Draft window. The other two commands let you open a Sculpt mode window and a Render mode window for the document.

Closing windows and documents

Each Hide command in the Window menu closes a single window, the same as clicking the window's close box. This is in contrast to the Close command in the File menu. The Close command closes the current document and all of its open windows.

However, if you use a Hide command when only one document window is open, DENEBACAD closes the document, the same as if you used the Close command. If the document has been changed but has not been saved, DENEBACAD asks if you want to save the changes before closing the document.

To save changes before closing the document, click Save. To close the document without saving changes, click Don't Save. To continue working in the document without closing the window, click Cancel.

Using multiple windows

You can open up to three windows to view a document in different modes — Draft, Sculpt, and Render — and different views — Top, Bottom, Front, Back, Left, and Right.

When you open a new window, or click a window to make it active, DENEBACAD switches to the mode of that window. The program operates in one mode at a time. Switching modes requires only that you switch to (or open) a window in a different mode.

For example, if you are working in Draft mode, and you choose the Show Sculpt command to open a Sculpt window, this makes the new Sculpt mode window active and switches you to working in Sculpt mode.

Changing modes in DenebaCAD changes the tools available in the toolbox, as well as changing the available menu commands and the available Action buttons (if the Action buttons are displayed).

Changing window modes

As mentioned previously, you can change modes by opening a new window or switching to another open window. However, you can also change modes by using the Mode pop-up menu at the bottom of a window to change the window's mode (*figure 293*).

You need to use this technique if you want to have more than one Draft, Sculpt, or Render window open at one time. By choosing a mode from a window's mode pop-up menu, you change the mode of that window, and the mode in which you are working, without affecting other open windows.

Fig. 293 Window mode pop-up menu

If you change the mode of any open windows, the Hide commands in the Window menu will correspond to the modes of the open windows. In other words, if there are three Sculpt mode windows open, the three commands at the top of the Window menu will be Hide Sculpt commands.

Tiling windows

By default, DENEBACAD arranges open windows so they do not overlap. This feature is called "Auto-tile windows." You can change this preference setting in the Preferences dialog box. If you deselect the "Auto-tile windows" checkbox, each new window that you open will fill the drawing window area.

You can use the Tile command in the Window menu to arrange open windows so they do not overlap.

HIDE DRAFT / SHOW DRAFT

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+Opt+1
Windows Shortcut: Ctrl+Alt+1

The Show Draft command opens a Draft mode window. The Hide Draft command closes the Draft mode window.

The Show Draft command is available if you have not opened a Draft mode window for the current document. The Hide Draft command is available if a Draft mode window is open for the current document.

◆ **To open a Draft window:** Choose Show Draft in the Window menu. DENEBACAD opens a Draft mode window in Top view.

◆ **To close a Draft mode window:** Choose Hide Draft in the Window menu or click the Window's close box. DENEBACAD closes the Draft mode window.

If you use the Hide Draft command when only one document window is open, DENEBACAD closes the document, or asks if you want to close the document if it has not been saved. To save changes before closing the document, click Save.

HIDE SCULPT / SHOW SCULPT

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+Opt+2
Windows Shortcut: Ctrl+Alt+2

The Show Sculpt command opens a Sculpt mode window. The Hide Sculpt command closes the Sculpt mode window.

The Show Sculpt command is available if you have not opened a Sculpt mode window for the current document. The Hide Sculpt command is available if a Sculpt mode window is open for the current document.

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

◆ **To close a Sculpt mode window:** Choose Hide Sculpt in the Window menu or click the window's close box. DENEBACAD closes the Sculpt mode window.

If you use the Hide Sculpt command when only one document window is open, DENEBACAD closes the document, or asks if you want to close the document if it has not been saved. To save changes before closing the document, click Save.

HIDE RENDER / SHOW RENDER

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+Opt+3
Windows Shortcut: Ctrl+Alt+3

The Show Render command opens a Render mode window. The Hide Render command closes the Render mode window.

The Show Render command is available if you have not opened a Render mode window for the current document. The Hide Render command is available if a Render mode window is open for the current document.

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

◆ **To close a Render mode window:** Choose Hide Render in the Window menu or click the window's close box. DENEBACAD closes the Render mode window.

If you use the Hide Render command when only one document window is open, DENEBACAD closes the document, or asks if you want to close the document if it has not been saved. To save changes before closing the document, click Save.

TILE PROJECT

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+T
Windows Shortcut: Ctrl+T

The Tile Project command fits open windows within the screen area and arranges them so they don't overlap.

With two windows open, Tile Project places one window on the left and the other on the right of the screen. With three windows open, the Tile Project command puts one window on the left and the other two windows on the right of the screen. If you're working with Draft, Sculpt, and Render windows open, DENEBACAD puts the Draft window on the left, the Sculpt window on the top right, and the Render window on the bottom right.

You can also use the Tile Project command to reposition document windows if you change the resolution of your monitor while you work.

◆ **To use the Tile Project command:** Choose Tile Project in the Window menu, press Ctrl+T (Windows) or press Cmd+T (Mac).

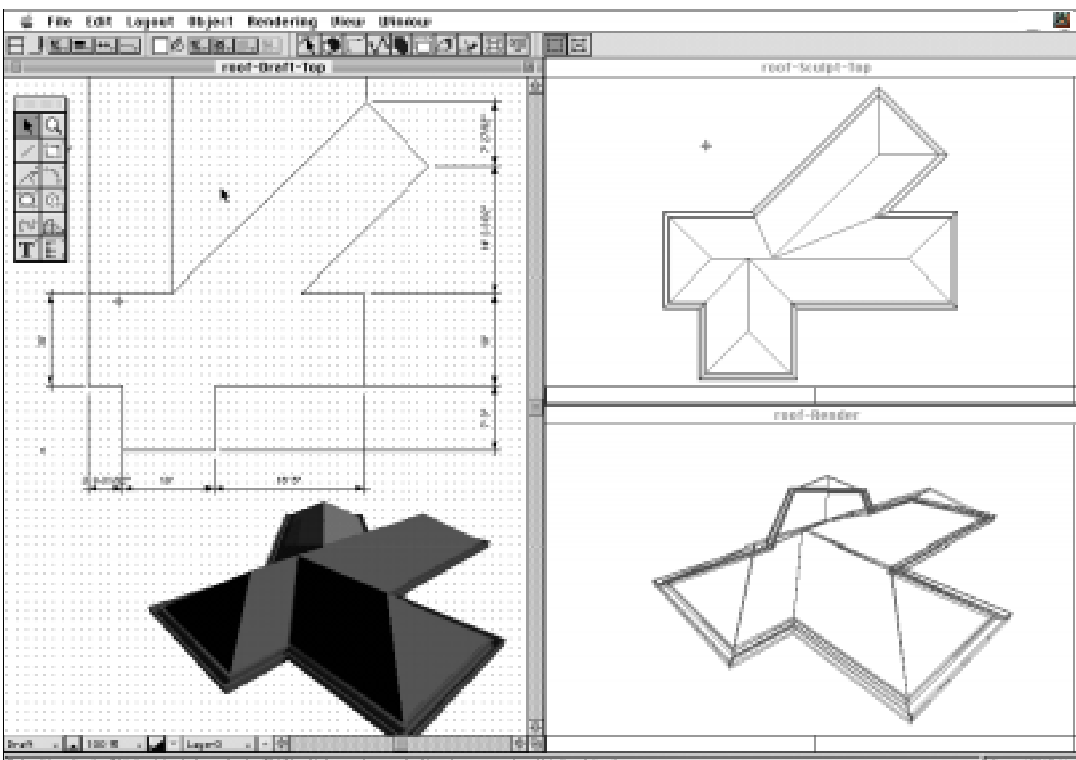


Fig. 294 Tile command window arrangement

Auto-tiling preference

By default, DENEBACAD arranges windows as if you had chosen the Tile command whenever you open a new window.

This behavior is controlled by the "Auto-tile windows" option in the Preferences dialog box. As long as this option is selected, you do not need to use the Tile command. If you deselect the "Auto-tile windows" option, you can use the Tile command to arrange multiple windows.

ANALYSIS MANAGER

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Shift+I
Windows Shortcut: F3

The Analysis Manager command displays the Analysis Manager. You can use the Analysis Manager to create summary reports on the objects in a DENEBACAD document. You can export this information as a text file and then use a spreadsheet or database application to make calculations and print reports.

The Analysis Manager can remain open while you work. It floats on the desktop, appearing in front of drawing windows.

◆ **To display the Analysis Manager:** Choose Analysis Manager in the Window menu, press F3 (Windows), or Cmd+Shift+I (Mac OS). DENEBACAD displays the Analysis Manager. When first displayed, the Analysis Manager appears in a two-column format.

◆ **To close the Analysis Manager:** Click the close box at the upper-left corner of the Analysis Manager.

DISPLAYING DATA IN THE ANALYSIS MANAGER

The Analysis Manager window (*figure 295*) contains a row of pop-up menu buttons near the top. Below the menus is a resizeable scrolling list area where report data is displayed.

You can think of the Analysis Manager as a report generator and data display window that is organized like a spreadsheet.

To create a report in the Analysis Manager, you select data types from the pop-up menus and choose the Update command. The scrolling list area displays object data in rows and columns.

Data field menus

Continuing the spreadsheet metaphor, the column headings in the Analysis Manager are data types that you choose from the pop-up menus. For example, if you choose the data type "Tool" in the "Data Field 1:" pop-up menu, the list area displays in the first column the names of tools used in the document.

Data rows

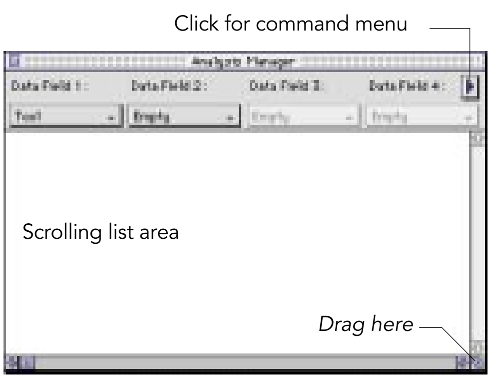
Each row displays information on one object or a collection of objects. A row in the Analysis Manager is similar to a record in a database. For example, if you choose a Class name as the first column heading, each row displays the name of a Class object defined in the document. If the next column displays the “Total” data type, each row displays the total number of associated objects.

The Analysis Manager window

When you first open the Analysis Manager, two pop-up menu buttons appear above the scrolling list. You can expand the manager by dragging the lower-right corner of the window to the right. As you expand the window, additional pop-up menu buttons appear.

You can also use the horizontal scroll bar at the bottom of the Analysis Manager to view columns to the right of those visible in the window.

The labels above the buttons identify the pop-up menus (from left to right) as Data Field 1, Data Field 2, Data Field 3, and so on through Data Field 8. When the manager window is fully extended, eight buttons appear.



Drag the corner to extend the Analysis Manager or use the scroll bar to view additional columns.

Fig. 295 Using the Analysis Manager window

Analysis Manager commands

A pop-up menu in the Analysis Manager contains commands to update the items displayed; to save the current report; and to select objects in the document based on a selected item in the scrolling list.

This pop-up menu is located under the arrow button at the upper-right corner of the Analysis Manager, as shown in (*figure 296*).

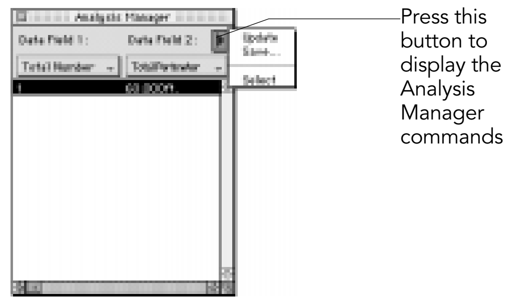


Fig. 296 Analysis Manager commands

The following commands appear in the pop-up menu:

Update Choose the Update command to update the items displayed in the scrolling list. You need to choose this command whenever you change a data field.

Save Choose Save to save the current listing in a text file.

Select Choose Select to select objects in the document based on the row of data that is selected in the scrolling list.

To use the Analysis Manager

1. Choose Analysis Manager in the Window menu to display the Analysis Manager.
2. Extend the Analysis Manager to the right by dragging the lower-right corner if necessary. You can extend the window to display up to eight columns of data.

3. In the Data Field 1 pop-up menu (above the first column on the left), choose the data item you want to display.

4. To display additional columns of data, choose data items from the other Data Field pop-up menus.

5. Choose Update in the pop-up menu under the arrow button. The Analysis Manager updates the data displayed in the scrolling list area.

Exporting data from the Analysis Manager

Once you compile a report in the Analysis Manager, you can export the data as a tab-delimited text file.

Exporting the Analysis Manager data lets you work with the data in any spreadsheet or database program that can import tab-delimited text.

You can use the calculation and data analysis features of a spreadsheet or database program to manipulate Analysis Manager data.

For example, you can use an Analysis Manager report in a spreadsheet program to calculate a project's cost of materials. You can do this by multiplying a column of price data by the item totals in another column.

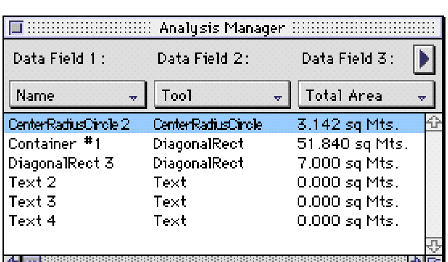


Fig. 297 An Analysis Manager report

When you select a different heading or modify the document, the window in the Analysis manager clears. So, to keep a report visible in the Analysis Manager window, make sure you're finished with the report before making changes.

To export an Analysis Manager report

1. Compile a report in the Analysis Manager by following the steps under "To use the Analysis Manager" on page 355.

2. Choose Save in the pop-up menu under the arrow button. A directory dialog box appears.

3. Select a location for saving the exported report and type a file name in the "Save a copy in" text box.

4. Click Save to save the report file.

Selecting objects with the Analysis Manager

You can use the Analysis Manager to select objects in a document. This feature can help you identify a particular item or to select a series of objects so you can apply a command to the entire selection.

When you use the Select feature in the Analysis Manager, you can make a selection based on one row in the Analysis Manager scrolling list.

- If you highlight a row that describes a collection of objects (such as a Class or Container) and choose the Select command in the pop-up menu, DENEBACAD selects all the objects in the collection.

- If you highlight a row that describes one object, DENEBACAD selects that one object when you choose the Select command. You can then highlight a different row and choose Select again to add to the current selection.

To select objects with the Analysis Manager

1. Compile a report in the Analysis Manager by following the steps under "To use the Analysis Manager" on page 355.
2. Click a row of data you want to use to make a selection. The row you click becomes highlighted.
3. Choose Select in the pop-up menu under the arrow button.
4. To make additional selections, click another row of data to highlight the row, and choose Select in the pop-up menu under the arrow button. DENEBACAD selects the objects described by the highlighted row, adding these objects to the current selection.

SELECTING DATA TYPES

The pop-up menus in the Analysis Manager define the column headings for the display of data in the scrolling list area. The first pop-up menu defines the data type for the first column from the left in the scrolling list. The next pop-up menu defines the data type for the next column of the scrolling list, and so on.

You can select or change the data displayed in any column by choosing a data type from the pop-up menu above the column.

Each pop-up menu contains four items: Empty, Computing, General, and Object Class.

Choosing Empty turns off the display of data in a column and all columns to the right. The pop-up menus to the right of the "Empty" column are dimmed and not available. Data won't appear in these columns when you choose the Update command.

The other three items — Computing, General, and Object Class — display submenus of data types. Drag to a data type in the pop-up submenu to choose it. The data type you choose appears in the pop-up menu button until you choose another data type.

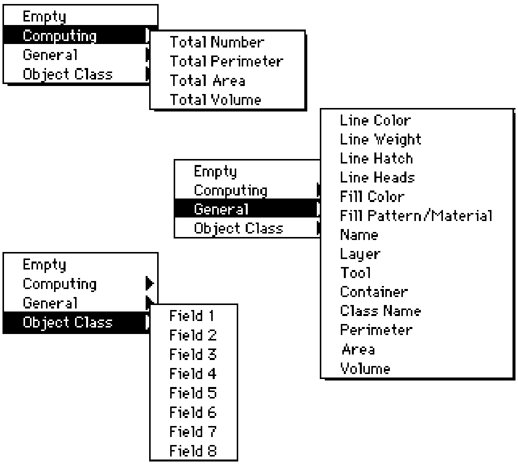


Fig. 298 Data categories in the Analysis Manager

The data types available in the pop-up menus are described below.

Computing data types

The Computing submenu contains four data types: Total Number, Total Perimeter, Total Area, and Total Volume.

- If you select a computing value in a column, the pop-up menu for the next column allows only the use of the remaining computing values. If you select all computing values, you can't make anymore selections.

Total Number Counts the number of items listed in each row and displays the total count.

Total Perimeter Calculates the total perimeter measurement of the objects listed in each row and displays the perimeter measurement. If an object is a polygon, its perimeter is computed. If the object is a group, the sum of the perimeters of the individual objects is displayed.

Total Area Calculates the total area of the objects listed in each row and displays the area measurement. If an object is a polygon, its area is computed. If the object is a group, the sum of the areas of the individual objects is displayed. If an object is an open polyline, its area is zero.

Total Volume Calculates the total volume of the objects listed in each row and displays the volume measurement. Volume is computed for 3D objects only.

General data types

The General submenu contains the following data types: Line Color, Line Weight, Line Hatch, Line Heads, Fill Color, Fill Pattern, Name, Surface material, Layer, Tool, Container, Class Name, Perimeter, Area, and Volume.

Line Color This data type displays the numeric RGB color values of an object's Pen color. This data is formatted as RGB: nnn,nnn,nnn (three, three-digit numbers). These numbers are the color values of the red, green, and blue components of the object's Pen color. The color values are measured on a scale of 0 to 100 percent. For example, the data for a pure red Pen color appears as RGB:100,000,000, indicating that the Pen color is 100 percent red, zero percent green, and zero percent blue in the RGB color system.

This method of displaying color data is also used on the Pen color tab in the Pen palette.

Line Weight This data type displays the Pen weight of an object. The data appears as Pen:n. The number following Pen is the position of the object's Pen weight in the Pen weight tab, counting from the top of the tab. The same number appears at the bottom of the tab when you select a Pen weight in the Pen palette.

Line Hatch This data type displays the name of the Fill hatch assigned to an object. The name of the Fill hatch also appears in the Name box on the Fill hatch tab.

Line Heads This data type identifies the Arrowheads assigned to an object. The data appears as two numbers separated by a comma. The first number identifies the Arrowhead at the first endpoint of the line; the second number identifies the Arrowhead at the second endpoint of the line. The number 0 indicates that no Arrowhead is assigned. Numbers 1 and higher indicate the Arrowhead's position in the Arrowhead tab of the Pen palette, counting from the top of the tab. For example, if a line has no Arrowhead at its first endpoint, and the Arrowhead at the line's second endpoint is the third Arrowhead from the top of the tab, this data appears as 0,3.

Fill Color This data type identifies the numeric RGB color values of an object's Fill color. This data is formatted as RGB: nnn,nnn,nnn (three, three-digit numbers). These numbers are the color values of the red, green, and blue components of the object's Fill color. The color values are measured on a scale of 0 to 100 percent. For example, the data for a pure blue Pen color appears as RGB:000,000,100, indicating that the Pen color is zero percent red, zero percent green, and 100 percent blue in the RGB color system.

This method of displaying color data is also used on the Fill color tab in the Fill palette.

Fill Pattern This data type displays the name of a selected object's Fill pattern, if one has been applied to the object. When you view a Fill pattern in the Fill palette, the pattern's name appears in the Name text box.

Surface material This data type displays the name of a selected object's Surface material, if one has been applied to the object. The name of a Surface material appears in the Properties Manager when the object is selected.

Name This data type displays the name of an individual object. DENEBACAD assigns a name to every object when it is created, based on the object type and a number. The name of an object appears in the Properties Manager when the object is selected. You can change the object's name by typing in the Name text box on the Info tab in the Properties Manager.

Layer This data type displays the name of the layer containing an object. You can assign layer names in the Layer Manager.

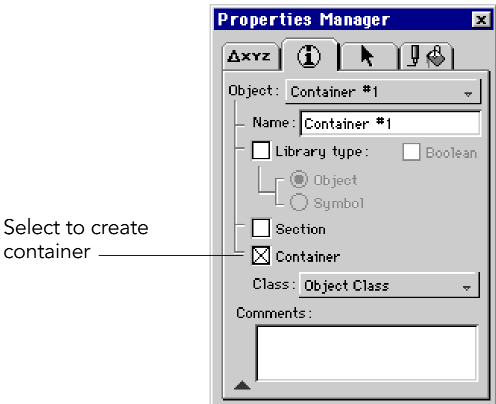
Tool This data type displays an abbreviation that identifies the tool used to create an object. The following table lists the tool abbreviations that appear in the Analysis Manager and the corresponding DENEBACAD tool names.

These tool data type abbreviations also appear in the Tool text box on the Tool tab in the Properties Manager, to identify the tool that created a selected object, and also in DENEBACAD's Parametric Text Format (PTF) language.

Tool data type abbreviation	DenebaCAD Tool name
3PointsArc	Arc 3 Points
3PointsCircle	Circle 3 Points
CenterEllipse	Ellipse Center to Corner
CenterRadiusArc	Arc Radius
CenterRadiusCircle	Circle Radius
CenterRect	Rectangle Center to Corner
CenterRoundRect	Rounded Rectangle Center to Corner
DiagonalEllipse	Ellipse Diagonal
DiagonalRect	Rectangle Diagonal
DiagonalRoundRect	Rounded Rectangle Diagonal
ElliptArc	Arc Elliptical
FreeEllipse	Ellipse 3 Points
FreeRect	Rectangle 3 Points
Line	Line
OffsetPoly	Double Polyline
Polyline	Single Polyline
QB_Spline	Curve
PolygonMidpoint	Polygon Midpoint
PolygonVertex	Polygon Vertex
Text	Text
ChainDimm	Chain Dimension
BaselineDimm	Baseline Dimension
LeaderDimm	Leader Dimension
DoubleLeaderDimm	Constrained Dimension
AngleDimm	Angle Dimension
Walkthrough	Camera Path
VectorText	Text Rotated

Container The Container data type lets you construct reports based on the container objects you have defined in a document.

Containers are 2D objects created in Draft mode that define areas within their borders. You designate an object as a container by selecting the object and then selecting the Container check box in the Properties Manager.



When you use the Container data type, the Analysis Manger reports on all objects within the Container object. For example, if six rectangles are inside a polygon that is a Container, and you display the Container data type and the Total Number data type (which is in the Computing submenu) in the Analysis Manager, the two columns will display the Container name and the number 6, because the Container holds six objects.

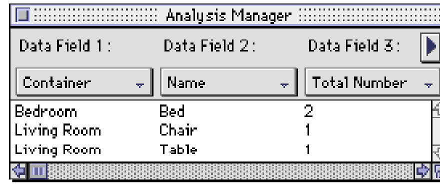


Fig. 299 Sample Container data

Class Name This data type displays the name of an object class that you have defined in the document. You define classes using the Class Manager.

Perimeter This data type displays the measurement of the perimeter of all objects.

Area This data type displays the measurement of the area covered by all objects.

Volume This data type displays the volume of all objects. DENEBACAD calculates volume for 3D objects only.

Object Class categories

The Object Class submenu displays eight data fields. These data fields are based on the fields you define for an object class using the Class Manager.

CLASS MANAGER

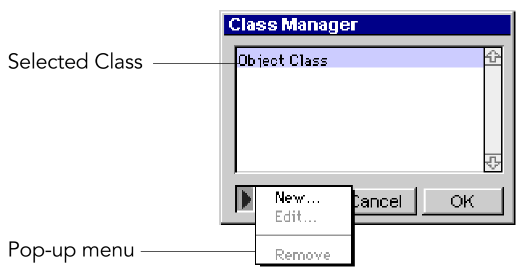
Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+Opt+I
Windows Shortcut: Shift+F3

The Class Manager command displays the Class Manager. The Class Manager is a dialog box you use to create Classes and define fields in a Class.

To create a Class

1. Choose Class Manager in the Window menu, press Shift+F3 (Windows), or Cmd+Opt+I (Mac OS). DENEBACAD displays the Class Manager dialog box.
2. Choose New in the pop-up menu below the right-arrow button. DENEBACAD displays the Class Editor.
3. Type a name for the Class in the Class Name text box.
4. Enter a field name in the field text boxes you want to use. Each field name becomes a category that you can quantify.

   ▲ For example, if you created a Class called "chair" that you applied to certain objects in your plan, you could enter values for the categories you have defined for the chair Class. Some category names for a chair Class might be: manufacturer, model, serial number, price, and color.
5. Click OK to close the Class Editor and return to the Class Manager.

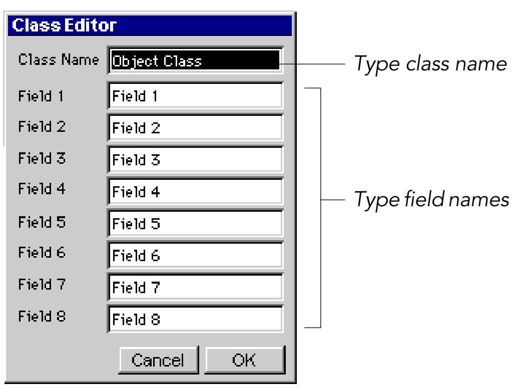


To edit a Class

1. Choose Class Manager in the Window menu, press Shift+F3 (Windows), or Cmd+Opt+I (Mac OS). DENEBACAD displays the Class Manager dialog box.
2. Select the Class you want to edit.
3. Choose Edit in the pop-up menu below the right-arrow button. DENEBACAD displays the Class Editor with the information for the selected Class in the text box and fields.
4. You can enter new information in any of the fields or add new fields to the Class.
5. Click OK to close the Class Editor and return to the Class Manager.

To remove a Class

1. Choose Class Manager in the Window menu, Ctrl+Alt+I (Windows), or Cmd+Opt+I (Mac OS). DENEBACAD displays the Class Manager dialog box.
2. Select the Class you want to remove.
3. Choose Remove in the pop-up menu below the right-arrow button. When you choose Remove, DENEBACAD deletes the class only if there are no objects in the document that belong to the selected Class. Otherwise, DENEBACAD displays a warning that you cannot delete the Class.
4. Click OK to close the Class Editor and return to the Class Manager.



Copying Class items to other documents

When you paste an object that has been assigned to a Class to another document, DENEBACAD treats the object in one of two ways:

- If the object's Class name exists in the new document, DENEBACAD transfers the object's class field data to the new document.

- If the object's Class name doesn't exist in the new document, DENEBACAD creates the object's class, its Class fields, and then transfers the objects's Class field data to the new document.

Organizing library objects and symbols

The Class Manager makes it easy to organize library objects and symbols. You can create custom classes, and then use the Info tab in the Properties Manager to insert information in fields you define for a Class.

This helps keep your project organized by making it easy to obtain information about all of the library objects and symbols in a Class.

To use the Class Manager with library objects and library symbols

1. Use the Class Manager to create a Class for a library object or library symbol. To do this, read "To create a Class" on page 361.
2. Select (or create) a library object or library symbol.
3. Press Shift+F3 (Windows) or Cmd+Opt+I (Mac OS) to open the Properties Manager, or choose Properties Manager in the Window menu. Click the Info tab to bring it to the front.
4. Click the triangle to expand the Info tab.
5. Type the data you want to assign to the object in the text boxes in the Class Manager.

To help keep your project organized, use this procedure for all library objects and symbols.

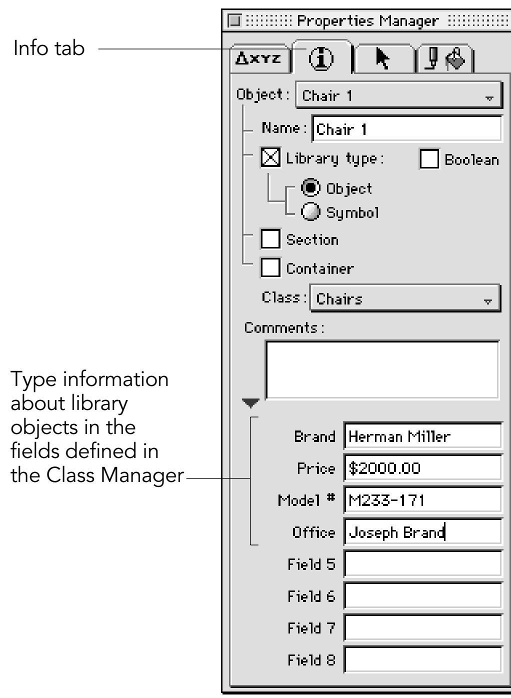


Fig. 300 Info tab with class information

LAYER MANAGER

Modes: Draft, Sculpt, Render
Mac OS Shortcut: Cmd+L
Windows Shortcut: Ctrl+L

The Layer Manager command displays the Layer Manager. Using the Layer Manager, you can create and manage flexible layer structures in DENEBACAD documents.

The Layer Manager command is available whenever a DENEBACAD document is open. The Layer Manager is a floating palette — a dialog box that can remain open in any mode.

Layers help you organize a project. You can use the Layer Manager to add layers, delete layers, name layers, assign colors to layers, create layer sets, and change the order of layers. You can also hide, lock, and protect layers with passwords.

LAYERS IN DRAFT AND SCULPT MODES

Draft and Sculpt modes have significantly different layer structures; this distinction is very important.

- In Draft mode, you can create six independent layer structures. In each view (Top, Bottom, Front, Back, Left, and Right) you can create up to 255 layers for 2D objects.
- In Sculpt mode, you can create one layer structure only for all six views. You can create up to 255 layers in Sculpt mode.

Draft mode layers

In Draft mode, you can create up to 255 layers in each view — Top, Bottom, Front, Back, Left, and Right.

To visualize the layer structure in Draft mode, imagine 256 layers of acetate overlaying each side of a cube. The sides of the cube correspond to the standard views of the DENEBACAD workspace.

As you draw 2D objects, you can put them on different layers to organize your drawing.

Keep in mind that there is a layer sequence or stacking order for layers in Draft mode. Objects on layers that are higher in the stack obscure objects on lower layers. However, don't confuse the stacking order of layers in the Layer Manager with the stacking order of objects on each individual layer.

As you change views in Draft mode, the list of layers in the Layer Manager also changes.

For example, create 10 layers in Top view in Draft mode, and DENEBACAD adds 10 new layers, numbered from Layer 0 to Layer 10, to the Layer Manager. Switch to Front view, and the layer list contains only Layer 0 if there are no other layers in Front view. If there are other layers in Front view, the Layer Manager displays the layer structure of Front view.

Sculpt mode layers

Sculpt mode is a 3D modeling environment in which you work with 3D objects. You can create up to 255 Sculpt mode layers. These layers function as a single layer structure in the Layer Manager for all six views in Sculpt mode.

For example, create 15 layers in Top view in Sculpt mode. The same layers exist in Bottom, Front, Back, Left, and Right views. Switch from one view to another, and you will see the same layer list in the Layer Manager.

Layers in Sculpt mode can help you organize 3D objects in your model in the same way that layers in Draft mode assist you in organizing a plan or elevation.

You can place elements of a 3D model on different layers in Sculpt mode. For example, you can create layers for walls, furniture, electrical fixtures, and plumbing. This gives you the flexibility that layers offer in the 2D environment in the 3D environment as well.

In Sculpt mode as well as Draft mode, you can lock and unlock layers, hide or show layers, assign colors to layers, and rearrange layers in the Layer Manager.

Layer order in Sculpt mode

The effect that layers have on the spatial arrangement of objects in Sculpt mode is more subtle than in Draft mode. Moving a layer higher in sequence in the Layer Manager does not make 3D objects on that layer obscure 3D objects on a “lower” layer.

However, the sequence of layers in Sculpt mode is significant when you combine 3D objects using Combine submenu commands. For these operations, layer sequence influences the way objects combine in the same way that stacking order affects the resulting shapes when you combine 2D objects.

USING THE LAYER MANAGER

Although Draft and Sculpt modes have different layer structures, the procedures for using the Layer Manager to create and organize layers are the same in both modes.

The Layer Manager displays a layer list for the active document window. You can expect the layer list to change as you change mode or view.

◆ **To open the Layer Manager:** Choose Layer Manager in the Window menu, or use the Layers pop-up menu at the bottom of the window. The Layer Manager appears.

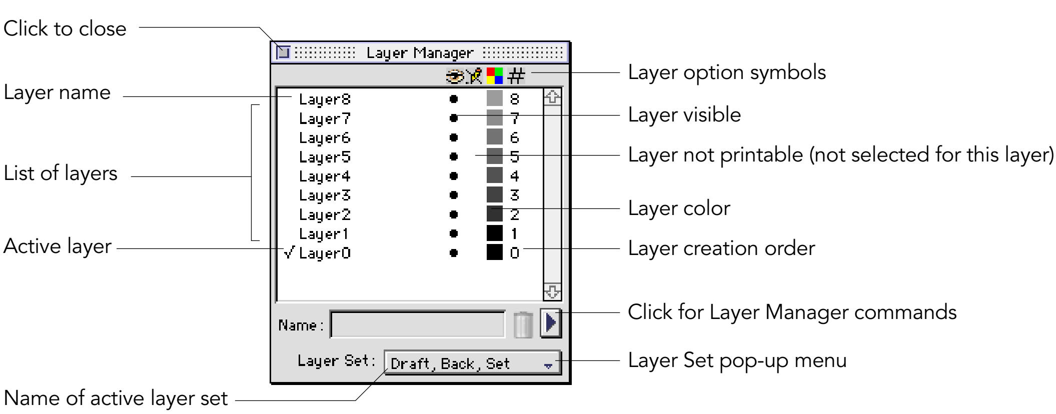


Fig. 301 Layer Manager

Layer Manager window

The Layer Manager windows floats, or appears in front of, all drawing windows. When you first open the Layer Manager, it appears in the center of the screen.

The Layer Manager window can appear in front of or behind other palettes, such as the toolbox. To bring the Layer Manager to the front of other palettes, click the Layer Manager window.

If the Layer Manager is already open, the Layer Manager command does not bring the Layer Manager to the front of other palettes.

When the Layer Manager first appears, the layer list shows the last layers created (those with higher creation order numbers) and no layer name is selected in the layer list. The layer list also indicates the active layer and a selected object's layer.

- One layer in the layer list has a check mark to indicate that it is the active layer.
- If an object is selected, a gray box encloses the name of the layer of the selected object. If more than one object is selected, the gray box encloses the layer name of the selected object that's highest in the layer sequence.

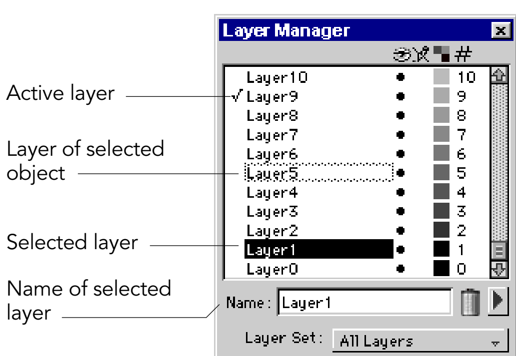


Fig. 302 Layer status indicators

ACTIVATING AND SELECTING LAYERS

The active layer

Only one layer at a time is the active layer, which can also be called the current layer.

- The active layer has a check mark next to its name in the Layer Manager.
- When you create a new object, it is on the active layer.

Note: More than one layer can be visible, and objects on many layers can be selected and printed. The layer called active is the one on which you draw new objects.

When you combine objects on different layers with Boolean operations, DENEBACAD also places the combined object on the active layer.

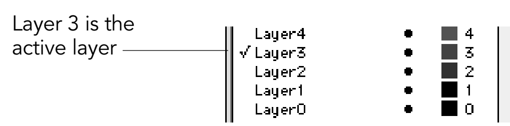


Fig. 303 Active layer indicator

Selecting layers

You can select one or more layers in the Layer Manager to configure layer options. When you select a layer, its name is highlighted in the Layer Manager.

The active layer does not have to be among a selection of multiple layers, unless you are creating a layer set.

Usually, you select a layer when you want to change its name, or change options in the Layer Options dialog box. Also, you can select multiple layers in the Layer Manager to delete the layers or define them as a layer set.

◆ **To select or deselect a layer:** Click the name of the layer in the layer list.

▲ To select (or deselect) a continuous range of layers, press Shift and click the first and last layers' names.

▲ To select (or deselect) several non-contiguous layers, press Ctrl (Windows) or Cmd (Mac OS) and click each layer's name.

Layers 1, 2, 3, 4 selected

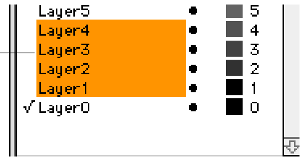


Fig. 304 Selected layers in the Layer Manager

Reordering layers in the Layer Manager

You can change the order of layers shown in the layer list in the Layer Manager.

When you move a layer higher in the layer list, you reposition that layer higher in the sequence of layers.

- In Draft mode, objects on higher layers block objects that they overlap on lower layers.
- In Sculpt mode, objects on higher layers are considered higher in the sequence of objects when you apply Combine submenu commands to them.

◆ **To reorder a layer:** Drag a layer to move it up or down in the list of layers in the Layer Manager.

▲ You can drag one layer at a time in the layer list.

▲ When you drag a layer, it appears underlined in the layer list until you drop it into a new position.

Dragging Layer 1 higher in the layer list

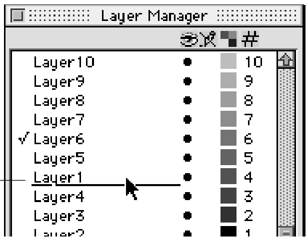


Fig. 305 Changing layer order

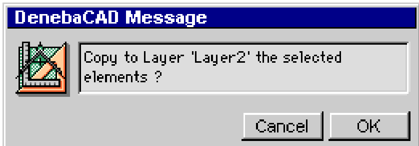
MOVING AND COPYING OBJECTS IN THE LAYER MANAGER

When the Layer Manager is open, you can use it to quickly move or copy selected objects to any visible layer. Using the Layer Manager is an alternative to using the Copy to Layer and Send to Layer submenus in the Object menu.

To copy objects to a layer

1. Select the objects you want to copy. You can select multiple objects on one or more layers.
2. Press Option and click the name of the layer where you want to copy the selected objects.

▲ A message asks you to confirm the copy operation. Click Yes in the message box to copy the selected objects to the layer you clicked.

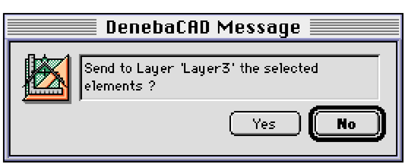


▲ If the layer you click isn't visible, a message tells you that you can't copy objects to a layer that isn't visible. Click OK in the message box to continue.

To send objects to a layer

1. Select the objects you want to send. You can select multiple objects on one or more layers.
2. Press Control and click the name of the layer to which you want to send the selected objects.

   ▲ A message asks you to confirm the operation. Click Yes in the message box to send the selected objects to the layer you clicked.



Note: If the layer you click isn't visible, a message tells you that you can't transfer objects to a layer that isn't visible. Click OK in the message box to continue.

LAYER OPTIONS

You can hide layers, lock layers, assign colors to layers, and change layer names. You can configure these layer options in the Layer Manager.

The scrolling layer list displays the status of several layer options in columns to the right of the layer names. Symbols at the top of the columns identify these options.

DENEBACAD assigns numbers to layers according to their creation order. The far right column in the layer list displays the creation order number of each layer. If you rearrange layers, this column still shows each layer's order in the creation sequence.

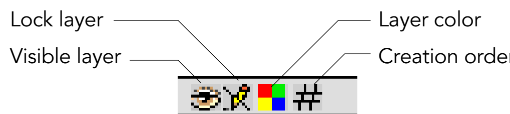


Fig. 306 Layer option symbols

Layer names

DENEBACAD assigns names to layers in the form *Layer#* when layers are created. You can change the layer names using the Layer Manager.

Layer names appear in the second column in the layer list. When a layer is selected, its name appears in the Name text box in the Layer Manager. When no layer is selected, the text box is blank.

To change a layer's name

1. Click a layer name in the layer list to select the layer. The name of the selected layer is highlighted, and its name appears in the Name text box in the Layer Manager.
2. Click in the Name text box to edit the layer's name.

   ▲ To replace the name, double-click in the text box to highlight the existing name.

3. Type a new name for the layer in the Name text box. A layer name can be 29 characters or less.

▲ If you try to type more than 29 characters, a message tells you that you can't type any more characters. Click OK in the message box to return to the Layer Manager.

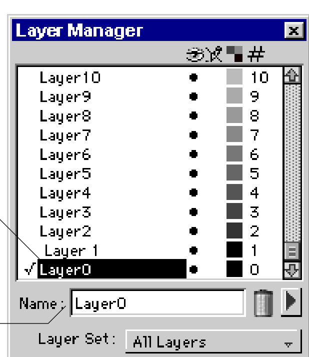
▲ You can type any characters, including spaces and Mac OS special characters (such as Option+8 for a bullet [•]), for a layer name. If you type only spaces, you will not be able to see the layer name in the layer list.

▲ If you type no characters in the text box and press Enter, a message tells you that empty names are not allowed. Click OK to return to the Layer Manager and type a new name.

4. Press Enter, or select another layer in the layer list to apply the new layer name.

Selected layer

Name of selected layer



Visible layer option

To hide or display a layer and all objects on the layer, you can change the visible option in the Layer Manager. New layers are visible when you create them.

With the Layer Manager open, you can use the visible layer option to quickly display or hide different parts of your plan or model, such as walls, floors, or furniture. This makes it easy to isolate the elements you want to edit.

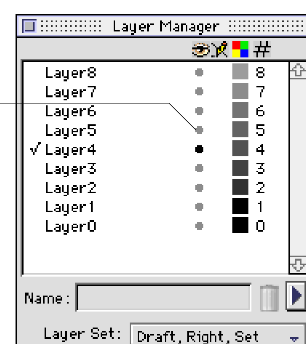
Printing When a layer is not visible, none of the objects on the layer are printed when you print the document.

◆ **To set a layer's visible option:** Click in the Visible column.

- A black dot in the column indicates that the layer and its objects are visible.
- No black dot indicates that the layer and its objects are not visible.
- A gray dot indicates that the layer is set to be visible, but is temporarily hidden because Display Active Layer is in effect.

Note: When you make a layer not visible, any selected objects on the layer are deselected.

Gray dots indicate visible layers that are temporarily hidden



In addition to the Visible option in the Layer Manager, you can hide objects using the Display Options submenu.

For example, when you choose Display Active Layer, DENEBACAD displays objects on the active layer only. When you choose Display Selected Object, DENEBACAD displays only selected objects on any layer.

Locked layer option

By locking a layer, you can prevent changes to the layer's objects. You cannot select, move, delete, or edit objects on locked layers.

In the Layer Manager, the column under the pencil symbol indicates whether a layer and its objects are locked.

◆ **To set a layer's locked option:** Click in the Locked column. You can't lock the active layer.

- A black dot in the column indicates that the layer and its objects are locked.
- No black dot indicates that the layer and its objects are not locked.

Layer color option

DENEBACAD assigns colors to layers when you create them. You can change a layer's color in the Layer Manager.

The Display Layer Colors command in the Display options submenu lets you distinguish objects on different layers.

The layer list shows each layer's color in the column under the color block.

To change a layer's color

1. Click the layer's color in the Color column. A color dialog box appears.
2. Click in the color wheel, or enter color values in the text boxes to specify a color.
3. Click OK to assign the color to the corresponding layer and return to the Layer Manager.

Clicking in the Color column lets you change a single layer's color. You can change the color of multiple layers by using the Layer Options command in the Layer Manager.

Layer creation order

When you create layers, DENEBACAD assigns a number to each layer based on its creation order. DENEBACAD maintains the creation order numbers for all layers, no matter where you move layers in the layer list. Therefore, you can't change the creation order numbers displayed in the far right column in the layer list.

You can use the command "Display by Layer Creation Order" in the Display Options submenu to display objects in the drawing windows based on the original creation order of their layers.

USING LAYER MANAGER COMMANDS

You can choose several commands from a pop-up menu in the Layer Manager. These commands let you create and delete layers and layer sets, select objects based on layers, and assign options to layers.

◆ **To use a Layer Manager command:** Press the arrow button near the bottom-right corner of the Layer Manager. Drag into the menu to choose a command.

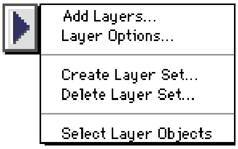


Fig. 307 Layer Manager pop-up menu

Adding and deleting layers

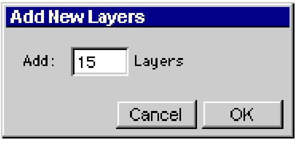
You can add layers to each of the six views in Draft mode, and to any view (all views share the same layers) in Sculpt mode.

To add layers

1. Select the mode in which you want to add layers (Draft or Sculpt) by activating the appropriate window.

   ▲ In Draft mode, go to the view in which you want to use the new layers. In Sculpt mode, you can use any view.

2. Press the arrow button in the Layer manager and choose Add Layers in the pop-up menu. The Add New Layers dialog box appears.



3. In the text box, type the number of new layers you want to create. You can create up to 255 layers.

   ▲ Because one layer always exists, you can type up to 255 in the text box if no other layers have been added.

   ▲ If you type a number greater than the number of layers that DENEBACAD can create, a message appears. Click OK in the message box to return to the Add New Layers dialog box.

4. Click OK. DENEBACAD adds the specified number of layers to the document and updates the layer list in the Layer Manager.

To delete layers

1. In the Layer Manager, select the layers that you want to delete.

2. Click the Trash icon in the Layer Manager. DENEBACAD deletes the selected layers, unless one of the following situation occurs:

   ▲ If you try to delete a layer that contains objects, a warning message appears. If you want to delete the layer whose name appears in the message, click No to return to the drawing. Then delete all of the objects on that layer. If you want to delete only the selected layers that do not contain objects, click Yes. If you click Yes and other layers contain objects, the warning message appears for each of these layers.

   ▲ If you try to delete the active layer, a warning message appears. If you want to delete selected layers other than the active layer, click Yes. If you do not want to delete any layers, click No.

Setting layer options in the Layer Options dialog box

In addition to using the layer list in the Layer Manager to set options for layers, you can set layer options in the Layer Options dialog box. This dialog box lets you set options, such as visibility, for multiple layers at once.

Also, you need to use the Layer Options dialog box if you want to use password protection for layers or change a layer's password.

To use the Layer Options command

1. In the Layer Manager, select the layers whose options you want to change.

2. In the Layer Manager, press the arrow button and choose Layer Options in the pop-up menu.

   ▲ To change options for a single layer, you can double-click the layer.

3. The Layer options dialog box appears. Depending on the options you want to change, do the following.

   ▲ In the Name text box, type a name for the selected layers. If multiple layers are selected, you apply the same name to the layers.

   ▲ To make objects on the selected layers visible, click the Visible checkbox.

   ▲ To lock objects on the selected layer, click the Locked checkbox. DENEBACAD locks the objects on that layer, and activates the Password text box.

   ▲ To assign a color to selected layers, press the color pop-up menu and drag into the color palette to select a color.

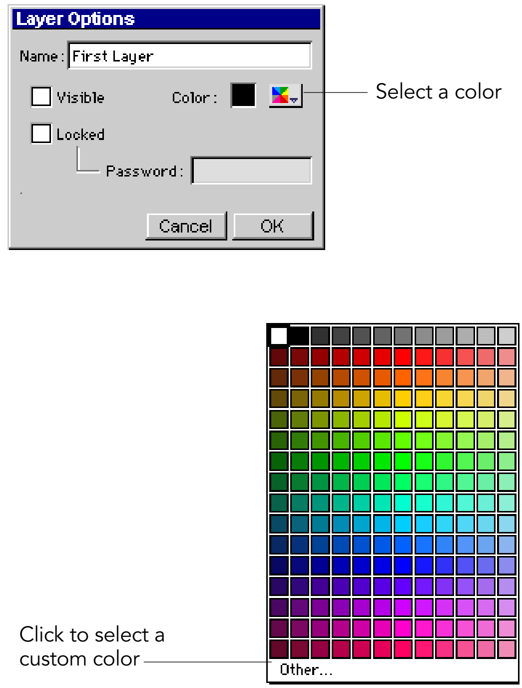


Fig. 308 Selecting a layer color

   ▲ To password protect a layer, select the Locked checkbox and type a password in the Password text box. A dialog box appears. To confirm the password, type the same password, and click OK. To return to the Layer Options dialog box without reconfirming the password, click Cancel. When you password protect a layer, a lock icon appears in the Locked column in the Layer Manager.

4. To implement the settings in the Layer Options dialog box, click OK.

Changing options for a layer locked with a password

When a layer has been locked and protected with a password, you can change the password or unlock the layer only if you type the correct password when prompted to by DENEBACAD.

However, you can change other options for a layer locked with a password by using the columns in the Layer Manager. By clicking in the appropriate column, you can change a layer's color or visibility. You can also drag a layer to change its position in the layer list.

To unlock a password-protected layer in the Layer Manager

1. Click in the Locked column for the layer you want to change.
2. A dialog box prompts you to enter the layer's password.
3. Type the correct password in the message box and click OK. The dialog box closes. In the Layer Manager, the lock symbol disappears from the Locked column for the layer.

To unlock password-protected layers using the Layer Options dialog box

You can use the following procedure to remove or change the password assigned to one or more layers, and to unlock one or more layers that have been password protected.

1. Select one or more layers and choose Layer Options from the pop-up menu in the Layer Manager.
2. A dialog box prompts you to enter the layer's password.
3. Type the correct password in the message box and click OK. The Layer Options dialog box appears.

   ▲ The password is removed. To assign a password, follow the procedure "To use the Layer Options command" on page 371.

   ▲ To unlock the layer, deselect the Locked checkbox.
4. Click OK to implement the Layer Options settings and close the dialog box.

USING LAYER SETS

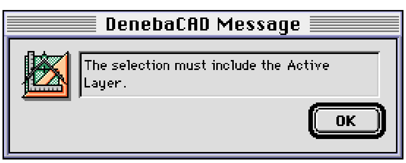
DENEBACAD lets you create layer sets that you can activate in the Layer Manager. When you activate a layer set, the layers in that set are the only ones that appear in the Layer Manager. Objects on layers that are not in the active set are not visible in the drawing windows.

A layer set is a subset of the total number of layers you create in the Layer Manager. When you create a layer set, you should assign it a name that indicates its purpose. DENEBACAD updates the layer list in the Layer Manager to display the layers that are included in the layer set. The layer set's name appears in the Layer Set pop-up menu at the bottom of the Layer Manager.

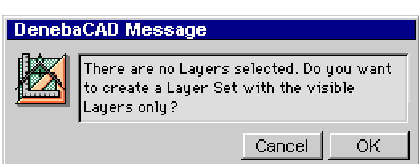
Layer sets have many applications. For example, suppose you create 12 layers in Draft mode, Top view for 12 different elements of your plan. You might put walls, electrical outlets, and plumbing, on three of the layers.

When you want to work with only the wall, electrical, and plumbing layers, you can create a layer set out of those layers. When you choose this layer set in the Layer Set pop-up menu, DENEBACAD hides objects on layers not included in the set, and shows only the layer set in the Layer Manager.

You must include the active layer when you create a layer set. The active layer is the layer in the Layer Manager with a check mark next to its name. Otherwise, this message appears:



You must select layers before you can create a layer set. Otherwise, a message appears, asking you if you want to make a layer set from visible layers. If you often want to make layer sets from the currently visible layers, this can be a useful shortcut.



To create a layer set

1. In the Layer Manager, select the layers you want to make into a set.
2. Choose the Create Layer Set command. A dialog box appears.

3. Type a name for the layer set in the text box. In general, you should try to assign a name that indicates the purpose of the set.
4. Click OK to save the selected layers as a layer set. The name of the layer set appears in the Layer Set pop-up menu in the Layer Manager.

Working with layer sets

You can display one layer set at a time in the Layer Manager. When you display a layer set, its name appears in the Layer Set pop-up menu. If you open the Layer Set pop-up menu, a check mark appears next to the current layer set.

◆ **To select a layer set:** Choose *layer set* in the Layer Set pop-up menu where *layer set* is the name of the set you want to select. In the Layer Manager, DENEBACAD displays the layers in that set only.

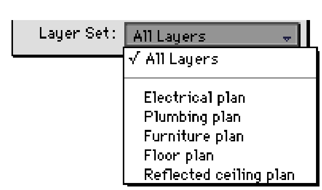


Fig. 309 Layer Set pop-up menu

To delete a layer set

1. In the Layer Set pop-up menu at the bottom of the Layer Manager, choose the layer set you want to delete (you can't delete the All Layers set). DENEBACAD activates the layer set you select.
2. Press the arrow button in the Layer Manager, and choose Delete Layer Set in the pop-up menu. A message appears.

3. Click Yes to delete the selected layer set. The layer set name disappears from the Layer Set pop-up menu.

Important: You cannot use the Undo command to restore a layer set once you delete it.

USING THE LAYER MANAGER TO SELECT OBJECTS

When the Layer Manager is displayed, you can select objects based on the layer structure in Draft and Sculpt modes.

To use the Select Layer Objects command

1. In the Layer Manager, select one or more layers containing the objects you want to select. The layers you select become highlighted in the layer list. You don't have to select the active layer.
2. Press the arrow button and choose Select Layer Objects in the pop-up menu. On the visible, unlocked layers that are highlighted in the layer list, DENEBACAD selects all objects.

SELECTING OPEN DOCUMENTS

When you have more than one DENEBACAD document open at once, you can switch from one document to another using the Window menu.

At the bottom of the Window menu, the names of all open DENEBACAD documents appear. A check mark appears next to the active document.

◆ **To select an open DENEBACAD document:** Choose *document name* in the Window menu, where d*ocument name* is the name of the document you want to open.

The active document

When multiple documents are open, only one document is active at a time. Any commands that you choose, attributes that you select, and other operations that you perform in DENEBACAD apply to the active document only.

The one exception to this rule is that some operations affect the entire program, so they do not apply to the active document only. These operations include changes you make to settings in the Preferences dialog box.

PROPERTIES MANAGER

Modes: Draft, Sculpt
Mac OS Shortcut: Cmd+I
Windows Shortcut: F7

The Properties Manager is a versatile palette that displays information about selected objects and lets you edit an object's settings and coordinates.

The Properties Manager is a floating palette that can remain on screen while you work.

Whenever you create or select an object, DENEBACAD updates the information displayed in the Properties Manager.

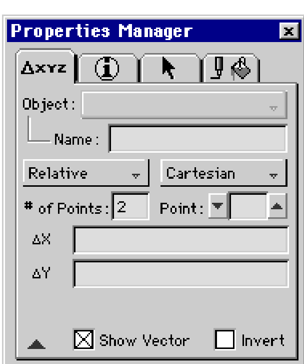
You can use the Properties Manager to do any of the following:

- View settings, options, attributes, and coordinates for an object by selecting the object when the Properties Manager is open.
- Change the settings, options, data, or coordinates of a selected object by editing these items in the Properties Manager.
- Create library objects, library symbols, Sweep Sections, and Containers by setting options in the Properties Manager.
- Create a database by entering data for selected objects in the text boxes in the Properties Manager. Then use the Analysis Manager to display reports using the data that you've entered.

Common and unique options

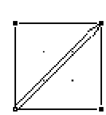
The Properties Manager has many options and settings in common with other parts of the DENEBACAD interface. For example, you can view and edit coordinate data in the Info bar, and the same data appears on the Coordinates tab in the Properties Manager. When you double-click the Chain Dimension tool in the toolbox, a dialog box of options appear; the same options appear on the Tool tab in the Properties Manager when you select a chain dimension object.

Some options and settings are unique to the Properties Manager. These settings include object names and the text in dimension objects, which DENEBACAD sets when the objects are created. To change the settings, you must use the Properties Manager when the object you want to change is selected.



When no objects are selected, most areas in the Properties Manager are blank.

When an object is selected, the Properties Manager displays data and settings for the selected object.



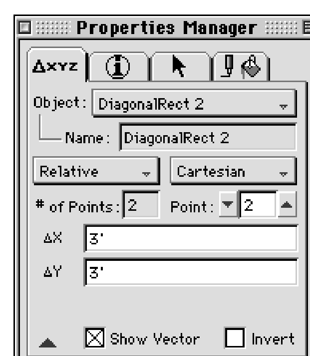


Fig. 310 Properties Manager

Displaying the Properties Manager

Properties Manager command Choose Properties Manager in the Window menu, press F4 (Windows), or Cmd+I (Mac OS) to display the Properties Manager.

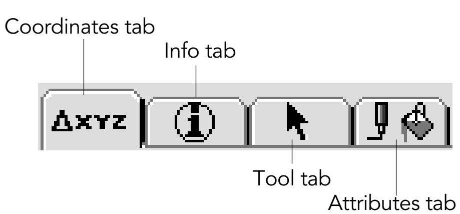
Note: If the Properties Manager is already displayed, choosing the Properties Manager command does not close the Properties Manager.

Double-clicking an object If you double-click an object in the Draft or Sculpt window, DENEBACAD selects the object and displays the Properties Manager (if it isn't already on screen). Each tab in the Properties Manager displays data regarding the object that you double-clicked.

Closing the Properties Manager Click the Close box at the upper-left corner of the Properties Manager to remove the Properties Manager from the screen.

Tabs in the Properties Manager

The Properties Manager contains four tabs: Coordinates, Info, Tool, and Attributes. A symbol at the top of each tab identifies its function. Each tab displays settings or options for the currently selected object.



Default tab The Coordinates tab is active each time you display the Properties Manager. To use another tab, click the tab to bring it to the front of the Properties Manager.

The following sections in this chapter describe each tab in the Properties Manager.

Attributes tab This tab displays Pen color, Fill pattern, and other attributes for the selected object. The settings on this tab are not editable. See page 377 for information about this tab.

Coordinates tab This tab displays the selected object's name, and the coordinates of its creation points. You can choose the coordinate system used to display this data, and you can edit the position of creation points. See page 380 for information about this tab.

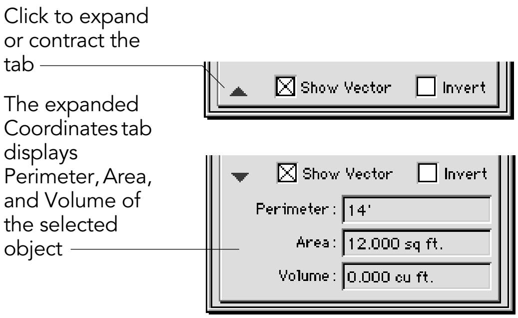
Info tab This tab lets you edit settings for the selected object, including its name. You can use the Info tab to designate a library object, library symbol, Sweep Section, or Container object. You also use this tab to assign an object to a Class, and to enter data for Analysis Manager reports. See page 392 for information about this tab.

Tool tab This tab shows the data type of the tool that created the selected object. When expanded, the Tool tab displays tool-specific settings, such as those used to create a dimension object. See page 394 for information about this tab.

Expanding the Properties Manager

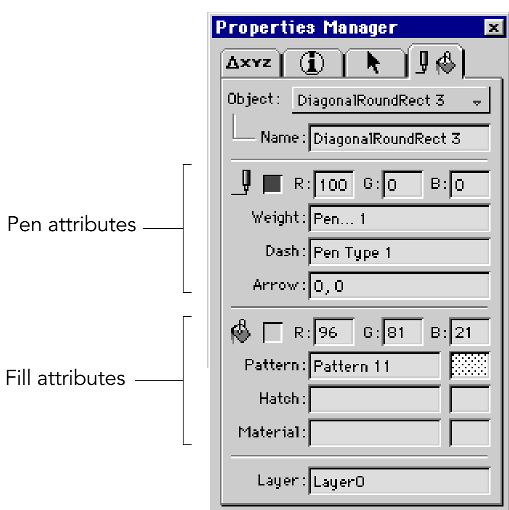
The Properties Manager can be expanded to display additional data or options when the Coordinates tab, Info tab, or Tool tab is active; the Attributes tab doesn't expand.

◆ **To show or hide expanded tabs:** Click the triangle at the bottom-left of the tab in the Properties Manager.



ATTRIBUTES TAB

You can use the Attributes tab of the Properties Manager to view the attributes of selected objects.



The Attributes tab displays the name, Pen and Fill settings, and layer name of the selected object

Fig. 311 The Attributes tab

◆ **To view an object's attributes:** Do one of the following to display in the Properties Manager the attributes of an object:

▲ Double-click the object whose attribute settings you want to view. This selects the object and displays the Properties Manager. Click the Attributes tab to view the attributes settings of the object.

▲ If the Properties Manager is displayed, and the Attributes tab is active, select the object whose attributes you want to view.

▲ Select the object whose attributes you want to view. Choose Properties Manager in the Window menu, press F4 (Windows), or Cmd+I (Mac OS), and then click the Attributes tab.

Using the Object pop-up menu

When multiple objects are selected in the document, you can use the Object pop-up menu to pick one object; its attribute settings are then displayed on the Attributes tab.

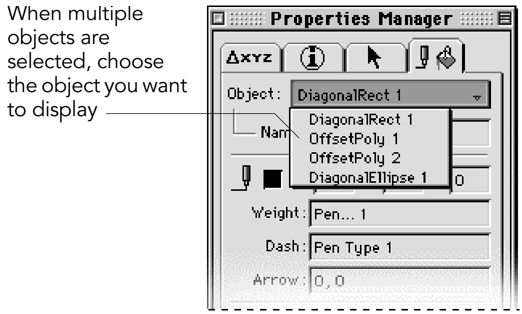
The Object pop-up menu contains the names of all objects that are currently selected in the document. When you choose an object's name in the pop-up menu, a gray bounding box appears around the object in the document, and the Attributes tab displays its attribute settings.

To view attributes if multiple objects are selected

If multiple objects are selected, the Attributes tab displays settings of one object at a time.

1. Select the objects whose attribute settings you want to view.
2. Choose Properties Manager in the Window menu to display the Properties Manager and click the Attributes tab to bring it to the front, if necessary.

3. In the Object pop-up menu at the top of the tab, choose the name of an object to display its attribute settings.

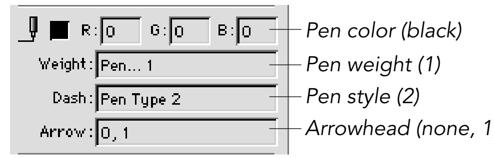


Settings displayed on the Attributes tab

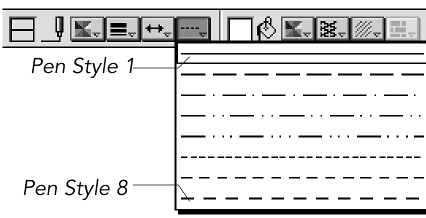
Attribute settings appear on the Attributes tab of the Properties Manager when an object is selected. These settings are not editable in the Properties Manager. To change the attributes of selected objects, use the Attributes buttons on the Attributes bar, or the Pen palette and Fill palette.

Name The name of the selected object appears in the Name box. DENEBACAD assigns names when you create objects using tool data types and object creation order. To change an object's name, use the Name text box on the Info tab in the Properties Manager. See "Info tab" on page 392 for more information.

Pen attribute settings



The Pen area of the Attributes tab displays the Pen color values, Pen weight index, Pen style index, and Arrowhead indexes.

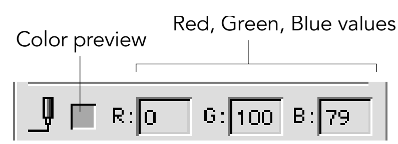


Index values for Pen weight, Arrowhead, and Pen style correspond to the position of the setting in the attributes pop-up palettes and on the tabs in the Pen palette.

Fig. 312 Pen attributes settings

Pen color The RGB color values of the selected object's Pen color appear at the top of the Pen area. A small preview box next to the Pen icon displays the Pen color.

Pen color is an attribute of 2D objects and 3D objects.



Pen weight The Weight text box in the Pen area displays the Pen weight of the selected object as an index value. The index value corresponds to the position of the object's Pen weight in the Pen weight palette, with 1 indicating the Pen weight setting at the top of the palette, 2 indicating the next Pen weight setting, and so on.

Pen weight is an attribute of 2D objects only. When a 3D object is selected, the Pen weight text box is blank.

Arrowhead The Arrow text box in the Pen area displays the Arrowhead settings of the selected object as two index values separated by a comma. The first value represents the Arrowhead at the object's first endpoint; the second value represents the Arrowhead at the object's second endpoint.

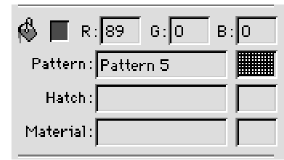
The Arrow index values correspond to the position of the object's Arrowhead in the Arrowhead palette, with 1 indicating the Arrowhead at the top of the palette, 2 indicating the next lower Arrowhead, and so on. A zero index value indicates that no Arrowhead appears at the object's corresponding endpoint.

Arrowheads are an attribute of 2D objects only. When a 3D object is selected, the Arrow text box is blank.

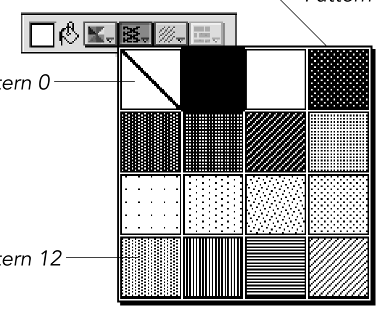
Pen style The Dash text box displays the Pen style setting of the selected object as an index value. The Dash index values correspond to the position of the object's Pen style in the Pen style palette, with 1 indicating the Pen style at the top of the palette (the default undashed Pen style), 2 indicating the next Pen style, and so on.

Pen style is an attribute of 2D objects only. When a 3D object is selected, the Dash text box is blank.

Fill attribute settings



The Fill area of the Attributes tab displays Fill color values, Fill pattern index, Fill hatch name, and Surface material name for the selected object.



Index values for Fill pattern correspond to the position of the pattern in the Fill pattern pop-up palette and on the Pattern tab in the Fill palette.

Fill color The RGB color values of the selected object's Fill color appear at the top of the Fill area. A small preview box next to the Bucket icon displays the Fill color.

Fill color is an attribute of 2D objects only. When a 3D object is selected, the RGB text boxes display the number zero and the preview box is blank.

Fill pattern The Pattern text box in the Fill area displays a Fill pattern index value if the selected object is a 2D object with a Fill pattern. The index value corresponds to the position of the object's Fill pattern in the Fill pattern palette, with zero indicating the first pattern tile at the upper-left corner of the palette (the no Fill pattern tile), and 1 indicating the Fill pattern in the next position in the Fill pattern palette, and so on.

A small preview box next to the Pattern text box displays the Fill pattern.

Fill pattern is an attribute of 2D objects only. When a 3D object is selected, the Pattern text box is blank.

Fill hatch The Hatch text box in the Fill area displays the name of the Fill hatch assigned to an object if the selected object is a 2D object with a Fill hatch.

A small preview box next to the Hatch text box displays the Fill hatch.

Fill hatch is an attribute of 2D objects only. When a 3D object is selected, the Hatch text box is blank.

Surface material The Material text box displays the name of an object's Surface material when a 3D object that has a Surface material is selected.

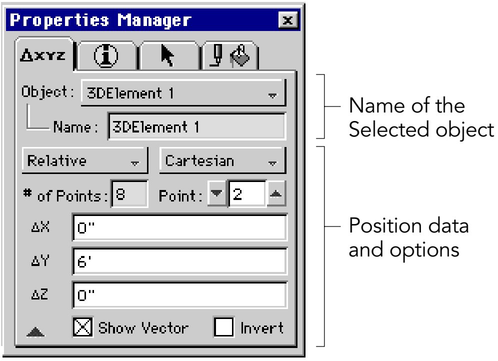
A small preview box next to the Material text box displays the Surface material.

Surface material is an attribute of 3D objects only. When a 2D object is selected, the Material text box is blank.

Layer Displays the name of the selected object's layer. This information is not editable in the Properties Manager; use the Layer Manager to arrange objects on layers and assign names to layers. See "Layer Manager" on page 363 for more information.

COORDINATES TAB

You can use the Coordinates tab of the Properties Manager to view and edit position data for a selected object.



◆ **To view an object's position data:** Do one of the following to display data for an object:

▲ Double-click the object. This selects the object and displays the Properties Manager, with the Coordinates tab in front.

▲ Select the object and choose Properties Manager in the Window menu, press F4 (Windows), or Cmd+I (Mac OS).

▲ If the Properties Manager is displayed and the Coordinates tab is active, just select the object whose data you want to view.

COORDINATES TAB OPTIONS

The Coordinates tab provides a variety of functions for displaying and editing position data for selected objects.

- You can display position data for the selected object's creation points using any coordinate system.
- You can display creation point position data using relative or absolute coordinates.

- The Show Vector option lets you display a drawing vector that corresponds to the position data displayed on the Coordinates tab.
- The Invert option lets you reverse the order of an object's creation points for displaying position data; this also reverses the direction of the drawing vector.

The next sections describe how to use the options and controls that appear on the Coordinates tab.

Using the Object pop-up menu

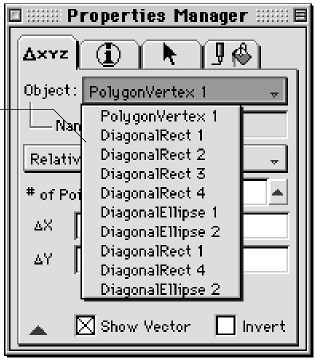
The settings and data displayed on the Coordinates tab apply to one object selected in the document. The Object pop-up menu and the Name text box display the name of the selected object.

The Object pop-up menu also lets you choose one object when multiple objects are selected in the document. In this case, the Object pop-up menu contains the names of all the selected objects. When you choose an object's name from the pop-up menu, the Coordinates tab displays its data.

If the Show Vector checkbox is selected, the drawing vector indicates which object is displayed on the Coordinates tab.

Name box The name of the object chosen in the Object pop-up menu also appears in the Name box. You can't edit the name in the Name box. To change the name of an object, use the Info tab in the Properties Manager. See "Info tab" on page 392 for more information.

When multiple objects are selected, choose one object to display its data



To choose from multiple selected objects

1. Select the objects and choose Window > Properties Manager to display the Properties Manager.
2. In the Object pop-up menu at the top of the Coordinates tab, choose the name of an object to display its position data.

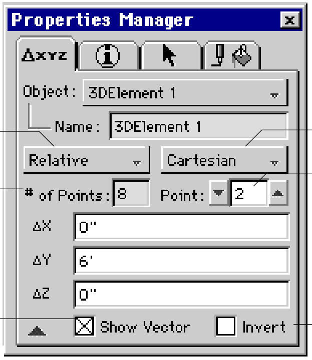


Fig. 313 Options in the coordinates data area

Coordinates data area

The coordinates data area is under the Name box on the Coordinates tab. You can use the options in this area to display position data for any one of the points that define the selected object.

The coordinates of the current point appear in the text boxes. The pop-up menu on the left switches between relative and absolute coordinate data. The pop-up menu on the right specifies the coordinate system for the point data.

The coordinates text boxes and pop-up menus are explained in more detail later in this section.

Position data "points" The "points" referred to on the Coordinates tab are creation points or vertices of the selected object.

- For a 2D object, a "point" is a creation point.
- For a 2D object that has been converted to a polygon, a "point" is a vertex of the polygon.
- For a 3D object, a "point" is a vertex of a polygonal section of the 3D object.

# of Points When you select an object, the "# of Points" box shows the object's total number of points. This value is not editable.

The number of points an object contains depends on the object type and mode.

For example, a rectangle drawn with the Rectangle 3 Points tool has three creation points, so "3" appears in the "# of Points" box when it is selected.

A cube, drawn as a rectangle in Sculpt mode, has eight vertices, so "8" appears in the "# of Points" box when the cube is selected.

Point The Point text box displays the number that identifies the current point. The position data that appears in the coordinates text boxes applies to the current point.

You can use the Point box to set any point in the selected object as the current point. To change the current point, type a number in the text box, or click the arrow buttons to change the current number.

Click the arrow button on the left of the box to select the next lower-numbered point. Click the arrow button on the right of the box to select the next higher-numbered point. When point 1 is the current point, the position data displayed is always absolute.

- Coordinates are measured from the document's Centerpoint, rather than a previous point in the selected object.
- "Absolute" appears in the pop-up menu, which is dimmed to indicate that the Relative option is not available.

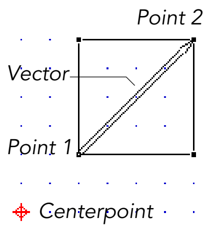
Show Vector When the Show Vector checkbox is selected on the Coordinates tab, a drawing vector appears in the document. The vector points to the current point specified in the Point text box. The origin of the vector arrow depends on whether absolute or relative data is displayed.

- For absolute data, the vector points from the Centerpoint to the current point.
- For relative data, the vector points from the previous point to the current point.

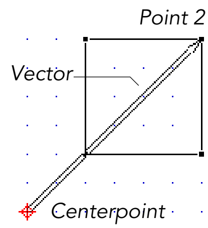
Invert In the numbering of points, the "previous" point depends on how the object was drawn, and whether the Invert option is selected.

- If Invert is not selected, the previous point is the point that was defined in the object prior to the current point.
- If Invert is selected, the order of the points is reversed: The "previous" point is

the point that was defined in the object after the current point.



When relative data is displayed, the vector runs from the previous point (Point 1) to the current point (Point 2).



When absolute data is displayed, the vector runs from the Centerpoint to the current point (Point 2).

Relative and Absolute data

To refer position data to the document's Centerpoint or to another creation point, you can choose Relative or Absolute in the pop-up menu on the Coordinates tab.

When Show Vector is selected, an arrow visually represents absolute or relative position vectors.

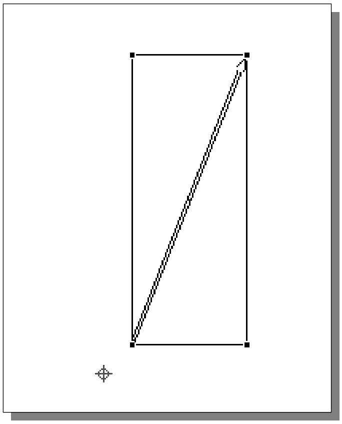
Relative position data is expressed as the change in distance, coordinates, or angle from the previous point to the current point. If relative data is displayed, the delta symbol (Δ) precedes the labels of the text boxes.

By default, the Coordinates tab displays relative position data, with Point 2 set as the current point. Each time you open the Properties Manager, it returns to the Relative setting and Point 2.

Absolute position data is expressed as the distance, coordinates, or angle measured from the document's Centerpoint to the current point.

When an object's first point is the current point (1 appears in the Point text box), the displayed position data is always absolute. The pop-up displays Absolute, and the menu is dimmed to indicate that the Relative option is not available.

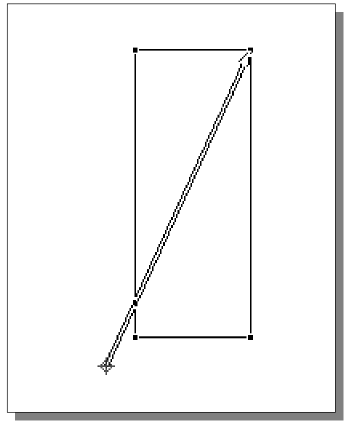
Position data and the current point



Relative position data

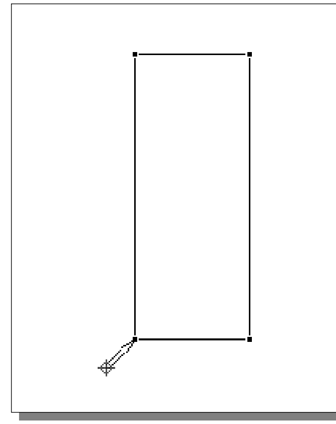
With Relative selected on the Coordinates tab, the drawing vector runs to the current point from the previous point (point 1 to point 2 here).

This vector also appears when you draw a rectangle from corner to corner.



Absolute position data

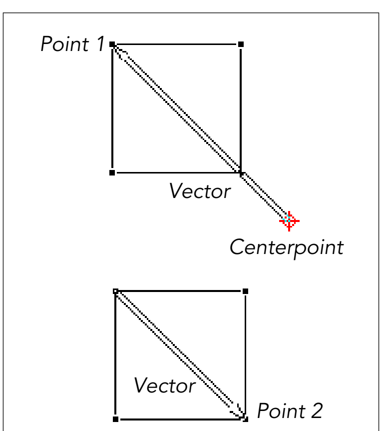
With Absolute selected on the Coordinates tab, the drawing vector runs from the Centerpoint to the current point (point 2 here).



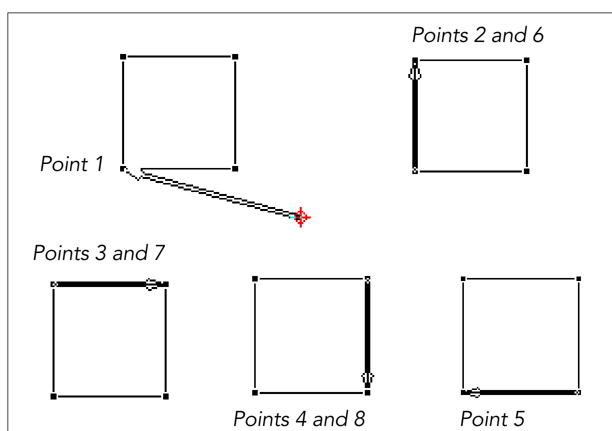
Point 1 position data

When Point 1 is the current point, position data is always absolute. The drawing vector runs from the Centerpoint to Point 1.

The pop-up menu is dimmed because you can't choose Relative position data if Point 1 is specified.



Current points: When a 2D object is selected, you can view each creation point. The vector points to the current point.



With a 3D object selected, some creation points are coincident in each view, so the vector can appear to point to the same position when either of two points is the current point.

Editing position data

The text boxes on the Coordinates tab display the coordinates of the current point of the selected object, based on the current coordinate system.

The text box labels change to reflect the current coordinate system and view.

When a 2D object is selected, two text boxes display position data on the Coordinates tab. When a 3D object is selected, three text boxes appear.

Note: A third text box appears if you select a dimension object. The third text box lets you edit the dimension text without changing the object.

You can edit the data in the text boxes. Changing a value changes the length or position of the vector, and, consequently, the position of the current point.

To edit position data in the text boxes

1. To change a value, select the text and type a new value in the text box.
2. Press the Tab or Enter key to input the new data. DENEBACAD moves the current point to the new coordinates, and the drawing vector reflects the new position data.

Relative Views and absolute position data

When you switch from an orthogonal view to a Relative View, the absolute position data, for each point of an object, changes. This is because in a Relative View, the axes rotate around the document's Centerpoint, but the object does not. The absolute relationship of the object to the axes changes based on this rotation. The relative position data, the relative measurements between points in an object, do not change.

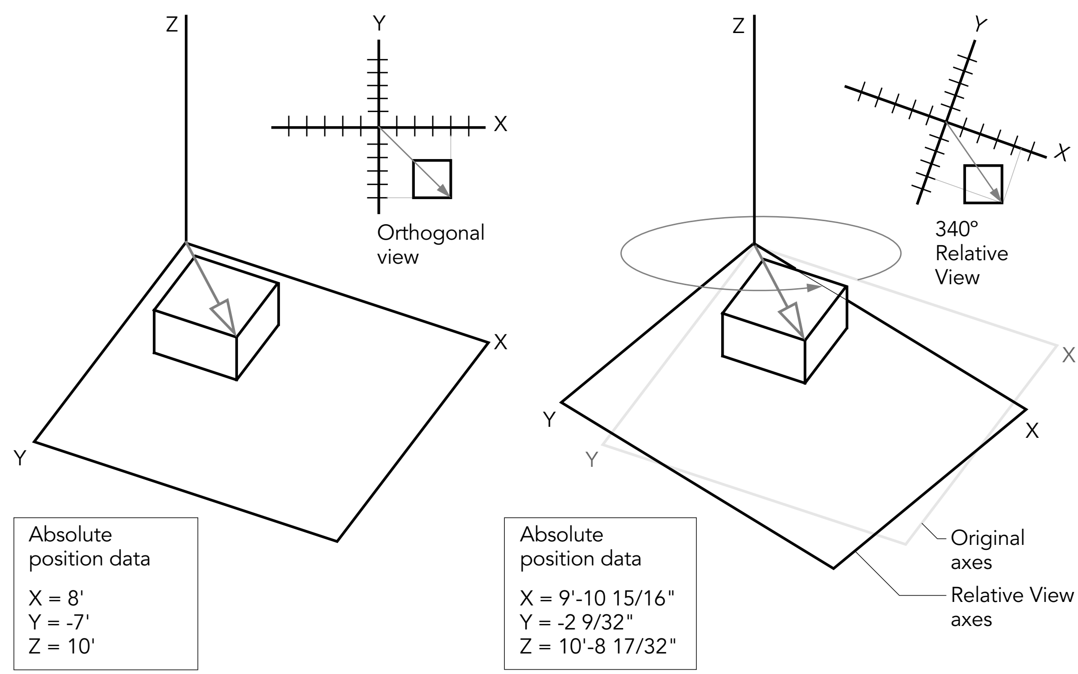
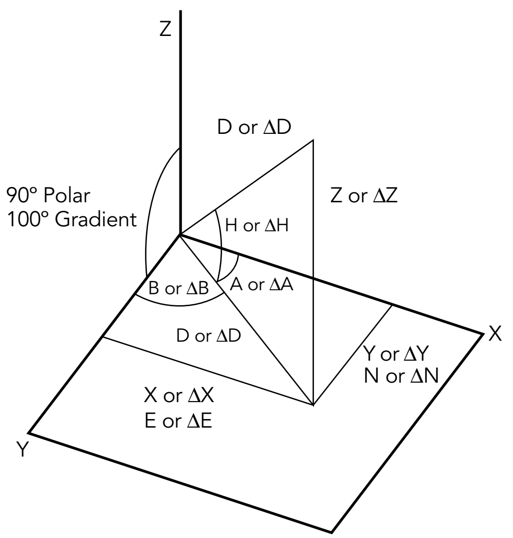


Fig. 314 The effect of a 340° horizontal Relative View on absolute position data in the Cartesian coordinate system

Coordinate systems for position data

There are six coordinate systems in DENEBACAD: Cartesian, Polar, Relational, Bearing, Geographic, and Gradient. By default, DENEBACAD selects Cartesian when you open the Properties Manager.

The coordinate system affects the object position data that appears in the text boxes. If you choose Cartesian, object position data is expressed in X, Y, and Z values. If you choose Geographic, object position data is expressed in North and East values, and an X, Y, or Z, value.



The Coordinates tab displays a point's position as the length, angle, or grid coordinates of a vector that runs to the point. The lines in this illustration represent various vector positions. The labels identify the position data in the text boxes on the coordinates tab.

For example, X, Y, and Z are the data labels when you select Cartesian on the Coordinates tab.

Fig. 315 Coordinate system data labels

If Relative is selected in the pop-up menu, the text box labels are preceded by the delta symbol (Δ) to indicate relative position data.

You can change the coordinate system you want to work in at any time by choosing the coordinate system name in the pop-up menu on the Coordinates tab.

Note: If you change the coordinate system on the Coordinates tab, it has no effect on the coordinate system selected on the Info bar.

Cartesian coordinate data

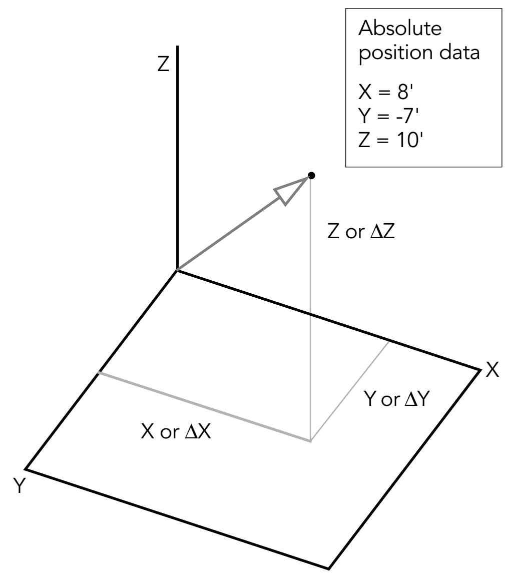


Fig. 316 Cartesian coordinate system

In the Cartesian coordinate system, the Coordinates tab displays a horizontal, vertical, and a depth coordinate (for a 3D object).

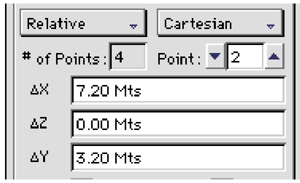


Fig. 317 Cartesian coordinates

X The position of the current point measured on the X axis from the Centerpoint.

ΔX The position of the current point measured on the X axis from the previous point.

Y The position of the current point measured on the Y axis from the Centerpoint.

ΔY The position of the current point measured on the Y axis from the previous point.

Z The position of the current point measured on the Z axis from the Centerpoint.

ΔZ The position of the current point measured on the Z axis from the previous point.

Polar coordinate data

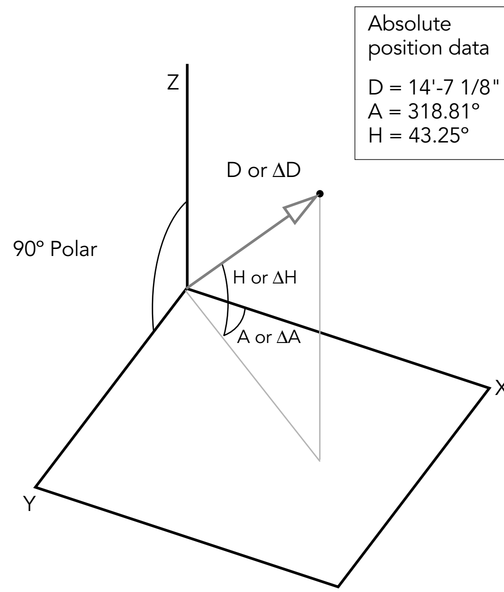


Fig. 318 Polar coordinate system

In the Polar coordinate system, the Coordinates tab displays distance, angle, and an elevation (if a 3D object is selected).

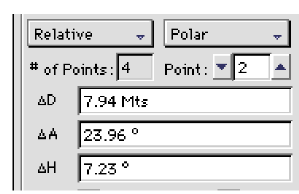


Fig. 319 Polar coordinates

D The distance to the current point, measured from the Centerpoint.

ΔD The distance to the current point, measured from the previous point.

A The angle to the current point in measured from the Centerpoint. The angular measurement is based on a 360º circle (*figure 320*).

ΔA The angle to the current point measured from the previous point. The angular measurement is based on a 360º circle (*figure 320*).

H The angle to the current point measured from the Centerpoint.

ΔH The angle to the current point measured from the previous point.

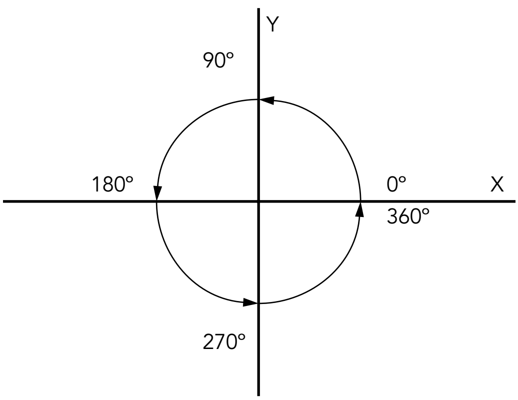


Fig. 320 Polar coordinate system

Relational coordinate data

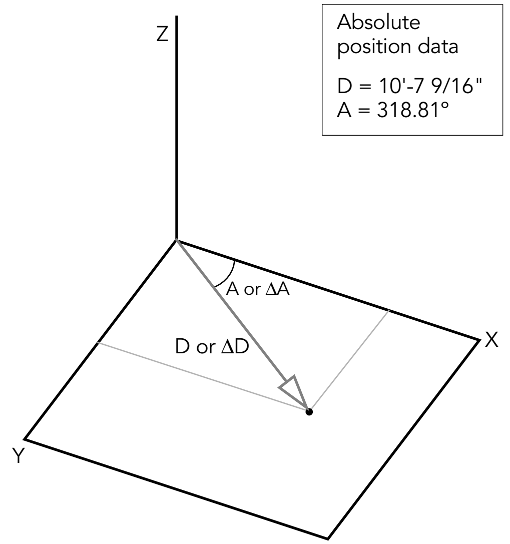


Fig. 321 Relational coordinate system

In the Relational coordinate system, the Coordinates tab displays distance and angle. The Relational coordinate system is available in the pop-up menu when a 2D object is selected. When a 3D object is selected, Relational is not available.

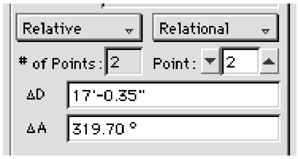


Fig. 322 Relational coordinates

D The distance to the current point, measured from the Centerpoint.

ΔD The distance to the current point, measured from the previous point.

A The angle to the current point, measured from the Centerpoint.

ΔA The angle to the current point, measured from the previous point.

Bearing coordinate data

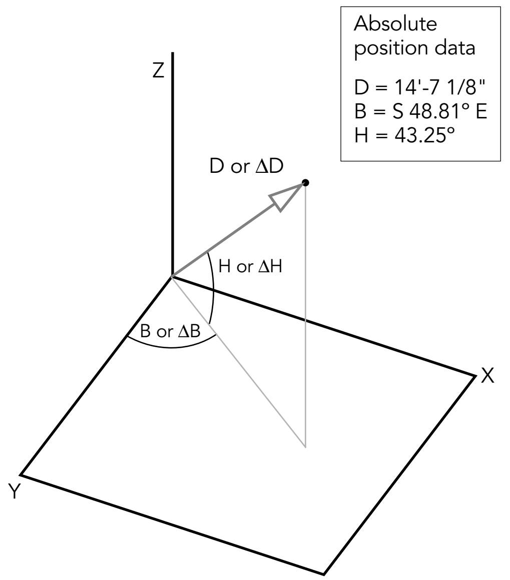


Fig. 323 Bearing coordinate system

In the Bearing coordinate system, the Coordinates tab displays distance, angle, and elevation angle (for a 3D object).

D The distance to the current point, measured from the Centerpoint.

ΔD The distance to the current point, measured from the previous point.

B The bearing angle to the current point, measured from the Centerpoint.

ΔB The bearing angle to the current point measured from the previous point.

H The elevation angle to the current point, measured from the Centerpoint.

ΔH The elevation angle to the current point, measured from the previous point.

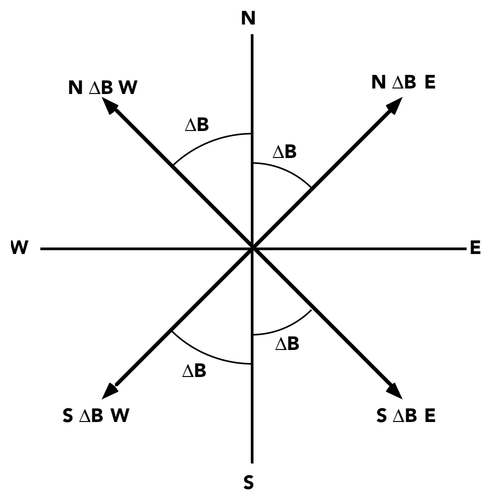


Fig. 324 Bearing coordinate system

Geographic coordinate data

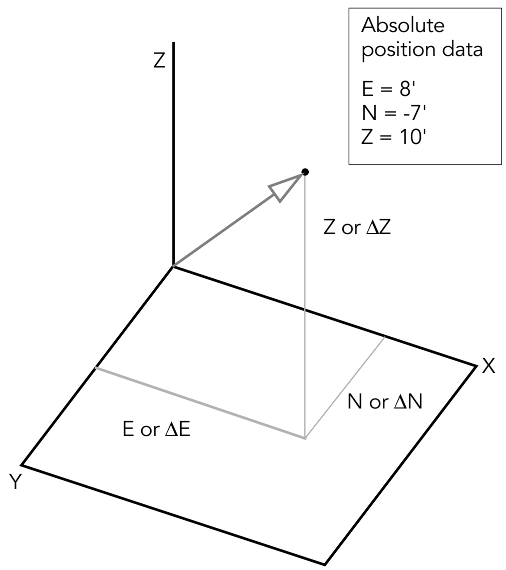


Fig. 325 Geographic coordinate system

Geographic coordinates refer to two geographic axes N (north), and E (east), and, for 3D objects, an elevation axis labeled X, Y, or Z.

In the Geographic coordinate system, the Coordinates tab displays the distance on the first geographic axis, the distance on the second geographic axis, and the elevation distance (for a 3D object).

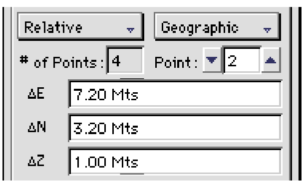


Fig. 326 Geographic coordinates

E The distance to the current point on the East geographic axis, measured from the Centerpoint.

ΔE The distance to the current point on the East geographic axis, measured from the previous point.

N The distance to the current point on the North geographic axis, measured from the Centerpoint.

ΔN The distance to the current point on the North geographic axis, measured from the previous point.

X, Y, or Z The elevation angle to the current point, measured from the Centerpoint.

ΔX, ΔY, or ΔZ The elevation angle to the current point, measured from the previous point.

Gradient coordinate data

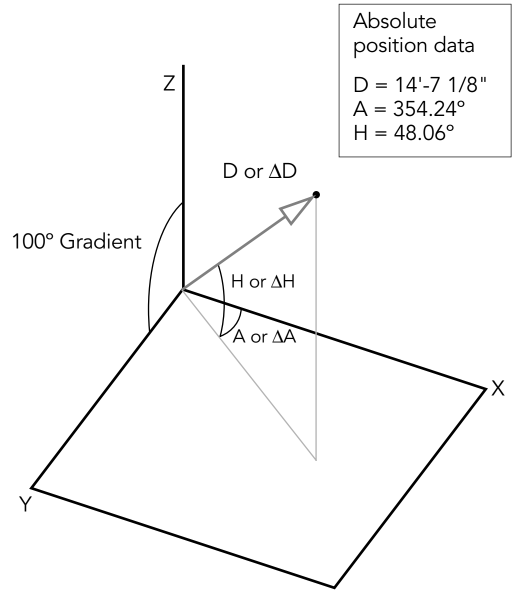


Fig. 327 Gradient coordinate system

In the Gradient coordinate system, the Coordinates tab displays distance, angle, and elevation (for a 3D object).

In the Gradient coordinate system, angles are measured in centesimal degrees, designated “grads.” In this system, a right angle is 100 grads, and a complete circle is 400 grads.

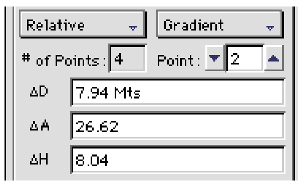


Fig. 328 Gradient coordinates

D The distance to the current point, measured from the Centerpoint.

ΔD The distance to the current point, measured from the previous point.

A The angle to the current point, measured from the Centerpoint.

ΔA The angle to the current point measured from the previous point.

H The elevation angle to the current point, measured from the Centerpoint.

ΔH The elevation angle to the current point, measured from the previous point.

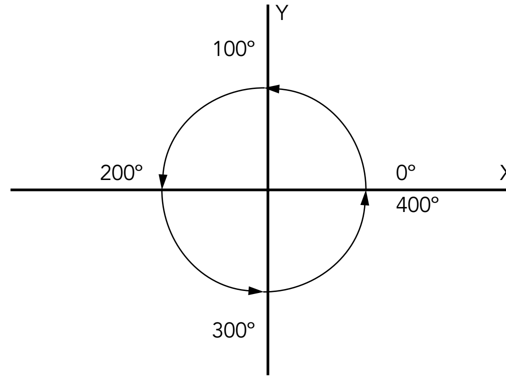
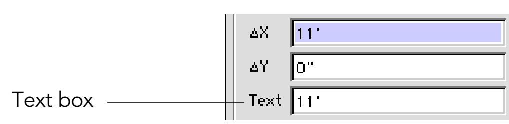


Fig. 329 Gradient coordinate system

Editing dimension text

A text box labeled “Text” appears on the Coordinates tab when a dimension object is selected. You can type new text in this text box to replace the dimension text of the selected dimension object. Entering a new numerical value does not change the length of the dimension line.

You can enter letters or numbers in the text box.



To use the Text text box

1. Type a numeric value in the Text text box. Or, type a word or short phrase.
2. Press the Tab key to replace the dimension object’s text with the new data.

Perimeter, area, and volume data

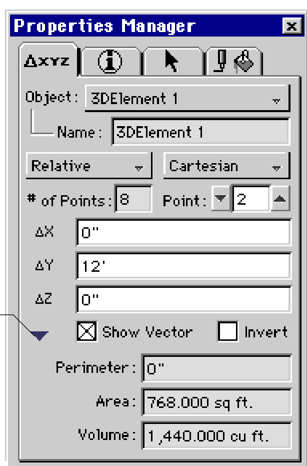
Expanding the Coordinates tab displays perimeter, area and volume text boxes. Click the triangle at the lower-left corner of the tab to expand or contract the tab.

2D objects When a 2D object is selected, the object's perimeter and area data are displayed; the Volume text box displays zero. The area and volume data are displayed as zero when a line is selected, and the perimeter measurement is the same as the length of the line.

3D objects When a 3D object is selected, the object's area and volume data are displayed; the Perimeter text box displays zero.

You cannot edit the values in the Perimeter, Area, or Volume text boxes.

Click to show (or hide) perimeter, area, and volume data



INFO TAB

You can use the Info tab of the Properties Manager to set and view information about selected objects.

To set object information with the Info tab

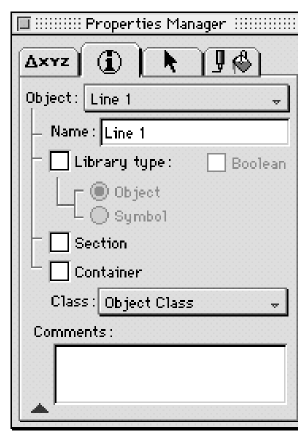
Select the objects for which you want to change or view information.

If necessary, open the Properties Manager by double-clicking an object or by choosing the Properties Manager in the Window menu.

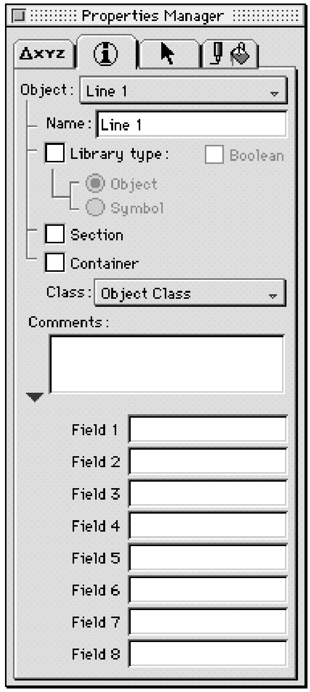
Click the Info tab. The Properties Manager now displays the Info tab options.

Choose the options you want. The options take effect immediately.

You can close the Properties Manager when you are done changing options or you can leave it open while you work.



The Info tab when a a line is selected



The expanded Info tab showing the class fields

INFO TAB OPTIONS

Object

If you have multiple objects selected, you can choose which object's information is displayed on the Info tab by choosing the object's name in the Object pop-up menu.

DENEBACAD displays a gray bounding box around the object whose name is selected in the Object pop-up menu on the Info tab.

Name

You can view and change the name of an object using the Name text box on the Info tab. To execute the Name change, press the Tab key or the Enter key after typing the new name.

DENEBACAD gives each object a default name based on the tool used to create the object and the order in which the objects are created.

Library type

Select the Library Type checkbox to make the current object a Library item.

Object Choose the Object option to make the current object a library object. This option is available only if you first select the Library option.

Symbol Select this option if you want the selected object to be a library symbol. This option is available only if you first select the Library option.

Boolean Select this option to designate the last object in the Library group as a subtractive boolean shape, which will punch through objects when you use a Combine command. This option is available only if you first select the Library option.

Section

You can select this option to make the selected object, or group of objects, available as a sweep section.

Container

Select the Container option to make the current object a container object. You can obtain information on the objects inside a container object in the Analysis Manager.

Class

You can choose a class for the selected object from the Class pop-up menu. You set up classes in the Class Manager. For more information, see "Class Manager" on page 360.

Comments

You can type comments in the Comments text box that you want associated with the selected object.

Fields

If you expand the Info tab by clicking on the arrow at the bottom-left corner of the dialog box, DENEBACAD displays the class fields. You can enter class information in the fields. The field names are set in the Class Manager.

TOOL TAB

You can use the Tool tab of the Properties Manager to view the tool settings of selected objects.

To view tool settings with the Tool tab

1. Select the objects for which you want to view settings.
2. If necessary, open the Properties Manager by double-clicking an object or by choosing the Properties Manager in the Window menu.
3. Click the Tool tab, and click the arrow to expand the tab. The Properties Manager displays the settings of the selected object.
4. You can close the Properties Manager when you are done changing options or you can leave it open while you work.

Tool tab options

The options that appear on the Tool tab in the Properties Manager are described in this section.

Object

If you have multiple objects selected, you can choose which object's information is displayed on the Tool tab from the object's names in the Object pop-up menu.

DENEBACAD displays a gray bounding box around the object whose name is selected in the Object pop-up menu on the Tool tab.

Name

You can view the name of an object using the Name box on the Tool tab.

DENEBACAD gives each object a default name based on the tool used to create the object and the order in which the objects are created.

You can change the object name using the Info tab of the Properties Manager. For more information, see "Info tab" on page 392.

Tool

The Tool text box displays a tool data type abbreviation that identifies the tool that created the selected object. These tool data types are listed in the "DenebaCAD tools and settings," next.

Settings

If you expand the Tool tab by clicking the arrow in the bottom-left corner of the dialog box, DENEBACAD displays the tool settings of the selected object. You can view and change the settings. The settings available for each tool appear in the table, "DenebaCAD tools and settings," next.

A tool's settings correspond to the settings in the dialog box for that tool. To open a tool's dialog box, double-click the tool icon in the toolbox.

Not all tools have a dialog box of options. A downward pointing double-arrow in the lower-right corner of a tool icon indicates the tool has a dialog box.

DENEBACAD TOOLS AND SETTINGS

The following table lists all DENEBACAD tools. The table shows each tool's name and data type. The data type is the name DENEBACAD assigns to objects created by the tool.

The Settings column lists settings that appear on the expanded Tool tab in the Properties Manager, when an object created by the tool is selected.

Tool name	Data type	Settings
Arc 3 Points	3PointsArc	None
Circle 3 Points	3PointsCircle	None
Ellipse Center to Corner	CenterEllipse	None
Arc Radius	CenterRadiusArc	None
Circle Radius	CenterRadiusCircle	None
Rectangle Center to Corner	CenterRect	None
Rounded Rectangle Center to Corner	CenterRoundRect	Radius of Rectangle Corner (Draft mode)
Ellipse Diagonal	DiagonalEllipse	None
Rectangle Diagonal	DiagonalRect	None
Rounded Rectangle Diagonal	DiagonalRoundRect	Radius of Rectangle Corner (Draft mode)
Omnidirectional Light Source	DirectLightSource	Power, Size (Sculpt mode)
Arc Elliptical	ElliptArc	None
Ellipse 3 Points	FreeEllipse	None
Rectangle 3 Points	FreeRect	None
Directional Light Source	LightSource	Power, Size, Angle (Sculpt mode)
Line	Line	None
Double Polyline	OffsetPoly	Width, Offset (Draft mode)
Single Polyline	Polyline	None
Curve	QB_Spline	None
Polygon Midpoint	PolygonMidpoint	Number of Sides (Draft mode)
Polygon Vertex	PolygonVertex	Number of Sides (Draft mode)

Tool name	Data type	Settings
Text	Text	None
Chain Dimension	ChainDimm	Chain Dimension dialog box.
Baseline Dimension	BaselineDimm	Baseline Dimension dialog box.
Leader Dimension	LeaderDimm	Leader Dimension dialog box
Constrained Dimension	DoubleLeaderDimm	Constrain dimension dialog box
Angle Dimension	AngleDimm	Angle Dimension dialog box
Camera Path	Walkthrough	None
Text Rotated	VectorText	None

PARAMETRIC TEXT FORMAT (PTF) A

Parametric Text Format (PTF) is DENEBACAD's plain-text language interface.

PTF lets you save the contents of DENEBACAD documents in files that are compatible with external programs such as spreadsheets, word processors, and programming language text editors.

You also can create PTF files for import into DENEBACAD using such programs. The import feature lets you import objects as well as execute scripted commands.

Introduction

What is PTF?

PTF [P*arametric* T*ext* F*ormat*] ***adj.*** **1** a tabbed text file format that expresses the content of a DenebaCAD™ document *[This is a PTF file]* **2** a public interchange file format, native to DenebaCAD, that can be used to translate other CAD file formats to the DenebaCAD file format *[This is a PTF exchange file]* ***—n.*** A language with a syntax that lets you create DenebaCAD files from textual or numerical data *[This object is described in PTF]*

PTF files can be created in several ways:

Quick and easy You can use DENEBACAD's Save As command in the File menu to create a PTF file from a DENEBACAD document. The PTF file describes in plain text the objects in the DENEBACAD document, and can later be imported, with or without modifications, into another DENEBACAD document.

> **Pros:** Fast and easy. Ensures file transportability even in tough environments such as the Internet.
>
> **Cons:** Resulting file is flat. Modifications are difficult because objects are hard to locate within the file.

Slow and dirty You can use a text editor to write, line by line, the commands required to create a collection of objects.

> **Pros:** A text editor is probably the most common tool in any computer environment. Ensures file transportability even in tough environments such as the Internet.
>
> **Cons:** Resulting file is flat. Modifications are difficult because objects are hard to locate within the file. Error-

prone. Files are hard to debug. This method, while viable, is not recommended.

Smart and savvy You can use a spreadsheet application to create every command required to reproduce a given collection of objects.

Pros: Besides text editors, a spreadsheet application is probablythemostcommontoolinanycomputerenvironment. You can use your spreadsheet knowledge to create functions and macros to automate your object design process, effectively improving your professional productivity.

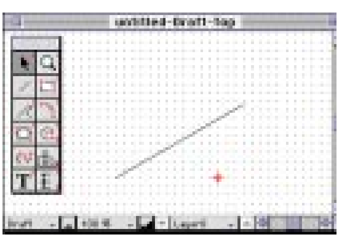
— Ensures file transportability even in tough environments such as the Internet.

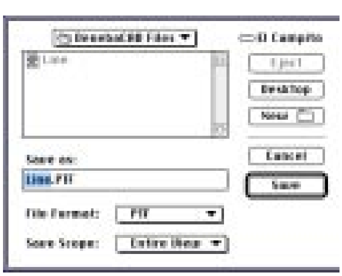
— Files are easy to debug because you have a better control at the source.

— Builds your self esteem and ego.

Cons: Knowledge of spreadsheet operations is required.

Creating and using PTF files

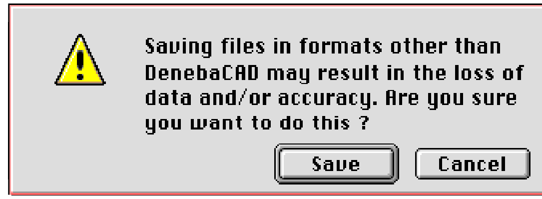


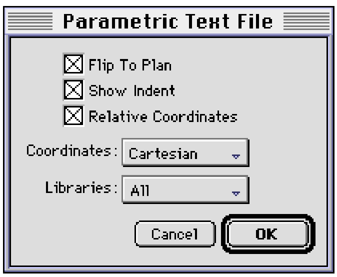


You create PTF files by using the Save As command in the File menu. This allows you to save an object, a group of objects, or a complete document using the PTF file format.

To create a simple PTF file in DENEBACAD

1. Draw a line in a new DENEBACAD document.
2. Choose Save As in the File menu.
3. Type a name for the file in the Save As text box.
4. In the File Format pop-up menu, select PTF. DENEBACAD adds “.PTF” to the file’s name.
5. Choose a location to save the file, and then click Save.
6. A message appears, warning that saving documents in other formats can result in lost data. Click Save in the message box to continue.





7. The Parametric Text File dialog box appears.
8. Click OK to use the default settings and save the PTF file.

To view and modify the PTF file

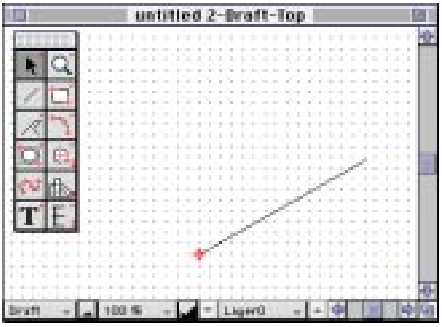
1. Launch your favorite spreadsheet program.
2. Choose Open in the File menu. In the dialog box that appears, select the PTF file that you saved and click Open. The information in the PTF file appears in the spreadsheet.

	A	B	C	D	E
1	Begin PTF				
2	Distance Units	DecInches			
3	Angular Units	DecDegrees			
4	Coordinates	Cartesian			
5	VectorCoordSys	XY	dXdY		
6	View	Top			
7	Mode	Draft			
8	Layer #	0			
9	Pen Color (R G B)	0	0	0	
10	Line Type (Thick Dash Eccen)	1	1	0	
11	Arrow Heads (Begin End)	0	0		
12	Fill Color (R G B)	0	0	0	
13	Fill Pattern	0			
14	Object	Line			
15		Name	Line 1		
16		# Points	2		
17		Pt# X Y	1	-108	48
18		Pt# dX dY	2	156	-72
19	End PTF				

3. Change the value of the X coordinate (cell D17 in the example) to the value you want. In this case, we'll change the value from -108 to 0. In your spreadsheet, you can use formulas to define this value.
4. Choose Save As in the File menu and save a copy of the file as a tab-delimited text file.

To insert the edited PTF file into DENEBACAD

1. Launch DENEBACAD.
2. Choose Import in the File menu. The PTF Place Options dialog box appears. Click OK to implement the default settings and import the file. Notice that the line you’ve drawn appears in a new position on screen.



When you import the PTF file, the changes you made in the spreadsheet program appear in the DENEBACAD document

In addition to changing position data, you can change color, dash settings, line thickness, and any other parameter in the description of an object by editing a PTF file.

Error messages If you change something in the PTF file so that DENEBACAD can’t interpret the file correctly, an error message appears. If this happens, note that the message lists the line in the file where the error occurs. Following the label “Line Text,” the message box displays the text of the line containing the error.

Click OK in the message box to continue. Then, check the line in the PTF file that was listed in the error message.

Options for saving PTF files

You have several options you can choose when you save files in the PTF file format:

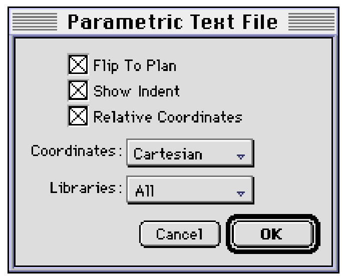
Flip to Plan Uses the Cartesian coordinate system in Top view. Positive coordinates are up and right, negative coordinates are down and left. If this option is not selected, DENEBACAD uses the current view.

Show Indent Saves the text tabs in the file to show the hierarchy of objects. If this option is not selected commands and object descriptions will not be indented.

Relative Coordinates Choose this option to define points by their relationship to the first point in the object or command. If you clear this check box, all coordinates will be saved as absolute and refer to a common fixed origin point

Coordinates pop-up menu Choose the coordinate system you want DENEBACAD to use when saving the PTF file.

Libraries pop-up menu Choose which Library Objects and Symbols you want to include in the PTF file. You can include all Library Objects and Symbols listed in the Library palette, Library Objects and Symbols used in the document only, or no Objects and Symbols.



Options for saving PTF files appear in the Parametric Text File dialog box

PTF Place Options

When importing a PTF format file, you have two options:

Use Top Coordinates System Uses the Cartesian coordinate system in Top view. Positive coordinates are up and right, negative coordinates are down and left.

Show Only Placed Layers Displays objects in the drawing that belong to imported layers only.

Creating PTF files with a text editor

Just fire up your text editor and type away! Keep the following issues in mind:

PTF is not case-sensitive "Fill Pattern," "FILL PATTERN" and "fILL pATTERN" all represent the same information.

The field separator is the Tab character Always use the Tab key to separate commands from their parameters, and the parameters from each other.

Space to the left is irrelevant The following two samples are the same, even though the space to the left is different. Although space to the left is not relevant, it helps organize your information. Notice that the second sample is easier to read and follow than the first one.

```
Begin PTF
Distance Units                      DecInches
Angular Units                       DecDegrees
View                                Top
Mode                                Draft
Layer #                             0
Pen Color (R G B)                   89              0           0
Line Type (Thick Dash Eccen)        1               1           0
Arrow Heads (Begin, End)            0               0
Fill Color (R G B)                  0               0           0
Fill Pattern                        0
Object                              DiagonalRect
Name                                DiagonalRect 1
# Points                            2
Pt# X Y                             1        -228.000     144.000
Pt# DX DY                           2         192.000     -96.000
End PTF
```

```
Begin PTF
    Distance Units                  DecInches
    Angular Units                   DecDegrees
    View                            Top
    Mode                            Draft
    Layer #                         0
    Pen Color (R G B)               89              0           0
    Line Type (Thick Dash Eccen)    1               1           0
    Arrow Heads (Begin, End)        0               0
    Fill Color (R G B)              0               0           0
    Fill Pattern                    0
    Object                          DiagonalRect
            Name                    DiagonalRect 1
            # Points                2
            Pt# X Y                 1        -228.000     144.000
            Pt# DX DY               2         192.000     -96.000
End PTF
```

Space above "Begin PTF" is irrelevant To annotate or describe your work, you can use as many lines as you want above the "Begin PTF" command.

Space below "End PTF" is irrelevant To annotate or describe your work, you can use as many lines as you want below the "End PTF" command.

Empty lines are irrelevant You can insert blank lines as needed. In general, this makes PTF files easier to read and follow.

Creating PTF files in a spreadsheet program

Using a spreadsheet program is by far the smartest and easiest way to generate PTF text files.

Because the final file structure is the same, the same issues apply when you use a spreadsheet to create a PTF file as when you create a PTF file with a text editor.

You can use all of the available spreadsheet functions to generate values and strings which can then be used to create and modify DENEBACAD objects.

When you finish creating or modifying a PTF file, remember to save the file in tab-delimited text format.

PTF Reference

This section describes the commands and object descriptions that are implemented in the PTF file format.

Simple Object descriptions begin on page 407.
Boolean Object descriptions begin on page 414.
Grouped Object descriptions begin on page 414.
Container descriptions begin on page 417.
Dimension Object descriptions begin on page 417.
One-time commands begin on page 420.
Environmental commands begin on page 421.
General Attributes commands begin on page 421.
Object Selection commands begin on page 424.
Object Position commands begin on page 425.
Object Combine commands begin on page 427.
Miscellaneous Object commands begin on page 428.
Object Extrusion commands begin on page 428.

The PTF file structure

When you save a file in PTF format, it appears in spreadsheet applications as shown under “Sample PTF file structure,” next. The exact appearance depends on the spreadsheet you use.

The PTF information is arranged in lines, with keywords and parameters separated from each other by tab characters.

There are three general areas in a PTF file: a Header, a Body, and a Footer.

The PTF Header The PTF Header contains commands that identify the beginning of the PTF file. It sets values that will affect the entire DENEBACAD document. The PTF Header can be set one time only.

The PTF Body The PTF Body contains all of the other commands that relate to object creation and manipulation, and to the DENEBACAD work environment settings.

The PTF Footer The PTF Footer contains a single line that identifies the end of the PTF file:

```
End PTF
```

Sample PTF file structure

`Begin PTF`			
`Distance Units`	DecInches		
`Angular Units`	DecDegrees		
`View`	Top		
`Mode`	Draft		
`Layer #`	0		
`Pen Color (R G B)`	89	0	0
`Line Type (Thick Dash Eccen)`	1	1	0
`Arrow Heads (Begin, End)`	0	0	
`Fill Color (R G B)`	0	0	0
`Fill Pattern`	0		
`Object`	DiagonalRect		
`Name`	DiagonalRect 1		
`# Points`	2		
`Pt# X Y`	1	-228.000	144.000
`Pt# DX DY`	2	192.000	-96.000
`Pen Color (R G B)`	0	40	100
`Object`	DiagonalRect		
`Name`	DiagonalRect 2		

`# Points`	2		
`Pt# X Y`	1	-84.000	84.000
`Pt# DX DY`	2	180.000	-84.000
`Pen Color (R G B)`	0	0	0
`Object`	DiagonalRect		
`Name`	DiagonalRect 3		
`# Points`	2		
`Pt# X Y`	1	60.000	12.000
`Pt# DX DY`	2	204.000	-84.000
`End PTF`			

Typographic conventions

Several typographic conventions are used in the PTF Reference section to distinguish commands, parameters, and related information.

Commands and keywords These words must be typed as shown (although they are not case-sensitive). Commands and keywords tell DENEBACAD to create objects, use specific settings, and apply commands.

Commands and keywords are shown in `monospace` type:

`Line Type (Thick Dash Eccen)`

Parameters Parameters are the values or data used by commands. In some cases, parameters are optional. In other cases, as noted in the text, a missing parameter is assumed to be zero.

Most parameters shown in this section are merely "metasymbols" in the PTF descriptions. In other words, the parameters shown are placeholders to be replaced by actual data or values.

Parameter metasymbols are shown in *italic* type:

Num_Points_Integer

Data Data used in this section represent actual numbers or other values that you specify. For example, when coordinates appear in an object description, they represent actual coordinates that you specify in a PTF object description. Examples of actual values that can be used are shown in roman type.

General syntax for object descriptions

The following syntax is used for object descriptions in PTF format. Keywords are separated from parameters, and parameters are separated from each other, by a tab character.

Keywords	Parameters		
`Object`	*Tool_Name*		
`Name`	*Default_Name* | *User_Name*		
`# Points`	*Num_Points_Integer*		
`Pt# X Y`	*Ordinal_Integer*	*X_Real*	*Y_Real*
`Pt# DX DY`	*Ordinal_Integer*	*dX_Real*	*dY_Real*
`Pt# D A`	*Ordinal_Integer*	*Distance_Real*	*Angle_Real*
`Pt# DD DA`	*Ordinal_Integer*	*dDistance_Real*	*dAngle_Real*

Parameter definitions

The parameters used in object descriptions and other commands are represented by the following metasymbols (metasymbols are terms that will be replaced by actual parameter values in a PTF file):

Tool_Name	The name of the tool used to generate the object
Default_Name	The object name assigned by DENEBACAD
User_Name	An object name assigned by the user
Num_Points_Integer	The number of creation points in the object
Ordinal_Integer	An integer indicating the position of the point within the object definition list
X_Real, Y_Real	Coordinate real values (absolute coordinates)
Distance_Real	Distance real value
Angle_Real	Angle real value
dDistance_Real	Change in distance real value (relative to previous point)
dAngle_Real	Change in angle real value (relative to previous point)
n	An integer value
a b c ...	Coordinate values

Notes When creating PTF documents using text editors or spreadsheet applications you can use the *User_Name* parameter when naming objects. The *Default_Name* parameters are used by DENEBACAD when you save a DENEBACAD document in PTF format.

In PTF keywords and parameters, point coordinates can be expressed as absolute coordinates (*X Y*), relative coordinates (*dX dY*), absolute distance and angle (*D A*) or relative distance and angle (*dD dA*).

Note: The letter "D" or "d" is used in place of the delta symbol to indicate a relative distance or angle. For example, *dX_Real* is a relative X coordinate value.

In a PTF file, an object's definition ends when the indicated number of points (*Num_Points_Integer*) is read, or when another object description is initiated, or when a general or particular attribute command is encountered.

If the number of points read is different than the number of points specified by the *Num_Points_Integer* parameter, the object is not drawn and an error message is generated.

Simple Object descriptions

Simple objects are objects you would create in DENEBACAD using a single tool, without additional object operations.

Simple Object descriptions lets you create these same objects in PTF files. A Simple Object description starts with the `Object` keyword, followed by keywords for the object's name, number of points, and the point coordinates.

Simple Object descriptions

3PointsArc

Describes an arc drawn by the Arc 3 Points tool, defined by three points: two endpoints and a point on the arc's periphery.

`Object`	3PointsArc		
`Name`	3PointsArc *n*		
`# Points`	3		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*
`Pt# DX DY`	3	*e*	*f*

3PointsCircle

Describes a circle drawn by the Circle 3 Points tool, defined by three points on its periphery.

`Object`	3PointsCircle		
`Name`	3PointsCircle *n*		
`# Points`	3		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*
`Pt# DX DY`	3	*e*	*f*

CenterEllipse

Describes an ellipse or circle drawn by the Ellipse Center to Corner tool, defined by its center point and one corner of its bounding rectangle.

`Object`	CenterEllipse		
`Name`	CenterEllipse *n*		
`# Points`	2		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*

CenterRadiusArc

Describes an arc drawn by the Arc Radius tool, defined by three points: its center, a starting point on the periphery and an ending point on its periphery

`Object`	CenterRadiusArc		
`Name`	CenterRadiusArc *n*		
`# Points`	3		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*
`Pt# DX DY`	2	*e*	*f*

Simple Object descriptions

CenterRadiusCircle

Describes an ellipse or circle drawn by the Circle Radius tool, defined by two points: its center and a point on its periphery.

`Object`	CenterRadiusCircle		
`Name`	CenterRadiusCircle *n*		
`# Points`	2		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*

CenterRect

Describes a rectangle drawn by the Rectangle Center to Corner tool, defined by two points: the center point and one corner point.

`Object`	CenterRect		
`Name`	CenterRect *n*		
`# Points`	2		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*

CenterRoundRect

Describes a rounded rectangle drawn by the Rounded Rectangle tool, defined by two points: its center and one of its corners.

`Object`	CenterRoundRect		
`Name`	CenterRoundRect *n*		
`# Points`	2		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*

DiagonalEllipse

Describes an ellipse or circle drawn by the Ellipse Diagonal tool, defined by two opposite corners of its bounding rectangle.

`Object`	DiagonalEllipse		
`Name`	DiagonalEllipse *n*		
`# Points`	2		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*

DiagonalRect

Describes a rectangle drawn by the Rectangle Diagonal tool, defined by two opposite corner points.

`Object`	DiagonalRect		
`Name`	DiagonalRect *n*		
`# Points`	2		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*

Simple Object descriptions

DiagonalRoundRect

Describes a rounded rectangle drawn by the Rounded Rectangle Diagonal tool, defined by two opposite corner points.

`Object`	DiagonalRoundRect		
`Name`	DiagonalRoundRect *n*		
`# Points`	2		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*

DirectLightSource

Describes a directional light source drawn by the Directional Light tool, defined by its position and a point located on its illumination axis. The Power setting defaults to 100. You cannot define the Power (brightness) setting.

`Object`	DirectLightSource		
`Name`	DirectLightSource *n*		
`# Points`	2		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*

FreeRect

Describes a rectangle drawn by the Rectangle 3 Points tool, defined by the end points of one side and an end point of the opposite side.

`Object`	FreeRect		
`Name`	FreeRect *n*		
`# Points`	3		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*
`Pt# DX DY`	3	*e*	*f*

LightSource

Describes a light source drawn by the Omnidirectional Light tool, defined by its position. The Power setting defaults to 100. You cannot define the Power (brightness) setting.

`Object`	LightSource		
`Name`	LightSource *n*		
`# Points`	1		
`Pt# X Y`	1	*a*	*b*

Simple Object descriptions

Line

Describes a line drawn by the Line tool, defined by two endpoints.

`Object`	Line		
`Name`	Line *n*		
`# Points`	2		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*

PolygonVertex

Describes a polygon drawn by the Polygon Vertex tool, defined by two endpoints.

`Object`	PolygonVertex		
`Name`	PolygonVertex *n*		
`#Sides`	6		
`# Points`	2		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*

PolygonMidPoint

Describes a polygon drawn by the Polygon Midpoint tool, with six sides.

`Object`	PolygonMidpoint		
`Name`	PolygonMidpoint *n*		
`#Sides`	6		
`# Points`	2		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*

QB_Spline

Describes a curve drawn by the Curve tool, defined by 5 points. The first and last points are the endpoints and the other points define the curvature of the curve.

`Object`	QB_Spline		
`Name`	QB_Spline *n*		
`# Points`	*n*		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*
`Pt# DX DY`	*n*	*e*	*f*

Simple Object descriptions

OffsetPoly

Describes an offset polyline drawn by the Double Polyline tool, defined by "n" points. The Poly Width parameter defines the distance between the parallel lines. The Offset parameter defines the distance from the center of the parallel lines to the line described by the set points. The Poly Ends parameter must be set to Close Close.

`Object`	OffsetPoly				
`Name`	OffsetPoly *n*				
`Poly Ends`	Close Close				
`Poly Width / Offset`	4.00	0.00			
`# Points`	5				
`Pt# X Y`	1	*a*	*b*		
`Pt# DX DY`	2	*c*	*d*		
`Pt# DX DY`	*n*	*e*	*f*		
`#Lines Eccen List`	*n*	*a*	*b*	*c*	*d*

Polyline

Describes a polyline drawn by the Single Polyline tool,defined by its "n" endpoints.

`Object`	Polyline		
`Name`	Polyline *n*		
`# Points`	n		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*
`Pt# DX DY`	*n*	*e*	*f*

3DElement

Describes a three-dimensional object defined by "n" points. DENEBACAD must be set to Sculpt mode before you define any 3D object.

`Object`	3DElement			
`Name`	3DElement *n*			
`# Points`	4			
`Pt# X Y Z`	1	*a*	*b*	*c*
`Pt# DX DY DZ`	2	*d*	*e*	*f*
`Pt# DX DY DZ`	3	*g*	*h*	*i*
`Pt# DX DY DZ`	*n*	*j*	*k*	*l*
`#Face #Pts List`	*n*	*n1*	*n2*	*n3*

Simple Object descriptions

2DLibrary

Describes a 2D Library Object.

`Object`	Group		
`Name`	Object Name		
`ObjectLibrary`			
`HiddenBoolean`			
`Insertion Point`	*X*	*Y*	
`# Objects`	2		
`Pen Color (R G B)`	*r*	*g*	*b*
`Line Type (Thick Dash Eccen)`	1	1	0
`Arrow Heads (Begin End)`	0	0	
`Fill Color (R G B)`	0	0	0
`Fill Pattern`	0		
`Object`	DiagonalRect		
`Name`	DiagonalRect *n*		
`# Points`	2		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*
`Object`	DiagonalEllipse		
`Name`	DiagonalEllipse *n*		
`# Points`	2		
`Pt# X Y`	1	*a*	*b*
`Pt# DX DY`	2	*c*	*d*

3DLibrary

Describes a 3D Library Object.

`Object`	3DElement			
`# Points`	*n*			
`Pt# X Y Z`	1	*a*	*b*	*c*
`Pt# DX DY DZ`	2	*d*	*e*	*f*
`Pt# DX DY DZ`	3	*d*	*e*	*f*
`Pt# DX DY DZ`	*n*	*d*	*e*	*f*
`#Face #Pts List`	1	*1*	*3*	*2*
`#Face #Pts List`	2	*1*	*4*	*3*
`#Face #Pts List`	3	*1*	*2*	*3*
`#Face #Pts List`	4	*2*	*1*	*4*
`#Face #Pts List`	5	*2*	*3*	*4*
`#Face #Pts List`	*n*	*3*	*4*	*2*

Boolean Object descriptions

Boolean Object descriptions specify composite objects derived from the application of a Boolean operation to several objects.

Boolean object descriptions

Boolean

`Object`	Boolean
`Name`	*Boolean_Element_Name*
`Boolean Mode`	*Boolean_Mode_String**
`# Objects`	*n*
`Pen Color (R G B)`	*R G B*
`Line Type (Thick Dash Eccen)`	*T D E*
`Arrow Heads (Begin End)`	*B E*
`Fill Color (R G B)`	*R G B*
`Fill Pattern`	*FP*
`Object`	*Object_Definition_1*
`Object`	*Object_Definition_2*
`Object`	*Object_Definition_n*

* *Boolean_Mode_String* can be one of five values: None, Add, Substract, Intersect, or Hide

Grouped Object descriptions

Grouped Object descriptions specify composite objects derived from the grouping of simple objects. A grouped object can be one of the following types:

- Simple group
- Section group
- Library Object group
- Library Symbol group

Library Objects and Library Symbols contained in PTF files can be loaded into standard DENEBACAD documents and used in exactly the same way as Library Objects and Library Symbols that are contained in standard DENEBACAD documents.

PTF Libraries

When you add a Library Object or Library Symbol to a DENEBACAD document, the directory dialog box includes a pop-up menu you can use to select the type of library host file to use.

The directory dialog box also includes a “Show All Files” check-box.

Grouped Object descriptions

Group

`Object`	Group		
`Name`	*Group_Name*		
`# Objects`	*n*		
`Object`	*Object_Definition_1*		
`Object`	*Object_Definition_2*		
`Object`	*Object_Definition_n*		

Group: Library

`Object`	Group		
`Name`	Library 1		
`Object`	Library		
`Insertion Point`	-4.20 2.70		
`# Objects`	2		
`Pen Color (R G B)`	0	0	0
`Line Type (Thick Dash Eccen)`	1	1	0
`Arrow Heads (Begin End)`	0	0	
`Fill Color (R G B)`	0	0	0
`Fill Pattern`	0		
`Object`	DiagonalRect		
`Name`	DiagonalRect 1		
`# Points`	2		
`Pt# X Y`	1	-5.40	4.00
`Pt# DX DY`	2	1.80	-2.00
`Object`	DiagonalEllipse		
`Name`	DiagonalEllipse 1		
`# Points`	2		
`Pt# X Y`	1	-4.40	2.60
`Pt# DX DY`	2	1.40	-1.20

Grouped Object descriptions

Group: Symbol

`Object`	Group		
`Name`	Symbol 1		
`SymbolLibrary`			
`Insertion Point`	3.10	2.30	
`# Objects`	3		
`Object`	DiagonalEllipse		
`Name`	DiagonalEllipse 2		
`# Points`	2		
`Pt# X Y`	1	2.20	3.20
`Pt# DX DY`	2	1.80	-1.80
`Object`	Line		
`Name`	Line 1		
`# Points`	2		
`Pt# X Y`	1	3.10	4.10
`Pt# DX DY`	2	0.00	-3.60
`Object`	Line		
`Name`	Line 1		
`# Points`	2		
`Pt# X Y`	1	1.30	2.30
`Pt# DX DY`	2	3.60	0.00

Group: Section

`Object`	Group		
`Name`	Section 3		
`Section`			
`Insertion Point`	-4.30	-2.50	
`# Objects`	3		
`Object`	DiagonalEllipse		
`Name`	DiagonalEllipse 2		
`# Points`	2		
`Pt# X Y`	1	-5.20	-1.60
`Pt# DX DY`	2	1.80	-1.80
`Object`	Line		
`Name`	Line 1		
`# Points`	2		
`Pt# X Y`	1	-4.30	-0.70
`Pt# DX DY`	2	0.00	-3.60
`Object`	Line		
`Name`	Line 1		
`# Points`	2		
`Pt# X Y`	1	-6.10	-2.50
`Pt# DX DY`	2	3.60	0.00

Container descriptions

A Container is a non-printing object used to group unrelated objects for reporting purposes. Any object can be used as a Container. In the example below, a DiagonalRect is used as a Container.

Container

Object	DiagonalRect		
Name	*Object_Name_String*		
Container			
# Points	2		
Pt# X Y	1	*X*	*Y*
Pt# DX DY	2	*Dx*	*Dy*

Dimension Object descriptions

Dimension Object descriptions specify dimension objects that can be drawn with the Dimension tools in DENEBACAD.

Dimension object descriptions

ChainDimm

Describes a dimension drawn by the Chain Dimension tool.

Object	ChainDimm				
Name	ChainDimm 1				
Dimm Arrowhead	8				
Witness (Position Length)	1	2			
Dimm Text (Font Size Face Just)	3	9	0	1	
Dimm Distances (A B C D E)	1.6	3.2	1.6	1.6	1.6
Dimm Precision (Dec Places +Tol -Tol)	2	0.000	0.000		
# Points	4				
Pt# X Y	1	-7.80	3.20		
Pt# DX DY	2	2.40	0.00		
Pt# DX DY	3	2.00	0.00		
Pt# DX DY	4	2.20	0.00		
Dimm Line	-7.80	4.20	6.60	0.00	
Dimm Label# Dh Dv Text	1	0.83	0.16	2.40	
Dimm Label# Dh Dv Text	2	0.63	0.16	2.00	
Dimm Label# Dh Dv Text	3	0.73	0.16	2.20	

Dimension object descriptions

BaseLineDimm

Describes a dimension drawn by the Basline Dimension tool.

Object	BaseLineDimm				
Name	BaseLineDimm 1				
Dimm Arrowhead	8				
Witness (Position Length)	1	2			
Dimm Text (Font Size Face Just)	3	9	0	1	
Dimm Distances (A B C D E)	1.6	3.2	1.6	1.6	1.6
Dimm Precision (Dec Places +Tol -Tol)	2	0.000	0.000		
# Points	4				
Pt# X Y	1	-7.80	0.00		
Pt# DX DY	2	2.40	0.00		
Pt# DX DY	3	2.00	0.00		
Pt# DX DY	4	2.20	0.00		
Dimm Line	-7.80	0.60	6.60	0.00	
Dimm Label# Dh Dv Text	1	0.83	0.16	2.40	
Dimm Label# Dh Dv Text	2	1.83	0.16	4.40	
Dimm Label# Dh Dv Text	3	2.93	0.16	6.60	

LeaderDimm

Describes a dimension drawn by the Leader Dimension tool.

Object	LeaderDimm				
Name	LeaderDimm 1				
Dimm Arrowhead	8				
Dimm Text (Font Size Face Just)	3	9	0	1	
Dimm Distances (A B C D E)	1.6	0.0	0.0	0.0	0.0
Dimm Precision (Dec Places +Tol -Tol)	2	0.000	0.000		
# Points	2				
Pt# X Y	1	-7.80	-1.20		
Pt# DX DY	2	6.60	0.00		
Dimm Label# Dh Dv Text	1	2.93	0.16	6.60	

DoubleLeader Dimm

Describes a dimension drawn by the Constrained Dimension tool.

Object	DoubleLeaderDimm				
Name	DoubleLeaderDimm 1				
Dimm Arrowhead	8				
Dimm Text (Font Size Face Just)	3	9		0	1
Dimm Distances (A B C D E)	1.6	0.0	0.0	0.0	0.0
Dimm Precision (Dec Places +Tol -Tol)	2	0.000	0.000		
# Points	2				
Pt# X Y	1	-7.80	-2.20		
Pt# DX DY	2	6.60	0.00		
Dimm Label# Dh Dv Text	1	2.93	0.16	6.60	

Dimension object descriptions

AngleDimm

Describes a dimension drawn by the Angle Dimension tool. The angle measured is between two lines. The first two points define the first line. The second set of points define the second line.

`Object`	AngleDimm		
`Name`	AngleDimm 2		
`# Points`	4		
`Pt# X Y`	1	1.80	0.00
`Pt# DX DY`	2	4.80	0.00
`Pt# DX DY`	3	-2.00	2.80
`Pt# DX DY`	4	1.63	-0.96

PTF commands

The PTF commands described in this section are presented using the following scheme:

CommandName

Command description

Location	Where in the PTF file the command is allowed.
Switches	Command modifiers, if applicable.
Parameters	None (if the command requires no parameters), or *ParameterName_Type*
Values	Strings or data that can be entered as parameters (shown separated by commas).
Syntax	One or more examples of the command syntax in the following format: `CommandName` *Param_1 Param_2 Param_n*
Note	When applicable, any special behavior or additional information needed to correctly use the command.
Warning	When applicable, any special information that should be taken into account or that could lead to problems or unexpected results.

One-time commands

These commands should appear only once in any PTF file. If a One-time command is repeated, DENEBACAD exectutes only the first instance of the command in the PTF file.

Begin PTF

Indicates the beginning of the PTF file information.

Location	PTF Header
Parameters	None
Syntax	`Begin PTF`
Note	Any information preceding this command in the file is ignored

End PTF

Indicates the end of the PTF file information.

Location	PTF Footer
Parameters	None
Syntax	`End PTF`
Note	Any information following this command in the file is ignored

Distance Units

Specifies linear units used to dimension objects.

Location	PTF Header
Parameters	*Distance_Unit_String*
Values	DecInches, Meters
Syntax	`Distance Units` *Distance_Unit_String*
Note	All distance units will be inserted as signed decimals. U.S. System units are always decimal inches.

Angular Units

Specifies angular units used to measure angles. DecDegrees is the only value allowed for the *Angular_Unit_String* parameter.

Location	PTF Header
Parameters	*Angular_Unit_String*
Values	DecDegrees
Syntax	`Angular Units` *Angular_Unit_String*
Note	All Angular units will be inserted as signed decimal degrees.

Environmental commands

The Environmental commands can be inserted anywhere within the Body of the PTF file (see "The PTF Body" on page 404 for more information). An Environmental command affects the operation of all commands and tools initiated after its appearance.

Mode

Directs the following operations to either the Draft or Sculpt modes in DenebaCAD's drawing environments. See the User's Guide for more information on working in DENEBACAD modes. Draft is the default mode.

Parameters	*Mode_String*
Values	Draft, Sculpt
Syntax	`Mode` *Mode_String*
Notes	If omitted, PTF assumes that all operations will be executed within the current active drawing mode. In Sculpt mode all 2D object descriptions are extruded using the currently active extrusion method. In Draft mode DENEBACAD displays an error message if you define a 3D object.

View

Changes the environment to the view indicated in the parameter. Top is the default view.

Parameters	*View_String*
Values	Top, Bottom, Front, Back, Left, Right
Syntax	`View` *View_String*
Notes	If omitted, PTF assumes that all operations will be executed within the current active view.

General Attributes commands

The General Attributes commands can be inserted anywhere within the body of the PTF file (see "The PTF Body" on page 404 for more information). The requested setting affects the operation of all commands and tools initiated after its appearance.

Arrow Heads (Begin End)

Establishes the beginning and ending arrowheads to be applied to arrow lines.

Parameters	*Begin_Integer End_Integer*
Values	Positive non-zero integers

Syntax	`Arrow Heads` (Begin End) *Begin_Integer End_Integer*
Notes	*Begin_Integer* and *End_Integer* point to the relative position of the defined begin and end arrowheads in the arrowhead palette; 1 points to the first arrowhead at the top of the palette, 2 points to the second arrowhead in the palette, and so on.

VectorCoordSys

Establishes the coordinate definition standard for the DenebaCAD Drawing Vector. Vectors you define in other commands must be in the VectorCoordSys format you specified last.

Parameters	*Start_Point_String End_Point_String*
Values	*Start_Point_String* *XY:* Cartesian absolute coordinates *DA:* Polar absolute coordinates *End_Point_String* *XY:* Cartesian absolute coordinates *DA:* Polar absolute coordinates *dXdY:* Cartesian relative coordinates *dDdA:* Polar relative coordinates
Syntax	`VectorCoordSys` *Start_Point_String End_Point_String*
Notes	The command affects all coordinate input that occurs after its appearance.
Warning	Double-check input coordinate values. PTF will interpret an intended 25', 45 input as 25', 45' if the *Start_Point_String* is set to 'XY' and as 25' at 45° if set to 'DA'.

Layer

Specifies the current layer, which is the layer that will contain the next object created.

Parameters	*Layer_Number_Integer*
Values	Positive non-zero integer
Syntax	`Layer #` *Layer_Number_Integer*
Notes	If *Layer_Number_Integer* indicates a non-existent layer, a layer with that number is created.

Pen Color (R G B)

Defines a pen color based on RGB (Red, Green, and Blue) color values.

Parameters	*Red_Integer, Green_Integer, Blue_Integer*
Values	Positive integers from 0 to 100
Syntax	`Pen Color` (R G B) *Red_Integer Green_Integer Blue_Integer*
Notes	Any missing value is interpreted as zero

Line Type (Thick Dash Eccen)

Specifies the line type, using integer values indexed to the tabs in the Pen palette.

Parameters	*Thick_Integer, Dash_Integer, Eccentricity_Integer*
Values	Positive non-zero integers
Syntax	`Line Type` (Thick Dash Eccen) *Thick_Integer Dash_Integer Eccentricity_Integer*
Notes	*Thick_Integer* is indexed to the Pen weight tab; *Dash_Integer* is indexed to the Pen style tab; *Eccentricity_Integer* is indexed to the eccentricity options on the Pen weight tab. If a value exceeds the largest index in any list, an error is produced and the last active value is reinstated.

Fill Color (R G B)

Defines a fill color using RGB (Red, Green, Blue) color values.

Parameters	*Red_Integer, Green_Integer, Blue_Integer*
Values	Positive integers from 0 to 100
Syntax	`Fill Color` (R G B) *Red_Integer Green_Integer Blue_Integer*
Notes	A missing value is interpreted as zero

Fill Pattern

Specifies a Fill pattern, using values indexed to the Fill Pattern tab.

Parameters	*Fill_Integer*
Values	Positive Integer
Syntax	`Fill Pattern` *Fill_Integer*
Notes	A missing value is interpreted as zero.

Fill Hatch

Specifies a Fill Hatch, using values indexed to the Fill Hatch tab.

Parameters	*Fill_Integer*
Values	Positive Integer

Syntax	`Fill Hatch` *Fill_Integer*
Notes	A missing value is interpreted as zero.

Fill Material

Specifies a Surface material, using values indexed to the Surface material tab.

Parameters	*Fill_Integer*
Values	Positive Integer
Syntax	`Fill Material` *Fill_Integer*
Notes	A missing value is interpreted as zero.

Object Selection commands

These commands are the same as the commands in the Edit menu of DENEBACAD. Careful use of selection commands in PTF is important. The selection status of objects affects almost every other PTF command.

If you define a new object, all selected objects become deselected, and DENEBACAD selects the new object. There is no PTF command to deselect objects.

Select

Selects objects to later operate on them.

Switches	Last (selects the most recent objects created; the number of objects selected is user defined)
	All (selects all objects created up to the point where this command appears in the file)
	ByName (selects specifically named objects
Parameters	*Number_of_Objects_integer*
Values	Positive non-zero integers
Syntax	`Select` Last *Number_of_Objects_integer*
	`Select` All
	`Select` ByName *Name_String*

AddToSelection

Selects objects and adds them to a previous selection.

Switches	All (selects all objects created up to the point where this command appears in the file)
	ByName (selects specifically named objects)
Parameters	*Name_String*

Syntax `AddToSelection` All

`AddToSelection` ByName *Name_String*

Object Position commands

The Object Position commands are the same as the commands in the Position submenu in the DENEBACAD Object menu.

Note: A PTF file created from the DENEBACAD environment does not contain these commands, but contains the final description of objects created or modified by the commands.

Mirror

Flips the selected object(s) symmetrically with reference to a user-defined mirror line.

Parameters *MirrorLine_Vector*

Values Refer to the VectorCoordSys command

Syntax `Mirror` *MirrorLine_Vector*

MirrorACopy

Flips a copy of the selected object(s) symmetrically with reference to a user-defined mirror line.

Parameters *MirrorLine_Vector*

Values Refer to the VectorCoordSys command

Syntax `MirrorACopy` *MirrorLine_Vector*

Move

Moves the selected object(s) to a new position defined by a user-defined vector.

Parameters *Move_Vector*

Values Refer to the VectorCoordSys command

Syntax `Move` *Move_Vector*

MoveACopy

Moves a copy of the selected object(s) to a new position defined by a user-defined vector.

Parameters *Move_Vector*

Values Refer to the VectorCoordSys command

Syntax `MoveACopy` *Move_Vector*

InclusiveLArray

Takes the selected object(s) and distributes a user-defined number of copies along a user-defined vector. An off-plane distance may also be specified if Mode is set to Sculpt.

Parameters	*NCopies_Integer* *Direction_Vector* *OffPLane_Real* (optional; zero if ommited)
Values	*NCopies_Integer:* Positive non-zero integer *Direction_Vector:* Refer to the VectorCoord Sys command *OffPLane_Real*: Signed real number
Syntax	`InclusiveLArray` *NCopies_Integer Direction_Vector* [*OffPLane_Real*]

OffsetLArray

Takes the selected object(s) and places a user-defined number of copies along a user-defined vector separated by a distance taken from the length of that vector. An off-plane distance may also be specified if Mode is set to Sculpt.

Parameters	*NCopies_Integer* *Direction_Vector* *OffPLane_Real* (optional; zero if omitted)
Values	*NCopies_Integer:* Positive non-zero integer *Direction_Vector:* Refer to the VectorCoordSys command *OffPLane_Real:* Signed real number
Syntax	`OffsetLArray` *NCopies_Integer Direction_Vector OffPLane_Real*

Rotate

Rotates the selected object(s) a user-defined number of degrees around a user-defined centerpoint.

Parameters	*Center_Point_A_Real* (first coordinate of the centerpoint) *Center_Point_B_Real* (second coordinate of the centerpoint) *Angle_Angle*
Syntax	`Rotate` *Center_Point_A_Real Center_Point_B_Real Angle_Angle*

RotateACopy

Rotates a copy of the selected object(s) a user-defined number of degrees around a user-defined centerpoint.

Parameters	*Center_Point_A_Real* (first coordinate of the centerpoint) *Center_Point_B_Real* (second coordinate of the centerpoint) *Angle_Angle*

Syntax `RotateACopy` *Center_Point_A_Real Center_Point_B_Real Angle_Angle*

PolarArray

Takes the selected object(s) and places a user-defined number of copies around a user-defined center point separated by an user-defined angle. An off-plane distance may also be specified if Mode is set to Sculpt.

Parameters *NCopies_Integer*
Direction_Vector
Center_Point
Angle_Angle
OffPLane_Real (optional)

Values *NCopies_Integer:* Positive Non-zero Integer
Direction_Vector: Refer to the VectorCoordSys command
Center_Point: Coordinate pair
OffPLane_Real: Signed real number

Syntax `OffsetLArray` *NCopies_Integer Direction_Vector Center_Point Angle_Angle OffPLane_Real*

Scale

Scales the selected object(s) by a user-defined horizontal and vertical proportion.

Parameters *Scale_Center:* Coordinate pair
HScale_Real: Positive real
VScale_Real: Positive real

Syntax `Scale` *Scale_Center HScale_Real VScale_Real*

Object Combine commands

The Object Combine commands in PTF perform Boolean operations on selected objects. The commands are the same as those in the Combine submenu in the DENEBACAD Object menu.

Please refer to the DENEBACAD documentation for more information about any command.

Note: A PTF file created from the DENEBACAD environment does not contain Combine commands, but contains the final description of the composite objects created or modified by the commands.

The general syntax for these commands is:

Command_Name

Where *Command_Name* is one of the following:

```
Unite
Intersect
PunchFromBack
PunchFromBack&Trim
PunchFromFront
PunchFromFront&Trim
```

Miscellaneous Object commands

The Miscellaneous Object commands in PTF perform operations on selected objects. The commands are the same as those found in the Object menu in DENEBACAD. Please refer to the DENEBACAD documentation for particulars about any command.

Note: A PTF file created from the DENEBACAD environment does not contain these commands, but contains the description of objects created or modified by the commands.

The general syntax for these commands is:

Command_Name

Where *Command_Name* is one of the following:

```
Chain
Unchain
Group
Ungroup
```

Object Extrusion commands

The PTF Object Extrusion commands are the same as those found in the Extrusion submenu in DENEBACAD. Please refer to the DENEBACAD documentation for particulars about these commands.

Note: A PTF file created from the DENEBACAD environment does not contain these commands, but contains the final descriptions of objects created or modified by the commands.

3DAxis

Defines a 3D Spin axis to be used for extrusions with the Spin command.

Parameters	*View_String* *Center_Point_*
Values	*View_String:* Vertical | Lateral | Front *Coord1 Coord2*
Syntax	`3DAxis` *View_String Coord1 Coord2*

3DPlane

Defines a set of 3D planes to be used for extrusions with the Linear command.

Parameters	*View_String* *Distance1_Real* *Distance2_Real*
Values	*View_String:* Vertical | Lateral | Front *Distance1_Real:* Real *Distance2_Real:* Real
Syntax	`3DPlane` *View_String Distance1_Real Distance2_Real*

Angled3DPlane

Defines an inclined set of 3D planes to be used for further Linear Extrude commands.

Parameters	*View_String* *Distance1_Real* *Distance2_Real*
Values	*View_String:* Vertical | Lateral | Front *Distance1_Real:* Real *Distance2_Real:* Real
Syntax	`Angled3DPlane` *View_String Vector1 Vector2*

Solid Faces

Specifies which faces of an extruded object are to be generated.

Parameters	*NearCap_Boole* *FarCap_Boole* *Sides_Boole*
Values	True | False
Syntax	`Solid Faces` *NearCap_Boole FarCap_Boole Sides_Boole*

SpinOptions

Specifies the parameters to be applied by the next Spin command. Please refer to “Spin extrusion tab” on page 286 for information on Spin Options.

Parameters	*Nsteps_Integer* *StepAngle_Angle* *AxialStep_Real* *SpiralStep_Real*
Values	*Nsteps_Integer:* Positive non-zero integer *StepAngle_Angle:* Decimal Angle Value *AxialStep_Real:* Real *SpiralStep_Real:* Real

Syntax `SpinOptions` *Nsteps_Integer StepAngle_Angle AxialStep_Real SprialStep_Real*

SweepOptions

Specifies the parameters to be applied by the next Sweep command. Please refer "Sweep extrusion tab" on page 288 for information on Sweep Options.

Parameters *DirectionString_String*
HScale_Real
VScale_Real

Values *DirectionString_String:* Vertical | Perpendicular
HScale_Real: Positive real
VScale_Real: Positive real

Syntax `SweepOptions` *DirectionString_String Hscale_Real Vscale_Real*

SweepSection

Specifies the name of the section object to be used for the next Sweep command. Use this command before you execute the Extrude Sweep command.

Parameters *Section_String*

Syntax `SweepSection` *Section_String*

ExtrudeLinear

Extrudes the currently selected 2D object between the current set of 3D planes with the current Solid Faces options.

Parameters None

Syntax `ExtrudeLinear`

ExtrudeSpin

Spin extrudes the currently selected 2D object around the current 3D axis with the current Solid Faces options.

Parameters None

Syntax `Extrude Spin`

Extrude Sweep

Sweep extrudes the current Sweep Section along the currently selected 2D object with the current sweep options. DENEBACAD displays an error if you do not have a Sweep Section defined when you use this command.

Parameters None

Syntax `Extrude Sweep`

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.

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

Analysis Manager A palette that displays 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.

attributes Graphical modifiers such as color and line width, which can be applied to objects. *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.

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

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 reference point in a document at the Cartesian coordinates X=0, Y=0, Z=0. You can set Centerpoints and save Centerpoints so you can assign the X=0, Y=0, Z=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.

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.

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.

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 The last Centerpoint set by a user using the Set Position command designating the coordinates x=0, y=0, z=0 in a drawing.

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 Centerpoint	The Centerpoint that is in effect when you create a new document or choose the Default Centerpoint command.
default spin axis	The spin extrusion axis that is in effect when you create a new document or choose the Default 3D Axis command.
default extrusion planes	The extrusion planes 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.
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.
dimension object	An object containing dimensions created with a Dimension tool.
drawing vector	A vector represented by an 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.
extrude	The process of adding a third dimension to a 2D object using the Linear, Spin, or Sweep extrusion method.
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.
extrusion planes	A set of two planes that define the boundaries of linear extrusions. Some objects, such as walkthrough paths and light sources, are always drawn on the upper or lower 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.

first plane The plane you set first, regardless of position, when you set extrusion planes.

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.

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.

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 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.
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.
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.
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.

polyline An open shape drawn by setting endpoints of connected line segments, using the Single Polyline tool or Double Polyline tool.

Properties Manager A palette that displays information about a selected object. You can display an object's attributes, position, name, creation points and class in the Properties Manager, and use it to designate a selected object as a Library Object, Library Symbol, Container, or Sweep Section.

QuickTime A software component 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 walkthrough path of a rendering.

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

relative view A plane set at any position 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.

reshape mode A mode initiated by the Reshape command, or the use of the Point Selection tool, that allows you to reposition creation points or vertices 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.

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.

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

snap To limit the movement of the drawing vector, the pointer, or an object to a specific location in a drawing by using various constraints. DENEBACAD provides multiple methods of constraining: the Snaps pop-up menu, and the Snap to Grid, Snap to Objects, and Create Snap Objects commands.

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 (Mac OS) or the right mouse button (Windows). 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 extrusion axis A line around which a 2D object is rotated to create a 3D object using Spin extrusion method.

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

Status bar An area at the top of the screen that displays five pop-up menus from which you choose display options, Full View or Sectioned View, extrusion 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 in the Properties Manager for extrusion into three dimensions using 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. You can designate 2D objects as Symbols. 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 extrusion 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 — *See* extrusion planes

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.

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.

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